Reliability

可靠性 Confiabilidade

موثوقية Fiabilité Affidabilità

信頼性 Αξιοπιστία Zuverlässigkeit

Confiabilidad Niezawodność Надежность

USER'S GUIDE

Reliability

SYNTHESIS

Elements ®

User's Guide

BLOCKSIM®

Version 9

RENO®

Version 9

ReliaSoft®

ReliaSoft Publishing
A DIVISION OF RELIASOFT CORPORATION

Tucson São Paulo Warsaw Chennai Singapore

BlockSim/RENO 9 User's Guide
Part Identification: RPUGBRE9

ReliaSoft Corporation
Worldwide Headquarters
1450 South Eastside Loop
Tucson, Arizona 85710-6703, USA
Tel: 1.520.886.0410
Fax: 1.520.886.0399
Sales and Information: 1.888.886.0410 (Toll-free in the U.S. and Canada)
ReliaSoft@ReliaSoft.com
http://www.ReliaSoft.com

© 1992-2015 ReliaSoft Corporation, ALL RIGHTS RESERVED.

Notice of Rights and Limited Rights to Print this Document
If you are a licensed user of the software you are hereby granted the right to print this document in whole or in part, as needed for your exclusive use in conjunction with the use of the software. Except for the limited print rights outlined above, no part of this document may be reproduced or transmitted, in any form or by any means, for any purpose, without the express written permission of ReliaSoft Corporation, Tucson, AZ, USA.

Disclaimer
Information in this document is subject to change without notice and does not represent a commitment on the part of ReliaSoft Corporation.

Companies, names and data used herein are fictitious unless otherwise noted.

Use of the software and this document are subject to the terms and conditions set forth in the accompanying License Agreement.

This software and documentation were developed at private expense; no portion was developed with government funds.

Trademarks
ReliaSoft, Synthesis Platform, Weibull++, ALTA, DOE++, RGA, BlockSim, RENO, Lambda Predict, Xfmea, RCM++ and XFRACAS are trademarks of ReliaSoft Corporation.

Product names and services identified in this document are trademarks of their respective trademark holders, and are used for illustration purposes. Their use in no way conveys endorsement or other affiliation with ReliaSoft Corporation.

10 9 8

Contents

1 Introduction 1
1.1 Technical Support 2
1.1.1 When Requesting Support 2
1.2 ReliaSoft Consulting 3
1.3 Minimum System Requirements 3
1.3.1 A Note About Enterprise Database Servers 5
1.4 Installing Synthesis 5
1.5 License Registration & Activation 6
1.5.1 Activating Synthesis Applications 7
1.5.1.1 Basic Instructions with Direct Communication to the License Server 7
1.5.1.2 Problems Communicating with the Server 8
1.5.1.3 Multiple Ways to Activate 8
1.5.2 Deactivating Synthesis Applications 12
1.6 Internet Connectivity 13
1.7 Region and Language Settings 13
1.7.1 Viewing/Changing the Region and Language Settings for Your Computer 14
1.7.2 Setting the Default Language When You Install the Software 15
1.7.3 Defining Date and Time Formats 15
1.7.4 Changing the Decimal Symbol and the List Separator 16
1.7.5 Setting the Measurement System in the Page Setup 16
1.8 Introduction to the Synthesis Platform 16
1.9 BlockSim Features Overview 18
1.10 RENO Features Overview 22
1.11 What's New in the Synthesis Version? 24
2 Working in Synthesis 27
2.1 Synthesis Repositories 28
2.1.1 Standard Repositories 28
2.1.2 Enterprise Repositories 29
2.1.2.1 Creating a New Enterprise Repository 30
2.1.2.2 Connecting to an Existing Enterprise Repository 30
2.1.2.3 SQL Server Logins or Using Windows Impersonation 32
2.1.2.4 Enterprise Repository Connection Speed 35
2.1.2.5 Upgrading an Enterprise Repository from Version 8 35
2.2 Projects 36
2.2.1 Creating and Managing Projects 37

2.2.2 Project Properties ... 38
2.2.2.1 MPC's Project Properties ... 39
2.2.3 Project Owner ... 40
2.2.4 Private and Public Projects ... 41
2.2.5 Locked and Unlocked Projects ... 42
2.2.6 Check In and Check Out Projects ... 42
2.2.7 Manage Projects ... 44
2.2.8 Recycle Bin ... 45
2.3 Security Options ... 46
2.3.1 Applying Login Security ... 47
2.3.2 Planning Your Security Approach ... 47
2.3.3 Managing User Accounts ... 50
2.3.3.1 Importing Users from Active Directory ... 51
2.3.3.2 Creating Alternative Credentials ... 52
2.3.3.3 User Login and Contact Information ... 53
2.3.4 Managing Security Groups ... 54
2.3.4.1 Synthesis Permissions ... 55
2.3.5 Setting Item Permissions ... 58
2.3.6 Status Indicators ... 59
2.3.7 Reset "In Use" Flags ... 61
2.3.8 Prior Logins ... 61
2.4 Repository Settings ... 62
2.4.1 E-mail Settings ... 62
2.4.1.1 Notification Groups ... 63
2.4.2 Manage Units ... 64
2.4.3 Default Name Formats ... 65
2.4.4 Global Identifiers ... 66
2.4.5 Project Categories ... 67
2.4.6 XFRACAS Connection ... 68
2.5 Backups and Database Maintenance: Protecting Your Data ... 69
2.5.1 Restore Points ... 71
2.6 Import, Export and Data Conversion ... 72
2.6.1 Importing from an Existing Repository ... 72
2.6.2 Importing from Prior Version Files ... 73
2.6.2.1 Importing to a New Standard Database (File > Open Repository) ... 73
2.6.2.2 Importing to an Existing Database and Project (Import > Other file) ... 74
2.6.2.3 Converting from BlockSim 6 or 7 Files ... 74
2.6.2.4 Converting Xfmea/RCM++ 5 Databases ... 77
2.6.2.5 Converting MPC 3 Databases ... 79
2.6.3 Importing/Exporting Projects ... 82
2.6.4 Importing/Exporting Project Items or Resources ... 82
2.6.5 Importing from Excel or Delimited Text Files ... 84
2.6.5.1 Selecting Which Data Sheets Will Be Imported ... 84
2.6.5.2 Using the Control Panel to Map the Columns That Will Be Imported ... 85
2.6.5.3 Setting the Import Template Directory ... 85
2.6.5.4 Creating a New Import Template ... 86
2.6.5.5 Applying a Saved Import Template ... 86
2.6.5.6 Importing the Data ... 86
2.6.6 Using XML in BlockSim and Lambda Predict ... 87
2.7 Importing XFRACAS Data to Weibull++ or RGA ... 88
2.7.1 Transferring Data from XFRACAS to the Reliability Data Warehouse (RDW) ... 88

2.7.2 Transferring Data from the Reliability Data Warehouse (RDW) to a Standard Folio ... 89
2.7.3 RDW Interfaces: Configuration and Data Filtering Tools ... 91
2.7.4 Dashboard Viewer ... 92

3 Interface ... 93

3.1 Show, Tile, Dock and Pin Panels ... 94
3.2 Project Manager ... 97
3.2.1 Filter Properties Window ... 99
3.2.2 Current Project Explorer ... 100
3.3 My Portal ... 101
3.3.1 Message Window ... 103
3.3.2 Actions ... 103
3.3.2.1 Actions as Resources ... 103
3.3.2.2 Action Statuses ... 104
3.3.2.3 Action Properties ... 104
3.3.2.4 Approve, E-mail and Outlook Commands ... 105
3.3.2.5 Action Notifications Window ... 106
3.4 Data Sheets ... 107
3.5 Change Units ... 108
3.6 General Spreadsheets ... 109
3.7 Control Panels ... 109
3.7.1 Switching Between Pages ... 109
3.7.2 Hiding or Displaying a Control Panel ... 110
3.7.3 Control Panel Identifiers Page ... 111
3.7.4 Control Panel Notifications Page ... 111
3.7.5 Control Panel Utilized Resources Page ... 112
3.8 Results Window ... 112
3.9 System Panel and Analysis Panel ... 113
3.10 Select Existing Window ... 114
3.11 Check Spelling ... 116
3.12 Find Tool ... 117
3.12.1 Searching FMEA Records in Xfmea, RCM++ and RBI ... 118
3.13 Quick Parameter Estimator (QPE) ... 119
3.13.1 Wizard View ... 119
3.13.2 Expert View ... 120
3.13.3 ALTA Version of the QPE ... 121
3.14 History Logs ... 124
3.14.1 Project History Log ... 124
3.14.2 Record History Log ... 125
3.14.3 Creating and Managing Archives ... 125
3.15 Attachments ... 126
3.15.1 Attachments Window ... 126
3.15.2 Add/Edit Attachment Window ... 127
3.15.3 Attachment Locations ... 127
3.15.3.1 Project Attachments ... 127
3.15.3.2 Synthesis Resource Attachments ... 128
3.15.3.3 Hierarchy Attachments ... 128
3.15.3.4 Block Attachments ... 128

4 Ribbon 129
4.1 Quick Access Toolbar 129
4.2 Backstage View (File Tab) 130
4.2.1 ReliaSoft Online Pane 132
4.2.2 Help Center (File > Help) 133
4.2.3 License Manager 134
4.3 Home Tab 135
4.4 My Portal Tab 138
4.5 Project Tab 138
4.6 Insert Tab 141
4.7 View Tab 142
4.8 Help Tab 144
4.9 Active Tabs (Alphabetical) 145
4.9.1 Allocation Tab 145
4.9.2 Analysis Tab 145
4.9.3 Diagram Tab 146
4.9.4 Fault Tree Tab 148
4.9.5 Flowchart Tab 151
4.9.6 FMRA Tab 153
4.9.7 Format Tab 155
4.9.8 FRED Tab 156
4.9.9 Phase Tab 157
4.9.10 Plot Tab 159
4.9.11 Sheet Tab 160
4.9.12 Simulation Tab 162
4.9.13 Workbook Tab 163
5 Synthesis Resources 165
5.1 Types of Resources 165
5.1.1 Local and Global Resources 167
5.2 Creating and Selecting Resources 168
5.3 Universal Reliability Definitions (URDs) 170
5.4 Models 172
5.4.1 Publishing Models 174
5.5 Resource Manager 175
5.5.1 Choosing and Configuring Columns 177
5.5.2 Finding and Filtering Records 177
5.5.2.1 Auto Filter Row 178
5.5.2.2 Find Panel 178
5.5.2.3 Advanced Filters 178
5.5.2.4 Filter Editor 179
5.5.3 Removing Unused and Duplicate Resources 180
5.5.4 Select Resource Window 181
5.6 Trace Usage: Dependency Viewer 181
5.6.1 Dependency Viewer - Diagram View 183
6 Building and Customizing Diagrams 187
6.1 Techniques for Building Diagrams 188
6.1.1 Arranging Blocks 189

6.1.2 Connecting Blocks ... 190
6.1.3 Annotations ... 191
6.1.4 Keyboard and Mouse Combinations ... 192
6.2 Block Properties ... 194
6.3 Diagram Settings ... 197
6.3.1 Diagram Style Window ... 197
6.3.1.1 Background & Grid Page ... 198
6.3.1.2 Connector Handles and Indicators Page ... 198
6.3.1.3 Block Handles and General Indicators Page ... 199
6.3.1.4 Block Corner Indicator Pages ... 199
6.3.1.5 Connector Page ... 200
6.3.1.6 General Page ... 201
6.3.1.7 Description Page ... 202
6.3.1.8 Image Page ... 202
6.3.2 Diagram Skins ... 203
6.3.3 Block Style Window ... 204
6.3.3.1 Block Tab ... 204
6.3.3.2 Description Tab ... 205
6.3.3.3 Image Tab ... 205
6.3.4 Connector Style Window ... 205

7 Building RBDs and Fault Trees ... 207

7.1 Reliability Block Diagrams ... 208
7.1.1 RBD Block Types ... 208
7.1.2 RBD Configurations and Constraints ... 210
7.1.3 Standard Blocks and Basic Configurations ... 211
7.1.4 State Change Triggers ... 216
7.1.5 Node Blocks and k-out-of-n Configurations ... 217
7.1.6 Subdiagrams ... 218
7.1.7 Standby Containers ... 220
7.1.7.1 Contained Standby Blocks ... 222
7.1.7.2 Switches ... 224
7.1.7.3 Standby Without Containers ... 225
7.1.8 Load Sharing Containers ... 226
7.1.8.1 Contained Load Sharing Blocks ... 227
7.2 Fault Trees ... 228
7.2.1 Fault Tree Block Types ... 230
7.2.2 Fault Tree Analysis Constraints ... 232
7.2.3 Fault Tree Gates ... 232
7.2.3.1 AND Gates ... 232
7.2.3.2 OR Gates ... 233
7.2.3.3 NOT Gates ... 233
7.2.3.4 NAND Gates ... 234
7.2.3.5 NOR Gates ... 235
7.2.3.6 Voting Gates ... 235
7.2.3.7 Inhibit Gates ... 236
7.2.3.8 Standby Gates ... 237
7.2.3.9 Load Sharing Gates ... 238
7.2.3.10 Priority AND (PAND) Gates ... 239
7.2.3.11 Sequence Enforcing (SEQ) Gates ... 240
7.2.4 Fault Tree Events ... 242
7.3 Mirroring (Using Blocks in Multiple Locations) ... 243

7.4 Block Details ... 246
7.5 Item Properties Table ... 246
7.6 Quick Calculation Pad (QCP) ... 249
7.7 Publishing Fitted and Analytical Models ... 251
7.8 Creating Diagrams from Synthesis Data ... 252
7.8.1 Building Diagrams from Xfmea/RCM++/RBI ... 252
7.8.2 Building Diagrams from Lambda Predict ... 258
7.8.3 Building Effect Fault Trees from Synthesis ... 261

8 BlockSim Analytical Diagrams ... 263

8.1 Analytical Diagram Control Panel ... 264
8.1.1 Control Panel Analytical Page ... 264
8.1.2 Control Panel Analysis Settings Page ... 266
8.1.3 Distribution Estimator ... 266
8.2 Equation Viewer ... 267
8.3 Cut Sets ... 268
8.4 Analytical Quick Calculation Pad ... 270
8.5 Analytical Plots ... 272
8.5.1 Setting Confidence Bounds ... 274

9 BlockSim Simulation Diagrams ... 275

9.1 Available Analyses for Simulation Diagrams ... 276
9.1.1 Throughput Analysis ... 277
9.2 Maintenance Groups ... 279
9.3 Universal Reliability Definitions (URDs) in Simulation Diagrams ... 281
9.4 Tasks ... 281
9.4.1 Corrective Tasks ... 284
9.4.2 Preventive Tasks ... 286
9.4.3 Inspection Tasks ... 288
9.4.4 On Condition Tasks ... 289
9.4.5 Task Scheduling ... 290
9.4.6 Multiple Tasks on the Same Block ... 292
9.5 Crews ... 292
9.6 Spare Part Pools ... 294
9.6.1 Spare Part Pools - Simulation Assumptions ... 296
9.7 Task Packages ... 297
9.8 Simulation Diagram Control Panel ... 298
9.8.1 Control Panel Simulation Page ... 299
9.8.2 Control Panel Analysis Settings Page ... 301
9.9 Maintainability/Availability Simulation Window ... 302
9.9.1 Simulation Status Display ... 305
9.9.2 Repeatability of Simulation Results ... 307
9.9.3 Simulation Batch Mode ... 307
9.10 Simulation Results Explorer ... 308
9.10.1 Ribbon Commands ... 309
9.10.2 Set Report Colors Window ... 310
9.10.3 General Summary Results ... 311
9.10.4 System Results ... 311
9.10.4.1 System Overview ... 311

9.10.4.2 System Point Results ... 313
9.10.4.3 System Costs ... 314
9.10.4.4 System Throughput ... 315
9.10.5 Block Results ... 315
9.10.5.1 Block Summary ... 315
9.10.5.2 Block Costs ... 316
9.10.5.3 Block Throughput ... 317
9.10.5.4 Block Details ... 318
9.10.5.5 Last Run Details ... 321
9.10.6 Crew Results ... 321
9.10.6.1 Crew Summary ... 321
9.10.6.2 Crew Details ... 322
9.10.7 Spare Part Pools Results ... 323
9.10.7.1 Spare Part Pool Summary ... 323
9.10.7.2 Spare Part Pool Details ... 323
9.10.8 Simulation Summary ... 324
9.10.9 Relevant Summaries ... 325
9.11 Simulation Quick Calculation Pad ... 326
9.12 Simulation Plots ... 328
9.13 Dashboard Viewer ... 333

10 BlockSim Phase Diagrams ... 335

10.1 Introduction to Phase Diagrams ... 336
10.2 Phase Diagram Control Panel ... 337
10.2.1 Control Panel Simulation Page ... 338
10.2.2 Control Panel Analysis Settings Page ... 339
10.3 Operational Phase Blocks ... 340
10.4 Maintenance Phase Blocks ... 342
10.4.1 Maintenance Phases and Maintenance Tasks in Diagrams ... 343
10.4.2 Maintenance Templates ... 344
10.5 Subdiagram Phase Blocks ... 345
10.6 Node Blocks and Stop Blocks ... 346
10.7 Phase Diagram Simulation Assumptions ... 347
10.8 Results and Plots ... 349
10.9 Throughput Analysis - Phase Diagrams ... 351

11 RENO Flowcharts ... 353

11.1 Using Time Units in RENO ... 354
11.2 Flowchart Block Types ... 355
11.2.1 Flowchart Standard Blocks ... 357
11.2.2 Result Storage Blocks ... 358
11.2.3 Conditional Formatting ... 359
11.2.4 Conditional Blocks ... 362
11.2.5 Binary Nodes ... 364
11.2.6 Logic Gates ... 366
11.2.7 Branch Gates ... 369
11.2.8 Summing Gates ... 372
11.2.9 Flag Marker and Go-to-Flag Blocks ... 373
11.2.10 Stop Flags ... 375
11.2.11 Counter Blocks ... 375

11.2.12 Reset Blocks 377
11.2.13 UI Blocks 377
11.2.14 Subchart Blocks 379
11.3 RENO Resources 380
11.3.1 Variables 381
11.3.1.1 Using Variables to Store Output Values 382
11.3.2 RENO Functions and Static Functions 383
11.3.3 RENO Simulation Definitions 386
11.3.4 RENO Tables 389
11.4 Function Selector and Equation Editor 390
11.4.1 Function Selector 390
11.4.2 Equation Editor 391
11.5 RENO Standard and Special Operators 391
11.6 Flowchart Configurations and Execution 393
11.6.1 Example 1: Blocks with Multiple Outgoing Paths 393
11.6.2 Example 2: Blocks that Require a Certain Number of Executed Incoming Paths 395
11.6.3 Example 3: Flowchart with Multiple Starting Paths 396
11.7 Debugging a Flowchart 396
11.8 Flowchart Control Panel 397
11.9 Flowchart Simulation Window 399
11.9.1 RENO Simulation Status Display 402
11.10 Simulation Results and Plots 402
11.10.1 Flowchart Plots 403

12 Plots 405

12.1 Plot Sheet Control Panel 405
12.2 Show/Hide Plot Items Window 407
12.3 Overlay Plots 407
12.4 Tips on Working with Plots 409

13 Plot Setup 411

13.1 Plot Titles Page 412
13.2 Axis Titles/Labels Page 413
13.3 Plot Labels Page 414
13.4 Legend Page 415
13.5 Canvas Page 416
13.6 Grid Page 418
13.7 Bars Page 419
13.8 Slices Page 419
13.9 Plot Items Page 420
13.10 Offsets Page 421
13.11 Plot Defaults Window 422
13.11.1 Plot Defaults: Titles Page 423
13.11.2 Plot Defaults: Plot Item Display Page 424

14 ReliaSoft Draw 425

14.1 Linking RS Draw 425
14.2 RS Draw Menu Bar 426
14.2.1 File Menu 426

14.2.2 Edit Menu ... 426
14.2.2.1 Preferences Window ... 427
14.2.3 Canvas Menu ... 427
14.2.4 Shape Menu ... 427
14.2.5 Help Menu ... 428
14.3 RS Draw Toolbars ... 428
14.3.1 Standard Toolbar ... 428
14.3.2 Edit Toolbar ... 428
14.3.3 Zoom Toolbar ... 428
14.3.4 Draw Toolbar ... 429
14.3.5 Customizing Bars in the RS Draw Interface ... 430
14.3.5.1 Moving Toolbars ... 430
14.3.5.2 Hiding and Showing Toolbars ... 430
14.4 Additional Tools ... 430
14.5 Object Properties — The Tool Options Area ... 431
14.5.1 Lines Properties ... 431
14.5.2 Fill Properties ... 431
14.5.3 Points Properties ... 431
14.5.4 Angles Properties ... 432
14.5.5 Text Properties ... 432
14.5.6 Color Properties ... 432
14.6 Working with Objects: Examples ... 433
14.6.1 Drawing a Line ... 433
14.6.2 Editing an Object ... 433
14.6.2.1 Example: Editing a Line ... 433
14.6.2.2 Example: Resizing a Rectangle ... 433
14.7 Saving an Image ... 434

15 System Improvements ... 435

15.1 Allocation Analysis ... 435
15.1.1 About Reliability Importance ... 439
15.2 Optimum Replacement Window ... 440

16 Failure Modes and Reliability Analysis (FMRA) ... 443

16.1 Viewing the FMRA ... 443
16.2 Working with Records in the FMRA ... 445
16.3 Setting FMRA Record Properties ... 447
16.3.1 Setting Reliability Policy Types ... 447
16.3.2 Using Xfmea/RCM++/RBI Rating Scales in the FMRA ... 450
16.3.3 Batch Update Reliability Policy Window ... 451
16.3.4 Add to Mirror Group Window (FMRA) ... 452
16.4 FMRA Diagrams and Reliability-Wise Configurations ... 453
16.5 Performing FMRA Calculations ... 455
16.5.1 Calculation Options Window (Xfmea/RCM++/RBI) ... 459
16.5.2 Changing Constant Probability Models to Distributions ... 459

17 Simulation Worksheets ... 461

17.1 Create the Variables and the Diagram in BlockSim/RENO ... 462
17.2 Define the Factor Settings ... 463
17.3 Define the Simulation Worksheet Columns in BlockSim/RENO ... 465

17.4 Simulate the Worksheet in BlockSim/RENO 466
17.5 Analyze the Results in DOE++ 467
17.6 Verify the Results in BlockSim/RENO 469
18 Analysis Workbooks 471
18.1 Analysis Workbook Wizard/Word Report Template Wizard 472
18.2 Associated Data Sources 472
18.3 Saved Workbook/Report Templates 474
18.4 Analysis Workbook Control Panel 475
18.5 Function Wizard 477
18.5.1 Data Entry Tips for Functions 479
18.5.2 Using Functions to Return Confidence Bounds 482
18.6 Chart Wizard and Designer 483
18.7 Tools for Working with Spreadsheets 484
18.7.1 Spreadsheet Components 485
18.7.2 Referencing Cells in the Spreadsheet 485
18.7.3 Defined Names 486
18.7.4 Format Cells - Number Formats 488
18.7.5 Sheet Options Window 489
18.7.6 Calculation Options 490
18.7.7 Protecting Cells 492
18.7.8 Sort 493
18.7.9 Exporting the Spreadsheet 495
18.8 Analysis Workbook Example 495
19 Word Report Templates 499
19.1 Word Report Template Control Panel 500
19.2 Plot Wizard 501
19.3 Tools for Working with Word Report Templates 502
19.3.1 Formatting Paragraphs 503
19.3.2 Find and Replace Text or Special Characters 504
19.3.3 Inserting Pictures and Symbols 505
19.3.4 Inserting Bookmarks and Hyperlinks 506
19.3.5 Inserting Tables and Setting Properties 507
19.4 Word Report Template Ribbon 509
19.4.1 Home Tab 510
19.4.2 Document Tab 511
19.4.3 Insert Tab 513
19.4.4 View Tab 513
19.4.5 Table Tab 514
19.5 Word Report Example 515
20 FRED Reports 519
20.1 Analytical FRED Reports 521
20.2 Simulation FRED Reports 522
21 Printing 525
21.1 Page Setup 525
21.1.1 Page Setup: Page Tab 526
21.1.2 Page Setup: Margins Page 526

21.1.3 Page Setup: Header/Footer Page 527
21.1.4 Header/Footer Format Codes 528
21.2 Print Preview 529
21.3 Print Layout 529
21.4 Print Window 530
22 Setup Windows 533
22.1 Synthesis Setup 533
22.2 Application Setup 535
22.2.1 Application Setup: User Page 536
22.2.2 Application Setup: Calculations Page 537
22.2.3 Application Setup: Plot Page 538
22.2.4 Application Setup: Display Page 539
22.2.5 Application Setup: Other Page 540
22.2.6 Application Setup: Reset Settings Page 541
22.2.7 Application Setup: Conversion Page 542
23 Container Algorithm Setup 543
23.1 Algorithm Setup Window 543
23.1.1 Container Integration Parameters Wizard 544

Introduction 1

The capabilities of ReliaSoft's BlockSim and RENO software tools have been integrated into a single user interface with the ability to save both types of analyses together in the same project. This User's Guide provides detailed documentation for both products. The features that are available on your computer will depend on the license that you have purchased from ReliaSoft.

ReliaSoft's BlockSim software provides a comprehensive and flexible platform to model systems and processes using reliability block diagrams (RBDs) or fault tree analysis. An extensive array of RBD configurations and fault tree gates and events are supported, including advanced capabilities to model complex configurations, load sharing, standby redundancy, phases and duty cycles. Using the exact system reliability function and/or discrete event simulation, BlockSim supports a wide variety of analyses for both repairable and non-repairable systems that will be of use to both product designers and asset managers. This includes reliability, maintainability, availability, optimum reliability allocation, throughput, maintenance planning, life cycle costs and other analyses.

RENO is a visual stochastic event simulator -- a powerful and flexible platform for visualizing and dynamically simulating nearly any kind of physical, financial or organizational system. Using the familiar flowcharting concept, you can intuitively build systems and/or scenarios and then run simulations with RENO. In addition to being a flexible visual modeler, RENO can also be described as a "visual spreadsheet" — a way to work with data and equations graphically, clearly showing the connections. Simulation confers the added benefit of being able to evaluate the evolution of systems and scenarios over time.

Both products have also been integrated into the ground-breaking new Synthesis Platform, which offers reliability practitioners the best of both worlds: analysis tools that are optimized to fully meet the individual user's needs for a particular set of reliability engineering methodologies, plus integration into a shared repository (database) that is accessible by multiple users throughout the organization and facilitates integration between all Synthesis applications. This powerful new platform makes it feasible to more effectively leverage and integrate the information from different types of reliability activities throughout the organization.

This chapter provides a wealth of information to help you learn how to use the software to its maximum potential. Before starting we recommend reading the following sections:

- Technical Support - Section 1.1 (p. 2)
- ReliaSoft Consulting - Section 1.2 (p. 3)
- Minimum System Requirements - Section 1.3 (p. 3)
- Installing Synthesis - Section 1.4 (p. 5)
- License Registration & Activation - Section 1.5 (p. 6)
- Internet Connectivity - Section 1.6 (p. 13)
- Region and Language Settings - Section 1.7 (p. 13)

- Product Features
  - Introduction to the Synthesis Platform - Section 1.8 (p. 16)
  - BlockSim Features Overview - Section 1.9 (p. 18)
  - RENO Features Overview - Section 1.10 (p. 22)
  - What's New in the Synthesis Version? - Section 1.11 (p. 24)

1.1 Technical Support

ReliaSoft's reliability software products are renowned for their ease of use and unparalleled after sale support. For standard shrink-wrap software, we provide free technical support for software-related issues via an established (and growing) network of regional offices and partners/distributors throughout the world.
(See http://Support.ReliaSoft.com for details and exclusions.) You can request assistance directly from within the software by choosing **File > Help > E-mail Support**. This option creates an e-mail message that is pre-populated with information about your license and operating system, which the technical support representative will need for troubleshooting the issue.

You can also contact us directly via phone, fax, e-mail, mail or live chat. To find the office that serves your region, choose **File > Help > Contact Global Support Center** or visit http://www.ReliaSoft.com/help.htm. Note. Alternatively, and regardless of your region, you can always request assistance from ReliaSoft's Corporate Headquarters in Tucson, Arizona. For details, choose **File > Help > Contact ReliaSoft USA**.

1.1.1 When Requesting Support

When you contact ReliaSoft to request technical support, please be prepared to provide the following information:

- Your phone number and e-mail address.
- The product name and the Compile Release Version number of your application.[1]
- Your product license key.
  - To determine the license for the application on your computer, you can choose **File > Application Setup** and look on the User page of the Application Setup window.
- The operating system (e.g., Windows 7), RAM and hard disk space on your computer.
- Describe what you were doing when the problem occurred and exactly what happened. Please include the specific wording of any message(s) that appeared.

Note: ReliaSoft's technical support representatives are not reliability consultants, and their assistance is limited to technical issues that you may encounter with the software tools. To get assistance with your analysis from a subject matter expert, please contact ReliaSoft Consulting Services. ReliaSoft also offers a comprehensive selection of training courses that cover both the underlying principles and theory, as well as the applicable software tools. For details, visit http://Seminars.ReliaSoft.com.

[1] To determine the Compile Release Version (CRV) of the application on your computer, choose **File > Help**. The CRV is displayed in the **About** area. Note that the CRV will also indicate whether you are running the 32-Bit version or the 64-Bit version of the software.

1.2 ReliaSoft Consulting

If your organization does not have sufficient time, expertise or objectivity in-house to accomplish specific reliability goals, turning to ReliaSoft's expert reliability consultants can prove to be the most effective and economical solution. Whether you need a quick statistical analysis, a complete assessment of your reliability program plan or something in between, ReliaSoft Consulting Services (RCS) is ready to help.

- Our reliability consulting services team has combined expertise in almost **all areas of reliability and quality engineering** with experience that spans **a broad spectrum of product types**, from micro-electronics and appliances to advanced weapons systems and off-shore oil well drilling equipment.
- Unlike engaging a consultant who works independently, RCS consultants have **direct access to all of ReliaSoft's global resources, expertise and contacts**.
- Our **team-based approach to consulting**, combined with ReliaSoft's global reliability engineering organization, allows us to provide you with reliability consultants who understand your culture and speak your language while ensuring that the appropriate reliability expertise can be applied to each and every project.
- RCS is structured to **accommodate requests of any size or complexity**, from short telephone consultations to multiple experts at a client's site for an extended time period.

Please visit http://Consulting.ReliaSoft.com for a published list of commonly requested services, answers to frequently asked questions, and other useful information about this service.

1.3 Minimum System Requirements

This section describes the minimum system requirements for all Synthesis applications.

ReliaSoft provides a tool that can help you determine whether you should install the 32-bit or 64-bit version of the Synthesis Platform. The tool will also tell you whether your computer and Internet connection meet the requirements for running and activating the Synthesis Platform. To download the tool, refer to Technical Support Document RS40025.

System Hardware Requirements	
CPU Speed	1.5 GHz or higher
Processor	Intel Pentium or higher
Memory/RAM	2 GB or higher (32-bit) 8 GB or higher (64-bit)
Display Properties	24-bit color depth
Screen Resolution	1024 x 768 or higher at normal size (96 dpi)
Swap Space	Determined by the operating system, 500 MB or higher
Disk Space	1 GB for installing the Platform and Synthesis applications
Video/Graphics Adapter	64 MB RAM or higher
Internet Access	Internet access is required for the License Manager and optional for using the Internet-enabled features in the software interface. *See Section 1.6 on page 13.*

<table>
<tr><th colspan="2">Operating Systems Supported</th></tr>
<tr><td rowspan="4">Microsoft Windows®</td><td>XP - 32-bit (Service Pack 3 or higher). The 64-bit version of Windows XP is not supported.</td></tr>
<tr><td>Vista - 32-bit and 64-bit (Service Pack 1 or higher)</td></tr>
<tr><td>Windows 7 - 32-bit and 64-bit</td></tr>
<tr><td>Windows 8 - 32-bit and 64-bit</td></tr>
<tr><td rowspan="4">Microsoft Windows Server® (Requires a license that specifically allows server installations/deployments.)</td><td>2003 R2</td></tr>
<tr><td>2008 - 32-bit and 64-bit (Service Pack 2 or higher)</td></tr>
<tr><td>2008 R2</td></tr>
<tr><td>SBS (Small Business Server) 2011</td></tr>
<tr><th colspan="2">Microsoft Office® Integration</th></tr>
<tr><td colspan="2">Microsoft Office 2000 or higher (Excel®, Word® and Outlook®) are required for automated exports, report generation and e-mail/calendar integration. ReliaSoft recommends Microsoft Office 2010.</td></tr>
<tr><th colspan="2">IMPORTANT NOTE: 32-bit and 64-bit Versions</th></tr>
<tr><td colspan="2">As shown in the following table, you can choose which version of Synthesis to install if you have a 64-bit version of Windows and any version of Microsoft Office except Office 2010 (32-bit). When the installation program gives you a choice, the decision should be based on the capabilities of your computer (processor and RAM) and how you plan to use the software.

<table>
<tr><th colspan="2" rowspan="2"></th><th colspan="2">Microsoft Office Version</th></tr>
<tr><th>32-Bit</th><th>64-Bit</th></tr>
<tr><td rowspan="2">Windows Version</td><td>32-Bit</td><td>32</td><td></td></tr>
<tr><td>64-Bit</td><td>32 or 64†*</td><td>32 or 64*</td></tr>
</table>

[See our 32-bit vs. 64-bit FAQ at http://www.ReliaSoft.com/synthesis/installation.htm for more details.]

† If you have Microsoft Office 2010 (32-bit), you must install the 32-bit version of Synthesis.
* If you plan to use the optional Synthesis API (Application Programming Interface) to call Synthesis functions from a Microsoft Office application (e.g., use VBA in Excel to call the Weibull++ calculation engines), you must have the same version of Office and Synthesis (i.e., 32-32 or 64-64).</td></tr>
<tr><th colspan="2">Enterprise Database Platforms</th></tr>
<tr><td>Microsoft SQL Server®</td><td>2005, 2008 and 2012
32-bit and 64-bit versions</td></tr>
<tr><td>Microsoft SQL Server Express Edition</td><td>2005, 2008 and 2012
32-bit and 64-bit versions</td></tr>
<tr><td>Oracle®</td><td>10g and 11g
32-bit and 64-bit versions</td></tr>
</table>

Oracle Express	11g 32-bit and 64-bit versions

1.3.1 A Note About Enterprise Database Servers

Although the enterprise database platform (Microsoft SQL Server or Oracle) could be installed on the same computer where the Synthesis applications reside, most organizations will choose to set up a separate server to host the database. Both the 32-bit and 64-bit versions of the Synthesis applications will work with either the 32-bit or 64-bit versions of a back-end database/server. The minimum hardware/software requirements for each server hosting the database should be obtained directly from the selected database vendor (i.e., Microsoft or Oracle). In general, ReliaSoft recommends a minimum CPU speed of 2 GHz and 4 GB of RAM.

Tip: If you're considering the option to use the free "Express" edition of Microsoft SQL Server, the http://www.ReliaSoft.com/synthesis/sql_express.htm file provides some information that may help you to get started.

1.4 Installing Synthesis

To install Synthesis, open the installation program (e.g., "Synthesis9.exe") and follow the prompts. This single *.exe will install all Synthesis desktop applications and then you can choose which application(s) to activate on your computer. Please note the following:

- Before running the installation program, make sure your computer meets the **minimum system requirements** for Synthesis. *See Section 1.3 on page 3*.
- To perform the installation, you must be logged in with a user account that has **administrative rights**.
- It is strongly recommended that you **close all other applications** during the installation.
- After the installation, all Synthesis desktop applications will be located in the same application directory and available for activation. *See Section 1.5 on page 6*. Depending on your preferences, the applications that you choose to activate can be accessible via the Windows Start menu, via shortcut icons on the desktop and from the Synthesis Launcher.

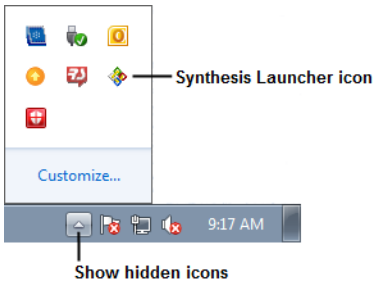
Deactivation and/or Uninstall

If you wish to stop using some or all of the Synthesis applications on this computer, you have two options:

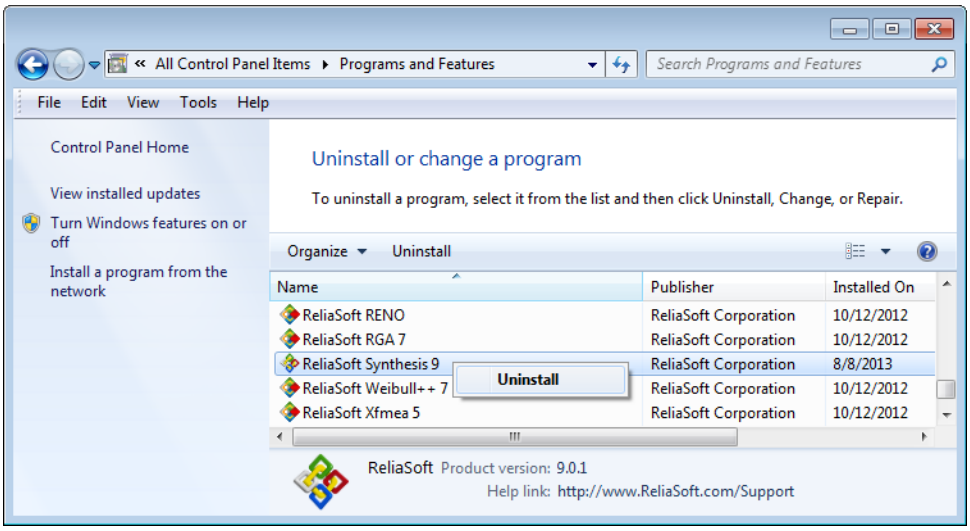
- **Deactivate Without Uninstall**. This will free up the specified activation(s) for your product license key so you can activate on another computer or for another user. However, you can continue to use any other Synthesis applications that remain activated on this computer. If you decide to start using the deactivated application(s) again, you will not need to reinstall (as long as the new activation would not exceed the limits of your license key). Simply launch the application and follow the prompts to activate again. *For details see Section 1.5.2 on page 12*.
- **Full Uninstall**. This will completely uninstall all Synthesis desktop applications from your computer. If you have a network connection to ReliaSoft's License Server, you will be able to deactivate all applications at the same time. If you are not able to complete the deactivations during uninstall, you will need to contact ReliaSoft

Technical Support for assistance to make the license available for later activation with another computer/user. To perform the uninstall, do the following:

1. Make sure you do not have any Synthesis applications currently open.
2. Make sure the Synthesis Launcher is not currently running. To check this, click **Show hidden icons** in the Windows notification area (i.e., the "system tray" located next to the clock in the Windows taskbar) and look to see if the Synthesis Launcher icon appears. If it does, click the icon and choose **Exit Synthesis Launcher**.



3. Open the Windows Control Panel and click **Programs and Features** (or use the search box in the Windows Start menu to find and select **Add or Remove Programs**).
4. In the Uninstall or change a program window, double-click **ReliaSoft Synthesis 9** and follow the prompts.



1.5 License Registration & Activation

The first time you run each Synthesis application, you will be prompted to register and activate the product on your computer. The License Manager wizard will guide you through the steps to provide your registration details and product license key. If you have an active Internet connection, you can complete the activation automatically via the wizard. If you do not currently have an active Internet connection, you can choose to implement the free 7-day trial or you can request an activation code from ReliaSoft and enter it manually when prompted by the wizard. *For details, see Section 1.5.1 on page 7.*

When the activation process is complete, you will receive a confirmation e-mail that includes your ReliaSoft ID and password. Please store this information in a safe place. It may be required if you later need to update your registration details or manage the products that are currently activated for your computer. If you are unable to locate this

information when you attempt to use the License Manager, you can request to have the ID and password sent to the e-mail address on file for your registration.

If you do not plan to continue to use a Synthesis application on a particular computer, it is very important to deactivate it. ReliaSoft's License Server will prohibit future activations that would exceed the limits of your license key. *For details, see Section 1.5.2 on page 12*.

1.5.1 Activating Synthesis Applications

Activation is a required process that associates your ReliaSoft products with the computer(s) on which they're installed. The process helps you maintain better control of your software assets (through activation and deactivation on different computers that you may use over time). It also allows you to recover lost keys and licenses, prevents unauthorized use and copying of the software, and protects you against the risks of counterfeit software. This is usually a quick and simple automated process done through our secure server at **https://Validate.ReliaSoft.org** (ReliaSoft License Server).[2]

If the computer has an active Internet connection and your firewall allows the software to access the ReliaSoft License Server, only a few simple steps are required. If you are unable to connect to the server, several alternative options are available. This document contains the following information:

- Basic instructions when the activation wizard can communicate directly with the server - Section 1.5.1.1 (p. 7)
- General tips to address problems communicating with the server - Section 1.5.1.2 (p. 8)
- Other ways to activate the application if the wizard cannot communicate directly with the server - Section 1.5.1.3 (p. 8)

1.5.1.1 Basic Instructions with Direct Communication to the License Server

1. The first time you launch an application, the activation wizard will appear. On the first page, select whether you want to activate a license or start with the free expiring demo.

   Note that if you choose to start with the demo first, it will be easy to upgrade to the full license when you're ready — no further downloads or installs required! The next time you launch the application after the demo expires, you will be prompted to repeat this process so you can enter the license key that you received from ReliaSoft.

2. On the next page, enter a valid e-mail address to serve as your **ReliaSoft ID**. This is the e-mail address where you will receive confirmation e-mails from the ReliaSoft License Server. It will also identify all of the products registered to you.
   - **One-Time Registration**: If this is the first time that you have registered a Synthesis application on this computer with this e-mail address, the next page will request your name and basic contact information.
3. If you are activating a product license key provided by ReliaSoft, you will be prompted to enter it on the next page. *Tip: If you are able to copy the key from a product delivery e-mail, the Paste icon saves time by automatically entering each section into the appropriate input box.*
   - **For Unlimited User Licenses**: If you have an Unlimited User License (e.g., a site-wide, division-wide, corporate-wide or corporate partner license), the wizard must confirm that your ReliaSoft ID matches one of the e-mail domains that have been authorized for the license (e.g., a corporate-wide license for ReliaSoft might be configured to allow access by any user with an e-mail account on @ReliaSoft.com or @weibull.com,

---

2. Specifically, the information transmitted to the ReliaSoft License Server is your ReliaSoft ID (e-mail address) and Product License Key. If this is your first time registering, your name and the basic contact information will also be transmitted. The information is encrypted by our software first and then transmitted using SSL to the secure server. We do not sell or provide your information to any third parties (see http://www.ReliaSoft.com/privacy.htm). The information sent back is simply a code that tells the software that it is now authorized to run on that computer.

etc.). If it meets the requirements, the next page requires you to copy/paste a code that you will receive from ReliaSoft via e-mail. *Tip: If the e-mail does not appear in your Inbox within a few minutes, check your Junk mail or SPAM folders*.

- **Activate Multiple Applications**: If your license key includes other Synthesis applications that have not yet been activated on this computer, the next page gives you the opportunity to activate any or all of them at the same time.

4. When you see the "Your product has been activated" message, click **Finish** to start using the software.

For Concurrent Network Licenses: If your organization has a Concurrent Network License, your access to the application at any given time depends on how many other users who share the license are accessing the application at the same time. After the initial activation (i.e., the steps described above), the software will automatically check the server each time you try to access the application. Therefore, the computer must have an active Internet connection and must be able to access the ReliaSoft License Server whenever you wish to use the software. You may need to adjust your Internet and/or firewall settings and/or set the server as a trusted location in order to use the application. To determine whether your computer will be able to connect with the ReliaSoft License Server each time you use the application, you can download a test application from http://www.ReliaSoft.com/support/rs40025.htm.

1.5.1.2 Problems Communicating with the Server

The software must communicate with the ReliaSoft License Server at **https://Validate.ReliaSoft.org** each time you activate a particular application on a particular computer. For Concurrent Network Licenses, the software must also connect each time you attempt to access and use the software. Personal or corporate firewalls can block this communication.

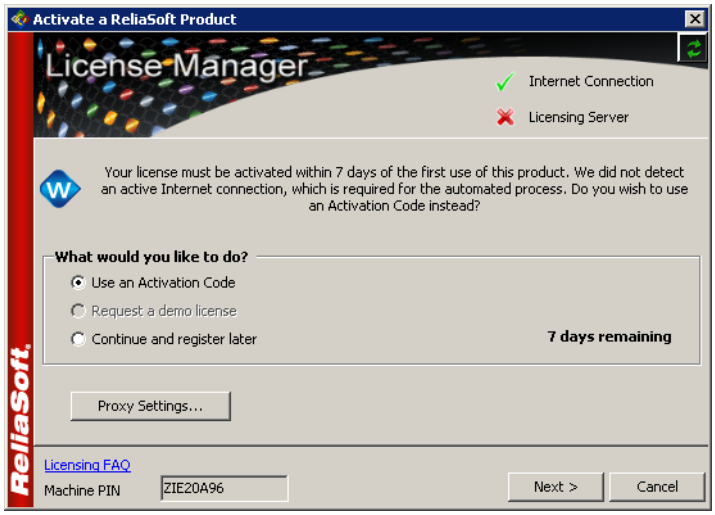
Possible Solutions:

- Personal firewalls (Norton Firewall, ZoneAlarm, BlackIce, Outpost, McAfee Firewall, etc.) need to be configured to allow the software to communicate with the server. Generally, these products will ask for your authorization when the software first tries to establish communication with the server (e.g., "BlockSim.exe is attempting to access the Internet. Do you wish to authorize it?" or something similar). You should authorize this and also make this authorization permanent if possible.
- Some firewall configurations block all new programs without any warning. To activate the software, it will be necessary to manually add the software name to the authorized list. Provide for unlimited access for the software.
- If all of this still fails, try to deactivate the firewall temporarily until the software has been activated. The majority of firewalls have an option to suspend their action. If not, you will have to terminate the firewall. Activate the software and then reboot your machine to restore firewall action.
- For network or corporate firewalls, you may need to ask the network manager to make the required settings to allow the software to communicate with the ReliaSoft License Server and/or add this domain to a trusted list.

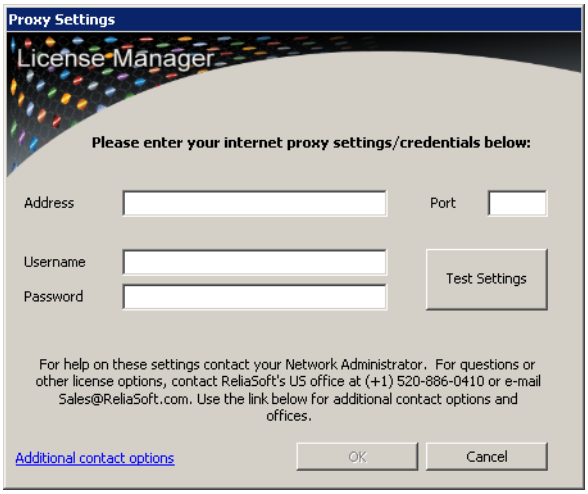
1.5.1.3 Multiple Ways to Activate

1. The software will first attempt to communicate with the ReliaSoft License Server directly. At this time, it checks both your network connection and the ability to connect to our server. In the image below, the connection is detected (green check mark) but it is unable to communicate with our server (red X). (Note that some licenses require additional information, such as a Confirmation Key, password, user or seat information, etc. In such cases,

you will not be able to obtain an Activation Code automatically. If you have trouble with the automatic process, you can refer to step 3 for instructions on obtaining a code manually.)

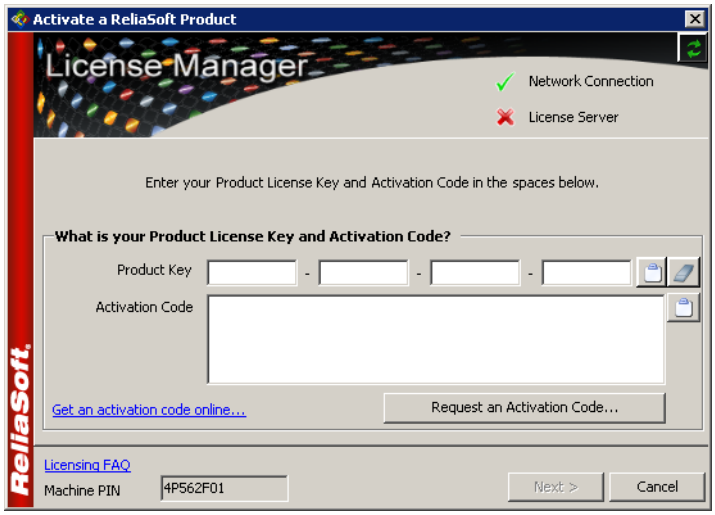


2. In the case that the server connection fails but the software detects network access, the wizard will first prompt you to indicate whether a proxy server is being used. If you choose to configure the software to communicate through a proxy, the Proxy Settings window looks like the one shown next. Enter the information required (you may need the assistance of your network administrator to complete this) and click **Test Settings**. The **OK** button will become enabled if the test is successful.

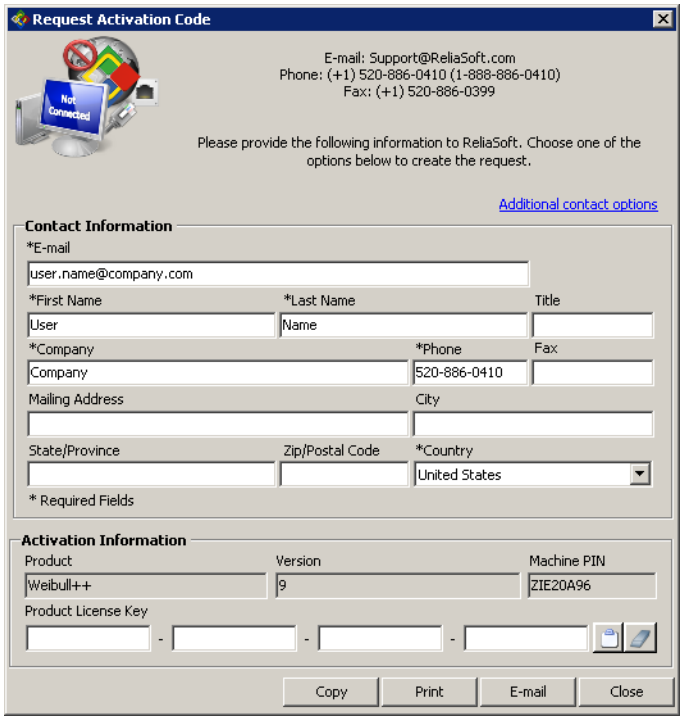


Once a proxy is successfully set, you can continue with the automated process. If the proxy fails, you may want to try again and confirm that the entries are valid. If the test is still not successful, you can exit the window using the **Cancel** button and then try one of the two methods for requesting an Activation Code: from within the wizard or from a web form. Both methods are described next.

3. To request the Activation Code from within the wizard, choose **Use an Activation Code** and continue until you reach the screen shown next.



If you click **Request an Activation Code**, the window shown next will appear.



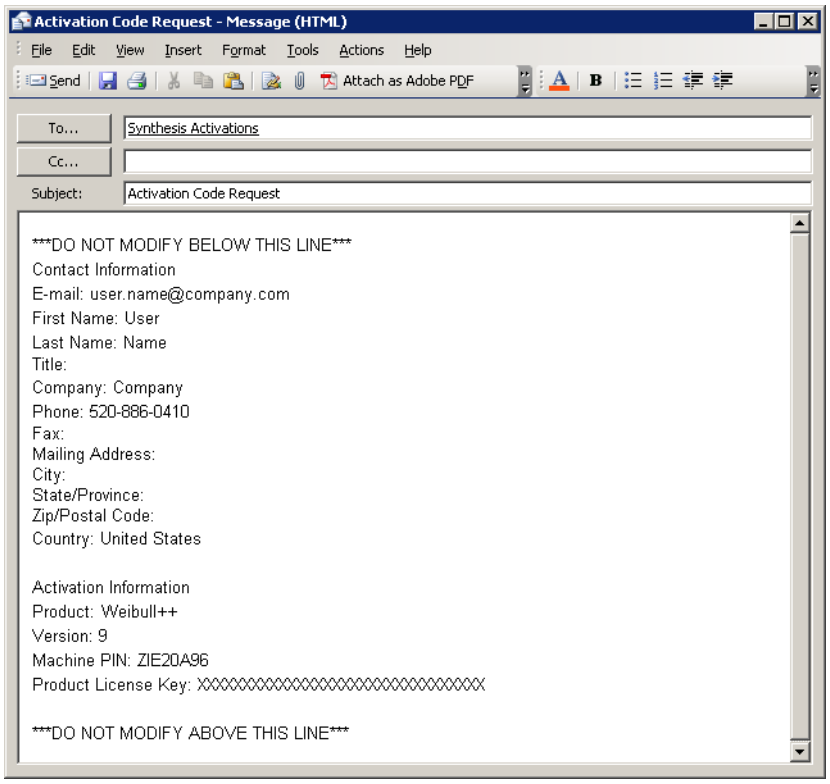
Once you enter the required information, you have the following options:

A. Send an e-mail to our server (if you have an e-mail connection).

B. Print a form to fax or mail to us.

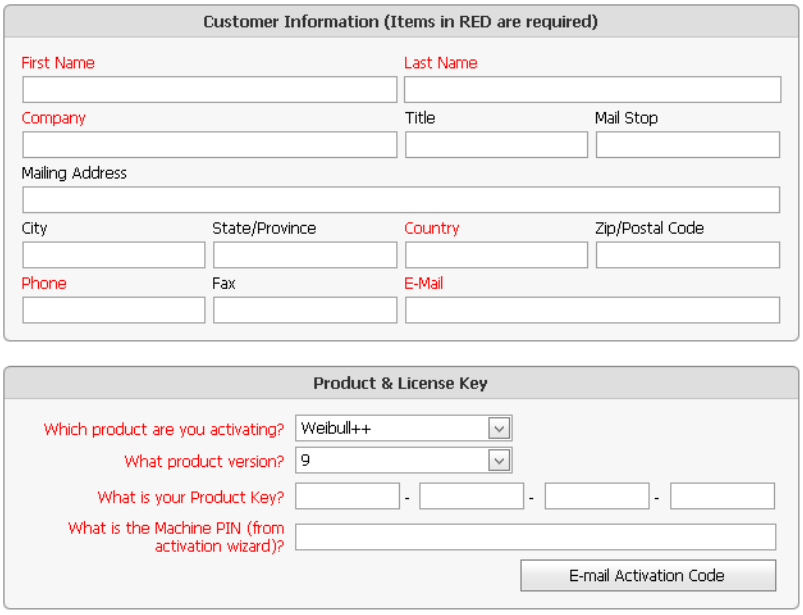
C. Copy the information to a text file so you can submit it to ReliaSoft at a later time via e-mail, mail or phone.

If you have an e-mail connection, the best option is to click the **E-mail** button. This will activate your e-mail client and automatically create an e-mail formatted so that it can be read by our server. Please do not alter any of the text in this e-mail; simply send the message exactly as it is.



When this is received by our server, the server will automatically reply with a block of text that you need to copy and paste in the Activation Code area of the wizard (i.e., below your Product License Key).

4. To request the Activation Code from the web form, go to https://validate.ReliaSoft.org/UserManagement/ActivationCode.aspx, enter the required fields and click **E-mail Activation Code**.



If your license is eligible for activation in this manner, the server will send an e-mail with a block of text that you need to copy and paste in the Activation Code area of the wizard (i.e., below your Product License Key).

5. Lastly, if all else fails, you can contact ReliaSoft via, phone, e-mail and/or fax to obtain an Activation Code. In cases where an automated activation is not possible, you can select the **Continue and register later** option on the first page of the License Manager wizard to continue without activating during a seven-day grace period; this allows you sufficient time to activate via alternative methods.

1.5.2 Deactivating Synthesis Applications

Unless you have an Unlimited User License, there will be a limit to the number of computers and users for which a Synthesis application can be activated at the same time with the same license key. (To review these limits for different license types, see http://www.ReliaSoft.com/license_compare.htm.)

Therefore, if you do not plan to continue to use a Synthesis application on a particular computer (or for a particular user), **it is very important to deactivate it**. The License Server will prohibit future activations that would exceed the license key's limits. If you do not free up activations that are no longer being used, the license may be unavailable for later activation on other computers and/or for other users.

For example, a Standard Network License permits each named user to activate the application on one computer at any given time. If you need to change computers, you must deactivate from the old computer before you will be able to activate on the new one.

There are three ways to deactivate Synthesis applications:

1. **Upon Uninstall**: When you uninstall Synthesis, you will be prompted to deactivate all applications for all users on that computer. If you are not able to complete the deactivation at that time (e.g., because you do not have a network connection to ReliaSoft's License Server), you will need to contact ReliaSoft Technical Support for assistance. *See Section 1.1 on page 2*.
2. **Deactivation Without Uninstall**:
   - From within the application you wish to deactivate, choose **File > Help** and click the **License Manager** link.
   - Enter your password or click **Cancel** to proceed without it. (A password is not required for deactivation.)
   - Click the **Deactivate** button in the center of the License Manager window.
   - You will be prompted to confirm. When you do, the application will immediately close and you will not be able to run it on this computer without activating again.
   - Repeat the process for any other applications that require deactivation.
3. **Contact ReliaSoft**: If you were unable to deactivate during the uninstall, or if you did not uninstall and no longer have access to the computer, you will need to contact ReliaSoft Technical Support for assistance.

1.6 Internet Connectivity

One of the many new enhancements for applications that are built on the Synthesis Platform is the ability to take advantage of an active Internet connection to obtain the most up-to-date announcements, documentation and examples. Internet access is not required to use the Synthesis applications, but once you have registered and activated the software on your computer, an active Internet connection will allow you to take advantage of several useful new features, including:

- **ReliaSoft Online**: When you are using any page in the Backstage view (accessed by clicking the **File** tab on the Ribbon), the ReliaSoft Online pane is displayed on the right side of the MDI. When you have an active Internet connection, this panel provides a variety of useful information, updated in real time so you always have access to the latest information. This includes quick tips for using the software, upcoming training seminars and other recent announcements.
- **Reliability Web Notes**: The Reliability Web Notes page of the My Portal window provides access to information about the tool that you are currently working with. This usually includes a brief description of the item, as well as links to examples and theory articles.
- **Help Center**: When you are using the new Help Center in the Backstage view (accessed by clicking **File > Help**), you will have quick access to a variety of support tools. Many of the tools require Internet access, such as the ability to download software updates and the option to generate an e-mail with the technical details about your computer that may be needed for support.
- **Latest Help File**: We will always install a copy of the help file on your computer so you can access it any time, even when you're working offline. But when you have an active Internet connection, the help topics that you see when you press **F1** or choose **File > Help > Help File Contents** will always be the most up-to-date versions available. This allows us to keep updating the product documentation to provide the information that users are looking for, and it ensures that you will always have access to the latest information about the software.

1.7 Region and Language Settings

The Region and Language settings for your particular computer will have some impact on the way information is displayed in Synthesis applications. Specifically, they will affect:

- Which language is selected by default when you install the software.
- How dates and times will be entered and displayed.
- How decimal values and currency will be entered and displayed.

Users with different regional and language settings can work together on the same analysis projects because, in most cases, the basic information is stored in the database and the software simply displays it in the format preferred by each user. For example, if the date September 25, 2012 is stored in the database, User A might see it as “9/25/2012” while User B might see “25-Sep-12.”

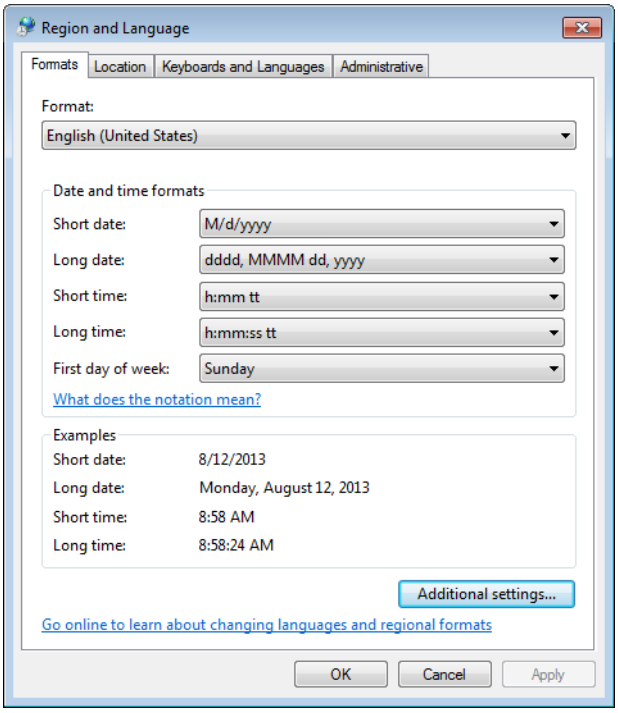
This section first explains how to view or change the Region and Language settings on your computer and then discusses some specific considerations for Synthesis applications.

1.7.1 Viewing/Changing the Region and Language Settings for Your Computer

To open the Region and Language window, do the following:

- In Windows 8, move the pointer to the lower-left corner of the desktop, then right-click and choose **Control Panel**. Click the **Clock, Language, and Region** option then click the **Region** link.
- In Windows 7, choose **Start > Control Panel** and then click the **Region and Language** link.
- In Windows Vista, choose **Start > Control Panel** and then under **Languages and Region**, click the **Change Display Options** link.
- In Windows XP, choose **Start > Control Panel** and then click the **Regional and Language Options** link.

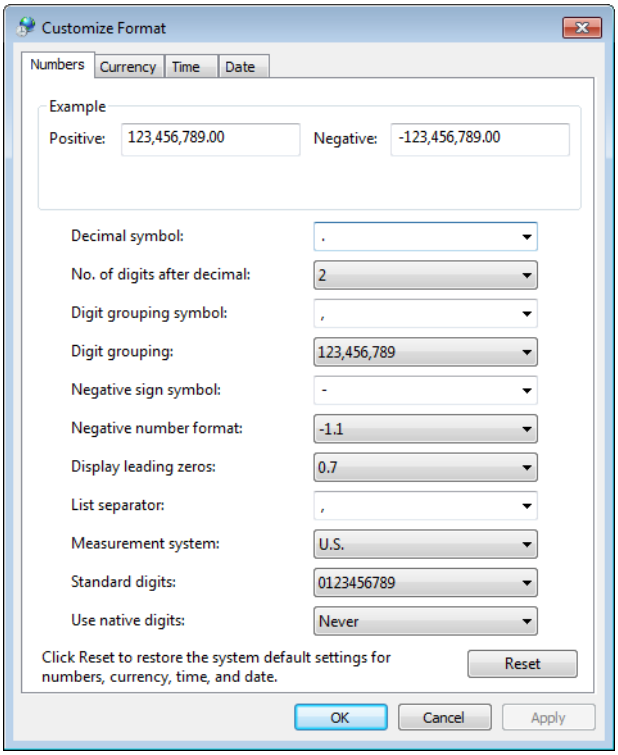
As an example, the Windows 7 interface is shown here. The Windows 8, Windows Vista and Windows XP versions look different but function similarly.



Other relevant settings are managed via the Customize Format window:

- In Windows 8, click the **Additional settings** button.
- In Windows 7, click the **Additional settings** button.
- In Windows Vista, click the **Customize Format** button.
- In Windows XP, click the **Customize** button.

As an example, the Windows 7 interface is shown next.



Note: To see the changed settings in the Synthesis application, you must close the application, and then restart it.

1.7.2 Setting the Default Language When You Install the Software

The user interface for Synthesis applications is available in several languages. You can change this language at any time by opening the Synthesis Setup window (**File > Synthesis Setup**) and selecting an option from the **Language** drop-down list.

When you first install the software, it will check your computer's current language as it is set in the **Format** field in the Windows Region and Language window. If that language is supported in Synthesis, Synthesis will use those settings; however, if that language is not supported, Synthesis will use the default setting of English.

1.7.3 Defining Date and Time Formats

Dates and times appear frequently throughout the Synthesis interface, including (but not limited to):

- The dates in the plot legend area of a plot in all Synthesis applications.
- The Created and Modified dates in many diagrams and in resources in all Synthesis applications.
- The History area of the record and item properties in Xfmea, RCM++, RBI and MPC.
- The dates in the worksheet view and in the record properties windows in Xfmea, RCM++, RBI and MPC.
- The dates in the "dates of failure" format and "usage" format of the Weibull++ warranty folios.

The **Short date** field from the Windows Region and Language window determines how dates are displayed. You can select any standard format (e.g., M/d/yyyy, dd/MMM/yy, yy/MM/dd) or you can create your own format using the available date notations.

The **Short time** field determines how times are displayed. You can select any standard format (e.g., h:mm tt, HH:mm, HH'h'mm) or you can create your own format using the available time notations.

The Windows settings do not apply to the following items:

- Dates and times in spreadsheet utilities (e.g., analysis workbooks, general spreadsheets, etc.).
- Dates and times displayed in the Weibull++ event log folio are created by the folio and are not affected by the computer's settings.

1.7.4 Changing the Decimal Symbol and the List Separator

How decimal values are displayed depends on the value of the Decimal symbol field in the Windows Customized Format window, which determines which character is used to indicate the decimal portion of a number (e.g., 85.25 or 85,25). In addition, the character used to separate the arguments in a list depends on the value of the **List separator** field. These settings affect how you enter functions in analysis workbooks and in general spreadsheets, either manually or by using the Function Wizard. For example, if the decimal symbol value is a comma, and the list separator value is a semicolon, you would enter a function like this: =RELIABILITY("Folio1!Data 1";500;100;0,9)). Whereas, if the decimal symbol is a period and the list separator is a comma, it would be
= RELIABILITY("Folio1!Data 1",500,100,0.9)).

1.7.5 Setting the Measurement System in the Page Setup

On the Margins tab of any Synthesis application's Page Setup window (**Home > Print > Page Setup**), the **Units** field determines which unit of measure (inches [U.S.] or centimeters [Metric]) is used for the page margins on print output. The field includes the **System's default** option. Selecting this option means that the measurement system used is determined by the value of the Windows Customize Format window's **Measurement system** field, and it will change if the computer's settings change.

Tip: If you want the page margins to always be the same, you can select a specific unit of measure and not use the **System's default** option.

1.8 Introduction to the Synthesis Platform

This section provides a brief introduction to the Synthesis Platform as it relates to using BlockSim and RENO.

Centralized Data Storage and Flexible Permissions

All Synthesis applications offer centralized data storage that allows multiple users to work cooperatively on analysis projects. As described in the "Working in Synthesis" chapter, you can choose which database platform will provide the back-end data storage for each Synthesis repository.

- A **Standard Repository** (*.rsr9) is easy to create and maintain without any special IT infrastructure or support, but there are limitations to the amount of data it can store and the number of users who can access the database simultaneously.
- An **Enterprise Repository** requires implementation of Microsoft SQL Server or Oracle (which requires separate licensing, support and maintenance), but it is a more robust platform that can store much more analysis information in the same database and supports access by many more simultaneous users.

For all enterprise databases, and when you have enabled login security for standard databases, authorized users will be able to create and maintain user accounts that determine which analyses and capabilities each individual user has

permission to access. You can use the basic security groups that have been predefined in the software or customize the access permissions to meet the specific needs of your organization.

Tip: Working with a database-driven, multi-user application such as the Synthesis version of BlockSim and RENO is a bit different from working with other types of software that you may be familiar with (such as Microsoft Excel or previous versions of BlockSim). For example, changes are saved automatically while you are updating the analysis (e.g., whenever you click **OK** or navigate away from a cell in a table or worksheet). In addition, you must be diligent about storing backups and performing adequate database maintenance to guard against data loss or corruption. Specific recommendations are provided on page 69 in Chapter 2.

Integration with Other Synthesis Applications

The Synthesis Platform offers a unique solution to the problem of integrating different types of reliability information without sacrificing the power and flexibility of each individual analysis tool. From the individual user's perspective, your interaction with each software product will be similar to your experience with previous versions — a full-featured tool that has been expertly designed to meet your needs for all aspects of the analysis methodology. However, from the overall reliability program perspective, multiple users will now have the ability to access the same centralized data repository from any Synthesis application and thereby share commonly needed resources between applications and analyses (such as the model that describes a component's failure behavior, the description of a particular maintenance strategy or the reliability-wise configuration of a system).

As described in the "Synthesis Resources" chapter, this integration is facilitated primarily through *Synthesis resources* that can be published from one type of analysis and utilized in other analyses where they may be relevant. When a resource needs to be updated in response to new information, the change can be reflected automatically in all analyses that rely upon it. Specifically for BlockSim and RENO, the Synthesis Platform gives you the ability to:

- Use models and/or universal reliability definitions (URDs) that were created in Weibull++, ALTA, Xfmea, RCM++ or RBI to define the reliability and/or maintenance characteristics of the blocks in your BlockSim RBDs and fault trees or to define some function within your RENO flowchart.
- Publish any RBD or fault tree analysis as a model that can be used in analyses performed in Synthesis applications.
- Work with system configuration and failure mode data from Xfmea, RCM++ or RBI as RBDs or fault trees in BlockSim (using the new "Failure Modes and Reliability Analysis" – FMRA).

Messages and Action Tracking

One of the many benefits of having multiple users working from the same centralized data repository is the opportunity to use a common interface for posting announcements and tracking the completion of assigned actions.

All Synthesis desktop applications offer the **My Portal** interface (*see page 101 in Chapter 3*), which facilitates the dissemination of **Messages** directed either to all system users or only to specific users/groups. My Portal can also be used to display the **Actions** that you are involved with, either as the person responsible for completing the assignment, the person who created the record or a person who simply needs to be kept informed about the progress.

If desired, the Synthesis repository can be configured to enable **E-mail Notifications** that serve to alert members of the team when a message or action that they're involved with is created or updated. You also have the option to send actions to your calendar in **Microsoft Outlook**.

Take Advantage of Internet Connectivity

One of the many new enhancements in the Synthesis Platform is the ability to take advantage of an active Internet connection to obtain the most up-to-date announcements, documentation and examples. Once you have registered and activated the application on your computer, Internet access is not required — you can still perform all of your analysis activities while working offline — but an Internet connection allows you to take advantage of several useful new features, including:

ReliaSoft Online for up-to-date tips and announcements.

Reliability Web Notes for context-sensitive access to theoretical background information for the feature you're currently working with.

Help Center for access to a variety of support tools – including the ability to download software updates and generate an e-mail with the details needed for technical support.

Online Help File for access to all of the latest updates that provide new information users have been looking for. (Of course we will always install a static version for the times when you're working offline.)

1.9 BlockSim Features Overview

This section provides a brief overview of the major analysis, data management and reporting capabilities provided by BlockSim.

Reliability Block Diagram (RBD) Configurations

BlockSim's interface for reliability block diagram (RBD) creation is the most intuitive, flexible and polished in the industry. Easy drag-and-drop techniques allow you to build RBDs for the simplest to the most complex systems. The software supports all of the following reliability-wise configuration types:

- **Simple Series and Parallel**: Simple series configurations assume that the failure of any one component causes the system to fail; while simple parallel configurations assume that the system continues to operate if any path succeeds.
- **Complex**: Complex configurations require a more advanced analytical treatment than a simple combination of series and parallel blocks. Such configurations may be required for analyzing network systems, competing failure modes, etc.
- ***k-out-of-n***: Node blocks can be used to define *k-out-of-n* redundancy, where a specified number of paths leading to the node must succeed in order for the system to succeed.
- **Load Sharing**: In a load sharing configuration, each block supports a percentage of the total load. In the Synthesis version, BlockSim now supports stress-independent distributions for load sharing blocks.
- **Standby Redundancy**: In a standby configuration, standby blocks are available to become active under specified circumstances. BlockSim can model *hot*, *warm* or *cold standby configurations*.
- **Mirrored Blocks**: Mirrored blocks allow you to put the exact same component in more than one location within the diagram. These blocks can be used, for example, to simulate bi-directional paths in a communications network. In the Synthesis version, BlockSim offers increased modeling flexibility by supporting mirrors across different diagrams.
- **Multi Blocks**: Multi blocks help you to save time (and space in the diagram) by using a single block to represent multiple identical components configured in series or parallel.

- **Subdiagrams**: BlockSim offers a virtually unlimited capability to link diagrams as components in other diagrams, which provides a variety of opportunities to encapsulate one analysis into another.
- **Publish Diagrams as Models**: Now in the Synthesis version, we have drastically increased BlockSim's capability for encapsulation by allowing users to publish an analyzed diagram as a model that can be used in any other Synthesis analysis (e.g., in another BlockSim diagram or in the reliability/maintainability calculations supported by Xfmea, RCM++ or RBI).

Fault Tree Analysis Configurations

BlockSim's fault tree analysis interface supports all of the traditional gates and event symbols that are applicable to system reliability and related analyses. In addition, only BlockSim allows you to expand the modeling capabilities with additional logic gates that represent load sharing and standby redundancy configurations. The available event symbols include **Basic**, **Undeveloped**, **Trigger**, **Resultant** and **Conditional**, while the supported fault tree diagram gates include:

- **AND** and **OR** gates
- **NOT**, **NAND** and **NOR** gates
- **Voting** gates
- **Inhibit** gates
- **Priority AND** and **Sequence Enforcing** gates
- **Load Sharing** and **Standby** gates (exclusively in BlockSim)

Now in the Synthesis version, fault tree diagrams can be configured to display intermediate results at each individual gate (in addition to the results calculated for the top-level gate).

Your BlockSim projects can contain both fault trees and reliability block diagrams together in the same analysis workspace. You can also integrate your fault trees and RBDs by linking a fault tree as a subdiagram to an RBD or vice versa, copying events from a fault tree diagram and pasting them as blocks in an RBD, and automatically converting any fault tree to an RBD.

Analysis Based on the Exact System Reliability Function

As described in the "BlockSim Analytical Diagrams" chapter, BlockSim uses an exclusive algorithm pioneered by ReliaSoft to algebraically compute the exact system reliability function for even the most complex systems. This makes it possible to perform a number of system reliability investigations analytically, without resorting to simulation. Some useful analytical results include:

- **System Reliability Metrics and Plots**: You can use the convenient Quick Calculation Pad (QCP) and plot sheets to calculate and visualize key system reliability metrics such as Reliability, Probability of Failure, Reliable Life (i.e., time for a given reliability), BX% life (i.e., time for a given unreliability), Mean Life and Failure Rate.
- **Minimal Cut Sets**: For each analytical diagram, BlockSim identifies the unique combinations of component failure that can cause system failure. These *Minimal Cut Sets* can be used to understand the structural vulnerability of a system.
- **Reliability Importance Measures**: BlockSim provides a set of "Reliability Importance" plots designed to show the relative importance of each component with respect to the overall reliability of a system. The software also offers "FRED Reports," which provide an intuitive graphical presentation of key reliability metrics, with color-coding to identify the ones that may be critical for system improvement.

As described in the "System Improvements" chapter, the Synthesis version of BlockSim also introduces a new allocation analysis tool dedicated to helping you find the most effective component **Reliability Allocation** to meet a system reliability goal. The new centralized utility makes it easier than ever to enter and manage all of the information required to use the innovative **Cost Optimized** allocation method that's still available exclusively in BlockSim. This feature now also supports **Equal** and **Weighted** reliability allocation methods.

Repairable System Analysis via Discrete Event Simulation

As described in the "BlockSim Simulation Diagrams" chapter, BlockSim's simulation capability for reliability, availability, maintainability and supportability (RAMS) analysis of repairable systems is more flexible and realistic than ever. For a new system, product engineers can use the simulation results to improve and optimize the design, and make projections about how the system may perform in the field. For existing equipment, asset performance managers, maintenance planners, risk/safety analysts and many other professionals can use the simulation results for maintenance resource planning, throughput estimates, life cycle cost analysis and much more.

When you utilize simulation, the analysis can consider:

- The **restoration factor** that captures the impact of repairs on the future reliability of the component (i.e., *as good as new*, *as bad as old* or partial restoration).
- **Duty cycles** for components that experience a different stress load than the rest of the system (e.g., a component that may run for only 10 minutes out of every hour that the system operates, or a component that works twice as hard during a particular phase of the overall mission).
- The **expected downtime** associated with corrective or scheduled maintenance (defined either as fixed durations or based on probabilistic distributions).
- The costs and logistical constraints associated with allocating the personnel (**repair crews**) and materials (**spare parts**) required to perform maintenance when needed.
- Components that will receive maintenance based on what happens to other components in a specified **maintenance group**. For example, if one component in the group fails, this might trigger preventive maintenance for other components while the system is already down. In the Synthesis version, BlockSim can now model even more complex scenarios, such as maintenance that occurs when another component is *restored* to operation, or components that are affected by *more than one* maintenance group.
- BlockSim now offers the ability to create **state change triggers** that activate or deactivate a block under certain conditions during the simulation. This provides increased modeling flexibility for highly complex dependency scenarios, such as standby configurations and other situations when you may need to divert the simulation onto an alternate path when a particular event occurs.

BlockSim's simulations generate a wide variety of results at the system and/or component level. This includes Uptime/Downtime, Mean Time to First Failure (MTTFF), Availability, Reliability, Number of Failures, Number of PMs/Inspections, etc. You can use these results for many different applications, including (but not limited to):

- Choosing the **most effective maintenance strategy** based on considerations of safety, cost and/or availability.
- Determining the **optimum preventive maintenance (PM) interval**.
- Managing the **spare parts inventory** based on considerations of cost, utilization rate, supply bottlenecks, etc.
- Identifying the components that have the **biggest impact on availability** (downtime).

The software's **Log of Simulations** feature provides the information you need to evaluate the variability in specific simulation results of interest. In the Synthesis version, you can now export these results to Microsoft Excel for greater flexibility.

BlockSim's **Throughput Analysis** can be used to identify bottlenecks, optimize resource allocation and otherwise improve the processing efficiency of the system. The software allows you to determine how the simulation will allocate the processed output (e.g., parts manufactured in an assembly line or volume of oil processed in a pipeline) across the paths defined in the diagram. The software also allows you to specify how the backlog will be processed. When the throughput varies over time (e.g., if the flow from an oil well drops over time as the reserves are depleted), BlockSim provides a choice of models to describe the time-dependent variability (linear, exponential or power).

Every successful organization understands that it is critical to understand the **Life Cycle Costs** (LLC) associated with their equipment. Whenever applicable, BlockSim allows you to specify the direct and indirect costs associated with the maintenance strategies that you have defined, including costs related to downtime, maintenance crews, spares, etc. This yields a wide array of simulation results that are instrumental in performing realistic LLC assessments. Now in the Synthesis version, any cost input can be entered as a probabilistic model, if desired.

Reliability Phase Diagrams

You can use **reliability phase diagrams** to model systems that go through different phases during the course of their operation. For example, some aircraft components (such as landing gear) operate only during the take-off and landing phases of a mission, while others (such as engines) may experience a higher failure rate during these phases due to higher stress. To model these types of situations in BlockSim, you can simply create separate RBDs to describe the system's operation during each distinct phase and then use a phase diagram to describe how the system proceeds through the phases over time. Reliability phase diagrams are discussed in the "BlockSim Phase Diagram" chapter.

In addition, the software uses *maintenance phases* to model scenarios in which a system goes directly to maintenance under specified conditions. For example, if a failure during the taxi phase sends an aircraft in for maintenance, it will start over from the beginning of the mission once repaired — not from the middle of the taxi phase where it was when the failure occurred, as other RBD analyses have been forced to assume. This flexibility provides a tremendous leap forward in the ability to simulate system operation more realistically.

Now in the Synthesis version, we have added **success/failure paths**, for situations where a system proceeds to one operational phase upon success and a different operational phase upon failure. **Node blocks** and **stop blocks** are also now supported.

Plots and Charts to Visualize Your Analysis Results

BlockSim makes it easy to create a complete array of plots and charts to present your analysis graphically. The **Plot Setup** allows you to completely customize the "look and feel" of plot graphics while the **RS Draw** metafile graphics editor provides the option to insert text, draw objects or mark particular points on plot graphics. You can save your plots in a variety of graphic file formats (*.jpg, *.gif, *.png or *.wmf) for use in other documents.

Overlay Plots (formerly called "MultiPlots") allow you to plot the results from multiple data sets together in the same plot. This can be an effective visual tool for many different purposes, such as comparing different analyses (e.g., Design A vs. Design B) or demonstrating the effects of a design change (e.g., Before vs. After).

Workbooks and Reports for Custom Analysis

As described in the "Reports" chapter, all Synthesis desktop applications offer powerful tools for custom analysis and reporting. If you want to create a custom analysis that integrates results from different diagrams, BlockSim now provides a choice of using an **Analysis Workbook** (with functionality similar to Microsoft Excel) or a new **Word Report Template** (with functionality similar to Microsoft Word).

Both tools allow you to use the **Function Wizard** to insert calculated results based on selected data sheet(s). You also have the option to configure the functions to use generically numbered "data sources" instead of named diagrams. This makes it easy to use the same template again for different analyses.

1.10 RENO Features Overview

This section provides a brief overview of the major analysis, data management and reporting capabilities provided by RENO. For a full discussion on RENO's features, see the "RENO Flowcharts" chapter.

Easy to Build Flowchart Models

With RENO, you use the familiar and intuitive flowcharting concept to build models for the scenarios that you are trying to understand, quantify and/or simulate. The software provides a full array of building blocks for these flowcharts to serve different functions within the model. For example, conditional blocks allow you to define "If/Then" statements that trigger one response if the condition is met (TRUE) and another if it is not (FALSE). You can also define various types of resources that can be used in multiple locations and shared among analyses. For example, a RENO function stores an equation that allows you to compute output based on given input(s). You employ easy point-and-click and drag-and-drop techniques to use these components to build simple or complex flowchart models.

"Intelligent" Features to Help You Build Equations

All equation-enabled windows in RENO provide "intelligent" features to help you save time and avoid errors when building equations. These include:

- **Function Selector**: Press **CTRL+SPACE** while building an equation to view a list of recognized resources, keywords and functions.
- **Equation Editor**: Press **SHIFT+SPACE** while building an equation to use RENO's Equation Editor to simplify the equation-building process.
- **Color-Coding**: As you type, RENO automatically color-codes recognized resources, functions and operators to make it easy for you to spot typing errors, undefined variables, unmatched parentheses, etc.
- **Resource Preview**: RENO provides a summary of the properties of each resource or function that you use in your equations and allows you to view/edit resources with the click of a button.

More Than 200 Internal Functions and Predefined Functions

The Function Selector and Equation Editor provide instant access to the RENO internal functions, resources and predefined functions that you will use in the equations for your analyses. This includes:

- **User-Defined Resources**: A list of the variables, RENO tables, RENO functions and static functions, RENO simulation definitions and models defined in the project, as well as any cell in any analysis workbook in the project.
- **RENO Internal Functions**: A list of functions that instruct the software to insert specific information into the equation, such as the value passed from the previous flowchart block (in), the total number of simulations performed (sims_total), the maximum value in a table column (table_max), etc. These also include functions that insert simulation results from BlockSim diagrams within the project, once the diagrams have been simulated via a simulation definition. For example, you can insert the amount of time that the system was down (sr_downtime), the costs associated with preventive maintenance (sr_costs_total_pm), etc.
- **Predefined Functions**: A list of built-in functions supported by the software. This includes:
  - Engineering functions such as Bessel, Error functions, etc.
  - Math and Trigonometry functions such as Sine, Derivative, Integral, Logarithm, etc.
  - Financial functions such as Future Value, Cumulative Interest Paid, Periodic Payment for an Annuity, etc.
  - Statistical functions such as the Weibull distribution, the Fisher Transformation, the Chi-Squared distribution, etc.
  - Logical functions such as IF statements.

Debugger Utility to Step Through Flowcharts

RENO includes an integrated utility to help you validate and "debug" your flowchart models. This feature allows you to move through each step in a flowchart and watch the values as each block is executed. You can also watch the progress as the software automatically performs a specified number of simulations in "debug" mode.

Sensitivity Analysis and Optimization Through Multiple Analyses

RENO's Sensitivity Analysis feature allows you to vary one or two constants across simulation runs. For example, if you are analyzing potential investment strategies, you may wish to vary the number of years that you will be investing and/or the percentage of your income that will be invested to see how different inputs will affect the final results. Likewise, you may wish to generate reliability or availability results for a range of times in order to examine the equipment's behavior over time. Potential applications are limited only by your imagination!

Simply enter the starting, ending and increment values for the variable(s) and then RENO automatically varies the inputs and performs runs under each set of conditions.

The Multiple Analyses feature takes the sensitivity analysis capability a step further by allowing you to configure the software to automatically vary the specified variable(s) in order to determine the value that minimizes or maximizes a specified result. For example, you could use this feature to automatically determine the optimum amount of inventory to maximize profits or the optimum preventive maintenance (PM) interval to minimize costs. The results of each analysis can be displayed in the plot sheet, with the optimum value marked on the plot.

Flexible Array of Results with Multiple Display Options

RENO can generate a wide variety of results for your analyses, including averages, sums, arrays, minimum values, maximum values, etc. Simply construct the flowchart model to generate the metrics of interest to you and then employ the software to use discrete event Monte Carlo simulation to execute the model. For models without random variables, a single execution will yield all results of interest. When random variables are present, simulation (repeating the model analysis multiple times) can be utilized. Results are presented in a variety of ways, including:

- **During the Simulation**: You can configure the RENO Simulation Window to display a specific result of interest while the simulation is running and watch the variation as the analysis progresses.
- **In the Simulation Results Explorer**: The Simulation Results Explorer displays the full results of the simulation in Excel-compatible spreadsheets that present the data in an organized fashion and make it easy to explore, copy/paste or export results of interest to your reports or customized analyses.
- **In the Flowchart**: After a flowchart has been analyzed/simulated, the values for all result storage blocks are displayed directly within the chart. You also have the option to show the last values for other constructs, if desired.
- **In the Plot Sheet**: RENO's integrated plots provide a graphical display of your simulation results with the click of a button. You can choose to display the data for any result storage block in the flowchart. The **Plot Setup** allows you to completely customize the "look and feel" of plot graphics while the **RS Draw** metafile graphics editor provides the option to insert text, draw objects or mark particular points on plot graphics. You can save your plots in a variety of graphic file formats (*.jpg, *.gif, *.png or *.wmf) for use in other documents.

1.11 What's New in the Synthesis Version?

With the release of the Synthesis versions, BlockSim and RENO offer a completely updated user interface that has many new and enhanced features. Some of the major additions and enhancements include:

New Synthesis Platform and Centralized Data Storage

BlockSim and RENO analyses are now stored in a centralized database that supports simultaneous access by multiple users and shares relevant reliability information between ReliaSoft's Synthesis applications. To give just a couple examples of the many integration opportunities, you can now:

- Seamlessly integrate Weibull++ and ALTA analyses to set the properties for a reliability block diagram or fault tree in BlockSim, then automatically update the diagram if the underlying data analysis changes.
- Work with system configuration and failure mode data from Xfmea, RCM++ or RBI analyses as RBDs or fault trees in BlockSim (using the new "Failure Modes and Reliability Analysis" – FMRA). *See Chapter 16*.

Technical Upgrades and Productivity Enhancements

All Synthesis desktop applications are now available in both 32-bit and 64-bit versions. We have added multi-thread support to take full advantage of the processing power on your computer, and also re-engineered/re-optimized much of the code, resulting in speed increases in excess of 10X. *For information about the Single-Thread and Multi-Thread versions of BlockSim, and to determine which version is right for you, refer to* http://www.ReliaSoft.com/BlockSim/multithread.htm.

The Synthesis version of BlockSim also introduces a new **Batch Mode** feature (*see page 307 in Chapter 9*) that allows you to schedule a series of simulations in advance, which can improve your productivity by allowing you to run lengthy simulations when you're away from your computer. Simply schedule the simulations to run overnight and find the results available when you return to your desk in the morning!

Reuse Reliability/Maintenance Definitions

The new universal reliability definition (URD) capabilities in the Synthesis version make it easier and more flexible than ever to manage the component reliability and maintenance characteristics required for system analyses. Now you can share the same resources (e.g., failure model, maintenance strategy, availability of repair crew/spare parts, etc.) with all users who have access to the project, and directly link the URD with other analyses performed in Weibull++, ALTA, Xfmea, RCM++ and RBI. *For more information on URDs, see page 170 in Chapter 5*.

For many of the BlockSim inputs (e.g., duration, cost, etc.), you now have the choice to enter a constant value (e.g., $100) or a probabilistic model (e.g., Normal distribution with Mean = $100 and Std = $16.67).

New Gates and Results for Fault Trees

BlockSim's fault tree analysis modeling capabilities have been expanded. The following gates have been added: NOT, NAND, NOR, Priority AND, Sequence Enforcing.

In addition, fault tree diagrams can now be configured to display intermediate results at each individual gate (in addition to the results calculated for the top-level gate).

New Interface for Optimum Reliability Allocation

BlockSim has always offered a unique capability to utilize feasibility and cost models to calculate the optimum component reliability allocation to meet a system reliability goal. Now we have made it easier than ever to enter and manage the information required for these calculations, and we've added support for two new allocation methods: Equal and Weighted.

Calculating the Optimum Replacement Time

The ability to calculate the optimum replacement time provides a powerful opportunity to reduce costs while maximizing uptime. In the Synthesis version, BlockSim makes it easy to enter and manage the information required for these calculations. Now you can perform the calculations for any number of blocks with a single click of the mouse, and the software also provides the option to find optimum times for performing maintenance on selected groups of blocks (i.e., for packaging maintenance tasks). *See page 440 in Chapter 15*.

State Change Triggers

BlockSim now offers the ability to create state change triggers that activate or deactivate a block under certain conditions during the simulation. This provides increased modeling flexibility for highly complex dependency scenarios, such as standby configurations and other situations when you may need to divert the simulation onto an alternate path when a particular event occurs. *See page 216 in Chapter 7*.

Enhanced Cost Calculations

Now BlockSim offers more control than ever over how costs are calculated and provides expanded cost-related results. You can:

- Specify, for each crew, what kinds of delays are included in cost calculations and what delays should be ignored.
- Specify costs associated with system failure, including cost per incident and downtime rate.
- Specify revenues, including system uptime revenue and revenue due to throughput, allowing calculation of opportunity costs.
- View new simulation results and metrics pertaining to costs and cost contributions, including new criticality metrics.

Automatic Conversion for Time Values Entered in Different Units

All Synthesis desktop applications now allow you to define conversion factors for entering time values in different units (*see page 64 in Chapter 2*). To give just one example, you can now define the component failure models in terms of hours but then use the QCP to calculate the system reliability for a year of operation — the software performs the conversion automatically based on the multipliers specified within the database.

Redesigned QCP

BlockSim's Quick Calculation Pad (QCP) has been updated and redesigned. The new interface provides a "Calculation Log" that works like the paper roll in an adding machine, allowing you to record the results from a series of different calculations and then copy/paste the results as needed.

New Template-Based Reporting Tools

BlockSim now offers two new template-based reporting tools. "Analysis Workbooks" are spreadsheets (similar to Microsoft Excel) that can be used for custom analysis and reporting. "Word Report Templates" have functionality similar to Microsoft Word. Both tools allow you to automatically insert calculated results from selected diagrams, and save templates that can be re-used again and again with different data sources. *See Chapter 19*.

Use BlockSim Simulation Results Within RENO Flowcharts

RENO and BlockSim are now directly integrated. You can view your RENO flowcharts conveniently in the same interface with BlockSim's diagrams. Plus, RENO's new simulation definition resources allow you to trigger simulation of a BlockSim diagram from within a RENO flowchart and then draw results from that simulation into the flowchart. This allows you to, for example, create a BlockSim RBD modeling a system that is used in your RENO flowchart and then use simulation results such as reliability, cost, etc. directly within the flowchart. *See page 386 in Chapter 11*.

Simulation Worksheets

Simulation worksheets allow you to vary values that are used in simulating a BlockSim simulation RBD or a RENO flowchart. This allows you to investigate the effect of one or more settings on the simulation results. The values used can be transferred from a DOE++ experiment, allowing you, for example, to design an experiment in DOE++, simulate the experiment in BlockSim or RENO and then analyze the simulation results in DOE++. Values can also be entered manually, giving you complete control over the combinations of settings used. This allows you to perform batch simulation of a simulation RBD, using different settings for each simulation. *See Chapter 17*.

Expanded Modeling Capabilities

RENO's former “Definitions” have been streamlined into Synthesis resources. Any resource can now be used in any project throughout the repository, saving you time and effort in creating related analyses. Additionally:

- The new binary node is used to determine the probability of an outcome, making it faster and easier than ever to build decision trees.
- The variable resource offers enhanced flexibility. You can use it as a constant, maintaining a fixed numerical value during simulation and/or varying it between runs for sensitivity analysis. Or you can use it to store the output value of a block during simulation.

Take Advantage of Internet Connectivity

One of the many new enhancements in the Synthesis Platform is the ability to take advantage of an active Internet connection to obtain the most up-to-date announcements, documentation and examples. Once you have registered and activated the application on your computer, Internet access is not required — you can still perform all of your analysis activities while working offline — but an Internet connection allows you to take advantage of several useful new features, including: **ReliaSoft Online** (up-to-date tips and announcements), **Reliability Web Notes** (context-sensitive access to theoretical background information for the feature you’re currently working with), **Help Center** (a variety of support tools – including the ability to download software updates and generate an e-mail with the details needed for technical support) and **Online Help File** (we will always install a static version for the times when you’re working offline, but the Internet version contains all of the latest updates to provide the information users have been looking for).

Working in Synthesis 2

With the Synthesis Platform, ReliaSoft's software tools have transitioned from a standalone document/file format, to a multi-user, database-driven approach. This offers enormous potential to integrate reliability program activities and tools, while simultaneously facilitating effective information sharing and cooperation between engineering teams of any size.

The experience of working in a database-driven, multi-user environment will be a bit different than using prior versions of ReliaSoft's applications (Version 7 or older) and other document-centered applications such as Microsoft Excel. For example, your organization (or team) will need to choose the most appropriate database type and security, implement shared settings that will effectively facilitate project management, and establish adequate backups and database maintenance to protect against data loss. In addition, individual users must be aware that changes are saved automatically as they work, and become familiar with the features that facilitate simultaneous access by multiple users.

This chapter addresses a variety of topics that will help you configure and use the Synthesis Platform to meet your particular needs. As a supplement to this information, we have also compiled some targeted tips and FAQs at http://Synthesis.ReliaSoft.com/tips/.

- The two types of Synthesis repositories- Section 2.1 (p. 28).
- Creating and managing projects - Section 2.2 (p. 36).
- Implementing repository security - Section 2.3 (p. 46).
- Repository settings - Section 2.4 (p. 62), including:
  - Using e-mail notifications to keep users up to date on the progress made in completing assigned actions.
  - Configuring settings that affect analyses in all projects throughout the database, including time units, default naming conventions, global identifiers, project categories and more.
- Protecting against data loss and corruption by establishing adequate backups and database maintenance and creating restore points - Section 2.5 (p. 69).
- Importing and exporting information - Section 2.6 (p. 72).
- Using the Reliability Data Warehouse to access XFRACAS data, and using the Dashboard Viewer to view Synthesis data in graphical form - Section 2.7 (p. 88).

2.1 Synthesis Repositories

Synthesis applications offer two types of databases for storing projects and analysis data. The type of database you choose will depend on the requirements of your organization.

Standard repositories (*.rsr9) are basic database files for single- or multi-user environments. Implementation of a database server is not required; however, there are limits on the amount of data and the number of simultaneous users.

Enterprise repositories (*.rserp) require implementation and support of Oracle or Microsoft SQL Server, but they are a more robust platform that can store much more analysis information in the same database and support access by many more simultaneous users. Synthesis applications are compatible with Microsoft SQL Server 2005 or later and Oracle 10g or later (including the free Express editions of all of these).

2.1.1 Standard Repositories

A standard repository is a basic database file for single- or multi-user environments. Implementation of a database server is not required; however, there are limits on the amount of data and the number of simultaneous users. (*If you prefer to use a more robust Oracle or SQL Server database, see Section 2.1.2 on page 29*.)

Creating a New Standard Repository

To create a new standard database, choose **File > New > Create a new standard repository**.

In the **Repository name** field, specify the filename for the new *.rsr9 file. The path where the file will be saved is shown below this field; to change the location, click the browse icon in the field.

The following options are available when you create a new standard database:

- Select the **Restricted Access** check box if you want the new database to be login secure. You can then select the **Immediately go to security window** check box if you want to add user accounts as part of the database creation process. (*To learn more about secure databases, see Section 2.3 on page 46*.)

  Note that you cannot automatically remove security from a database once it has been enabled. However, you can create a new non-secure database and use the **Import from existing repository** check box to automatically import all of the data from the secure database to the non-secure one.

- Select the **Import from existing repository** check box if you want to import entire projects and other data from an existing database, after the new database is created.

Opening an Existing Standard Repository

If you have opened the file recently, choose **File > Recent** and then select the file in the Recent Repositories list.

Otherwise, choose **File > Open Repository** and then browse for the *.rsr9 file.

Tip: If you usually work with the same repository, you can configure the Synthesis applications to attempt to connect with it automatically each time you start the application. Choose **File > New** and then select the **Always start by connecting with the last repository** check box.

If the repository has login security enabled and you are unable to connect, you may be encountering one of the following issues:

- **No access to the repository**. If you have not been given access to the repository, you will see a message stating that your account in the repository is not active or not assigned to at least one security group. You will need assistance from someone who can create and update user accounts (*see Section 2.3.3 on page 50*).
- **Windows authentication failed**. If your Windows login (domain and username) is different from what was specified for your user account, you will see a message to connect using alternative credentials. You will need assistance from someone who can enable the use of alternative credentials for your user account (*see Section 2.3.3.2 on page 52*).

2.1.2 Enterprise Repositories

Unlike standard repositories, enterprise repositories require an established database server with Oracle or Microsoft SQL Server. This allows you to store more analysis information in the same repository and support access by many more simultaneous users. Synthesis applications are compatible with Microsoft SQL Server 2005 or later and Oracle 10g or later (including the free Express editions of all of these).

To establish a database server, you will need to purchase an appropriate license package from Oracle or Microsoft. The license, as well the IT maintenance and support needed to establish the database server, is separate from the ReliaSoft license agreement and must be negotiated directly with Oracle or Microsoft.

As an alternative, you may choose to use the free versions of Oracle or SQL Server if the expected load for the database fits within the limited capabilities of the Express edition (as specified by Oracle/Microsoft). With the free editions, you can establish a functioning enterprise database on your own without making a large investment of time and resources. If your organization's needs grow beyond the capabilities of the Express edition, you can then upgrade to a more robust version with the appropriate IT infrastructure and support.

Tip: Although ReliaSoft cannot provide full documentation and support for third-party database platforms, we do provide a limited number of resources as a convenience for users who wish to explore the possibilities of an enterprise database implementation. For resources on SQL Server (including instructions for configuring the Express edition), see http://Synthesis.ReliaSoft.com/sql_server.htm. For resources on Oracle, see http://Synthesis.ReliaSoft.com/oracle.htm.

The following topics will be of interest to database administrators:

- Creating a new enterprise repository on an established database server - Section 2.1.2.1 (p. 30).
- Three ways that Synthesis user accounts may be recognized by SQL Server (individual logins, group logins or Windows impersonation) - Section 2.1.2.3 (p. 32).
- Upgrading an enterprise repository from Version 8 - Section 2.1.2.5 (p. 35).

The following topics will be of interest to all enterprise repository users:

- Creating and using an enterprise repository connection file (*.rserp) to connect to an existing repository - Section 2.1.2.2 (p. 30).
- Monitoring the enterprise repository connection speed - Section 2.1.2.4 (p. 35).

2.1.2.1 Creating a New Enterprise Repository

If your organization already has established a database server with Oracle or Microsoft SQL Server and you have the permissions necessary to create databases on the server, you can create a new Synthesis enterprise repository by choosing **File > Manage Repository > Repository Creation**.

You can choose to create the new database in either Oracle or Microsoft SQL Server.

If you choose **SQL Server**, you will be required to enter:

- **Server Name**: The name of the Microsoft SQL Server implementation where the new database will be created. Note that if you are using SQL Server Express, the server name is usually your login for that computer followed by \SQLEXPRESS (e.g., Username\SQLEXPRESS).
- **Database Name**: The name of the new database that will be created.

If you choose **Oracle**, you will be required to enter:

- **Port**, **Host** and **Service** identifiers for the Oracle server where the new database will be created.
- The **Schema** of the new Enterprise database.
- The **Password** for the new enterprise database schema.
- The administrative username and password for the Oracle server (entered in the **Admin Information** area).

For either server type, select the **Import from existing repository** check box if you want to import entire projects and other data from an existing database.

Click **OK** to create the database. The database will not open automatically; you must connect to it (see the next section).

2.1.2.2 Connecting to an Existing Enterprise Repository

If your organization already has created an enterprise database on Oracle or Microsoft SQL Server and you have an active user account, you can use an enterprise repository connection file (*.rserp) to connect with it. You can create this file yourself or use a file that has been created by someone else. Once this file is created, it can be used to connect to the database and to import from or export to the database.

Creating a Connection File

To create a connection file, choose **File > New > Create a new enterprise repository connection file**.

This command creates a Synthesis repository connection file (*.rserp) that is stored locally on your computer; the file contains all of the necessary information for connecting to the enterprise database.

Enter a name for the connection file, then choose the database type and version (i.e., Microsoft SQL Server 2005 or 2008, or Oracle 10 or 11).

- For SQL Server databases, enter the server name and database name. Select the **Use impersonation** check box if you want the new connection file to impersonate a Windows user account with a SQL Server login that can be shared by multiple users. This connection file can then be distributed to any user who does not have his/her own individual SQL Server login and is not part of an Active Directory group that has a login. *See Section 2.1.2.3 on page 32*.
- For Oracle databases, enter the port, host and service identifiers and the database schema. Your Oracle schema owner and password credentials are used for access to the database.

Using an *.rserp File to Connect to an Enterprise Repository

If you have used the connection file recently, choose **File > Recent** and then select the file in the Recent Repositories list.

Otherwise, choose **File > Open Repository** and then browse for the *.rserp file.

Tip: If you usually work with the same repository, you can configure the Synthesis applications to attempt to connect with it automatically each time you start the application. Choose **File > New** and then select the **Always start by connecting with the last repository** check box.

If you are unable to connect to an enterprise repository, you may be encountering any of the following issues:

- **No access to the repository**. If you have not been given access to the repository, you will see a message stating that your account in the repository is not active or not assigned to at least one security group. You will need assistance from someone who can create and update user accounts (*see Section 2.3.3 on page 50*).
- **Windows authentication failed**. If your Windows login (domain and username) is different from what was specified for your user account, you will see a message to connect using alternative credentials. You will need assistance from someone who can enable the use of alternative credentials for your user account (*see Section 2.3.3.2 on page 52*).
- **Cannot connect to the server or login failed**. Server-related issues may occur for several reasons, and you may see various messages pertaining to the situation. Common issues are:
  - You do not have a network connection or you may have entered the incorrect database name (i.e., Synthesis repository name) or server name in the connection file.
  - The server may not be configured to allow remote connections, has certain firewall settings or is experiencing other issues. In this case, you will need to contact the IT support group responsible for the server.
  - In SQL Server databases, a login issue may occur if your username is not associated with a SQL Server Login. *See Section 2.1.2.3 on page 32*.

2.1.2.3 SQL Server Logins or Using Windows Impersonation

Connecting with a SQL Server database via Windows authentication requires a "SQL Server login" that allows the database platform to recognize the user and gives access to the Synthesis repository. There are three ways that a Synthesis user account may be recognized by SQL Server:

- **Individual Login**: The user has an individual SQL Server login that is associated directly with his/her Windows username.
- **Group Login**: The user belongs to an Active Directory group that has a SQL Server login shared by all members of the group.
- **Use Impersonation for Connection File**: The user does not have an individual or group login but he/she connects to the database with an enterprise connection file (*.rserp) that impersonates another Windows user account that does have a SQL Server login.

Your organization may choose to use any or all of these methods for your Synthesis implementation (e.g., some users may have their own individual logins, while other users connect using Windows identity impersonation). The following sections provide overviews of all three options.

Option 1: Creating Individual SQL Server Logins

If you choose to create individual SQL Server logins for some or all of the Synthesis user accounts, you have two options:

1. A database administrator for SQL Server can create SQL Server logins in advance for every potential user and give the logins access to the application database (at least the db_datareader and db_datawriter roles are required). This would be performed directly in SQL Server (not via one of the Synthesis applications).
2. A database administrator for SQL Server can grant the appropriate level of database authority for creating SQL Server logins and database roles (e.g., securityadmin or sysadmin) to any user who has the ability to create user accounts in the Synthesis repository. The additional authority would be added directly in SQL Server. Then, when any of these administrative users creates a new user account via the Synthesis application, the required SQL Server login can be created and the application database roles can be assigned automatically at the same time.

The web page at http://www.ReliaSoft.com/synthesis/sql_server.htm provides links to instructions for performing certain tasks in SQL Server. This includes a link to an FAQ that discusses these two options in more detail and provides specific instructions for the actions that must be performed in the SQL Server Management Studio.

If you are using the first approach, you can clear the **Create SQL Server login** check box that is displayed when you are adding or importing a user account. If you are using the second approach, you must select this check box.

Tip: If the user already has a SQL Server login and access to the application database, it does not matter whether you select or clear the **Create SQL Server login** check box because the application attempts to create the login only if one does not already exist.

Furthermore, if the user who is creating the user account does not have the necessary level of database authority in SQL Server, the login will not be created even if the check box is selected.

Option 2: Using a Group Login

If the user belongs to an Active Directory group that has a SQL Server login shared by all members of the group and that group has access to the application database, you can clear the **Create SQL Server login** check box that is displayed when you are adding or importing a user account.

For example, base installations of Microsoft SQL Server Express 2005 and 2008 include the "Builtin\Users" Active Directory group as a SQL Server login by default. This means all users with a Windows account for that domain will be able to log in to the enterprise database with no need to create individual SQL Server logins in SQL Server Express. However, it will still be necessary to grant access for this group login to the application database (at least the db_datareader and db_datawriter roles are required).

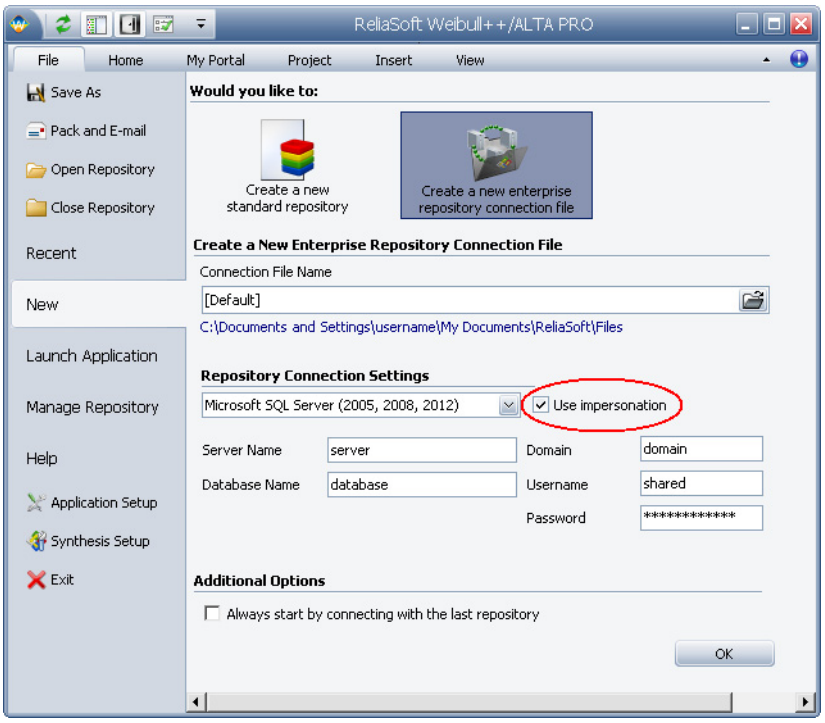
Option 3: Using Windows Impersonation for the Connection File

If you choose to have some (or all) users connect to the SQL Server database with a connection file that impersonates a shared Windows user account that has a SQL Server login, you must do the following:

- A Windows network administrator must establish the shared user account on Windows.
- A database administrator for SQL Server must create a SQL Server login for the shared Windows user account and grant this login access to the application database (at least the db_datareader and db_datawriter roles are required).

- A Synthesis user must create an enterprise repository connection file that impersonates the shared Windows user account:

  1. Choose **File > New** then click **Create a new enterprise repository connection file**.
  2. Under **Repository Connection Settings**, select **Microsoft SQL Server (2005, 2008)** from the drop-down list and then select the **Use impersonation** check box.
  3. Enter the server and database name for the SQL Server database, then enter the domain, username and password for the shared Windows account that users will need to impersonate.
  4. Click **OK** to create the connection file (*.rserp). It will be stored in the location specified under **Connection File Name**. Note that the default filename will be "SQL_(Server Name)_(Database Name)," but you can assign any name that fits the process your organization will use for distributing the file to users. (Note that while the window shown next is for Weibull++, the settings are the same for all Synthesis applications.)



Once you have created a connection file that impersonates the shared Windows user account, you can distribute the file to any Synthesis user who needs it. To connect to the repository using this file, the user can:

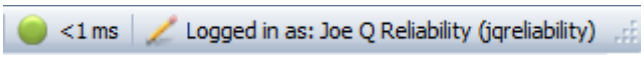
1. Choose **File > Open Repository** and browse for the connection file.
2. Click **Open** to connect with the repository.

After the first connection, this *.rserp file will be saved in the list of recent repositories, which can be accessed by choosing **File > Recent**.

Note: For the purpose of being recognized by SQL Server and accessing the application database, the user will be impersonating the shared Windows login. For the purpose of performing actions via the Synthesis applications, the user's actions will be governed by his/her own user account in Synthesis. In other words, multiple users can connect with the database using the same enterprise connection file, but their activities within the Synthesis applications will be governed by the permissions established in their own individual Synthesis user accounts, and any changes made to the analysis data will be recorded in Synthesis under their own usernames.

2.1.2.4 Enterprise Repository Connection Speed

If the Synthesis application is exhibiting slow performance when you're connected to an enterprise repository, the issue may be related to your network connection. The MDI status bar displays a connection speed indicator.



The following icons are used to indicate the connection speed. In cases of poor network performance, you will need to request assistance from the IT support group responsible for the server.

- < 30 ms: Good (acceptable performance)
- 30 - 70 ms: OK (may exhibit some delays in operations, opening/closing windows, etc.)
- 70 - 110 ms: Slow (will exhibit some delays in operations, opening/closing windows, etc.)
- 110 - 150 ms: Very slow (will exhibit significant delays in operations, opening/closing windows, etc.)
- > 150 ms: Extremely slow (will result in unacceptable performance and usability)

2.1.2.5 Upgrading an Enterprise Repository from Version 8

If you have an existing enterprise database that was created with any of the Version 8 Synthesis applications, an administrative user can transfer all of the existing data into a new database that is compatible with the Version 9 applications.

To open the Upgrade Version 8 Enterprise Repository window from any Version 9 application, choose **File > Manage Repository > Upgrade Repository**.

Synthesis 8 Repository to Be Upgraded

Use the left side of the window to specify the Version 8 enterprise database that you wish to upgrade. Note that you must have Admin level permissions in this database in order to be able to transfer all of the data. If you do not have sufficient permissions in the selected database, a message will be displayed when you attempt to perform the upgrade process.

Synthesis 9 Repository to Be Created

Use the right side of the window to define the new Version 9 enterprise database that the existing data will be transferred into. All of the same requirements for creating a new enterprise database apply here (e.g., you must have an established database server and the necessary permissions to create a database on the server).

You can choose to create the new database in either Oracle or Microsoft SQL Server, even if the existing Version 8 database resides in the other database platform. (In other words, it is possible to upgrade a database from SQL Server to Oracle, or from Oracle to SQL Server, if desired.)

If you choose **SQL Server**, you will be required to enter:

- **Server Name**: The name of the Microsoft SQL Server implementation where the new database will be created. Note that if you are using SQL Server Express, the server name is usually your login for that computer followed by \SQLEXPRESS (e.g., Username\SQLEXPRESS).
- **Database Name**: The name of the new database that will be created.

If you choose **Oracle**, you will be required to enter:

- **Port**, **Host** and **Service** identifiers for the Oracle server where the new database will be created.
- The **Schema** of the new Enterprise database.
- The **Password** for the new enterprise database schema.
- The administrative username and password for the Oracle server (entered in the **Admin Information** area).

Click **OK** to start the upgrade. When the process completes, the prior Version 8 database will remain unchanged and there will be a new Version 9 database that contains all of the existing data.

IMPORTANT for SQL Server Databases: The username must be associated with a "SQL Server Login" that allows the database platform to recognize the user and give access to the application database. This can be accomplished with an individual login, a group login or Windows impersonation. *See Section 2.1.2.3 on page 32*.

2.2 Projects

In Synthesis repositories, projects serve as a way to keep related analyses together. Each project can contain analyses (e.g., folios, diagrams, plots, FMEAs, etc.), resources that can be shared between analyses (e.g., models, tasks, actions, etc.) and attachments.

Any project can be opened in any Synthesis application; however, application-specific analyses (e.g., folios in Weibull++, diagrams in BlockSim, etc.) are visible only in the application(s) that can edit them.

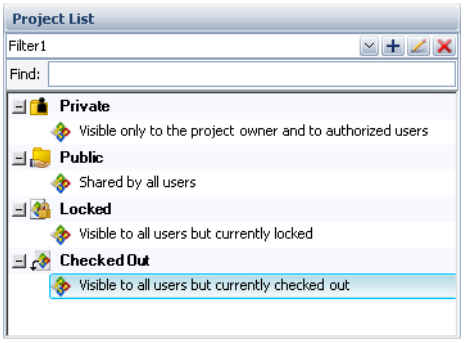
The following sections describe:

- How to create and manage projects - Section 2.2.1 (p. 37).
- Using project properties to view/edit a project's name, category, security settings (for secure databases) and other settings - Section 2.2.2 (p. 38).
- Assigning an owner to a project - Section 2.2.3 (p. 40).
- Working with private and public projects - Section 2.2.4 (p. 41).
- Working with locked and unlocked projects - Section 2.2.5 (p. 42).
- Working with checked in and checked out projects - Section 2.2.6 (p. 42).
- Using the Manage Projects window to perform administrative tasks on several projects - Section 2.2.7 (p. 44).
- Using the recycle bin to temporarily store deleted projects - Section 2.2.8 (p. 45).

2.2.1 Creating and Managing Projects

There are two interfaces you can use to view and manage the projects in a database:

- The project list (**View > Project Manager > Show Project Manager**) provides a list of the projects that are accessible to you (based on your permissions in the database and any security that has been defined at the project level). There are two types of projects you can create: private or public. In addition, a project may be locked or checked out at any given time. Each type of project is displayed under the appropriate heading.



- The Manage Projects window (**Project > Management > Manage Projects**) allows you to perform administrative tasks for multiple projects all at once. In a secure database, only users with the "Manage all projects" permission can access this window.

Creating a New Project

To add a new project to an existing database, go to the project list, right-click the heading of the desired project type (i.e., private or public), and then choose the **Create Project** command on the shortcut menu. Alternatively, you can select the heading and choose **Project > Management > Create Project**.

The Project Properties window opens, allowing you to specify the project name, category, security settings (if applicable) and other settings. *See Section 2.2.2 on page 38*.

Tip: If you later need to change the project type, select the project and choose **Project > Security >** [**Make Private** or **Make Public**]. The same commands are also available in the Manage Projects window.

Duplicating Projects

To duplicate an existing project, select it and choose **Project > Management > Duplicate Project**.

This command will create an exact duplicate of the original project with a name that contains an increment number (e.g., Project_1, Project_2, etc.). Note that duplicate projects do not retain the security settings of the original project.

Deleting Projects

To delete a project, select it and press **DELETE** or choose **Project > Management > Delete Project**.

To make sure that your analysis information is not deleted by mistake, you will always be prompted to confirm before proceeding. By default, the project will be moved to the recycle bin, which will give you a chance to recover it later if needed. If you want to permanently delete the project now (no way to undo unless you have a backup or restore point), clear the **Send project to recycle bin** check box in the confirmation window.

2.2.2 Project Properties

The Project Properties window will be displayed when you create a new project or when you choose **Project > Management > Edit Project Properties**.

The options available in this window will vary depending on which application you are using and whether the database has login security enabled. For MPC, see Section 2.2.2.1 on page 39. For all other applications, this may include:

- General Tab
- Global Identifiers Tab
- Security Tab (only if the database is secure and the project is public)
- Configurable Settings Tab (only in Xfmea/RCM++/RBI)

In addition, the status bar at the bottom of the window always displays the following information:

- The user who has been identified as the project owner. In a non-secure database, all users are able to edit the properties for any project. However, in a secure database, the ability to edit the project properties is restricted to the project owner and to users with the "Manage all projects" permission.
- The time and date the project was last changed/updated, and the user who made the change. In Version 9, this now considers any change to any of the analysis data in the project (not just the project properties). This same information is also visible in the Manage Projects window (*see Section 2.2.7 on page 44*), which provides an easy way to determine which projects have been updated recently.

General Tab

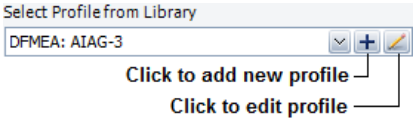
The General tab is available for all Synthesis applications except MPC. It always contains the following options:

- **Name** is the identifier for the project that will appear in the current project explorer and many other windows and reports throughout the software. This field is required and cannot contain any of the following characters: \ / : " * ? < > |.
- **Description** and **Remarks** can contain more detailed notes or information about the project.
- **Proprietary Label** can contain a copyright or distribution statement, if desired. In Xfmea/RCM++/RBI, this label appears in the footer of reports. In MPC, it is displayed in the footer of Standard reports but is not applicable for the Sukhoi or Dassault templates.
- **Project Category** displays a list of the categories that have been defined in the current database. These categories are configurable to meet your particular needs and they can be used to filter the projects displayed in the project list and in many other windows throughout the software. *See Section 2.4.5 on page 67*.

In Xfmea/RCM++/RBI, this tab also contains:

- **FMEA Structure** determines how the software will display the effect and cause records in the FMEA hierarchy. This is applicable only in Xfmea/RCM++/RBI. (See "Choosing the FMEA Structure" in the Xfmea, RCM++ or RBI documentation.)
- **Select Profile from Library** displays a list of all profiles that have been defined in the active library. This is applicable only in Xfmea/RCM++/RBI. When you choose a profile from the list, all of the configurable settings for the project will be set/reset based on the settings that have been predefined for that profile. If you are editing a

project, the current profile is shown in brackets. For users who are authorized to edit the active library, two additional icons will be enabled to the right of this drop-down list, as shown next.



Global Identifiers Tab

The Global Identifiers tab is available for all Synthesis applications except MPC. Identifiers are user-defined fields that can be used to search for certain types of records within the current project. The default names of the identifiers are defined at the database level via the Global Identifiers window; this tab allows you to specify different settings at the project levels. *See Section 2.4.4 on page 66*..

Security Tab

The Security tab is available only when you are working with a public project in a secure database. *For a full discussion on how to use these settings, see Section 2.3.2 on page 47*.

For quick access to the Security tab, you can select the project in the project list and choose **Project > Security > Project Security**.

Configurable Settings Tab

The Configurable Settings tab is available for Xfmea/RCM++/RBI only. It provides access to all of the configurable settings that have been defined for the current project, based on the profile that was selected on the General tab.

This tab allows you to make specific changes that will apply to the current project only. For example, if you are using a predefined profile but want to make a change to the interface style for this particular project only, you could click the **Edit** icon associated with the **Interface Style** drop-down list in the **Interface Settings** area.

Note: For RCM++ and RBI only, in the **Tasks** area, if you have chosen a task selection logic that presents different questions depending on the failure effect categorization, then you must choose a failure effect categorization (FEC) logic with the same number of categories. If the logics do not match, a message will be displayed when you attempt to save the project properties.

It is important to note that any change to the Configurable Settings page will update the settings for the current project, but it will not alter the original profile. If you want the project's current settings to be available as a new profile in the active library, click the **Send Settings to Library** button and then type the name and description for the new profile that will be created.

2.2.2.1 MPC's Project Properties

For users of MPC, the interface has been customized to display only the information that is directly relevant for MSG-3 analyses. This includes all of the same basic properties that are available on the General tab in other applications. In addition, you will be able to define the following:

- **Short Description** appears in the header of the Standard report template. It is not displayed in the Dassault or Sukhoi templates.
- **MSG-3 Guidelines** displays a list of the available MSG-3 guidelines that can be used for the systems and powerplant analysis. In most cases, the version that you select will not have any impact on the MSG-3 logic displayed in the interface and print-ready reports. The only substantive difference occurs when you select MSG-3

Revision 2002.1 because this older version of the guidelines uses slightly different titles for two of the maintenance significant item (MSI) questions, and lists the questions in a different order.

- **Model/Equipment/Effectivity** allows you to enter the information that will appear on the title page of the Dassault and Sukhoi templates.

2.2.3 Project Owner

For each project in a Synthesis repository, one user will be assigned as the *project owner*. By default, the owner will be the user who created the project, but this can be changed when needed.

In a secure database, being the project owner means that you have full permissions over the project. This includes the ability to edit the project properties, add/edit/delete project items and resources, lock and unlock, set security settings, create restore points and delete the project. These permissions are always in effect regardless of the project security settings or item permissions that may be in place. *For more information on security settings, see Section 2.3.2 on page 47*.

Changing the Project Owner

In a non-secure database, any user can assign a different user to be the owner for a selected project. In a secure database, only users with the "Manage all projects" permission can change the project owner.

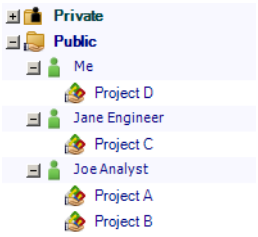
To change the owner, select the project in the project list and choose **Project > Security > Change Owner**.

If you want to change the owner for multiple projects simultaneously, use the Manage Projects window (*see Section 2.2.7 on page 44*).

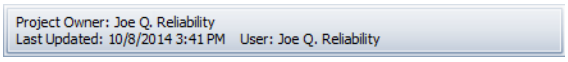
Identifying the Project Owner

There are three ways to identify the current owner of a project:

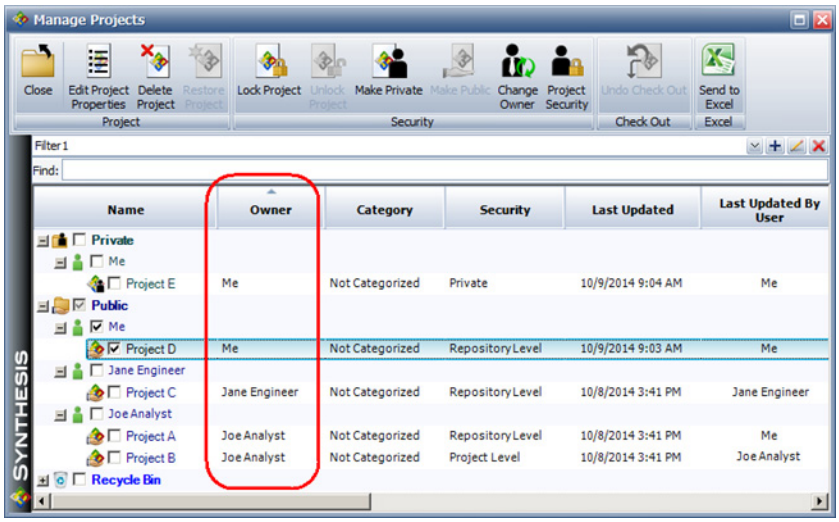
- Use the filters in the project list to filter and/or group projects based on the project owner. For example:



- The Project Properties window (**Project > Management > Edit Project Properties**) displays the name of the current owner in the status bar at the bottom of the window.



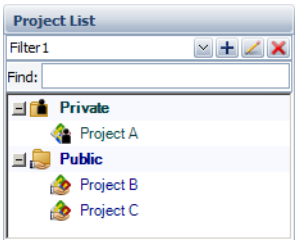
- The Manage Projects window (**Project > Management > Manage Projects**) displays a list of all projects and their owners in a table format. You can use custom filters to filter, sort and/or group the list based on the project owner. (In a secure database, only users with the "Manage all projects" permission can access this window.)



2.2.4 Private and Public Projects

In Synthesis repositories, there are two types of projects you can create: a private or public project. In a non-secure database, both types of projects are accessible to any user who has access to the database. In a secure database, a private project is accessible only to the project owner and to users with the "Manage all projects" permission. No other user can view or edit the project. On the other hand, public projects may be accessible to any user who has access to the database (depending on the security settings that have been implemented). *For more information on security settings, see Section 2.3.2 on page 47.*

In the project list, each type of project is displayed under the appropriate heading, as shown in the following example.



Creating New Private or Public Projects

To add a new project to an existing database, go to the project list, right-click the heading of the desired project type (i.e., private or public), and then choose the **Create Project** command on the shortcut menu. Alternatively, you can select the heading and choose **Project > Management > Create Project**.

The Project Properties window opens, allowing you to specify the project name, category, security settings (if applicable) and other settings. *See Section 2.2.2 on page 38.*

Changing the Project Type

To change the project type, select the project and choose **Project > Security** and then select one of the following options:

Make Private

Make Public

In addition, you can select multiple projects and change them to private or public projects all at once via the Manage Projects window (*see Section 2.2.7 on page 44*). In a secure database, only users with the "Manage all projects" permission can access this window.

2.2.5 Locked and Unlocked Projects

In all Synthesis repositories, a public project can be *locked* to prevent users from editing the data. If a project is locked, it will be moved from the **Public** heading to the **Locked** heading in the project list and it cannot be edited by any database user unless an authorized user unlocks it again.

In a secure database, a user can lock or unlock a project if he/she a) is the project owner, b) has the "Lock or check out project" permission for the project or c) has the "Manage all projects" permission.

Locking a Project

To lock a project, select it in the project list and choose **Project > Security > Lock Project**.

When a project is locked, all database users (including the user who locked the project) will have read-only access to the project. In addition, a locked project cannot be deleted or have its properties and public/private status edited.

Unlocking a Project

To unlock a project, select it in the project list and choose **Project > Security > Unlock Project**.

Tip: You can select multiple projects and lock or unlock them all at once via the Manage Projects window (*see Section 2.2.7 on page 44*). In a secure database, only users with the "Manage all projects" permission can access this window.

2.2.6 Check In and Check Out Projects

In all Synthesis repositories, both private and public projects can be *checked out* to temporarily allow one particular user to work on a project independently for a period of time and/or to work on a computer that is not connected to the network where the shared database resides (e.g., if you need to work on the project while you're out of the office). When a project is checked out, two things happen:

- An editable copy of the project will be saved into a standard database (*.rsr9) in the specified "Checked Out" folder on your local computer. This database will have the same name as the project.
- The original project will be set to read-only and moved under the **Checked Out** heading in the project list. This allows other users who would normally be able to access the project to see that it is currently being edited

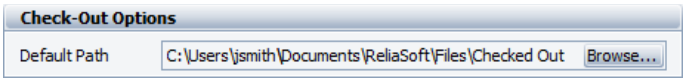
exclusively by one particular user. It also provides read-only access to the latest version of the project at the time it was checked out so those users can query, copy data, etc. while you're editing the project "offline."

You can then edit the project as needed in the standard database and check the project back in when you're finished.

Specifying the "Checked Out" Folder

By default, projects will be checked out to the default Documents folder on your computer (e.g., My Documents\ReliaSoft\Files\Checked Out).

To change this location, open the Synthesis Setup window (**File > Synthesis Setup**) and browse for a new default path under **Check-Out Options**. Keep in mind that anyone who has access to the folder will have full access to the checked out project; however, only the user who has the project checked out will have the ability to check in the project.



Tip: If you need to work on the project from a different computer after it has been checked out to the local folder, you will simply need to make sure that the latest copy of the standard database file is saved back to the exact same check-out location and filename before you check the project back in.

Checking Out a Project

In a secure database, a user can check out a project if he/she a) is the project owner, b) has the "Lock or check out project" permission for the project or c) has the "Manage all projects" permission.

To check out a project, select it in the project list then choose **Project > Management > Check In/Out > Check Out**.

Checking in a Project

To check in a project, select it under the **Checked Out** heading in the project list and choose **Project > Management > Check In/Out > Check In**.

Note that when you check in a project, the software creates a restore point, which is an exact replica (i.e., a backup) of the project before it was checked out. This allows you to restore the project to its prior state when and if needed. The restore point will include a description of the user who checked out the project, as well as the date and time of the check out. *For a discussion on how to use restore points, see Section 2.5.1 on page 71.*

Undo Check Out

In a secure database, a user can undo a check out if he/she a) is the user who checked out the project, b) is the project owner or c) has the "Manage all projects" permission.

To discard changes and restore the project to the state it was in before it was checked out, select the project under the **Checked Out** heading in the project list and choose **Project > Management > Check In/Out > Undo Check Out**.

2.2.7 Manage Projects

The Manage Projects window allows you to perform administrative tasks for multiple projects all at once. In a non-secure database, any user who has access to the database can access the Manage Projects window. In secure databases, only users with the "Manage all projects" permission can access this window.

To open the Manage Project window, choose **Project > Management > Manage Projects**.

This displays a list of all the projects that currently exist for the database. If you have created custom filters in the project list, you can apply those same filters in this window.

Tasks You Can Perform

Some of the tasks you can perform in this window include:

- Change the owner or type (private or public) for multiple projects all at once.
- Apply the same changes to the project properties for multiple projects all at once.
- Review and edit the security settings (if applicable) for multiple projects all at once.
- Delete, restore, lock and unlock, or undo check out for multiple projects all at once.
- Review and sort projects based on the user who made the last change to a project, the date/time of the last change and the last application that was used to make the change.

Tip: The Manage Projects window allows you to edit the properties of a project that is currently open or in use. When you make a change, the user who has the project open will see the change you have made when the database refreshes (the refresh happens automatically whenever the user makes a change, such as closing a window, selecting a different item, etc.).

Tools

The Manage Project window contains the following commands:

Project

Close closes the Manage Projects window.

Edit Project Properties allows you to view and edit the properties of the selected project(s). *See Section 2.2.2 on page 38*.

Delete Project permanently deletes the selected project(s) and bypasses the recycle bin. ***There is no undo for delete*** unless you have a stored backup (*Section 2.5 on page 69*) or restore point (*Section 2.5.1 on page 71*).

Restore Project is available only when you have selected project(s) under the Recycle Bin heading. Each selected project will be recovered from the recycle bin and restored to its original location in the project list.

Security

Lock Project moves the selected project(s) into the Locked heading of the project list. When a project is locked, all database users (including the user who locked the project) will have read-only access to the project.

In addition, a locked project cannot be deleted or have its properties and public/private status edited. To unlock project(s), choose **Unlock Project**. *See Section 2.2.5 on page 42*.

Make Private moves the selected project(s) into the Private heading of the project list. To move private project(s) to the Public heading, choose **Make Public**. *See Section 2.2.4 on page 41*.

Change Owner is available only for public projects in secure databases. It allows you to assign a different database user to be the owner for the selected project(s). *See Section 2.2.3 on page 40*.

Project Security is available only for public projects in secure databases. It opens the Project Properties window with the Security tab active, where you can specify the user accounts that can view/modify the selected project. *For a detailed discussion on how to configure the security setting for the projects in your database, see Section 2.3.2 on page 47*.

Check Out

Undo Check Out discards all changes made to a checked out project and restores it to the state it was at the time it was checked out. *See Section 2.2.6 on page 42*.

Excel

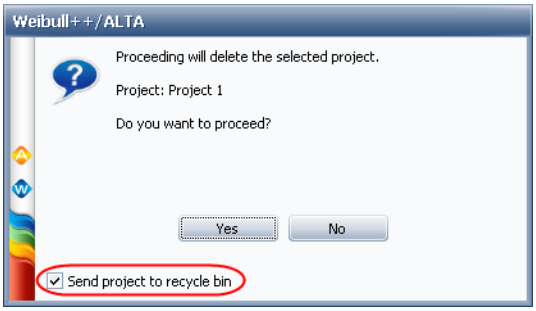
Send to Excel exports the data currently displayed in the Manage Projects window to an Excel spreadsheet.

2.2.8 Recycle Bin

The recycle bin is a temporary storage location for projects that have been deleted. It gives you a chance to restore the deleted project to its original location, if needed. Projects in the recycle bin are stored until you either empty the recycle bin or delete each individual project from the recycle bin.

Sending Projects to the Recycle Bin

Whenever you delete a project from a database, a confirmation window like the one shown next will appear.



If you want to move the project to the recycle bin, make sure the **Send project to recycle bin** check box is selected. If you want to immediately and permanently delete the project (no way to undo unless you have a backup or restore point), clear the check box.

Restoring and Deleting Projects from the Recycle Bin

In order to restore or delete a project from the recycle bin, you must have access to the project and have the permissions that are required to delete the project in general (i.e., for secure databases, you must either be the project

owner, have the project-level “Delete project” permission or have the repository-level “Manage all projects” permission).

To restore a project from the recycle bin, select it and choose **Project > Management > Recycle Bin > Restore Project**.

To permanently delete a project in the recycle bin (no way to undo unless you have a backup or restore point), select it in the recycle bin and choose **Project > Management > Recycle Bin > Delete Project**.

To permanently delete all projects currently in the recycle bin (no way to undo unless you have a backup or restore point), select the Recycle Bin header in the project list and choose **Project > Management > Recycle Bin > Empty Recycle Bin**.

2.3 Security Options

All enterprise databases use *login security*, meaning that the Synthesis applications use Windows authentication (or alternative credentials) to identify each user and control the user’s access via *security groups*. For a standard database, you can choose whether to apply login security; if you don’t, any user who has read/write access to the file will have full permissions throughout the database.

By default, each user in a secure database will have the same set of permissions for every public project in the database (e.g., Jane Engineer has read-write access to all projects, Bill User has read-only access to all projects, and so on). Alternatively, you can configure the database to provide different permissions for different public projects, if desired (e.g., Jane Engineer has read-write access to all of Department A’s projects, but she has read-only access to other projects). With either approach, you also have the option to further limit access for specific project items (e.g., folios, diagrams, system hierarchy items, etc.) if needed.

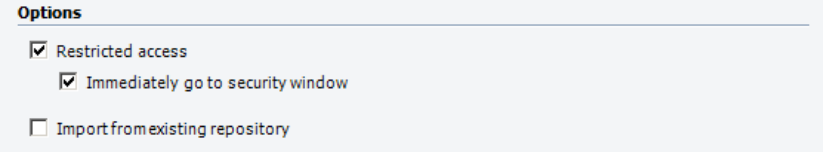
There are many different ways these options can be configured depending on your organization’s particular needs. The following sections discuss:

- Applying login security for a standard database - Section 2.3.1 (p. 47).
- Planning your security approach - Section 2.3.2 (p. 47).
- Managing user accounts - Section 2.3.3 (p. 50), which may also involve:
  - Importing users from Active Directory - Section 2.3.3.1 (p. 51).
  - Creating alternative credentials (if the user needs to log in from a different domain) - Section 2.3.3.2 (p. 52).
  - Editing user login and contact information - Section 2.3.3.3 (p. 53).
- Managing security groups (sets of permissions) - Section 2.3.4 (p. 54).
- Setting item permissions - Section 2.3.5 (p. 58).
- Tools for coordinating access by multiple users, including:
  - Status indicators (“in use” or “restricted access”) - Section 2.3.6 (p. 59).
  - Resetting “in use” flags (if an item is incorrectly locked for editing) - Section 2.3.7 (p. 61).
  - Keeping track of prior logins - Section 2.3.8 (p. 61).

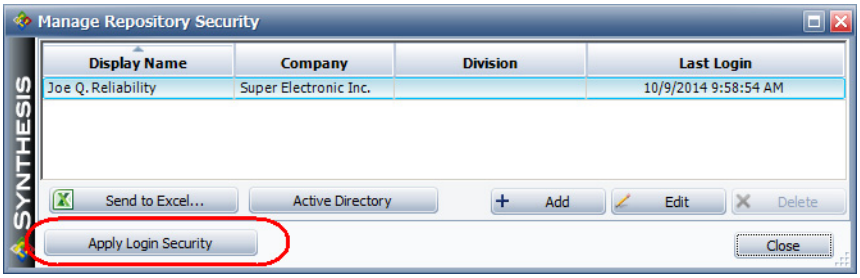
2.3.1 Applying Login Security

All enterprise databases use login security. For a standard database, there are two ways to apply login security if desired:

- Upon creating a new standard database (**File > New**), you can make it login secure by selecting the **Restricted access** check box.



- For an existing standard database, you can apply login security at any time by choosing **File > Manage Repository > Authorized Users** and then clicking the **Apply Login Security** button at the lower left corner of the window.



When you create an enterprise database or apply login security to a standard database, you will automatically be a member of the Admin security group, which has full permissions throughout the database. You can then use the Manage Repository Security window to add/edit/delete other user accounts (*see Section 2.3.3 on page 50*) and assign them to appropriate security groups (*see Section 2.3.4 on page 54*).

Note: You cannot automatically remove security from a database once it has been enabled. However, you can create a new non-secure database and use the **Import from existing repository** check box to automatically import all of the data from the secure database to the non-secure one.

2.3.2 Planning Your Security Approach

In secure databases, there are two basic factors that determine what a typical user can see and do in the database: the security group(s) that the user account belongs to and public project security settings.

This section discusses two general approaches you can use to configure the security groups and project security settings to fit your organization's specific needs:

- Same permissions for all public projects
- Different permissions for different public projects

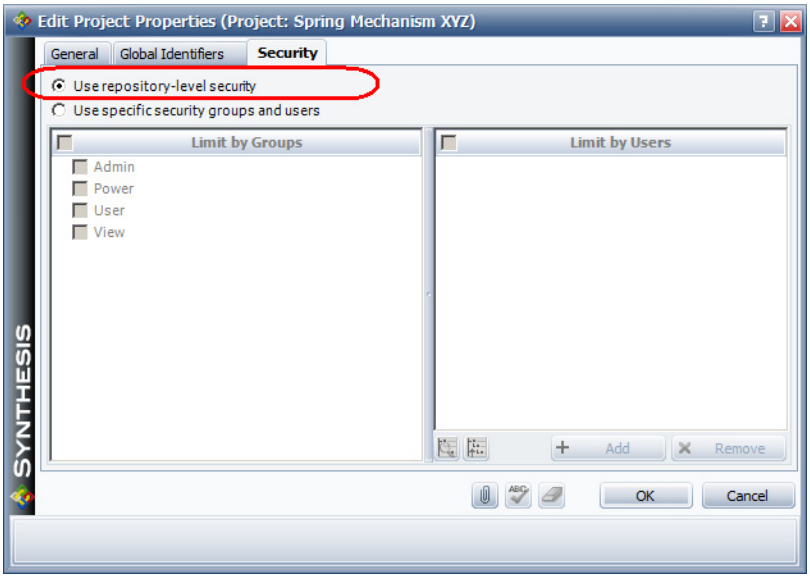
Tip: In addition to these considerations, it is also important to note the following: a) Users with the "Manage all projects" permission (in any of the security groups that they belong to) will always have full project-level permissions for all public projects in the database; b) The project owner will always have full project-level permissions within that particular project; and c) The item permissions can be used to further limit access to specific items within a project (e.g., folios, diagrams, system hierarchy items, etc.). *See Section 2.3.5 on page 58.*

Same Permissions for All Public Projects

If you want each user to have the same set of permissions for all public projects in the database (e.g., Jane Engineer has read-write access to all projects, Bill User has read-only access to all projects, and so on), follow these steps:

1. Assign each user account to an appropriate security group (*see Section 2.3.4 on page 54*). This can be one of the four security groups that are created by default in each new Synthesis Repository — Admin, Power, User (Read/Write) or View (Read-Only) — or you can configure new or existing security groups to meet your particular needs.

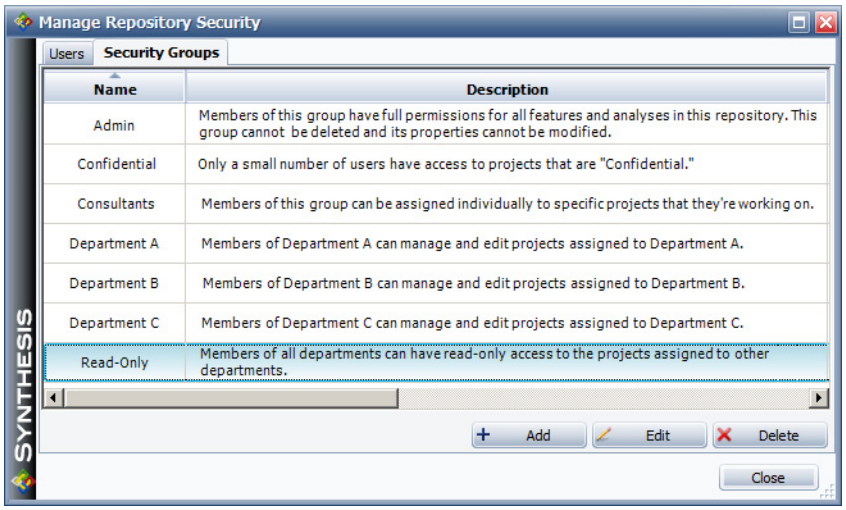
2. Accept the default option on the **Security** tab of the Project Properties for all public projects (**Project > Security > Project Security**). Note that if a user belongs to more than one security group, he/she will have the combined permissions of those groups in any project that is set to repository-level security.



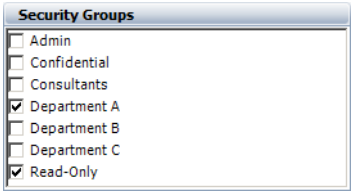
Different Permissions for Different Public Projects

If you want the same user to have different permissions for different public projects (e.g., Jane Engineer has read-write access to all of Department A's projects, but she has read-only access to other projects), you will need to:

1. Create a security group for each distinct type of access that users might need in any particular public project. Here's a simple example:



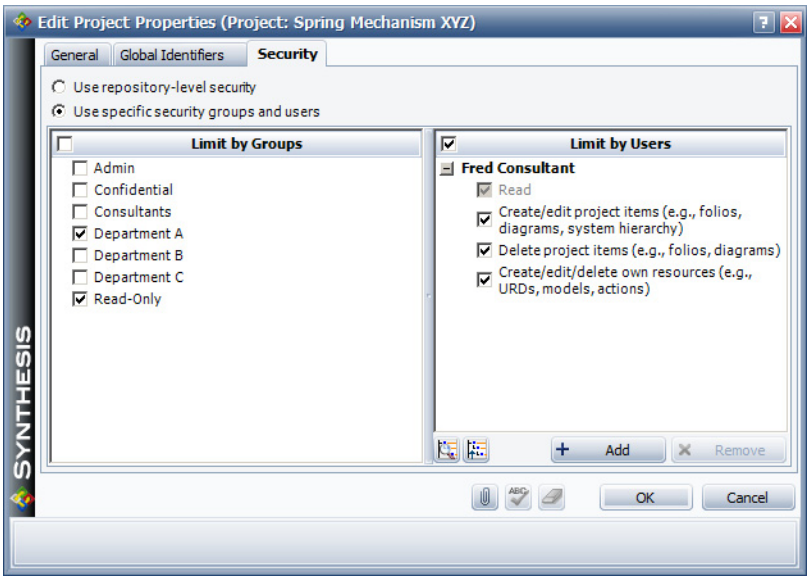
2. Assign the appropriate security group(s) to each user account. For the example shown below, the user will have read/write permissions in projects that are assigned to "Department A," and read-only access in projects that are assigned to "Read-Only."



3. Assign the appropriate security group(s) and/or specific user(s) for every public project in the database.
   - When you assign a security group, every user who belongs to that group will be able to access the project with the permissions that are specified in the group.
   - When you assign a specific user, the combined permissions from all of the security groups that the user belongs to will be displayed. Use the check boxes to select which of those permissions will be in effect for that user in this particular project.

For the example shown below, users from Department A will have read/write access (because they belong to the "Department A" security group), users from Departments B and C will have read-only access (because they belong to

the “Read-Only” security group) and Fred Consultant will have read/write access (because he belongs to the “Consultants” security group and has been specifically assigned to have those permissions in this project).



2.3.3 Managing User Accounts

Every person who will access a Synthesis repository or receive notification e-mails must have a personalized user account in the database.

- In **non-secure databases**, the software automatically creates an account for anyone who opens the file. Every user has full permissions throughout the database, including the ability to create, edit or delete other user accounts (e.g., so you can modify contact information or send notifications to someone who has not yet had an account created automatically).
- In **secure databases**, the accounts must be created and managed by users with the “Manage users and logins” permission. The security group(s) assigned to each account determines what the user can see and do in the database.

To view and manage the user accounts in a database, choose **File > Manage Repository > Authorized Users**.

This section describes how to use the Manage Repository Security window to create, edit or delete/deactivate user accounts. The same window can also be used to apply login security for a standard database (*see Section 2.3.1 on page 47*) and to manage security groups (permissions) for a secure database (*see Section 2.3.4 on page 54*).

Creating and Editing User Accounts

The Users tab of the Manage Repository Security window displays a table of all user accounts that have been created in the database. You can use the **Add** or **Edit** buttons below the table to create or modify individual accounts.

You can also import user accounts from Microsoft Active Directory® by clicking the **Active Directory** button. (*see Section 2.3.3.1 on page 51*).

Keep in mind the following requirements when creating new user accounts for a secure database:

- In order to use Windows authentication, the user must be logged in to a computer with the same domain/username that is defined in the Synthesis user account. If a user needs to connect to a database from a different domain, you can set up alternative credentials that will allow access without domain authentication. *See Section 2.3.3.2 on page 52*.
- In order to access the database, the user account must be assigned to at least one security group. *See Section 2.3.4 on page 54*. (If you simply wish to send notifications to the user, a security group is not required.)
- For SQL Server databases, the username must be associated with a "SQL Server Login" that allows the database platform to recognize the user and give access to the application database. This can be accomplished with an individual login, a group login or Windows impersonation. *See Section 2.1.2.3 on page 32*.

Deleting or Deactivating User Accounts

If an account has never been used and you want to permanently remove it, select the account name from the list and click **Delete**.

In a secure database, you also have the option to *deactivate* the account instead. This can be used if the account has already been logged in at least once, or if you only want to temporarily suspend access. If you attempt to delete an account that has previously been logged in, you will be prompted to deactivate it instead. If you want to manually deactivate an account:

- Select the account name from the list and click **Edit**.
- In the User Login and Contact Information window, clear the **Active** check box at the bottom of the window and click **OK**.
- When you return to the Manage Repository Security window, select the **Show active users only** check box to hide the accounts that are currently inactive.

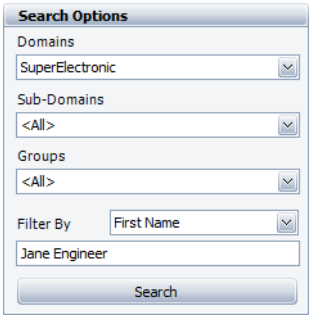
To reactivate the account again later, return to the User Login and Contact Information window and select the **Active** check box.

2.3.3.1 Importing Users from Active Directory

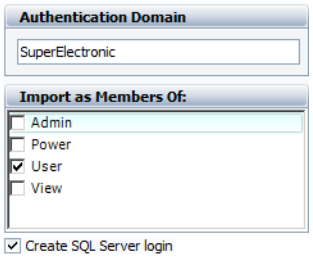
If your organization uses Microsoft Active Directory, it is easy to import username and contact information from the directory to create new user accounts in the Synthesis repository.

To import users from Active Directory, open the Manage Repository Security window (**File > Manage Repository > Authorized Users**), click the **Active Directory** button, and then follow these steps:

1. Use the **Search Options** to find the user(s) you want to create accounts for. When you enter the criteria and click **Search**, the utility will return only records that a) have the minimum information required to create a Synthesis user account (i.e., username, first name, last name and e-mail address) and b) do not already have a user account in the current database.



2. In the table of results, select the check box for each user you want to create an account for.
3. Use the remaining control panel options to set certain properties for the new user account(s) that will be created. This will vary depending on the type of database you're using. The following picture shows all possible options.



   - In the **Authentication Domain** field, specify the network domain name (i.e., the domain that the user's computer will be logged in to).
   - For secure databases only, use the **Import as Members Of** field to select the security group(s) that will be assigned to the new user account(s). If you skip this option now, you can assign security groups later for each individual account. *See Section 2.3.4 on page 54*.
   - For SQL Server databases only, select the **Create SQL Server login** check box if you want to create an individual SQL Server login for each new Synthesis user account. *See Section 2.1.2.3 on page 32*.
4. Click **Import** to create the account(s). After the process completes, the window will remain open to allow you to import additional users, if desired.

2.3.3.2 Creating Alternative Credentials

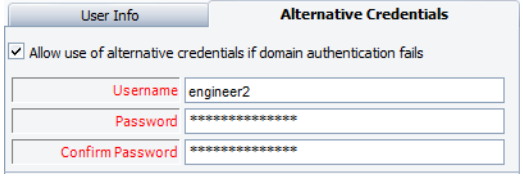
Secure databases use Windows authentication by default. This means that users must be logged in to a computer using the same domain and username specified for their Synthesis user accounts. You can, however, allow specific users to bypass Windows authentication as needed. This may be useful for situations where:

- A user needs to work on a copy of a secure standard database on a computer that is not connected to the company's network.
- A user needs to connect to an enterprise database from a different domain.

For these situations, you can set up alternative credentials for a user's account that will allow that user to access the database without domain authentication. These credentials must be set up in advance, before the user attempts the connection.

If you have the "Manage users and logins" permission, you can create alternative credentials for a user account by following these steps:

1. Open the Manage Repository Security window (**File > Manage Repository > Authorized Users**). On the Users tab, select the account name from the list and click **Edit**.
2. In the User Login and Contact Information window, click the **Alternative Credentials** tab and then select the check box to enable alternative credentials.
3. With the setting enabled, create an alternative username and password for the account. The username and password must be unique within the database, and the password is CaSe SeNsiTiVe.



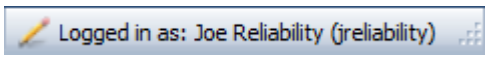
Once you have created the credentials, instruct the user to enter this login whenever the database prompts for it. The database will ask for the alternative credentials whenever the user account cannot be matched based on domain authentication.

Tip: The first time you connect to a database using alternative credentials, the application will automatically remember the login information on your computer. If you wish to clear the saved login, you can click the **Forget Alternative Login** button on the Synthesis Setup window (**File > Synthesis Setup**).

2.3.3.3 User Login and Contact Information

The User Login and Contact Information window allows you to view and edit the contact information for your own user account. If you have the "Manage users and logins" permission, you can also edit the information for any user account.

To view or edit your own user account, double-click your user name in the MDI status bar or choose **My Portal > Users > My Profile**.



or

In the User Login and Contact Information window, you can edit the following information:

- **User information**. At minimum, you must supply your first name, last name, display name and e-mail address. Your display name will appear in all projects, analyses and plots you create or modify in the database. Click the **Active Directory** button if you wish to update your contact details based on information stored in Microsoft Active Directory for your account.
- **User image**. You can associate a profile photo with your user account. Saved images are resized to 100 x 100 pixels.

If you have the "Manage users and logins" permission, you can also edit the following information for any user account. To edit an account, open the Manage Repository Security window (**File > Manage Repository > Authorized Users**) and then double-click the account name.

- **Domain and Username**. Windows authentication uses an account's domain and username to identify the user and give him/her access to the database.
- **Security groups(s)**. Security groups control the user's access to the database. *See Section 2.3.4 on page 54*.
- **Alternative credentials**. Select the check box on the Alternative Credentials tab to allow the user to bypass Windows authentication and connect to the database using an alternative username and password. *See Section 2.3.3.2 on page 52*.
- **Deactivate** or **Activate** the account. Clear or select the **Active** check box to deactivate or activate the account. A deactivated account will not have access to the database.

2.3.4 Managing Security Groups

In a secure database, security groups control what users can see and do in the database. By default, the software includes four predefined security groups: Admin, Power, User and View. The Admin group, which has full permissions throughout the database, can neither be deleted nor have its permissions modified. For the other predefined groups, you can edit their permissions or replace them with new groups that fit the specific way the database will be used. As discussed in Section 2.3.2 on page 47, there are two basic approaches you can use:

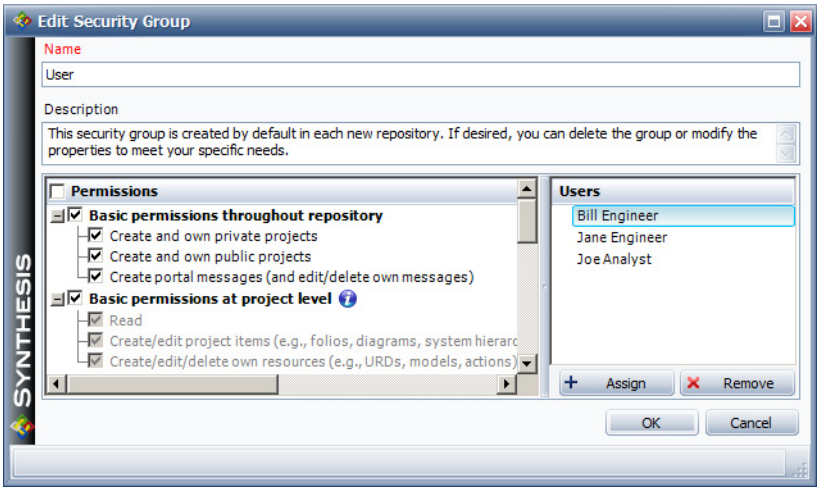
- **Same permissions for all public projects** – each user account is assigned to one security group and all public projects use the default security option (repository-level security).
- **Different permissions for different public projects** – each user account may be assigned to multiple security groups and each public project may be accessed only by specific security groups and/or users.

Creating, Editing or Deleting Security Groups

If you have the "Manage users and logins" permission in the database, you can manage the security groups by choosing **File > Manage Repository > Authorized Users**.

In the Manage Repository Security window, click the **Security** tab to display a table of all the security groups that have been created in the database. Use the **Add**, **Edit** or **Delete** buttons below the table to manage the groups.

When you edit a security group, the left side of the Security Group window allows you to choose the permissions, while the right side shows all the users currently assigned to the group, as shown in the following example. Use the **Assign** or **Remove** buttons to modify the list of users.



2.3.4.1 Synthesis Permissions

Here is a summary of all the permissions that can be granted to a particular security group.

Basic permissions throughout repository

These permissions apply throughout the database if they are in any of the security groups that the user belongs to. These permissions do not depend on the security settings for a particular project.

Create and own private projects	You can create and own *private* projects in the database that are accessible only to you and to users with the "Manage all projects" permission.
Create and own public projects	You can create and own *public* projects that other users can view and edit (depending on the project security settings).
Create portal messages	You can create and send messages to other users via the Messages page in My Portal.

Basic and advanced permissions at project level

A user can have these permissions in some projects but not others, depending on the project security settings (*see Section 2.3.2 on page 47*). Regardless of the project security settings, these permissions are always automatically granted to the current project owner and anyone with the "Manage all projects" permission.

Read	You can perform tasks that do not modify the data in the project (e.g., view the analysis, calculate metrics in a Quick Calculation Pad, export data, etc.).
Create/edit project items	You can create and edit items in a given project, such as folios in Weibull++/ALTA, diagrams in BlockSim, system hierarchy items in Xfmea/RCM++/RBI, and the like.
Create/edit/delete own resources	You can create Synthesis resources (e.g., URDs, models, etc.) and edit or delete any existing resources you have created.

Delete project items	You can delete any item in a given project (e.g., folios in Weibull++/ALTA, diagrams in BlockSim, system hierarchy items in Xfmea/RCM++/RBI, and the like). This permission cannot be assigned unless you also have the "Create/edit project items" permission.
Create/edit/delete all resources	You can create, edit and delete any local resources in the project (not just the ones that you created).
Set project security	You can control who can view and edit a given project. This permission allows you to configure both project security settings (*see Section 2.3.2 on page 47*) and item permissions (*see Section 2.3.5 on page 58*).
Edit project properties	You can use the Project Properties window to edit the name, description, category and other settings of a given project. *See Section 2.2.2 on page 38*.

Lock or check out project	You can: ● Lock and unlock a given project. *See Section 2.2.5 on page 42*. ● Check in and check out a given project. *See Section 2.2.6 on page 42*.
Create restore points	You can utilize restore points for a given project, which are exact replicas of the project at a particular point in time (i.e., backups). *See Section 2.5.1 on page 71*.
Delete project	You can delete a given project.
Manage change logs in Xfmea/ RCM++	You can enable and manage change logs within a given project. Change logs can be created for FMEAs and DVP&R analyses in Xfmea, RCM++ and RBI.
Approve change logs in Xfmea/ RCM++	You can implement electronic approval tracking for change logs within a given project. Change logs can be created for FMEAs and DVP&R analyses in Xfmea, RCM++ and RBI.

Administrative permissions throughout repository

These permissions apply throughout the database if they are in any of the security groups that the user belongs to. These permissions do not depend on the security settings for a particular project.

Manage all projects	You have all the basic and advanced project-level permissions for all projects in the database. This overrides any project security settings and item permissions. This permission also gives you the ability to change the project owner for any given project and the ability to use the Manage Projects window (*see Section 2.2.7 on page 44*).
Manage users and logins	You can: ● Use the Manage Repository Security window to add user accounts to the database and define security groups. *See Section 2.3.3 on page 50*. ● Use the Prior Logins window to view and export a history of database logins. *See Section 2.3.8 on page 61*. ● Use the Reset “In Use” Flags window to clear the “in use” status for selected database users. *See Section 2.3.7 on page 61*.
Manage e-mail notifications	You can: ● Use the E-mail Notifications window to enable e-mail notifications sent by the software and control when they are sent. *See Section 2.4.1 on page 62*. ● Use the Notification Groups window to add and manage the distribution groups for the e-mail notifications. *See Section 2.4.1.1 on page 63*.

Manage profiles and templates in Xfmea/RCM++	This permission is available only in enterprise databases. You can: ● Use the Profiles/Library Manager window in Xfmea/RCM++/RBI to configure the predefined settings stored in the active library of the enterprise database. ● Use the Templates Manager window in Xfmea/RCM++/RBI to configure report templates stored in the enterprise database.
Manage other repository settings	You can: ● Use the Manage Units window to define the time units available for use in any project within the database. *See Section 2.4.2 on page 64*. ● Use the Manage Warranty Units window in Weibull++ to set equivalencies between the time units used throughout the database and the time units used in some warranty folios. ● Use the Define Default Names window to specify how default names for resources and blocks are created. *See Section 2.4.3 on page 65*. ● Use the Task Types window in RCM++ and RBI to map the task types used in RCM++/RBI to the task classes in the universal reliability definition (URD). ● Use the Global Identifiers window to define the default names for identifier fields that can be used to search for blocks or resources within a given project. *See Section 2.4.4 on page 66*. ● Use the Project Categories window to define the categories that can be used for grouping and filtering the projects in the database. *See Section 2.4.5 on page 67*. ● Use the Restore Points window to manage restore points, which are exact replicas of the project at a particular point in time (i.e., backups) that can be restored when and if needed. *See Section 2.5.1 on page 71*.
Approve actions	You can review and approve actions, which are Synthesis resources that allow you to track progress made in a project. *See page 103 in Chapter 3*.
Manage all portal messages	You can edit or delete all messages you sent and messages addressed to you via the Messages page in My Portal.
Create/edit/delete global resources	You can create, edit and delete any global resources in the project (not just the ones that you created). In addition, you can transfer data from XFRACAS to the RDW (*see Section 2.7.1 on page 88*).

Manage Lambda Predict repository settings	You can: ● Use the Manage FIDES Settings window in Lambda Predict to configure the FIDES-related settings stored in the database. This applies to the FIDES prediction standard only. ● Use the Manage Custom Derating Standards window in Lambda Predict to create, edit or delete user-defined derating standards stored in the database. ● Use the MIL-217 Custom Connections window in Lambda Predict to define failure rates for user-defined connectors stored in the database. This applies to the MIL-217 prediction standard only.
Manage dashboard templates	You can create and save layouts for use in the Dashboard Viewer in Weibull++, BlockSim and RGA. *See Section 2.7.4 on page 92*.

2.3.5 Setting Item Permissions

In a secure database, all project items (e.g., folios, diagrams, system hierarchy items, etc.) are, by default, set to inherit their permissions from the project. This means that if a user has the "Create/edit items" permission for the project, he or she will be able to edit all of the items by default. If you want to prevent some or all of those users from editing a particular item (i.e., lock or limit access to the item), you can use the Item Permissions window.

To set item permissions, you must either a) be the project owner, b) have the "Set project security" permission and/or c) have the "Manage all projects" permission.

The window functions a little differently depending on whether you're working with items in a project explorer (e.g., folios, diagrams, etc.) or items in a system hierarchy.

Project Explorer Items

In Weibull++, ALTA, BlockSim, RENO, DOE++, RGA or Lambda Predict, select the item in the current project explorer and choose **Project > Current Item > Item Permissions**.

In the Item Permissions window, select the **Restrict editing to selected project users** option and then select which specific users will be able to edit this particular item. All others can view, but not edit, the item. Only users with editing rights to the project will appear in the list.

To remove item permissions and change the security settings back to the defaults, select the **Inherit from project** option.

System Hierarchy Items

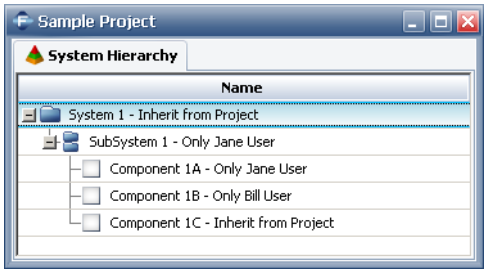
In Xfmea, RCM++ or RBI, select the item and choose **System Hierarchy > Security > Item Permissions**. In MPC, select an item and choose [**Systems**/**Structures**/**Zones**] **> Security > Item Permissions**.

In the Item Permissions window, you can choose to:

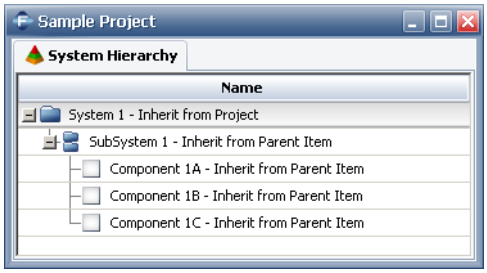
- Define the permissions for the item by selecting the **Restrict editing to selected project users** option and then selecting which specific users will be able to edit this particular item. All others can view, but not edit, the item. Only users with editing rights to the project will appear in the list.

- Set the item to inherit the same permissions as the next item above it in the system hierarchy by selecting the **Inherit from parent item** option.
- Remove item permissions and change the security settings back to the defaults by selecting the **Inherit from project** option.

If you select the **Apply to all dependents** check box, the software will automatically set the permissions for all next-level items in this branch of the hierarchy to match the item you are currently editing. For example, consider a case where each item in an Xfmea system hierarchy has different permissions, as shown next.



For a quick way to reset all of the item permissions to the defaults, you can open the Item Permissions window for SubSystem 1, set to **Inherit from parent item** and select the **Apply to all dependents** check box. The resulting system hierarchy is shown next.

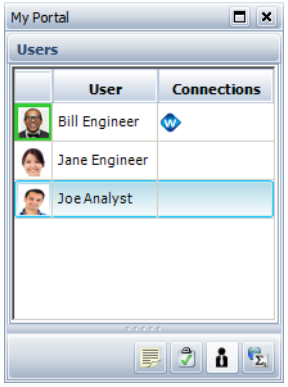


2.3.6 Status Indicators

Synthesis applications include status indicators that tell you which user accounts are currently logged in to the database and which project items (e.g., folios, diagrams, system hierarchy items, etc.) are in use or restricted.

Database Users

The names of all users who have accounts in the database are displayed on the Users page of the My Portal panel (**View > My Portal > Show My Portal**). A green status light around the user's profile image indicates that the user is currently connected to the database, as shown in the following example. The Connections column shows which Synthesis application(s) are currently connected.



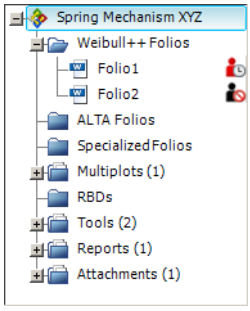
You can refresh the list of users by choosing **View > Refresh > Refresh**.

Project Items In Use or Restricted

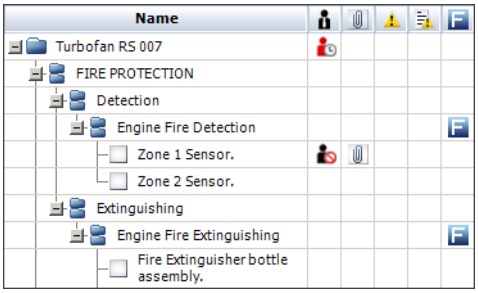
When you encounter an item that is read-only (e.g., OK button is disabled in the window or you are unable to type in the cells of a worksheet or table), you can check the status of the item to see if it is currently **in use** (i.e., you can't edit the item until the user closes the folio/diagram or selects a different item in the hierarchy) or **restricted** (i.e., you don't have permissions to edit the item). If the item is in use, move the pointer over the icon to identify the user.

The location of these status icons will vary depending on which Synthesis application you are using:

- In projects that use a current project explorer to manage analysis data, the status icons are displayed next to the item's name in the current project explorer, as in the example shown next.



- In projects that use hierarchical trees to manage analysis data (as seen in the Xfmea, RCM++, RBI and MPC applications), the status icons are displayed in the User Access column of the system hierarchy. (To hide or display columns, right-click the column headers, then click **Customize Columns** to select which columns you want to display.)



Tip: The application interface refreshes automatically whenever you make a change (e.g., close a project, create a resource, select a different item, etc.). If your computer will not let you edit an item that is in fact not currently being edited by another user, it could be because your computer has not been recently refreshed with the latest changes made by other database users. You can initiate the refresh manually by choosing **View > Refresh**.

In addition, the software employs "in use" flags within the database to record when an analysis is currently being edited by a particular user. If the refresh still does not show that the item has been released, then something might have occurred to prevent the flags from being reset correctly (e.g., a network interruption or if the software closes unexpectedly). Refer to Section 2.3.7 on page 61 for instructions on how to reset the flags throughout the database if this problem occurs.

2.3.7 Reset “In Use” Flags

Because Synthesis repositories allow simultaneous access by multiple users, it is necessary to store “in use” flags within the database to indicate when a particular portion of the analysis is currently being edited by a particular user. There are some circumstances when these flags might not be reset correctly when a user stops editing an analysis (e.g., if there is a network interruption or if the software closes unexpectedly). If that occurs, then the analysis will be locked for editing because the software receives an erroneous indication that it is still in use by another user.

To correct the problem, it is necessary to reset some or all of the “in use” flags within the database. To do this, you will need to have the “Manage users and logins” permission.

To reset the flags, choose **File > Manage Repository > Reset ‘In Use’ Flags**.

This opens the Reset 'In Use' Flags window, which shows all users who have an account in the database. (If you are working in a non-secure database, any user who has ever opened the database will have an account created automatically and will be shown in this list.) A status light is displayed for each user; if it is “lit up” (i.e., green), the user is currently logged in to the database. In addition, the Connections column shows the Synthesis application(s) that the user currently has connected to the database. You can select the check box for each user for whom you want to reset “in use” flags, then click **OK** to reset the flags.

IMPORTANT: It is important to make sure that no selected user is currently logged in to the database when you use this command. Users currently logged into the database can be viewed on the Users page of My Portal. *See Section 2.3.6 on page 59*.

2.3.8 Prior Logins

The Repository Logins window displays a record of the date and time the users in the database logged in. You will need to have the “Manage users and logins” permission to access the record.

To open the Repository Logins window, choose **File > Manage Repository > Prior Logins**.

The following options are available:

- Use the **Most Recent** filter to display in the table the last 10; 100; 10,00 or 10,000 users who logged in to the database.
- Use the **User** filter to specify whether the table will display the logs for all users or display only the logs for a selected database user.
- Click the **Send to Excel** button to export the data currently displayed in the table to an Excel spreadsheet.
- Click the **Clear Logins** button to clear the entire history of users who logged in to the database. Since this action cannot be undone, you will be prompted to confirm that you want to proceed before the records are erased.

Users currently logged into the database can be viewed on the Users page of My Portal. *See Section 2.3.6 on page 59*.

2.4 Repository Settings

Synthesis applications include a variety of tools to help authorized users manage settings that are shared by all users and all analysis projects in the same database. The following topics describe some of these features that are applicable for all (or most) Synthesis applications. This includes:

- **Notifications**
  - E-mail notifications and notification groups keep users up to date on the progress made in completing assigned actions - Section 2.4.1 (p. 62).
- **Time Units**
  - The Manage Units window allows you to define the "repository time units" that will be available for use in any project within the database - Section 2.4.2 (p. 64).
- **Names, Identifiers and Categories**
  - The Define Default Name Formats window allows you to specify the format for the default names of newly created resources and other items - Section 2.4.3 (p. 65).
  - The Global Identifiers window allows you to define identifiers (e.g., part number, version, supplier, etc.) that can be used to search for resources - Section 2.4.4 (p. 66).
  - Project Categories can be used to filter the projects displayed in the project list - Section 2.4.5 (p. 67).
- **Integration with XFRACAS**
  - Setting up an XFRACAS Connection allows you to access XFRACAS data from a standard database. (For an enterprise database, the Synthesis and XFRACAS data will be stored in the same repository.) - Section 2.4.6 (p. 68)

For information about other repository settings that are available only in specific Synthesis applications, please consult the documentation for the application(s) where those features are used.

2.4.1 E-mail Settings

E-mail notifications can be an effective tool to facilitate communication between database users and to track the status of assigned tasks.

The E-mail Settings window allows authorized users to enable e-mail notifications within the current database for:

- Actions created in an FMEA, DFR plan or My Portal. These notifications can be sent manually. They can also be sent automatically based on the settings established in this window.
- Messages created in My Portal. These notifications can be sent manually.
- Change logs in Xfmea/RCM++/RBI that are assigned to be approved. These notifications will be sent automatically, if applicable.

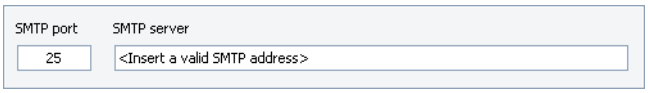
To access the window, choose **File > Manage Repository > E-mail Settings**.

(If you do not have the "Manage e-mail notifications" permission, this command will not be visible in the Backstage view.)

Enabling E-mail Notifications

To configure the database to enable e-mail notifications, do the following:

1. Select the **Enable Notification E-mails** check box.
2. Specify a valid **SMTP port** and **SMTP server**. (You may need to consult with the IT professionals who have configured the e-mail server used within your organization.)



3. Enter your e-mail address in the **Recipient address for test e-mail** field and then click **Send Test E-mail**. If the test e-mail cannot be sent, an error message will be displayed.

4. Once you have confirmed that the e-mail settings are valid, specify which automated notifications, if any, will be sent for action records:
   - **Upon creation**: Sends an e-mail to the Person Responsible and/or Approver when the action is first created.
   - **Upon modification**: Sends an e-mail to the Person Responsible and/or Approver each time an existing action is modified via the Action Properties window. Note that modifications applied via the Worksheet View in Xfmea/RCM++/RBI will not trigger an automated notification.
   - **Upon completion**: Sends an e-mail to the Approver when the completion date has been entered and the action is ready to be reviewed/approved.

   For all of these automated notifications, you can also include the user who created the action as a primary recipient by selecting the **Action creators receive notifications** check box in the E-mail Settings window. Additionally, if other users and/or notification groups have been defined in the Action Notifications window for a particular action, they will also receive courtesy copies (CCs). Note that courtesy copies will not be sent if there is not at least one primary recipient (Person Responsible, Approver and/or Creator) for the e-mail.
5. Use the **Introduction to each action notification e-mail** field to specify the default text (up to 1,000 characters) that will be used at the beginning of each action notification e-mail. The rest of the e-mail will include all of the details that have been defined for the particular action record.
6. Click **OK** to save these settings for the current database.

Saving Settings for Future Standard Databases

If you want to use the same e-mail settings for each new standard database that you create from the current computer/username, click **Set As Default**. This button is not available if you are working with an enterprise database.

2.4.1.1 Notification Groups

The Define Notifications Groups window allows authorized users to manage the notification groups that are available within the current database.

If a notification group has been assigned to a specific action or message, all of the users assigned to that group will:

- Be able to see the record in My Portal.
- Receive courtesy copies of automated action notification e-mails, if any.

To access the window, choose **File > Manage Repository > Notification Groups**.

(If you do not have the "Manage e-mail notifications" permission, this command will not be visible in the Backstage view.)

Adding or Editing a Group

The table displays a list of the notification groups that have already been defined in the current database.

- To add a new group, click **Add**.
- To view or edit an existing group, double-click the row or select the row and click **Edit**.

In the Notification Group window, a unique **Name** is required and a short **Description** of the intended purpose for the group is optional.

The **Available Users** list shows all of the user accounts in the current database that are not yet assigned to the group. At least one user account must be moved into the **Selected Users** list.

There are a variety of ways to move records between the lists:

- To move one record, double-click the name, or select the name and click > or <.
- To move multiple records at the same time, press the **CTRL** key and click each name, then click > or <.
- To move all records at the same time, click >> or <<.

Deleting a Group

To delete an existing group, select a row and click **Delete** or press the **DELETE** key.

If the notification group has been assigned to any existing actions and/or messages, it will be removed automatically. ***There is no undo for delete.***

2.4.2 Manage Units

Users with the "Manage other repository settings" permission can define the time units that will be available for use in any project within the database. This allows users to work with time-based inputs and results in the units that are appropriate for the situation. For example, you might have a data set in Weibull++ where times to failure were recorded in hours, but you want to obtain results from the QCP in terms of years. Similarly, you could accommodate a situation in RCM++, RBI or BlockSim where you need to define the duration of a maintenance task in terms of hours and to specify the total operating time of the system in terms of months. In MPC, using the predefined time units to set task intervals helps to ensure consistency and makes it possible to transfer data to Xfmea/RCM++/RBI if desired.

IMPORTANT: In is important to realize that making changes to the existing units can have implications for existing analyses throughout the database. Therefore, it is recommended to set up units once, upon creation of the database, if possible.

To define time units, choose **File > Manage Repository > Manage Units** to open the Define Units window.

The units are defined in relation to a common, or "base," unit so that the software can convert data from one unit to another. For example, the units defined by default assign a conversion factor of 1 to the Hour unit, which indicates that an hour is equal to 1 System Base Unit (SBU); the Year unit has a conversion factor of 8760, which indicates that a year is equal to 8,760 SBUs (i.e., 8.760 hours). Note, however, that the SBU does not have to be hours. You could consider days to be your base unit and define all other units in relation to a day (e.g., an hour would be 1/24th of a base unit, with a conversion factor of 0.04167).

Each unit must have a name, an abbreviation of up to three characters and a conversion factor. In addition, each unit must be assigned to any one of the two available categories: Time or Usage.

Note: If you will be using the usage format of the Weibull++ warranty folio, note that only the units assigned to the Usage category will be available for those analyses; units assigned in the Time category are not available in the warranty usage format.

You can set any unit as the default by selecting it in the Default column; the default unit will be used automatically each time the software requires a unit.

- To edit an existing unit, click inside the cell and edit the text.
- To add a new unit to the bottom of the list, click inside the last row and type the text.
- To delete a unit, select the row and click the **Delete** icon. ***There is no undo for delete***.

- To insert a new unit above an existing one, select the existing unit and click **Insert**.

- To move an existing unit up or down in the list, select the row and click the **Move Up** or **Move Down** icon.

2.4.3 Default Name Formats

When you add a resource to a project, it is given a default name unless you specify a different name. In all Synthesis applications other than MPC, users with the "Manage other repository settings" permission can specify how these default names are created in the current database via the Define Default Name Formats window, which is accessed by choosing **File > Manage Repository > Define Default Names**.

The Define Default Name Formats window consists of four tabs. The settings in the last three tabs will be used only for diagrams in BlockSim but they're visible when you open this window from any of the Synthesis applications.

- The **Resources** tab allows you to specify the default name formats for resources.
- The **RBD** tab allows you to specify the default name format for each type of block that you can add to a reliability block diagram in BlockSim. Note that this feature is not applicable for the blocks you create in a Weibull++ or ALTA diagram because those blocks are always named based on the calculated standard folio data sheet that they represent.
- The **Fault Tree** tab allows you to specify the default name format for each type of block (i.e., each type of gate or event) that you can add to a fault tree in BlockSim.
- The **Phase** tab allows you to specify the default name format for each type of block that you can add to a phase diagram in BlockSim.

All tabs work in the same way. Each row of the tab's table represents a type of resource or type of block. For each row:

- The **Resource Type** column displays the type of resource or block. This column cannot be edited.
- The **Resource Name** column displays the default name used for that type of resource or block (e.g., "Model" or "Block"). This will be the name represented by the \N code in the Name Format column described below. For example, if the Name Format for a model is set to \N \S (which is the default), then the default name for the first new model you create in the database will be "Model 1." If you change the Name Format to \T \N \D, then default name for a model that you create in BlockSim at a particular time will be "BKS Model 9/15/2011 2:29:23 PM." Click in the cell to edit the name; click outside the cell to accept your changes.
- The **Name Format** column allows you to specify the total default name format used for that type of resource or block, using the following elements:
  - \N returns the default name for the resource or block type, as specified in the Resource Name column.
  - \U returns the name of the user who created the resource or block.
  - \D returns the date when the resource or block was last modified.
  - \S adds a sequential number, or increment. For example, if the last URD created was URD 1, the next URD created will use 2 as the sequential number (i.e., URD 2).
  - \T returns the software tool used to create the resource or block.
  - \F returns the contents of each of the five identifier fields for the record (i.e., all global identifiers except the comments field). *See Section 2.4.4 on page 66*.
  - \G returns a summary of the properties of the resource or block.
  - \I inherits the name of the selected block when a resource is created from the Block Properties window. The default name for the resource type (i.e., \N) is also included. For example, if you create a new URD for Block 1 using the Block Properties window, the URD's name will be "Block 1 URD." When a resource is not created from the Block Properties window, \I returns the same format as \N (e.g., "URD," "URD_2," etc.).

2.4.4 Global Identifiers

In all Synthesis applications other than MPC, identifiers are user-defined fields that can be used to search for certain types of records within the current project. The names of the identifiers are defined at the database level; these identifiers will be used in all projects throughout the database unless different settings are specified at the project level. *See Section l on page 69*.

The Global Identifiers window, accessed by choosing **File > Manage Repository > Global Identifiers**, is used to define the identifiers at the database level.

For each type of record, six identifiers are available: five "user-defined fields" (UDFs) and one comments field. Outside of this window (e.g., in properties windows, the Resource Manager, filters, etc.), these fields are known by the display names defined within this window. By default, the fields have the following display names:

- UDF 1: Name
- UDF 2: Number
- UDF 3: Version
- UDF 4: Applies To
- UDF 5: Keywords

- Comments: Comments

Not all of the identifier sets are applicable for all Synthesis applications. For example, tasks are used only in BlockSim, RCM++ and RBI. The affected applications are listed in parentheses for each identifier set.

To change the display name for a field, click the relevant cell in the Property Display Name column. When you are finished making changes to the cell, click outside the cell to accept the changes. Display names can be up to 60 characters long. The identifier fields themselves can contain up to 1,000 characters.

The changes that you make here will apply to all new projects added to the database. If the database already contains projects, your changes will not affect the existing projects unless you select the **Apply to existing projects** option at the bottom of the window.

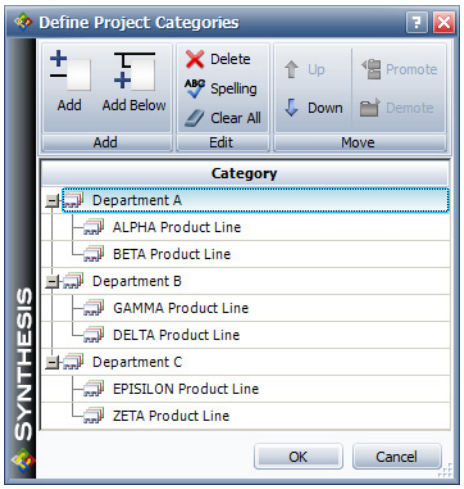
Tip: You can change identifiers at the project level on the Global Identifiers tab of the Project Properties window, which is accessed by choosing **Project > Management > Edit Project Properties**.

Selecting **Set as default for new databases** at the bottom of the window will save the settings that are currently displayed in the window as the default for all new standard databases that you create on your computer. If this check box is not selected, the default names given above (Name, Number, Version, etc.) will be used for new databases. This option is not applicable if you are working with an enterprise database.

2.4.5 Project Categories

Each Synthesis repository can contain a large number of analysis projects that may be accessed simultaneously by many different users. The flexible Project Categories feature allows you to group and manage the projects in a way that fits the specific needs of the users and the organization.

Just to give one simple example, the following picture shows a database that uses two category levels, where the projects are grouped first by department and then by product line. Of course, many other groupings could be defined depending on your particular objectives. You can add as many levels of indenture as needed.



This section describes how an authorized user can configure the categories for a particular database. These categories can then be applied to any project in the database (via the Project Properties window). They can also be used to filter any project in the list (via the Filter Properties window). For additional information, see http://Synthesis.ReliaSoft.com/tips/project_categories.htm.

Defining Project Categories

Users with the "Manage other repository settings" permission can access the Define Project Categories window by choosing **File > Manage Repository > Project Categories**.

This window displays a list of the project categories that have been defined for the current database, shown in the order in which they will appear throughout the application. If you rename or delete a category, any projects that are already assigned will be updated automatically (the next time the interface is refreshed).

- To edit an existing category, double-click inside the cell and edit the text.
- To add a new category in the same level as the one that is currently selected, click **Add**.

- To add a new category in the level below the one that is currently selected, click **Add Below**.

- To move a selected category up or down within the same level, select the row and click **Up** or **Down**.

- To move a selected category to a different level, click **Promote** or **Demote**.

- To delete a category, select the row and press the **Delete** key or click the icon. To delete all categories at the same time, click **Clear All**. ***There is no undo for delete***.

2.4.6 XFRACAS Connection

ReliaSoft's XFRACAS is a web-based, closed-loop, incident (failure) reporting, analysis and corrective action system designed for the acquisition, management and analysis of product quality and reliability data from multiple sources.

With the release of the Synthesis version, information in XFRACAS can be accessed via other Synthesis applications, allowing you to perform a variety of additional analyses on your XFRACAS data.

XFRACAS data is stored in a set of tables within a Synthesis enterprise repository. This means that when you are working with an enterprise database within other Synthesis applications, the XFRACAS data stored within the database is available. When you are working with a standard database, you can still work with XFRACAS data in any Synthesis application other than MPC. To do this, you simply specify where the XFRACAS tables of interest are located by choosing **File > Manage Repository > XFRACAS Connection**.

In the XFRACAS Connection Settings Window, you can specify that there are no associated XFRACAS tables for the current repository, or you can specify the connection settings for the enterprise repository that contains the XFRACAS tables of interest. Specifically:

- Choose the database type and version (i.e., Microsoft SQL Server 2005 or 2008, or Oracle 10 or 11).
- For SQL Server databases, enter the server name and database name. Select the **Use impersonation** check box if you want the connection settings to impersonate a Windows user account with a SQL Server login that can be shared by multiple users. This connection file can then be distributed to any user who does not have his/her own individual SQL Server login and is not part of an Active Directory group that has a login. *See Section 2.1.2.3 on page 32*.
- For Oracle databases, enter the port, host and service identifiers and the database schema. Your Windows login credentials are used for access to the database; enter your Windows password.

2.5 Backups and Database Maintenance: Protecting Your Data

Since each database may contain a large amount of valuable information that would be difficult to re-create, it is essential to make sure that you are diligent about storing adequate backups and performing the necessary maintenance activities to keep the database operating smoothly. The necessary procedures vary depending on the type of database.

Enterprise Database Maintenance

When you choose to store analysis information in an enterprise database, a database administrator must perform backups and database maintenance activities using the data management tools that are packaged with and/or designed for the database platform (e.g., SQL Server Enterprise Manager for SQL Server). Each individual organization typically establishes its own procedures for protecting the data stored in the Oracle or SQL Server databases. As a convenience for users who wish to explore the possibilities of an enterprise database implementation without making a large investment of time and resources, instructions on how to perform ReliaSoft's minimum database maintenance recommendations are posted on the ReliaSoft website:

See http://Synthesis.ReliaSoft.com/sql_server.htm for SQL Server.

See http://Synthesis.ReliaSoft.com/oracle.htm for Oracle.

Standard Database Maintenance

When you choose to store analysis information in a standard database, it will be subject to the same limitations and vulnerabilities as any other file that uses the Microsoft Access database file format. For example, the limitations (in terms of maximum file size and simultaneous users) are publicized by Microsoft at http://office.microsoft.com/en-us/access/HP051868081033.aspx and some of the database vulnerabilities are discussed in another Microsoft publication at http://support.microsoft.com/kb/283849/EN-US/. As this second publication states:

> "Microsoft Jet, the database engine that is used in Microsoft Access, is a file sharing database system. When Microsoft Jet is used in a multi-user environment, multiple client processes are using file read, write, and locking operations on a shared database. Because multiple client processes are reading and writing to the same database and because Jet does not use a transaction log (as do the more advanced database systems, such as SQL Server), **it is not possible to reliably prevent any and all database corruption**." [emphasis added]

Although ReliaSoft's developers have made every effort to reduce or eliminate the possibility that the software will induce a database error, there is no way to absolutely prevent corruption that might be caused by other factors, such as faulty network hardware, an unexpected "crash" on your PC or a network interruption. Therefore, this section provides some recommendations for standard precautions that all users can take to protect the data in their standard databases from this type of corruption and reduce the impact of the data loss if corruption is unavoidable.

1. **Create backups regularly**. As with any resource that contains a large amount of valuable information that would be difficult to re-create, it is essential to make sure that you are diligent about creating and storing backup files. There are a number of ways this can be accomplished:
   - If you select **Automatically back up database upon closing** from the Synthesis Setup, any Synthesis application that you use to open the database will back up the database every time you close the file.
   - If you have a database open and choose **File > Save As**, the application will create a copy of the database to a pathname/filename of your choosing.
   - If you browse to the database file (*.rsr9) in one of the Windows file management tools (such as My Computer or Windows Explorer), you can copy and paste the database file as needed.
2. **Compact and repair regularly**. Using the "Compact and Repair" feature will help to reduce the size of the database file and help to protect against problems with the operation of the database. If you have the database open and it is not currently in use by another user, you can initiate the process at any time by choosing **File > Manage Repository > Compact and Repair**.

3. Do not store the database in a shared network location if you suspect that your network connection and/or hardware may be unreliable. According to Microsoft, faulty network hardware is one of the main reasons why a file that uses the Microsoft Access database file format may become corrupted. As the Microsoft publication at http://support.microsoft.com/kb/283849/EN-US/ states:

   > "The cause can be one or more links in the hardware chain between the computer that the database resides on and the computer that has the database open. This list includes, but is not limited to, network interface cards, network cabling, routers, and hubs.
   >
   > Hardware-based corruption is typically indicated by .mdb files that cannot be restored through the use of compacting, repairing, or Jetcomp. **Hardware corruption will typically recur until the responsible hardware is repaired or replaced**." [emphasis added]

   If you have experienced this type of corruption for a standard database file, it is recommended that you take steps to correct the network problem or refrain from accessing database files over the network. In such cases, you may choose to use an enterprise database instead (i.e., Oracle or SQL Server), which would be less vulnerable to network interruptions. Alternatively, you could keep multiple analysis projects together in a single shared standard database file but ask users to export the analysis to a separate "working" database on their own computers when there is a need to make substantial modifications. Users could then import the data back into the shared database after the modifications have been completed.
4. **Do not allow the file size of the database to grow too large**. Performance will be affected by the size of the database and the number of simultaneous users. Therefore, it is important for users to monitor the sizes of their database files and take steps to export the data into several smaller and more manageable files if they become too large. Please be aware of the following factors, which can lead to very large database files:
   - Failure to compact and repair the database on a regular basis.
   - Using a very large number of attached documents. In some cases, using a link instead of an attachment may provide equivalent functionality with a much smaller impact on the size of the database file.

If you try to open a standard database via the software and receive a message that says "Unable to open the database," this is an indication that the database file may have become corrupted. Please contact ReliaSoft's Technical Support via phone, fax or e-mail and provide as much information as possible about exactly what you were doing when the corruption occurred. Whenever possible, please provide a copy of the corrupted file. In some cases, ReliaSoft may be able to provide assistance with salvaging some or all of the affected data. However, in many cases, the best recourse may be to restore the latest backup from before the corruption occurred.

2.5.1 Restore Points

One of the many data management capabilities offered by Synthesis is the option to create and manage restore points for any analysis project. Within the context of Synthesis, the phrase "restore point" refers to an exact replica of the project at a particular point in time (i.e., a backup) that can be restored when and if it is needed. A restore point will include all the data that the project contained at the time the restore point was created, including information about the project properties, security settings and project owner.

There are a variety of ways that this functionality could be used, depending on your particular analysis process and data management requirements. For example, if you are about to begin a major revision to an existing project, you could choose to archive the original version as a restore point and then proceed with updating the project. This would ensure that the active projects in the database contain only the most recent information but also provide easy access to a fully editable copy of the previous version of the analysis if it is ever needed.

The following sections describe how you can manage restore points.

What's Changed? In previous versions of the software, restore points were known as Project Baselines.

Create a Restore Point

If you are currently working in a project and you have any of the following permissions in the database:

- Project owner
- "Create restore points" permission
- "Manage other repository settings" permission

then you can create a restore point for the project at any time by choosing **Project > Management > Restore Point > Create Restore Point** or by right-clicking the name of the project in the project list and choosing the command on the shortcut menu.

IMPORTANT: When you create a restore point, any global resources used in the project are converted to local resources and stored with the backup. This ensures that you will have access to these resources upon restoration, regardless of what may have happened to the global resources in the interim.

Restore a Project

To restore all data from an existing restore point, use the **Restore Project** command. This opens the Restore Project window, which displays a list of all existing restore points for the project, if any. You have two options:

- **Overwrite existing project** allows you to roll back the current project to an earlier state. The restore process will complete as long as the project is not currently in use by another user. ***There is no undo for project overwrites***. Therefore, it would be prudent to create a new restore point for the project before you overwrite it with one of the older restore points.
- **Create new project** allows you to use the restore point to create a new project. You may enter a unique name for the new project. When the restore process completes, the new project will be accessible from the project list.

Manage all Restore Points that Exist for the Database

If you have the "Manage other repository settings" permission, you can manage all the restore points that have been created in the database by choosing **File > Manage Repository > Restore Points**.

This opens the Restore Points window, which displays the details for each of the restore points. This includes the date when the restore point was created, the name of the restore point creator, the project name and any additional notes that the restore point creator may have added to help identify the purpose or context of the restore point. In Xfmea/RCM++/RBI, the project profile that was used to set the configurable project settings is also displayed.

- To create a new restore point, click the **Create** button. The Create Restore Point window will appear, which displays a list of all projects in the current database. Choose a project from the list, enter any notes that are appropriate to describe the purpose or the circumstances of the restore point and click **OK**.
- To delete an existing restore point, select the row and click the **Delete** button or press **DELETE**. ***There is no undo for delete***.
- To create a new project that restores all of the data from an existing restore point, select the row and click the **Restore Project** button. You will be prompted to specify a unique name for the new project. When the process completes, you can close the Restore Points window and see that the new project is now accessible from the project list.

2.6 Import, Export and Data Conversion

Synthesis applications provide a variety of different tools for import, export and data conversion from external files and between Synthesis databases. The options will vary depending on which application you are currently using.

In all applications, you can:

- Import data from an existing database when you are creating a new one - Section 2.6.1 (p. 72)
- Convert and import data from prior version files - Section 2.6.2 (p. 73)
- Import/export selected projects - Section 2.6.3 (p. 82)

When applicable, you can also:

- Import/export selected project items or Synthesis resources - Section 2.6.4 (p. 82)
- Import from an Excel spreadsheet or delimited text file into a data folio - Section 2.6.5 (p. 84)
- Use XML files to import/export system configuration information in BlockSim or Lambda Predict - Section 2.6.6 (p. 87)

These features can be accessed either from the Import/Export Wizard (**Project > Management > Import/Export**) or from the Backstage View. *See page 130 in Chapter 4*.

In addition to the common functionality described in this chapter, some Synthesis applications provide other data transfer and import/export utilities that fit specific needs (e.g., the Import Bill of Materials feature in Lambda Predict, the Excel templates in Xfmea/RCM++/RBI, the ability to share analysis details between specific Synthesis applications.). For more information about these specialized tools, please consult the documentation for the particular application(s) involved.

2.6.1 Importing from an Existing Repository

When you are creating a new Synthesis repository, you have the option to import repository settings (e.g., user accounts, security groups, project categories, etc.) and/or entire analysis projects from another database.

If the **Import from existing repository** check box is selected when you start to create a new standard database or enterprise database, the Import Data from Existing Repository window will be displayed.

First, use the drop-down list or browse button (…) to select the database that you want to import from. This can be a standard database (*.rsr9) or an enterprise database connection file (*.rserp).

Then, use the check boxes to select the settings and/or projects you wish to import.

Finally, click **OK** to create the new database with the selected data imported.

Note: If you are creating a new enterprise database on SQL Server, the Create SQL Server login check box will be displayed at the bottom of the window. *See Section 2.1.2.3 on page 32*.

2.6.2 Importing from Prior Version Files

There are two ways to convert and import data from project files and standard database files that were created in previous versions of the software.

- Importing to a new standard database (**File > Open Repository**)
- Importing to an existing database and project (**Import > Other file**)

The options depend on which Synthesis application you're currently using:

- For Weibull++, ALTA, DOE++, RGA and BlockSim/RENO, you can use either method.
- For Lambda Predict, Xfmea/RCM++ and MPC, you must use the File > Open Repository method to import the data into a new project in a new standard database.

Finally, for an enterprise database created in Xfmea/RCM++ 5, there is a dedicated utility that you can access from the Backstage View. *See Section 2.6.2.4 on page 77*.

2.6.2.1 Importing to a New Standard Database (File > Open Repository)

For all Synthesis desktop applications, you can use the File > Open Repository method to import data from a project file or standard database that was created in a previous version of the software. This imports the data to a new project in a new standard database.

Do the following:

1. Choose **File > Open Repository**.
2. Browse for the file and click **Open**.
3. The conversion process will begin immediately for most applications. In BlockSim and MPC only, you will be prompted to specify some preferences and then click **OK** to proceed.
   - To convert from BlockSim 6 or 7 files, see Section 2.6.2.3 on page 74.
   - To convert from MPC 3 Databases, see Section 2.6.2.5 on page 79.

Note that although the conversion of Xfmea/RCM++ 5 databases does not require any user input, the process does apply some assumptions to address changes in the functionality and data structure between versions. For details, see Section 2.6.2.4 on page 77.

When the process completes, the original file will remain unchanged and the new standard database will be created in the same folder.

Tip: Once the new database has been created, you can use the Import/Export Projects feature to copy the data into an existing database, if desired. *See Section 2.6.3 on page 82*.

2.6.2.2 Importing to an Existing Database and Project (Import > Other file)

For the applications and file types shown in the following table, you can use the Import > Other file method to import data from a project file that was created in a previous version of the software. This imports the data to a selected project in an existing standard database or enterprise database.

Weibull++/ALTA	DOE++	RGA	BlockSim/RENO
ReliaSoft Office 7 projects (*.rso7)	ReliaSoft Office 7 projects (*.rso7)	RGA 7 projects (*.rga7)	BlockSim 7 projects (*.rbp)
Weibull++ 7 projects (*.rwp)	DOE++ 1 projects (*.rdoe)	RGA 6 projects (*.rga)	BlockSim 6 projects (*.rb6)
ALTA 7 projects (*.ralp)			RENO 1 projects (*.rnp)
Weibull++ 6 projects (*.rw6)			
ALTA 6 projects (*.ra6)			

Do the following:

1. Create a new project or open an existing project that you want to import the data into.
2. Choose **Project > Management > Import/Export > Import**.

3. In the Import wizard, choose **Other file** and click **OK**.
4. The conversion process will begin immediately for most applications. In BlockSim only, you will be prompted to specify some preferences and then click **OK** to proceed. *See Section 2.6.2.3 on page 74*.

Note that although the conversion of Xfmea/RCM++ 5 databases does not require any user input, the process does apply some assumptions to address changes in the functionality and data structure between versions. For details, see Section 2.6.2.4 on page 77.

When the process completes, the original file will remain unchanged and the converted data will be copied into the selected project.

2.6.2.3 Converting from BlockSim 6 or 7 Files

When you import from a BlockSim 6 (*.rb6) or BlockSim 7 (*.rbp) file, the Conversion Settings window allows you to specify:

- Whether you want the diagrams to be converted to analytical diagrams, simulation diagrams, or both.

- Whether the application will attempt to merge identical records when certain block properties are converted to Synthesis resources. For example, if the old diagram has two blocks with the same failure distribution, this can be imported as two separate but identical failure models, or as a single failure model that's linked from both blocks.

Tip: The default preferences for merging identical resources upon conversion are set on the Conversion page of BlockSim's Application Setup (**File > Application Setup**). Note that any changes you make in the Conversion Settings window will update your preferences in the Application Setup. In other words, the same options will be selected by the default the next time you attempt to convert a BlockSim 6 or 7 file on this computer.

Conversion Constraints and Differences

Numerous improvements have been made to the modeling capabilities and underlying analysis and simulation algorithms used in the Synthesis version of BlockSim. As a result, analysis or simulation of diagrams imported from previous versions of BlockSim may yield results that differ from the results originally obtained. In particular, results may differ in the following cases and/or for the following reasons:

- **Simulation diagrams**:
  - When multiple blocks are used in conjunction with subdiagrams, the underlying order of block expansion differs between versions, so results may differ.
  - In the Synthesis version, containers are treated as subdiagrams. The underlying order of block expansion differs between versions, so results may differ. However, some special cases will produce identical results. These include cases where:
    - The container is the only block in the diagram.
    - The container is at the end of the list in the original diagram (i.e., it was created last and has the highest block ID, which is automatically assigned by the software upon block creation).
  - The load on contained load sharing blocks is calculated differently. Version 7 required a life-stress relationship for such configurations and based the re-calculation of load after block failure on that relationship; the Synthesis version calculates load using the weight proportionality factor as a multiplier.
  - Normal/lognormal distributions have increased precision in the Synthesis version.
  - Maintenance tasks that are performed at specified intervals (based on item age or calendar time) for multiple blocks are performed in a different order, producing results that are not identical, although the difference will not be statistically significant.
  - Indirect cost is calculated differently. In version 7, the following was calculated at the end of the simulation:

    Indirect cost = Average stock level * Holding cost per item * Simulation end time

    In the Synthesis version, the indirect cost is calculated for each simulation and then averaged at the end to yield the indirect cost that is shown. This is because holding cost per item may be a distribution in the Synthesis version.
  - The order in which random numbers are assigned in general is by block ID. This means that if diagrams have the same blocks with the same IDs in both the previous version and the current version, the results will be identical. Standby containers represent an exception to this. In version 7, the order of blocks is overwritten by the standby priority. Therefore, results may not be identical with standby containers if the block IDs do not match the block standby priorities. The difference should not be statistically significant.
  - For mirrored blocks, in version 7, the block that fails is the source and all the other mirrors are assigned these failures. In the Synthesis version, the failures are assigned to the mirrored block with the lowest block ID. If the failure source is not the block with the lowest ID, then the results will be different between versions.

- Mirrored blocks inside subdiagrams are handled differently. In version 7, they were treated as different groups of mirrors; the Synthesis version treats them as the same group. This may produce results that are statistically different.
- Throughput is not available in fault trees in the Synthesis version.
- The throughput property **Send units to failed blocks** works differently for subdiagrams in the Synthesis version, in that it applies to the whole diagram. That is, if the current block is set to not send units to failed blocks and the next block is a subdiagram that is not operating, then the throughput will be re-routed if possible or the current block will accumulate backlog.
- Phases have different rules on how interrupted events are handled and may give different results when re-simulating.
- In the Synthesis version, containers do not exist in maintenance phases. Thus, the availability results for containers in phase simulation may be different than in previous versions.

- **Analytical diagrams**:
  - The load on contained load sharing blocks is calculated differently. Version 7 required a life-stress relationship for such configurations and based the re-calculation of load after block failure on that relationship; the Synthesis version calculates load using the weight proportionality factor as a multiplier.
  - Mirrored blocks inside subdiagrams are handled differently. In version 7, they were treated as different groups of mirrors; the Synthesis version treats them as the same group. This may produce results that are statistically different.
- **Other issues of interest when opening BlockSim 7 files**:
  - Nodes do not have failure/maintenance properties in the Synthesis version. Therefore, if a node has failure properties in BlockSim 7, it will be imported in the current version as two blocks: a node with the k-out-of-n and the throughput properties (if applicable) and a block with the failure properties positioned immediately after the node. If the node does not have failure properties, a second block will not be imported.
  - The load on contained load sharing blocks is calculated differently. Version 7 required a life-stress relationship for such configurations and based the re-calculation of load after block failure on that relationship; the Synthesis version calculates load using the weight proportionality factor as a multiplier. Because of this, if you convert a diagram that uses load sharing containers, you will need to manually configure the contained load sharing blocks after conversion.
  - Simulation FRED reports will be imported, but cannot be restarted or have levels appended/removed until the diagram is resimulated.

  In addition, certain rules apply when importing maintenance properties from BlockSim 7:
  - Preventive maintenance:
    - For each preventive maintenance setting, a new task will be created. For example, if the preventive maintenance policy is set to be performed upon system down and upon system age, two tasks will be created.
    - For each task created, the duration, crews, restoration factor, etc. will be identical.
  - Inspections:
    - For each inspection setting, a new task will be created. For example, if the inspection policy is set to be performed upon system down and upon system age, two tasks will be created.
    - For each task created, the duration, crews, restoration factor, etc. will be identical. Note that inspections do not use pools.
    - If a detection threshold is defined in version 7, an on condition task will be created instead of an inspection task in the Synthesis version. A threshold is defined in version 7 if a Failure Detection Threshold greater than 0 and less than 1 is specified or if a P-F Interval greater than 0 is specified.

- The inspection properties from version 7 will be transferred to the inspection properties of the on condition task in the Synthesis version.
- The threshold (i.e., detection) information from version 7 will be transferred to the failure detection properties of the on condition task in the Synthesis version.
- The preventive maintenance properties from version 7 will be transferred to the on condition task (upon detection) properties of the on condition task in the Synthesis version.

- If a detection threshold is defined in version 7 but no preventive maintenance properties are defined, then only an inspection task will be created (i.e., the threshold is ignored).

- PM/Inspection based on group:
  - A task will be added and assigned to the maintenance group that the block belongs to. For example, if Block 1 belongs to Item Group 1 and has an inspection policy based on group maintenance in version 7, then in the Synthesis version an inspection task will be created and will be set to be performed upon group maintenance. Maintenance Group 1 will be checked in the list of groups that will trigger a maintenance, and Block 1 will be assigned to Maintenance Group 1.
  - If the block does not belong to a group, a task will not be added.
- The corrective maintenance properties will be imported as corrective tasks.
- A preventive maintenance action or an inspection based on maintenance phase and associated with a block in a standard phase will not be imported.

2.6.2.4 Converting Xfmea/RCM++ 5 Databases

You can convert/import data from an Xfmea/RCM++ 5 database into a Synthesis repository. This section describes:

- How to convert an Xfmea/RCM++ 5 standard database (*.rx5).
- How to import selected data from an Xfmea/RCM++ 5 enterprise database.
- Some conversion assumptions and tips that you should be aware of.

For information about converting and importing data from an Xfmea/RCM++ 5 library, please consult the "Configurable Settings" chapter in the Xfmea/RCM++/RBI documentation.

Standard Databases (*.rx5)

For Xfmea/RCM++ 5 standard databases, you must convert the entire *.rx5 file to a new Synthesis standard repository. (Later, you can use the Import/Export Projects feature to copy specific projects into an existing database, if desired.)

Do the following:

1. In the Synthesis version of Xfmea, RCM++ or RBI, choose **File > Open Repository**.
2. Select **RCM++ 5/Xfmea 5 (*.rx5)** from the **Files of type** drop-down list.
3. Select the file you wish to convert and click **Open**.

The application will create a new standard database file in the same directory with the extension *.rsr9; the existing *.rx5 file will remain unchanged.

Enterprise Databases

For Xfmea/RCM++ 5 enterprise databases, you can select which analysis projects and settings you wish to import into a Synthesis enterprise repository that has already been created.

Do the following:

1. Connect with the enterprise database you want to import into.

2. Choose **File > Manage Repository > Enterprise > Import from Prior Version**. (Note that this command is visible only if your user account has Admin permissions in the database.)
3. In the area at the top of the Import from Version 5 Enterprise Database window, enter the connection information for the Version 5 enterprise database and click **Connect**.
4. The table shows all of the data and settings in the original database. Use the check boxes to select which data you wish to import and then click **OK** to start the transfer.

Note: If you are importing into a SQL Server database, the **Create SQL Server login** check box will be displayed at the bottom of the window. This is applicable only if you are importing user accounts. *See Section 2.1.2.3 on page 32*.

Conversion Assumptions and Tips

When you convert data from an Xfmea/RCM++ 5 database, please consider the following assumptions and tips:

- **Conversion to Synthesis Resources**

  Certain types of information in the Version 5 projects will be converted to *resources* in the Synthesis version. This includes reliability information (which will be converted to models), controls and actions. In these cases, records that have identical properties will be merged into a single resource. For example, if there were two identical actions in the original project, a single action would be created as a resource in the Synthesis project and used in both locations.

- **Conversion to Security Groups**

  If you are working with secure databases, the access levels and access groups in Version 5 will be converted to security groups in Synthesis. Specifically:

  - Repository-level security: User permissions assigned by access level in Version 5 will be assigned by the corresponding security group in the Synthesis version. For example, if a user had the Admin access level in the original database, then that user will be assigned to the Admin security group in the Synthesis version. If there is no corresponding security group in the Synthesis version, the user will be imported with no security groups assigned, and a user with "Admin" access level permissions will have to manually assign security groups to the imported user.
  - Project-level security: For each access group in the Version 5 database, a security group will be created in the Synthesis repository. Because the Version 5 access groups could have different access levels for different users in the same group, it is not possible to automatically determine which permissions should be assigned for the new security group in the Synthesis version. Therefore, these groups will initially be assigned the default permissions; you can edit the permissions via the Security Groups tab of the Manage Repository Security window. Projects that had access limited by access group in Version 5 will continue to have their security set by the appropriate security group(s) at the project level in the Synthesis version.

- **Configurable Settings for Converted Projects**

  When you convert an existing project from Version 5, the application will add new configurable settings for new features that were added in the Synthesis version. For example, the configurable PFD Worksheet settings will be added to the interface style, the quantitative values will be added to the occurrence rating scale, etc.

  It is important to note that these default settings may need to be modified after the conversion. For details, please consult the "Configurable Settings" section in the Xfmea/RCM++/RBI User's Guide.

2.6.2.5 Converting MPC 3 Databases

If you have an existing systems and powerplant analysis that was created in MPC 3, it is easy to convert the *.rsm file into a new Synthesis repository.

Choose **File > Open Repository**, select MPC 3 (*.rsm) from the **Files of type** drop-down list, and then browse for the desired file.

When prompted to enter a username and password, enter the administrative login from the old *.rsm file.

Note: By default, a converted Version 3 database is a secure database that transfers any user accounts that were defined in the old database. After the new database has been created, the administrator should review the automatically created user accounts and update as appropriate. *See Section 2.3 on page 46 for more information about database-level security.*

The final step is to use the Tasks Conversion window to review the task records that will be imported and make any updates that may be needed (see details below). Once you have completed the review, click **OK** to start the import.

When the process completes, there will be a new standard database file in the same folder and with the same name as the original *.rsm file. The new file will have the extension *.rsr9 and the original *.rsm file will remain unchanged.

Tasks Conversion Window

The Tasks Conversion window displays a list of all of the tasks defined in the original database and allows you to review how the records will be converted upon import. For any properties that are displayed with blue text, you have the option to change the task record data before it is imported.

Some task properties are handled a bit differently in Version 9 than they were in Version 3. Specifically:

- Some of the **Task Types** that were combined in Version 3 are now categorized separately in Version 9 (e.g., Operational Check (OPC) and Visual Check (VCK) are now separate task types).
- In Version 3, the task **Interval** was always stored as a text field. In Version 9, you can choose whether each task's interval will be stored as a text field or if it will instead be recorded as a number with an associated time unit (e.g., 2000 flight hours can now be stored as value=2000 and time unit = FHr). Numbers may be easier to sort and will also make it possible to perform simulation-based reliability calculations if you choose to import the analysis project to RCM++ or RBI.
- The **Zonal** field from Version 3 is called **Zonal Candidate** in Version 9. This is now a yes/no field that can be set to **Yes** only if the failure effect categorization (FEC) is set to 5 or 8 (i.e., a safety issue) and the task type is "General Visual Inspection (GVI)." Also in Version 9, the **Zone** field will be enabled only if Zonal Candidate is set to **Yes**.

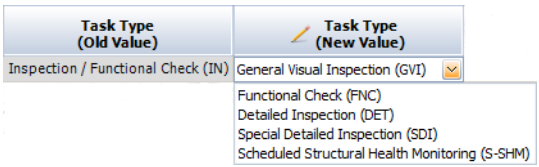
The following subsections describe the default conversion logic for Task Type, Interval and Zone.

Task Type

For the **Discard** and **Restoration** task types, there is no difference between Versions 3 and 9. The text in the Task Type (New Value) column will be displayed in italics to indicate that this property cannot be changed via the Tasks Conversion window.

Task Type (Old Value)	Task Type (New Value)
Discard (DS)	*Discard (DIS)*
Restoration (RS)	*Restoration (RST)*

For the remaining task types, the software will use the logic specified below to suggest a task type, but the property will be displayed in blue text to indicate that it can be edited. If you disagree with the default selection, you can choose one of the other eligible task types from the drop-down list, as shown in the following example.



- When the task type was set to **Inspection/Functional Check (IN)** in Version 3, the following logic will be applied in the specified order. For example, if the task description contains both the word "Visual" and the word "Function," the General Visual Inspection task type will be applied by default because the word "Visual" will be matched first.

If the task description contains:	The default option is to import the task as:
Visual	General Visual Inspection
Special Detail	Special Detail Inspection
Health Monitor	Scheduled Structural Health Monitoring
Function	Functional Check
none of the words/phrases listed above	Detailed Inspection (DET)

- When the task type was set to **Lubrication/Servicing (LU)** in Version 3, the default option is to import the task as Servicing (SVC) if the task description contains "Servic" and as Lubrication (LUB) if it does not.
- When the task type was set to Operational/Visual Check (OP) in Version 3, the default option is to import the task as Visual Check (VCK) if the description contains "Visual" and as Operational Check (OPC) if it does not.

Tip: The task types in MPC 9 are determined by the requirements of the MSG-3 guidelines and cannot be changed. However, the abbreviations can be configured to fit your particular preferences. If you want to change the default task type abbreviations (e.g., if you don't want to use DIS for Discard, RST for Restoration, etc.), click the **Task Type Abbreviations** button. The changes that you make in the Define Task Type Abbreviations window will automatically apply to all analyses in the current database. This window is also accessible from **File > Manage Repository > Task Types**.

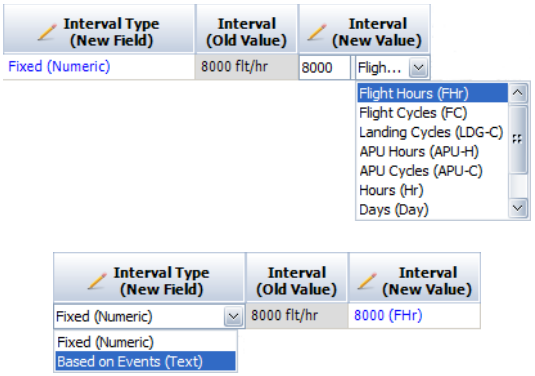
Interval Type and Interval

Two options are displayed at the top of the Tasks Conversion window to determine how the task intervals will be imported:

- If you choose **Transfer all intervals as text**, the interval type for all imported task records will be set to **Based on Events (Text)** and the interval will be stored as a text field (just like in MPC 3). The Interval (New Value) column will be displayed in blue text to indicate that you can change the text if desired before the import.

- If you choose **Transfer intervals as numeric if possible**, the application will check to see if the original task interval begins with a number and is followed by a space and at least one text character. If it does (e.g., "2000 flt/hr"), the default option is to import it as a number with an associated time unit. If not (e.g., "Per Mfg Life Limit"),

the default is to import as text. Either way, you have the option to change both the interval type and new value before the import. For example:



Tip: If you want to change the time units displayed in the drop-down list, click the **Time Units** button. The changes that you make in the Manage Units window will apply immediately to the tasks that you are currently importing, and also to all other analyses you may later add to this database. This window is also accessible from **File > Manage Repository > Manage Units**.

The Tasks Conversion window uses a variety of techniques to try to match the text field from Version 3 with one of the specified time units in Version 9. There will be a match if the name or abbreviation of a Version 9 time unit either starts with or contains the text in the Version 3 field. For some predefined time units, the software also recognizes other commonly used abbreviations for the time unit (e.g., the software will recognize any of the following abbreviations as flight hours: Flt/Hr, Flight H, Flt H or Flt/H).

If your data set contains a task interval that is not recognized, you can do any of the following:

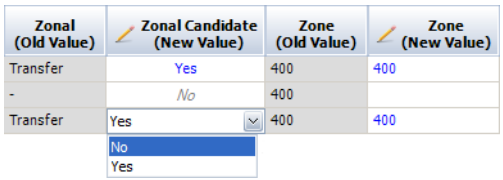
- Edit the text in the **Interval (New Value)** column so it will match either the name or the abbreviation of a predefined time unit. For example, if the Version 3 task record contains a misspelling for "2000 fliight hours," you can click inside the cell and remove the extra i.
- Click the **Time Units** button and define a new time unit for the database with a name or abbreviation that matches the data you want to import. For example, if the Version 3 task record has an interval of "5 Weeks," you can open the Manage Units window and create a new time unit with the name "Weeks."
- Click **Cancel** to close the Tasks Conversion window then edit the original tasks in MPC 3 before starting the import again.

Zonal Candidate and Zone

If all of the following conditions are met, the **Zonal Candidate (New Value)** column will be set to **Yes** by default and any information from the **Zone** field will be transferred during the import:

- The failure effect categorization (FEC) is set to 5 or 8 (i.e., it is a safety issue).
- The Task Type is set to General Visual Inspection (GVI).
- The Zonal property in the original MPC 3 analysis contains the word "Transfer."

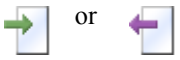
If you don't want the task to be considered for zonal analysis even though it meets the requirements, you can click inside the Zonal Candidate (New Value) column and choose No from the drop-down list. If this column displays No in italics, this indicates that at least one of these conditions is not met and the property cannot be changed.



2.6.3 Importing/Exporting Projects

All Synthesis desktop applications make it easy to import or export selected projects from one database to another.

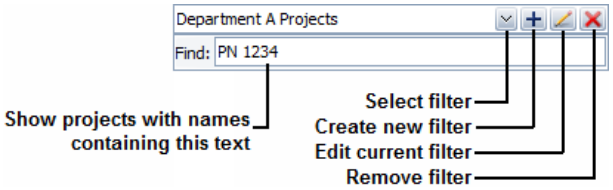
First, choose **Project > Management > Import/Export**.



In MPC, the import/export window opens directly. In all other applications, a wizard displays the options that are relevant for the current application. Select **Projects** and click **OK**.

Once the Import Projects or Export Projects window is open, do the following:

1. Use the drop-down list or browse button (…) to select the database that you want to import from or export to.
   - This can be any existing standard database (*.rsr9) or enterprise database connection file (*.rserp).
   - If you are exporting, you can also use the add button (+) to create a new standard database to export into.
2. The tree displays the projects that are available to be imported or exported. If desired, you can use the **Filter** and **Find** tools to limit the list of projects displayed. *For details, see page 97 in Chapter 3*.



3. Use the check boxes to select which project(s) you want to import/export then click OK to copy the data.

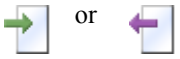
Tip: Project names must be unique within each database. If you attempt to import/export a project with a name that already exists in the destination database, the application will automatically increment the name. For example, if "Project1" already exists, the new project might be renamed to "Project1_1," "Project1_2," etc.

2.6.4 Importing/Exporting Project Items or Resources

When applicable, Synthesis applications make it easy to import/export selected project items (e.g., folios, diagrams, plot sheets, workbooks, etc.) or Synthesis resources (e.g., models, maintenance tasks, etc.) between existing projects. The projects can be in the same database or in different databases.

- The **Items** option is available for all desktop applications except Xfmea/RCM++/RBI and MPC.
- The **Resources** option is available for all desktop applications except MPC.

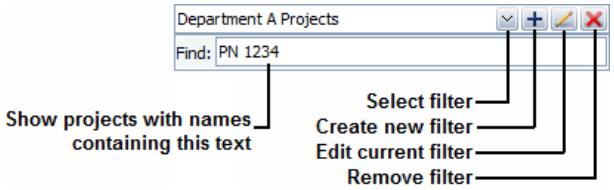
First, open the project that you want to import to or export from and choose **Project > Management > Import/Export**.



In the wizard, choose **Items** or **Resources** and click **OK**.

Once the Import Window or Export Window is open, do the following:

1. Use the drop-down list or browse button (…) to select the database that you want to import from or export to.
   - This can be the current database, or you can choose another existing standard database (*.rsr9) or enterprise database connection file (*.rserp).
   - If you are exporting, you can also use the add button (+) to create a new standard database and project to export into.
2. Use the tree in the **Source Project** or **Destination Project** area to select the project that you want to import from or export to. If desired, you can use the **Filter** and **Find** tools to limit the list of projects displayed. *For details, see page 97 in Chapter 3*.



3. Use the check boxes in the **Select Items** area to select which project items or resources you want to import/export.
   - For project items, this area will display the same folders that appear in the current project explorer. A +/- icon next to the folder indicates that it contains at least one project item that can be imported or exported.
   - For resources, this area will display a folder for each type of Synthesis resource that exists in the selected project.
4. Click **OK** to copy the data.

Tip: The names of project items and resources must be unique within each Synthesis project. If you attempt to import/export something with a name that already exists in the destination project, the application will automatically increment the name. For example, if “Folio1” already exists, the new folio might be renamed to “Folio1_1,” “Folio1_2,” etc.

Keeping Associated Items/Resources Together

It is important to note that some project items and resources link to other items or resources in the same project. For example, an overlay plot usually links to at least one analyzed folio or diagram. Likewise, a URD (universal reliability definition) resource usually links to a variety of different models, maintenance tasks, crews, etc.

In the case of project items, the application will automatically import/export the associated analyses, even if you did not specifically select to import/export them. For example, if you select to import an overlay plot, the folio(s) or diagrams(s) that were used to generate the plot will be automatically imported along with the plot.

In the case of Synthesis resources, the application will automatically copy any linked resources. For example, if you select to import a URD, all associated models, maintenance tasks, crews, etc. will be imported regardless of whether you selected each individual check box.

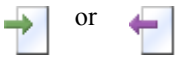
2.6.5 Importing from Excel or Delimited Text Files

In Weibull++/ALTA, RGA and DOE++, it is easy to import data from any of the following file types into a folio.

- Excel files (*.xls, *.xlsx)
- Tab, comma, space and semicolon delimited files (*.txt, *.csv, *.prn, *.smc)

Tip: Other Synthesis applications use different tools for importing/exporting via Excel (i.e., for importing Bill of Materials data in Lambda Predict and failure mode data in Xfmea/RCM++/RBI and MPC). For more information about those features, please consult the documentation for each particular application.

First, open the project that you want to import into and choose **Project > Management > Import/Export**.



In the wizard, choose **Other file** and click **OK**.

Browse for the file you wish to import from and click **Open**. Note that:

- In DOE++, the data will be copied into a new free form folio, where you can then designate columns for factors and responses and analyze the data. If you are importing from an Excel file that has multiple worksheets, the sheet that was active the last time the file was opened will be used.
- In Weibull++/ALTA and RGA, you will need to specify which data will be imported and how it will be mapped to the columns in the particular type of data folio. The following sections describe how to map the data for import into a Weibull++, ALTA or RGA data folio. This includes:
  - Selecting which data sheets will be imported
  - Using the control panel to map the columns that will be imported
  - Setting the import template directory
  - Creating a new import template
  - Applying a saved import template
  - Importing the data

2.6.5.1 Selecting Which Data Sheets Will Be Imported

The left side of the utility displays the data from the Excel or delimited text file that is currently selected. If you wish to open a different source file, click **Open**.

If you are importing from an Excel file that has multiple worksheets, use the tabs at the bottom of the window to view and set the import preferences for each sheet.

Use the options at the top of the control panel for each sheet to specify whether the data will be imported.

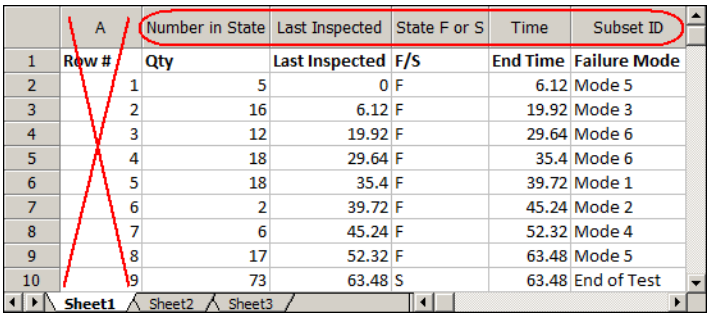
- **Do not import sheet**: The control panel will not contain any other options and the data will not be imported.
- **Import sheet**: The control panel displays the options you need to either manually map the columns or use a template to automatically apply the same mapping that was used for another file.

2.6.5.2 Using the Control Panel to Map the Columns That Will Be Imported

If you are not using a template, or if you need to modify the settings after an import template has been applied, do the following:

1. Use the **Data Type** drop-down list to specify which type of folio to import into.
   - This can be different for each sheet, and it determines which options will be available in the rest of the control panel.
2. Click inside each column that you wish to import and then click the corresponding button in the control panel to map it to a column in the data folio. This updates the column header and maps the data to a column in the data folio.
   - If the header displays a letter, the data in that column will not be imported.
   - If the header displays a name, the data will be imported to the column associated with that name.
   - To remove the column mapping, click inside the column and click the corresponding button again.

   As an example, the following picture shows data that will be imported into a Weibull++ standard folio. Column A will not be imported. The rest of the data has been assigned to columns that are used in the standard folios.



	A	Number in State	Last Inspected	State F or S	Time	Subset ID
1	Row #	Qty	Last Inspected	F/S	End Time	Failure Mode
2	1	5	0	F	6.12	Mode 5
3	2	16	6.12	F	19.92	Mode 3
4	3	12	19.92	F	29.64	Mode 6
5	4	18	29.64	F	35.4	Mode 6
6	5	18	35.4	F	39.72	Mode 1
7	6	2	39.72	F	45.24	Mode 2
8	7	6	45.24	F	52.32	Mode 4
9	8	17	52.32	F	63.48	Mode 5
10	9	73	63.48	S	63.48	End of Test

Sheet1 Sheet2 Sheet3

3. If you are importing to an ALTA standard folio, use the **Number of Stresses** field to specify how many stress columns will be created in the new folio (maximum = 8). Note that:
   - If you enter a value that is less than the number of columns that were mapped with the **Stress** button, the "extra" stress column data will not be imported.
   - If the number is greater, the additional stress columns will be created in the new folio and you can enter the data later.
4. If the source file contains column headings or other introductory material that should not be imported, use the **Start from Row** field to specify where the actual data begins. For example, in the picture above, the first row contains heading labels so the data import should begin from Row 2.

2.6.5.3 Setting the Import Template Directory

If you will be importing data from multiple files that have the same structure, you can use saved template files (*.waim in Weibull++/ALTA or *.rgaim in RGA) to automatically map the columns for all other similar files that you need to import from.

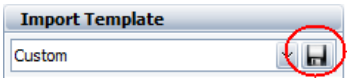
By default, template files will be stored at C:\Users\<username>\AppData\Roaming\ReliaSoft\<application>\Import Templates. This directory determines which templates will be displayed in the **Import Template** drop-down list in the control panel. It also sets the default path for saving any new templates you create.

If you want to access templates from a different location, click the **Import Template Directory** button at the top of the window and select a different folder.

2.6.5.4 Creating a New Import Template

1. Open a data file and use the control panel buttons to map the columns.

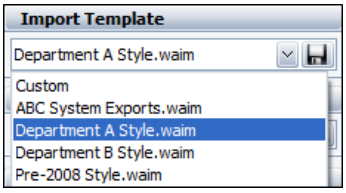
2. Click the **Save** icon in the Import Template area on the control panel.



3. Specify a name and click **Save**.

2.6.5.5 Applying a Saved Import Template

1. Open the data file and select **Import sheet** to display the rest of the control panel options.
2. Make sure the **Import Template Directory** is set to the folder that contains the applicable template file(s).
3. Use the drop-down list in the **Import Template** area to select the template you want to apply.



The utility will automatically assign the column mappings defined in the template. If desired, you can use the control panel to make further adjustments before import.

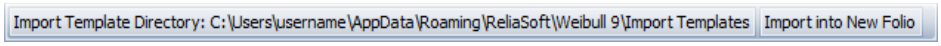
4. When you are satisfied with the mapping, click **Import** to create the new folio.

Tip: If you make adjustments after applying the template, you have the option to click **Save** in the **Import Template** area to either replace the existing template, or create a new template with a different name.

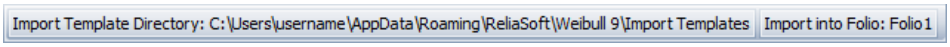
2.6.5.6 Importing the Data

When you are ready to import the data, click the **Import** button.

By default, the application will import the data into a new standard folio. The status bar will display "Import into New Folio," as shown in the following example.



If you prefer to import into an existing folio instead, click the **Import to Existing Folio** button and choose one of the available folios. When you return to the import window, the status bar will now display the name of the selected folio, as shown in the next example.



2.6.6 Using XML in BlockSim and Lambda Predict

In BlockSim and Lambda Predict, you have the option to use XML (extensible markup language) files to import or export system configuration data and item properties.

Both applications utilize open schemas published by ReliaSoft. You can use these files to transfer data to/from other applications or database systems.

In BlockSim, the XML files include the block properties and information about how the blocks are connected in a reliability block diagram or fault tree diagram. They do not include visual aspects such as diagram style settings, block style settings, etc. BlockSim supports both import and export via XML.

In Lambda Predict, the XML files include the structure of the system hierarchy in a prediction folio and some of the item properties. Lambda Predict supports export to XML.

Exporting to XML

To create an XML file, first open the project and then choose **Project > Management > Import/Export > Export**.

In the wizard, choose **Other file** and click **OK**.

Specify the desired pathname/filename and click **Save**. This will export the relevant information for all of the diagrams or prediction folios in the current project. Other project items (such as RENO flowcharts, plots, attachments, etc.) are not included.

Importing from XML

There are two ways to import from an XML file.

If you want to import to a new project in a new standard database:

1. Choose **File > Open Repository**.
2. Browse for the file and click **Open**.

If you want to import to a selected project in an existing standard database or enterprise database:

1. Create a new project or open an existing project that you want to import into.
2. Choose **Project > Management > Import/Export > Import**.

   In the wizard, choose **Other file** and click **OK**.
3. Browse for the file and click **Open**.

2.7 Importing XFRACAS Data to Weibull++ or RGA

Starting with Version 9, you can access XFRACAS data from within Weibull++ or RGA for life data analysis or reliability growth analysis. This functionality is available when you are working with an enterprise database that has XFRACAS data in it, or when you are working with a standard database that has a connection to external XFRACAS tables defined (*see Section 2.4.6 on page 68*).

Accessing XFRACAS data is a two-part process.

- First, you specify the components of interest and pull the data from XFRACAS into the Reliability Data Warehouse (RDW) - Section 2.7.1 (p. 88)
- Once the data set is in the RDW, you can:
  - View the data in a variety of graphical presentations via the Dashboard Viewer. This tool is also available for viewing BlockSim simulation results - Section 2.7.4 (p. 92)
  - Choose the incidents to include in the Weibull++ or RGA data set and extract the data to a standard folio - Section 2.7.2 (p. 89)

2.7.1 Transferring Data from XFRACAS to the Reliability Data Warehouse (RDW)

In Weibull++ and RGA, the Reliability Data Warehouse (RDW) allows you to view data from XFRACAS. Data in the RDW can be extracted to standard folios for data analysis. To bring data into the RDW, choose **Project > Synthesis > XFRACAS to RDW**.

Note: This command is available only in Weibull++ or RGA, and only when you are working with an enterprise database that has XFRACAS data in it, or when you are working with a standard database that has a connection to external XFRACAS tables defined (*see Section 2.4.6 on page 68*). It is available to users with the "Create/edit/define global resources" permission in Synthesis and the “Synthesis - Read Data” permissions for the associated XFRACAS tables.

If no XFRACAS entity has previously been selected for the current database, then you will be asked to select an entity to import from. Otherwise, the last selected entity will be used. Once an entity has been selected, the XFRACAS to RDW interface will open. You can select a different entity at any time by clicking the **Change Entity** icon on the control panel.

The **Select Parts** area displays all available parts in the XFRACAS entity, organized by system hierarchy. In this area, select the check box for each part that you want to pull incident (i.e., failure) data for.

Tip: You can configure the appearance of the window and filter the data in the table, as explained in Section 2.7.3 on page 91.

In Weibull++, the application will pull all incident data for the selected parts into the RDW data set. In RGA, you can use the **Additional Settings** area of the control panel to filter the data that you want to pull. You can decide:

- Whether to include only system down events, or all types of events.
- Whether to include only data for systems commissioned between specified dates, or data for all systems.

When you have selected all of the parts of interest and specified any relevant settings, click the **Send to RDW** icon on the control panel.

You will be prompted to name the data set. It is important to remember that each data set you create is a static record of the XFRACAS data at that point in time (i.e., the data set will never be updated). For this reason, you may want to name your data sets in a way that clearly indicates when they were created. When you click **OK**, the extraction process will begin. Depending on the amount of data that you are extracting, this may take some time.

2.7.2 Transferring Data from the Reliability Data Warehouse (RDW) to a Standard Folio

Once XFRACAS data has been pulled into the RDW, you can extract it to a Weibull++ or RGA standard folio at any time. To do this, choose **Project > Synthesis > RDW to Folio**.

or

The RDW to Folio window will appear, with the most recent data set displayed.

In this window, you will specify which data you want to extract and how the values in the data should be entered in the resulting standard folio.

Selecting the Data to Extract

You can work with one or more data sets at a time. Working with multiple data sets may be useful in certain circumstances. For example, if you have extracted an individual data set for each version of a part, you can select all of those data sets in order to analyze all versions together. To change the data set(s) that you are working with, choose **Data Set > Select Analytical Data Set**.

Choose the desired data set(s) in the Select Analytical Data Set window, then click **OK** to display the data set(s) in the RDW to Folio window.

Note: You can use the Auto Filter row, which appears immediately below the column headers, to help you find data sets of interest.

Additionally, you can double-click a data set name to rename it, and click the **Delete** icon to delete the selected data set(s). Each data set is a static record of the XFRACAS data at the moment of data set creation (i.e., the data set will never be updated). Therefore, you may want to ensure that data set names clearly indicate when they were created, and delete old data sets when they are no longer applicable.

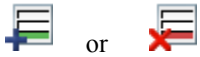
Each row in the table represents an incident from XFRACAS and can be extracted as a data point in the resulting data set.

- In Weibull++, incidents are considered failures if the Incident Category in XFRACAS is Chargeable and the replaced part has a failure type of "failure." All other incidents are considered suspensions.
- In RGA, the Top Level Serial Number column is used to determine the System ID for each system. The earliest data point for each system will be considered the start time and the latest data point will be considered the end time.

The Export Status column shows whether each row will be included in the extraction.

- **Included**: Incidents that will be included in the Dashboard chart and/or exported to the standard folio are shown in green.
- **Invalid**: Incidents with a State Time of less than 0 are shown in red.
- **Ignored**: Incidents with a State Time of 0 and marked as suspensions in the State FS column are shown in gray. This applies only in Weibull++.
- **Excluded**: Incidents that have been manually excluded are shown in blue.

By default, incidents that are marked as Invalid or Ignored will be excluded from the extraction. You can manually include such incidents, or manually exclude incidents that were marked as Included by default, by selecting the row(s) and choosing **Current Row > Include** or **Current Row > Exclude** (to select multiple rows, hold down the **CTRL** key while clicking the rows).

 or

If you are uncertain about whether you want to include a particular row in the extraction, you can select the row and choose **Current Row > View XFRACAS Incident** to open the incident in your web browser. This command is available only if you have permission to view incidents in the current XFRACAS entity.

You may also find it useful in this process to view a graphical presentation of the current data set(s). To open the Dashboard Viewer for this purpose, choose **Dashboard > Dashboard Viewer**.

To view only those rows that are included, choose **View > Show Only Selected**. You can choose this command again to show all records.

Tip: You can configure the appearance of the window and filter the data in the table, as explained in Section 2.7.3 on page 91.

Specifying the Extraction Settings

You can specify which columns in the data set will be used to populate the Time and Subset ID or Comments columns in the standard folio using the **Data Set > Set Time Column, Data Set > Set Subset ID Column 1** and **Data Set > Set Subset ID Column 2** commands.

For the Time column, you will choose a single column in the data set. For instance, choosing the State Time column will enter the values from that column in the Time to F or S column in the resulting Weibull++ standard folio or in the Time to Event column in the resulting RGA standard folio.

For each Subset ID column (in Weibull++) or Comments column (in RGA), you can choose one or more columns in the data set to include in the column. If you include information from multiple data set columns in a Subset ID or Comments column, the information will be separated with a dash. For example, choosing the Set Subset ID Column 1 command and choosing the Report Type and Category columns will result in values in the form “Report Type -

Category" (e.g., "Field Issue - Component Failure" or "In-House Testing - Installation Issue") in the Subset ID 1 column in the resulting Weibull++ standard folio or in the first Comments column in the resulting RGA standard folio.

Note: All columns will be available for selection, regardless of whether they currently appear in the data table. You can add columns to the data table via the Column Chooser. *See Section 2.7.3 on page 91for more information*.

Once you have specified the extraction settings, click **Extract Selected** to extract the "included" incidents from the RDW and create the standard folio.

2.7.3 RDW Interfaces: Configuration and Data Filtering Tools

In both the XFRACAS to RDW interface and the RDW to Folio window, you can configure the interface appearance and filter the data sets in numerous ways. Many of these tools are shared between the two interfaces.

To configure the appearance of the interface:

- To change the order of the columns, click a column name and drag the column to the desired location.
- To resize columns, you can:
  - Drag the edge of a column header to the desired position.
  - To automatically resize a column to fit the data, double-click the edge of the column header or right-click the column header and choose **Best Fit** on the shortcut menu.
  - To resize all columns in the table to fit the data that they contain, right click a column header and choose **Best Fit (all columns)**.
- To hide or show columns, right-click any column header and choose **Column Chooser** on the shortcut menu. You can then click the header of any column and drag it into the Column Chooser to hide it from the data set. To show additional columns, drag them from the Column Chooser to the desired location.
- To sort the data set by a column, click the column name. The sort direction is indicated by an arrow beside the column name; click again to reverse the direction of the sort. Note that in the XFRACAS to RDW interface, the data will be sorted by the specified field at each level of the hierarchy.
- In the RDW to Folio window, you can group the data set by one or more columns.
  - To quickly group the data set by one column, right-click the column header and choose **Group By This Column**.
  - To perform more advanced grouping, choose **Additional Options > Show Group By Box**. The Group By box will be displayed above the table. To group by a column, click the column name and drag it into the Group By box. You can drag multiple columns into the Group By box; the order in which they appear determines the way the records are sorted, and you can drag them within the Group By box to reorder them. Note that each column name in the Group By box has an arrow indicating the direction of the sort; you can click the column name to reverse the direction.

To filter the data set:

- The Auto Filter row appears by default immediately below the column headers. Enter text in any cell in this row to display only those rows that contain matching text in that field. You can hide or show the Auto Filter row by right-clicking any column header and choosing **Hide Auto Filter Row** or **Show Auto Filter Row**.
- In the XFRACAS to RDW interface, click the **Search** icon on the control panel to open the Search window.

On the Find tab of the Search window, you can specify text to search for, which column to search for the text in and whether the text should be located only at the start of the field or anywhere within the field. The Query tab allows you to search for multiple text strings in a given column by entering each text string and then adding it to the **Criteria** list.

- In the RDW to Folio window, the Find panel allows you to search for text within the data currently displayed in the table. Choose **Additional Options > Show Find Panel** or right-click any column header and choose the command to display the panel. Enter text in the field and click **Find** to filter the data displayed in the table. You can click **Clear** to remove the filter. To close the panel, click the **X** or clear the command.
- You can also use the Advanced Filters functionality, which is always enabled for these interfaces. *See page 177 in Chapter 5*.

2.7.4 Dashboard Viewer

The Dashboard Viewer allows you to view Synthesis data in a variety of graphical presentations. You can use it to create plots and charts based on:

- Data imported into the Reliability Data Warehouse (RDW)
- BlockSim simulation results

In Weibull++ and RGA, it is accessed by choosing **Dashboard > Dashboard Viewer** in the RDW to Folio window. In BlockSim, it is accessed by choosing **Simulation > Dashboard > Dashboard Viewer**.

The contents of the Dashboard Viewer will vary depending on the templates that have been created for your application. To choose a template, select it from the drop-down list at the top of the window.

Note: Dashboard templates are created via the Dashboard Designer and managed via the Dashboard Manager, both of which are available only to users with the “Create dashboard templates” permission.

Interface 3

Most Synthesis applications have their own Multiple Document Interfaces (MDIs) that provide full-featured support for the relevant analysis methods. (The exceptions are Weibull++/ALTA and BlockSim/RENO, which can display both applications in a shared MDI.)

When you are using multiple Synthesis applications simultaneously, different applications will open in separate MDIs but they can all connect to the same repository, and even to the same analysis project.

All Synthesis MDIs share a common structure, which includes the following elements.

Ribbon and Backstage View

The Ribbon at the top of the MDI replaces the toolbars and menus of previous versions of the software. The Ribbon is divided into tabs that organize commands into logical groups. Some tabs will always be available (such as File, Home, etc.) while others are *contextual* and will appear only when you are working on a particular task.

The first tab in the Ribbon, the File tab, provides access to a special type of interface called the Backstage view. This is where you will create and open databases, manage settings that apply to the entire database, access the Help Center, etc. For more information about the Ribbon, Backstage View and Help Center, see the "Ribbon" chapter.

Project Manager and My Portal

The **Project Manager** allows you to manage the projects within the current database and, for many Synthesis applications, it also allows you to manage all of the folios, diagrams, reports, attachments, etc. in the current project. This panel is docked on the left side of the MDI and pinned by default. ("Pinned" means that the entire panel is always visible unless you decide to hide or "unpin" it.) *See Section 3.2 on page 97*.

My Portal provides access to messages between users, actions that need to be performed, a list of users who are currently logged in to the same database and context-sensitive notes about the feature your are currently using. This panel is docked on the right side of the MDI and unpinned by default. ("Unpinned" means that only a small tab will be visible in the interface unless you move the pointer over the tab to make the entire panel visible.) *See Section 3.3 on page 101*.

For both of these MDI features, you can decide whether the panel will be visible and how it will be positioned in the interface. *See Section 3.1 on page 94*.

Tip: If desired, you can change the overall color scheme used in the MDIs for all Synthesis desktop applications on your computer. Choose **File >Synthesis Setup** to open the Synthesis Setup window and then use the **Skins** drop-down list to select the style you prefer.

Status Bar

The status bar at the bottom of the MDI displays the following:

- The software name and version.
- The path and filename (if applicable) of the database that you are currently using.
- The currently active project.
- If you are connected to an enterprise database, a connection speed indicator, as follows:

  < 30 ms: Good (acceptable performance)

  30 - 70 ms: OK (may exhibit some delays in operations, opening/closing windows, etc.)

  70 - 110 ms: Slow (will exhibit some delays in operations, opening/closing windows, etc.)

  110 - 150 ms: Very slow (will exhibit significant delays in operations, opening/closing windows, etc.)

  > 150 ms: Extremely slow (will result in unacceptable performance and usability)
- Your login information. Double-clicking this section of the status bar provides quick access to the User Login and Contact Information window, where you can view/modify your display name and other contact details.

3.1 Show, Tile, Dock and Pin Panels

In all Synthesis desktop applications, the MDI includes a Project Manager panel and a My Portal panel. You can decide whether each panel will be displayed and how it will be positioned in the interface at any given time.

Show or Hide

To show or hide the Project Manager panel, choose **View > Project Manager** and toggle the **Show Project Manager** command on or off.

For My Portal, toggle the **View > My Portal > Show My Portal** command.

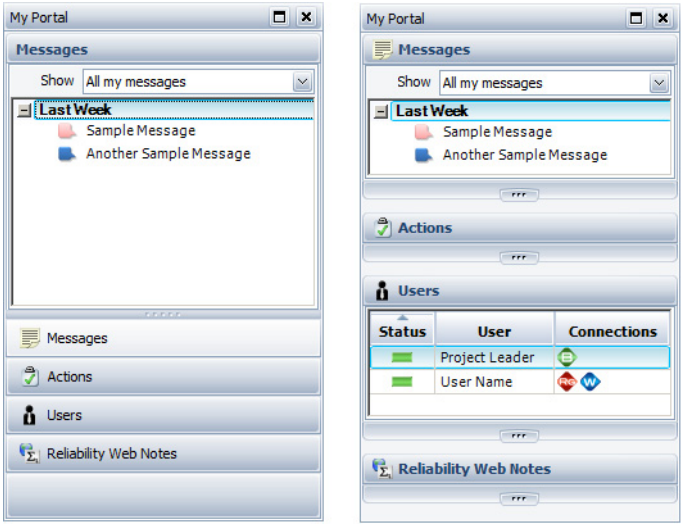
You can also hide either panel by clicking the X at the top of the window.

Tile or View One Page at a Time

In all Synthesis desktop applications, My Portal consists of four pages. You have the option to view one page at a time (and use large buttons or small icons at the bottom of the panel to switch between pages) or tile the pages so that more than one page can be visible at the same time. To specify your preference, choose **View > My Portal** and toggle the **Tile My Portal** command on or off.

When applicable, these same options are available for the Project Manager panel. Choose **View > Project Manager** and toggle the **Tile Project Manager** command on or off. (This command is not available in applications such as Xfmea, RCM++, RBI and MPC, which have only a single page in the Project Manager.)

As an example, the first picture shows the My Portal panel when it is not tiled. The second picture shows the tiled panel, with two of the four pages expanded to be visible at the same time.



When the panel is not tiled, you can drag the horizontal splitter bar up to display large buttons or down to display small icons.

When the panel is tiled, you can use the handle buttons to expand or collapse each page.

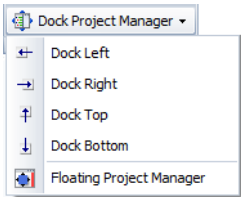
Dock or Float

For both panels, you have the option to dock the panel on the left, right, top or bottom side of the MDI. When the panel is docked, it cannot be moved but you can change the width (if docked left or right) or height (if docked top or bottom) by dragging the edge to the desired position.

Alternatively, you can choose to float the panel as a window that can be resized and dragged to any location on the desktop.

There are several ways to set the docking position:

- Choose **View > Project Manager > Dock Project Manager** and then choose the desired option.

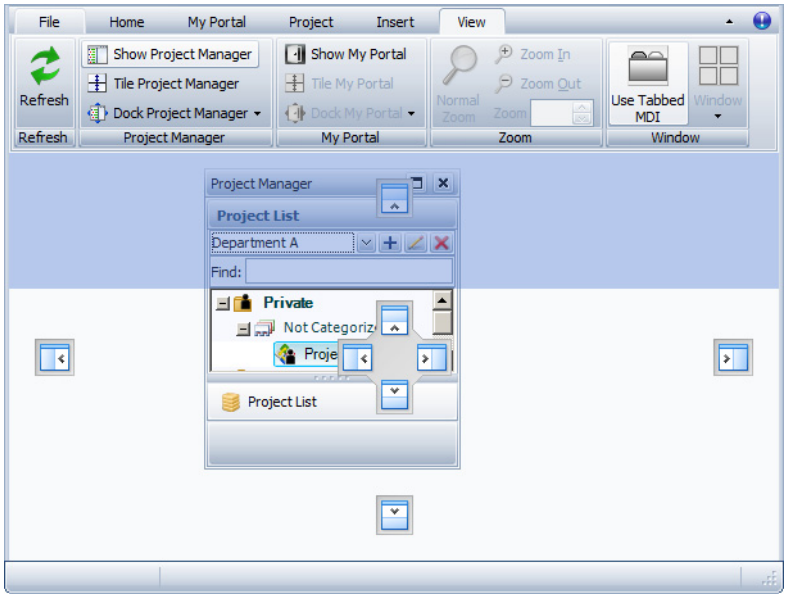


The same options are available when you choose **View > My Portal > Dock My Portal**.

- If the panel is currently floating:
  - Double-click the window's title bar to return it to the most recent docked position.

  OR

  - Click the window's title bar and drag it onto one of the location icons that become visible while you are moving the window. When the window is correctly positioned, blue shading will show where the window will be docked when you release the mouse button. (Note that the window's title bar must be exactly positioned above one of these location icons before the blue shading appears. If you don't see blue shading, the panel will not be docked when you release the mouse button.)



If you prefer to float the window, you can:

- Choose either:
  - **View > Project Manager > Dock Project Manager > Floating Project Manager**

  OR

  - **View > My Portal > Dock My Portal > Floating My Portal**
- Double-click the title bar of the docked panel.

Pinned or Unpinned

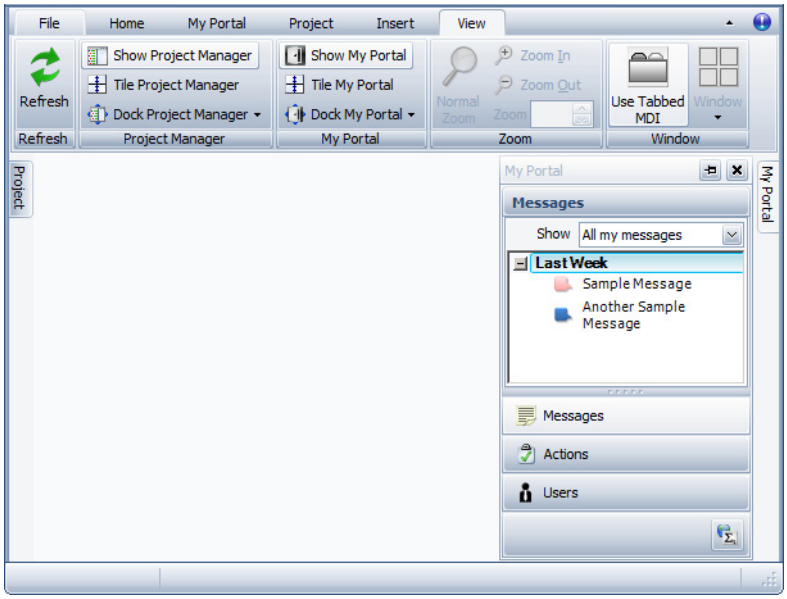
When docked, both panels can also be toggled between pinned and unpinned states by clicking the pushpin icon in the panel's title bar. When the pushpin in the icon is vertical , the panel is pinned and will be displayed at all times.

When the pushpin is horizontal , the panel is unpinned. A tab will be displayed at the docking location for each unpinned panel.

The behavior for unpinned panels is as follows:

- If you point to the tab, the panel displays only until you point to something else.
- If you click the tab, the panel stays displayed until you click something else.
- When a panel is unpinned, it cannot be undocked or moved.

As an example, the following picture shows the Project Manager panel docked on the left side of the MDI, unpinned and hidden, while the My Portal panel is docked on the right side of the MDI, unpinned and visible (either because the user moved the pointer over or clicked the tab).



3.2 Project Manager

The Project Manager provides the tools you need to browse or search for a particular project or analysis folio in a Synthesis repository. Depending on the application, the Project Manager consists of either one or two pages:

- In all Synthesis desktop applications, the project list displays all the projects in the current database. (For Xfmea, RCM++, RBI and MPC, this is the only page available in the Project Manager).
- In Weibull++, ALTA, BlockSim, RENO, RGA, Lambda Predict and DOE++, the Current Project page of the Project Manager displays all the analysis folios, reports and other items in the current project that apply to the software you are using. *See Section 3.2.2 on page 100*.

There may be up to four nodes in the project list, indicating the sharing status of the projects: Private and Public projects, Locked projects and Checked Out projects.

Each node may be further broken down by project category and/or by project owner. This ensures that projects will be organized in a logical, manageable way within the project list.

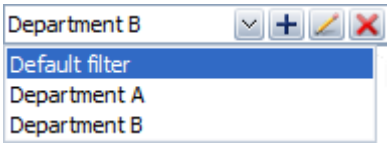
Opening a Project

To open an existing project, double-click the project name or select the project and choose **Project > Management > Open Project**.

Note: Remember that each project may contain items from any Synthesis application but you will only see the items that are relevant for the application you are currently working with (e.g., Weibull++ analysis folios are visible only when the project is opened in Weibull++). Common items, such as project attachments, are visible from all applications.

Filtering the List of Projects

The **Filter** area at the top of the project list allows you to refine which projects are displayed in the list. This feature might be particularly useful for an enterprise database that may be used to store analysis projects for the entire organization in a single centralized location. In such cases, the number of projects displayed in the project list could become overwhelming and these filters provide the ability to display only those projects that are of interest to you at any given time.



Use the drop-down list to select which filter will be applied. The **Default filter** option always shows all projects in the database. Other filters can be created and managed by each individual user. In other words, the filters that you have defined will be saved in the database and associated with your user account. When you use this account to log into the database from any computer, your filters will be available to you. However, they will not be visible to other users.

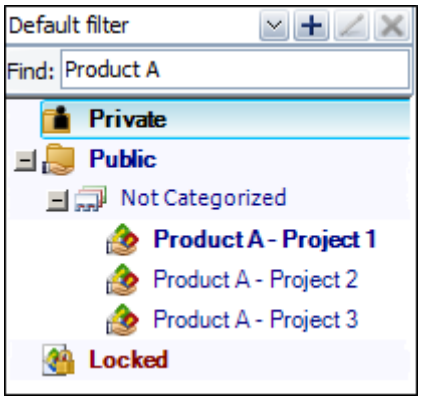
Click the **Create new filter** icon + to create a new filter or click the **Edit current filter** icon to edit the filter that is currently selected. Both commands open the Filter Properties window so you can view or modify the filter criteria.

To delete a filter, select it from the list and then click the **Delete current filter** . Remember that this will completely remove the filter from the database and this cannot be undone. If you want to keep the filter for future use and simply stop using it right now, choose a different filter from the drop-down list.

Searching for a Particular Project

You can also search for projects by entering text in the **Find** field. The project list updates dynamically to show only the projects with names that contain the text that you have entered. As an example, the following picture shows a

quick way to find all of the projects for a particular product line by using the project names instead of predefined categories or other filter criteria.



3.2.1 Filter Properties Window

The Filter Properties window is used to create new filters or edit existing filters. These filters can be used to limit the projects displayed in the project list, in the Manage Projects window or when importing/exporting entire projects or items/resources.

The custom filters that you create will be saved in the database and associated with your user account. When you use this account to log into the database from any computer, your filters will be available to you. However, they will not be visible to other users.

Specify a unique name for your filter and then use the following properties to define how the filter will limit the projects shown in the list:

- **Filtering Options**
  - **Filter by owner** filters by the name of the project owner. The available names are based on the user accounts defined in the current database.
  - **Filter by category** filters by the selected project category. The available categories are based on the project categories that have been defined in the current database.
  - **Filter by analysis** filters by the types of analyses contained in the project.
  - **Filter by 'last update' user** filters by the name of the person who last modified the project.
  - **Filter by 'last update' date** filters by the date when the project was last modified.
- **Other Options**
  - If **Show locked projects** is selected, the Locked node will be displayed in the project list and any locked projects will be displayed under it. If this check box is cleared, the Locked node and all locked projects will be hidden.
  - If **Group by category** is selected, the Public, Private and (if **Show locked projects** is selected) Locked nodes will each be subdivided into nodes for each project category. Only those categories that are currently in use by at least one project will be shown. If this check box is cleared, projects will not be divided by category.
  - If **Group by owner** is selected, the projects will be grouped according to project owner. If the **Group by category** check box is also selected, each project category will be grouped by category and then by owner.

3.2.2 Current Project Explorer

In all Synthesis desktop applications except Xfmea, RCM++, RBI and MPC (which use the system hierarchy to manage all of the different analyses in a given project), the Current Project page of the Project Manager allows you to manage all of the relevant *project items* (e.g., folios, diagrams, etc.) in the project that is currently open. Starting in version 9.0.9, this page is identified in the Project Manager by the project name.

Note: Remember that each project may contain items from any Synthesis application but you will only see the items that are relevant for the application you are currently working with (e.g., Weibull++ analysis folios are visible only when the project is opened in Weibull++). Common items, such as project attachments, are visible from all applications.

The current project explorer presents all of the project items in a hierarchical tree structure. The project name is displayed at the top of the tree. The folders that contain the analysis and reporting components of the current project, as well as project-level attachments, are displayed in nodes below the project name.

Click the plus (+) button to open a folder and display the items within that folder. Click the minus (-) button to close a folder.

The **Filter based on creator** drop-down list allows you to show only the items that were created by a particular user, if desired.

Opening Items

Double-click an item to open that item in the MDI or to bring that item to the front of the MDI if it is already open. (Note that if you double-click an attachment that appears in the current project explorer, it will open in its corresponding application, as long as that application is installed on your computer.)

Tip: In all Synthesis desktop applications, projects can be shared by multiple users and the current project explorer may display an “in-use” icon to indicate the status of an item that can't be edited because it is currently locked or being edited by another user.

Adding Items

You can add items to the project by choosing the relevant command on the Insert tab of the Ribbon. You can also right-click the relevant folder and choose an appropriate command from the shortcut menu that appears.

Editing, Renaming, Duplicating or Deleting Items

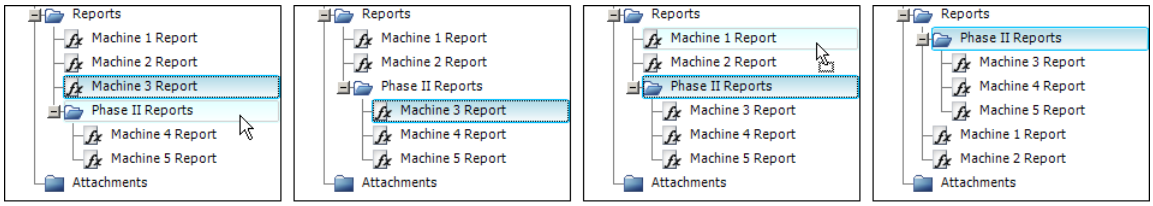
To manage existing items within the project (i.e., **Edit Item**, **Rename Item**, **Duplicate Item** and **Delete Item**), right-click the item and choose a command from the shortcut menu or select the item and use the commands in the Current Item group on the Project tab of the Ribbon.

Organizing Items

In addition to the top-level folders that are fixed for each application you also have the ability to create your own custom sub-folders to organize project items (the exception being the Attachments folder, which you cannot add sub-folders to). To add a custom sub-folder, right click any existing folder and choose **Add Folder** from the shortcut menu.

You can also arrange custom sub-folders and project items by dragging them to the desired location. Dragging a folder or project item onto a folder will place it into the bottom of that folder. Dragging a folder or project item onto a

project item will place it in the same level above that item. Using Weibull++ reports as an example, consider the following steps:



In Step 1, Machine 3 Report is being dragged onto the Phase II Reports folder. Step 2 shows that Machine 3 Report is then located at the top of the list within the Phase II Reports folder. In Step 3, the Phase II Reports folder is being dragged onto Machine 1 Report. Step 4 shows that the Phase II Reports folder is then positioned above Machine 1 Report.

3.3 My Portal

My Portal provides additional information relevant to your work within the database. Four pages are available within My Portal:

- The Messages page
- The Actions page
- The Users page
- The Reliability Web Notes page

When you first activate the software, My Portal will be "docked" as a panel on the right side of the MDI and will be in an unpinned state. My Portal can be manipulated in a way similar to the Project Manager. *See Section 3.1 on page 94*.

Messages

The **Messages** page allows you to communicate with other users who have access to the database. Note that all users have the ability to manage the messages sent to their user accounts, but the ability to create messages is based on the permissions assigned to the user accounts.

The messages displayed on this page depend on your selection in the **Show** field at the top. If you select **All my messages**, you will see all messages that you created and all messages that are directed to you, either individually or as part of the All Users group. You can also choose to view only messages that you created or only messages directed to you individually.

The messages will appear in the order in which they were created, and an icon will appear next to each message to indicate its specific priority level (blue = Low, pink = Medium and red = High).

- To create a message, choose **My Portal > Messages > Create Message**. The Message window will appear. See Section 3.3.1.
- To edit a message, double-click the message or select the message and choose **My Portal > Messages > Edit Message**.
- To delete a message, select the message and choose **My Portal > Messages > Delete Message**.
- To refresh the list of messages, choose **View > Refresh > Refresh**.

Actions

The **Actions** page allows you to view the actions that you are involved with. Actions are resources that allow you to ensure that appropriate progress is being made (*see Section 3.3.2 on page 103*). The actions displayed on this page depend on your selections in the filter options fields, which are accessible by opening the drop-down list at the top of the page. The following filters are available:

- The **Project** field allows you to specify the project for which you want to view actions. Select **All Projects** to view actions for all projects in the database.
- The **Show** field allows you to specify which actions are displayed based on your type of involvement with the actions. You can choose to view only the actions that you are assigned to review, only the actions that you created, only the actions that you are monitoring via notifications or only the actions that are approved by you.
- The **Status** field allows you to specify which actions are displayed based on the action status. You can choose to view only actions that are reviewed and approved, complete, overdue or due. To view all actions regardless of their status, select **All Statuses**.
- The **Priority** field allows you to specify which actions are displayed based on the action priority. To view all actions regardless of priority, select **All**.

The displayed actions will be categorized according to their status (i.e., Overdue, Not Started, In Progress, Completed and Approved). Within each category, they will be arranged in the order in which they were created. A due date will appear next to each action.

You can create, edit and delete actions in the same basic way that you work with messages, using the commands in the Actions group of the My Portal Ribbon tab. Note that you must have a project selected in the **Project** filter field in order to create an action.

If Microsoft Outlook is installed on your computer, you can choose **My Portal > Actions > Add Action to Outlook** to create a calendar event based on the action on your own Outlook calendar. By default, the event will be scheduled for the action's due date, but you can make modifications to the event in the properties window that appears.

Users

The **Users** page displays a list of all users who have an account in the database. (If you are working in a standard database without login security, any user who has ever opened the database will have an account created automatically and will be shown in this list.) A status light is displayed for each user; if it is "lit up" (i.e., green), the user is currently logged in to the database. In addition, the Connections column shows the Synthesis application(s) that the user currently has connected to the database. Clicking a column header in the list of users sorts the list by that column; click again to reverse the direction of the sort.

- Choose **My Portal > Users > My Profile** to open the User Login and Contact Information window, which allows you to edit the contact information associated with your user account.
- To refresh the list of users, choose **View > Refresh > Refresh**.

Reliability Web Notes

The **Reliability Web Notes** page provides access to further information on the interface that you are currently working with. Choose **My Portal > Reliability Web Notes > Get Notes for Active Item** or click the button at the bottom of the page to see basic theoretical information that may be of use in your current task, as well as links to further information and examples.

You can also select the **Automatically retrieve notes** check box to have the software automatically update the displayed notes depending on what item is currently active.

3.3.1 Message Window

The Message window allows you to add or edit messages that are displayed on the Messages page of My Portal. This window is accessed by clicking **Create Message** or **Edit Message** on the My Portal tab of the Ribbon, or by double-clicking a message.

To specify the user(s) who will see the message, click the arrow in the **Address To** field.

- Select **All Users** to address the message to all users associated with the project.
- To address specific members or notification groups:
  - Select the **Selected Groups/Users** option to enable the remaining address options.
  - Then select the check box for each desired notification group and/or individual user. (Note that you can click the + beside each notification group to view the members of the group.)
  - When you are done specifying who will see the message, click inside the **Address To field** again.

Enter a subject and the message details, then specify the priority level of the message. Messages marked as low priority will be displayed in My Portal with a blue indicator, medium priority messages with a pink indicator and high priority messages with a red indicator.

Finally, specify the date range during which the message will be displayed. If e-mail notifications are enabled for the database, you can select the **Also send this via e-mail** option to send an e-mail copy of the message to the recipient(s).

3.3.2 Actions

Actions are resources that enable you to track the progress for assigned tasks. The same action record can be used multiple times in different locations, if appropriate. Actions can be created, displayed and/or managed: a) via My Portal in all Synthesis applications, b) via the Resource Manager in all Synthesis applications except MPC and c) in FMEAs and DFR plans in Xfmea/RCM++/RBI.

This section focuses on general considerations that will be applicable regardless of which application you are using and where the action is being used. For specific instructions regarding how to create or edit an action in each location, please consult the documentation for that particular feature. The following topics are discussed here:

- Actions as resources - Section 3.3.2.1 (p. 103)
- Action statuses - Section 3.3.2.2 (p. 104)
- Action properties - Section 3.3.2.3 (p. 104)
- Using the Approve, E-mail and Outlook commands - Section 3.3.2.4 (p. 105)

3.3.2.1 Actions as Resources

As with any other Synthesis resource:

- Any changes to the action properties will be reflected in all locations where the resource is used.
- If you remove an action from an FMEA or DFR plan, it will remain in the project unless/until an authorized user deletes it from the database.
- If an action resource is not currently used in an FMEA or DFR plan, an authorized user (i.e., the project owner, the action creator or any user with the “Create/edit/delete all resources” permission) can delete it via My Portal or the Resource Manager.

In addition, the **Trace Usage** icon at the bottom of the window opens the Dependency Viewer, which allows you to see where the resource is used and what other resources are associated with it.

3.3.2.2 Action Statuses

The action status is determined automatically based on the dates defined for the action:

An action is **Overdue** if its due date is prior to today's date.

An action is **Not Started** if its start date is later than today's date.

An action is **In Progress** if its start date is today's date or earlier and its due date is today's date or later.

An action is **Completed** if a completion date has been entered and no approver has been assigned.

An action is **Pending Approval** if a completion date has been entered and an approver has been assigned.

An action is **Approved** if an approval date has been entered.

3.3.2.3 Action Properties

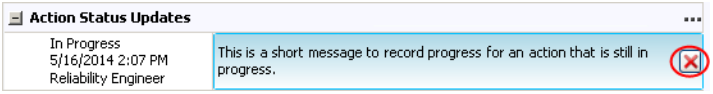
The properties enabled in the Action window (and their display names) will depend on the interface style settings that have been defined for the current project in Xfmea/RCM++/RBI. If you have one of those applications installed and have permission to edit the project properties, you can choose **Project > Management > Configurable Settings > Interface Style** to view or change the settings for the current project.

- In the **Action** area, note the following:
  - The **Action Description** is required.
  - The **Short Description** is a shortened version of the record description that will be used in locations where the full description would be too long to display (e.g., diagrams). If the field is blank when you save the record, the software automatically populates it based on the first 50 characters of the record description. This will not be automatically updated if you later change the full description.
  - For the **Person Responsible**, you can type a description (up to 150 characters) or select a specific database user from the drop-down list. If you select a database user, the action will be displayed in that user's My Portal and he/she can also receive automated e-mail notifications, if applicable.
- In the **Timeline** area, note the following:
  - The **Planned Start Date** and **Planned Due Date** are for bookkeeping purposes only; they are not used in determining the action status.
  - The **Start Date**, **Duration** and **Due Date** fields are used to determine the action status. In addition, once you have entered the start date:
    - If you enter/change the due date, the duration will be updated automatically.
    - If you enter/change the duration, the due date will be updated automatically.

- Entering a **Completion Date** will change the action's status from due/overdue to complete. It also triggers an automated e-mail notification if the database is configured to send e-mails "Upon completion."

- In the **Approval** area, note the following:
  - If you decide to assign an **Approver** to review and approve the action after it is completed, you must select a database user who has the "Approve actions" permission. The action will be displayed in that user's My Portal and he/she can also receive automated e-mail notifications, if applicable.
  - The **Approval Date** will be entered automatically when the assigned approver chooses the **Approve** command.
  - The **Approval Comments** field will not be available until the assigned approver has entered an approval date.
- The **Action Status Updates** area displays any notes that have been added to report progress toward the completion of the action.
  - To add a new note, click the heading and then click the **Add** icon that becomes visible.

    Note that if an action has not yet been completed, you can choose any action status update type from the drop-down list in the Action Status Update window. Choosing **Not Started** or **In Progress** will not affect the action's status. Choosing **Completed** will enter a completion date for the action and will change the action's status to complete. If an action has already been completed, the only action status update type that will be available is **Completed**.
  - To delete an existing note, click inside the field and then click the **Delete** icon that becomes visible. ***There is no undo for delete***.



- The **E-mail Notifications** area allows you to access the Action Notifications window, which identifies the database users who can receive notifications regarding the current action. *See Section 3.3.2.5 on page 106*. These settings apply to automated e-mail notifications (if any) and the Actions page of the My Portal window.
- The **History** area provides information about the creation and last modification of the resource. If the history log has been activated at the project level, you can click the **View Item History** icon to open the Record History Log for the resource.

3.3.2.4 Approve, E-mail and Outlook Commands

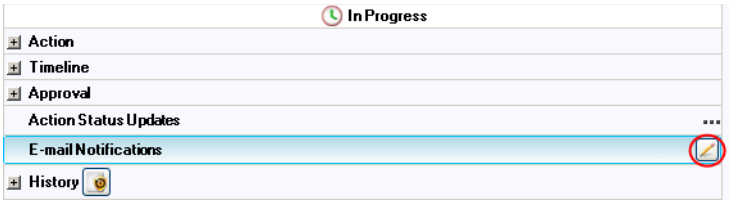
Three commands will be enabled only when applicable. For most applications, they will be available as buttons at the bottom of the window. If you are using Xfmea/RCM++/RBI and working with an action that's used in an FMEA or DFR plan, they will be available from the Ribbon at the top of the window.

- The **Approve** command will be enabled if you are assigned to review/approve the action and a completion date has been entered. When you choose the command, the software automatically enters the approval date, and the comments field becomes enabled so you can add any relevant notes.
- The **E-mail** command will be enabled if an authorized user has defined a valid SMTP server via the E-mail Settings window. This allows you to manually send an e-mail containing the current action details to any valid e-mail address.
- The **Outlook** command will be enabled if Microsoft Outlook is installed on your computer. This provides a quick way to add the task to your Outlook calendar. By default, the event will be scheduled for the action's due date, but you can make modifications before saving the new event to your calendar. Note that you may need to give focus to the Outlook application in order to see the window.

3.3.2.5 Action Notifications Window

The Action Notifications window allows you to specify individual users and/or groups of users who will be able to monitor the progress for the current action. These settings apply to automated e-mail notifications and the Actions page of My Portal.

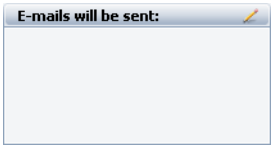
To view or configure these settings, first open the Action window and click the E-mail Notifications heading. Then click the **View/Edit E-mail Notifications** icon that becomes visible.



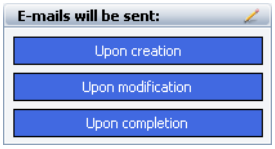
E-mails Will Be Sent

The **E-mails will be sent** area indicates which e-mail notifications, if any, have been enabled for all action records throughout the current database. (Notifications are enabled via the E-mail Settings window, which can be accessed by clicking the Edit icon in this area.) It may look like any of the following three pictures:

E-mail notifications are not enabled for this database



Automated e-mails are not enabled, but you can send e-mails manually



Some automated notifications are enabled; the specific buttons identify which ones

Users and Groups

The rest of the window identifies which users and/or groups are assigned to monitor the current action. These users receive automated e-mail notifications (if applicable) and will also be able to see the action displayed in the My Portal window.

Action Creator: If the **Action creators receive notification** check box is selected in the E-mail Settings window for the database, the user who created the action will receive a courtesy copy of all automated e-mail notifications.

Person Responsible: The user who is assigned to complete the action will receive any automated e-mail notifications that are sent "Upon creation" and/or "Upon modification."

Assigned Reviewer: The user who is assigned to review and approve a completed action (aka the "Approver") will receive any e-mail notifications that are sent "Upon creation," "Upon modification" and/or "Upon completion."

Notification Groups and **Other Users**: The selected notification groups and other individual users will receive courtesy copies (CCs) for all automated e-mail notifications related to this specific action, as long as there is at least one primary recipient (Person Responsible, Approver or Action Creator) for the e-mail.

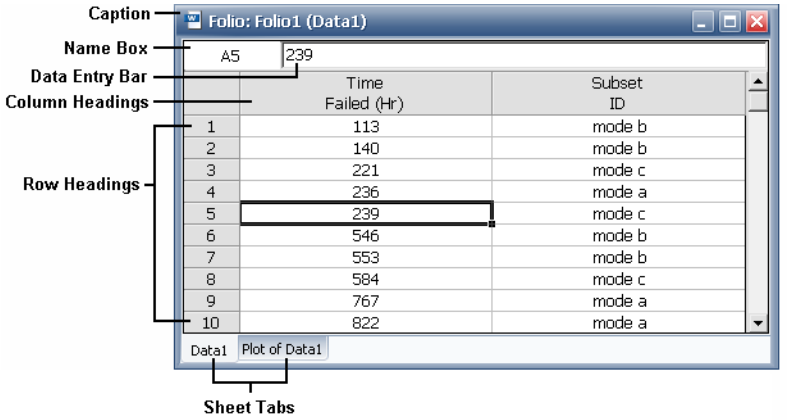
- The list of notification groups can be defined for the current database by user(s) with the "Manage e-mail notifications" permission.
- The list of other users contains all of the user accounts that are defined in the current database.

3.4 Data Sheets

Analysis folios in Weibull++, ALTA, RGA and DOE++ are made up of data sheets. This section describes the basic components of a data sheet and shows how you can resize the columns and rows, and enter formulas.

Components of a Data Sheet

Data sheets in most folio types share the same general components. The following picture shows the basic components of a data sheet using one particular format of the Weibull++ standard folio as an example.



- **Caption** displays the name of the folio and the name of the sheet that is currently displayed.
- **Name Box** displays the location of the currently selected cell by listing the column letter and row number that intersect at the cell's location. Note that the column letter represents the order of the column as it appears in the data sheet.
- **Data Entry Bar** displays the contents of the selected cell, which may be text, a numerical value or a formula. For text or numerical values, you can enter, or edit, the information directly inside the cell or inside the data entry bar. For a formula, you must use the data entry bar for editing; the cell will display only the calculated value.
- **Row Headings** identify the row numbers. You can change the name or number of a row by double-clicking its heading. The row will still retain its order number in the data sheet.
- **Column Headings** indicate the title for the entries in each column. These titles depend on the type of data you are entering. You can change the name of a column by double-clicking its heading.
- **Sheet Tabs** represent the different sheets within the folio. To switch between sheets, click the appropriate tab. You can change the name of a sheet by double-clicking its tab.

Each standard folio can contain up to 256 data sheets with up to 65,536 data rows per sheet. However, the application's speed of execution is inversely proportional to the number of data rows in the current data set. In addition, the number of folios that can be opened at once is dependent upon the amount of memory of your system.

Resizing Columns and Rows

To resize the columns for data sheets in folios in Weibull++, ALTA and RGA, there are two ways:

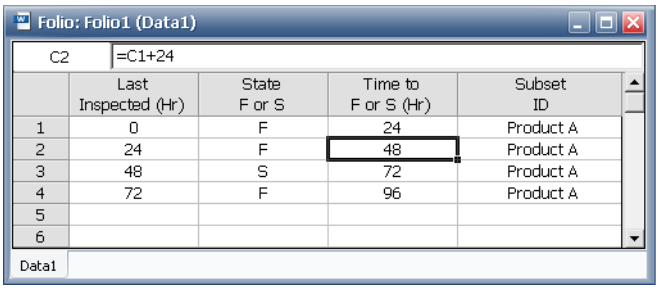
- To resize a single column, move the pointer over the line that borders the column in the column header. The pointer will change to a double-headed arrow. Click and then drag the double-headed arrow to the left or right to resize the column. To save the changes, choose **Home > Edit > Format Selection > Column Width Style > Keep as Set by User**.
- To span the columns across the spreadsheet with equal widths, choose **Home > Edit > Format Selection > Column Width Style > Span Evenly**. To keep the ratio of the column widths to span across the spreadsheet after they have been resized, choose **Span Proportionally**.

To resize the rows, there are two ways:

- To resize a single row, move the pointer over the line that borders the row in the row header. The pointer will change to a double-headed arrow. Click and then drag the double-headed arrow up or down to resize the row.
- To resize several rows all at once, select a cell in each of the rows to be resized and then choose **Home > Edit > Format Selection > Row Height**. This opens the opens the Row Height window, which allows you to set the row height of the selected cells and specify a unit of measurement for the height.

Entering Formulas in the Data Sheet

Except for DOE++, data sheets in Weibull++, ALTA and RGA allow you to enter formulas in columns that do not require entries to be dates, times or text (e.g., you cannot use formulas in the State F or S column in a Weibull++ standard folio, or in the Classification column in an RGA standard folio). For example, in a Weibull++ standard folio, if you have a data set where the units were inspected every 24 hours, you can speed up data entry by creating a formula to add 24 hours to the previous inspection time, as shown next.



There are two types of cell references (locations) when inputting a formula: absolute and relative. By default, the cell reference is relative, meaning that as a formula is copied and pasted to other cells, the cell references in the formula will be adjusted to reflect the new relative location. In the previous figure, the formula in cell C2 is copied and pasted into cell C4. This changes the cell reference in the formula from "=C1+24" to "=C3+24"; therefore, the time value in C4 is 72+24 = 96.

In contrast, an absolute reference does not change when the formula is copied to other cells. Absolute references are designated by placing a dollar sign ($) in front of the row and/or column to be made absolute. If the formula in the example were rewritten to "=$C$1+24", then copying the formula to cell C4 would retain the cell reference to C1 and the time value in C4 would result to 24+24 = 48.

3.5 Change Units

In the current versions of Weibull++, ALTA and RGA, you have the ability to define the time units used for the data set and calculations. For example, if your data set contains failure times in hours, you now have the option to perform calculations using different time units such as years, months, days, etc. The software performs the unit conversion automatically. Authorized users can define the time units that will be available for use in any project within the database and set up the conversion factors. (Refer to the "Manage Units" section in the "Working in Synthesis" chapter for more information.)

The units are displayed in the header of the relevant column in the data sheet. If you wish to change the time units of an existing data sheet, click the **Change Units** icon on the Main page of the control panel.

In the Change Units window, the units that are currently being used in the data sheet will be displayed at the top of the window. In the **Change Units To** field, you can choose any other defined time unit to be used in the data sheet. In the **Conversion Options** area, specify what will happen when the units are changed:

- You can convert the existing data to the new units. For example, if you had a time of 10 in a data sheet that was using hours, that time would become 600 if you converted the data to minutes (10 hours x 60 minutes per hour = 600 minutes). For standard folios, when this option is selected, you can select whether you want to copy the data to a new sheet in the folio and then perform the conversion, leaving the original data sheet unchanged, or convert the data in the current sheet.
- You can leave the data unchanged and just apply the new units. This would be appropriate if, for example, you had entered all of your data and then realized that the data sheet was using different units from the original units in which the times were measured.

3.6 General Spreadsheets

General Spreadsheets can be inserted into any of the standard folios in Weibull++, ALTA and RGA. These provide the same spreadsheet capabilities that are available in the analysis workbook, but they are stored together with an analysis folio. You may prefer to use this reporting tool if you are performing custom calculations based on the data sheets in the same folio and you wish to keep the analyses together with their source data.

To add a general spreadsheet to a folio, right-click the data sheet tab area (the area at the bottom of the window that shows the name of the data sheets in the folio) and choose **Insert General Spreadsheet** on the shortcut menu. This command also appears in the Ribbon tab for the particular type of data folio. For example, to insert a general spreadsheet into a Weibull++ standard folio, you can choose **Life Data > Folio Sheets > Insert General Spreadsheet**.

Unless explicitly stated as being restricted to an analysis workbook, all components and tools that are available in an analysis workbook are also available in a general spreadsheet.

3.7 Control Panels

Many of the analysis folios, diagram sheets and other interfaces in Synthesis applications utilize a control panel that allows you to make required inputs, initiate the desired analysis or simulation and view/access applicable results. (Control panels are not used in Xfmea, RCM++, RBI, MPC and Lambda Predict, which use the Analysis Panel; see Section 3.9.) This section describes some features that are common to most control panels, including:

- Switching between pages in a control panel - Section 3.7.1 (p. 109)
- Hiding/displaying the entire panel - Section 3.7.2 (p. 110)
- Using the Identifiers page to record information about the particular analysis - Section 3.7.3 (p. 111)
- Using the Notifications page to view the calculation/simulation status for the current folio or diagram, and read any relevant warnings or notes (e.g., calculation errors) - Section 3.7.4 (p. 111)
- Using the Utilized Resources page (BlockSim only) to view a list of the Synthesis resources that are being directly used by the blocks in the current diagram - Section 3.7.5 (p. 112)

For information about using the Publishing page to publish or update a model based on the current analysis, please refer to the “Publishing Models” section in the “Resources” chapter.

For information about the control panel commands and options that are relevant for a specific type of folio or diagram sheet, please consult the section for that particular interface.

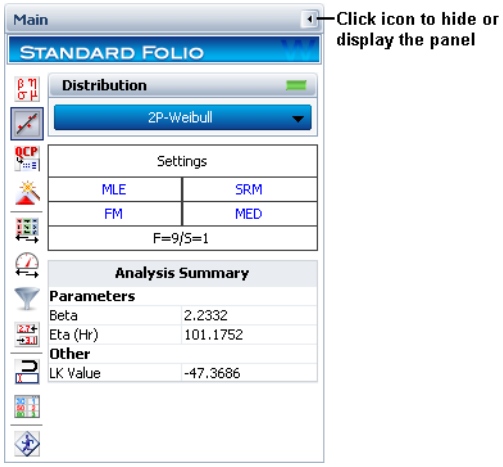
3.7.1 Switching Between Pages

Many control panels and navigation panels (e.g., the navigation panel on the left side of the Resource Manager) contain multiple pages that vary depending on the type of analysis. You can switch between pages by clicking either the large buttons or small icons at the bottom of the control panel.

The large buttons are displayed in a vertical column; the small icons are displayed in a horizontal row. To change between the button types, point to the horizontal splitter bar above the buttons and drag it up or down. If you drag the horizontal splitter bar to the bottom of the control panel, all of the pages will be accessed by small icons. If you drag it as far up as it will go, all of the pages will be accessed by large buttons. Positions in between allow you to use some large buttons and some small icons.

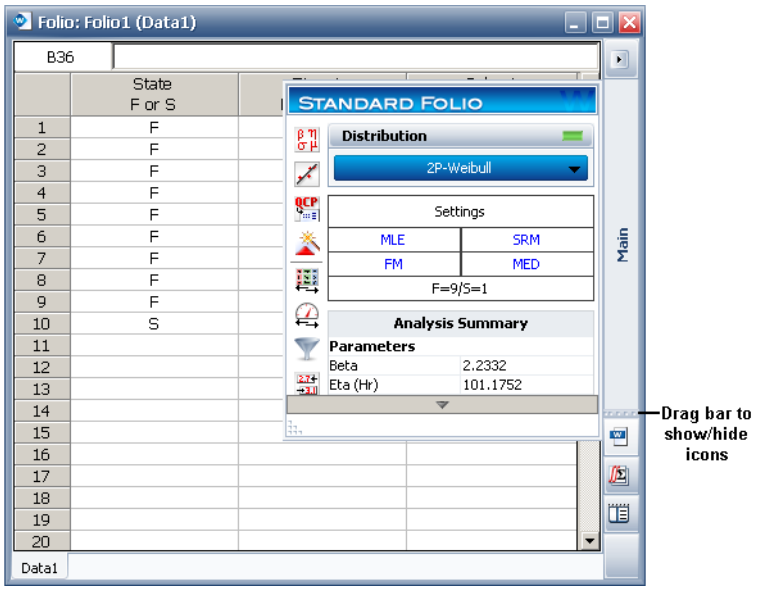
3.7.2 Hiding or Displaying a Control Panel

The control panel can be toggled between hidden and displayed states by clicking the **Hide** or **Show** icon in its title bar, as shown below using a Weibull++ standard folio as an example.



When the control panel is hidden, a bar representing the control panel will be displayed on the right side of the window, as shown next. The label in the vertical bar indicates which page of the control panel is currently active

("Main" in this case). When you click the bar, the page will be displayed temporarily. When you click anywhere outside of the control panel page, it will be automatically hidden again.



3.7.3 Control Panel Identifiers Page

As described in the "Publishing Models" section of the "Resources" chapter, certain analyses can be published as models that become resources available for other analyses. For example, you could perform a life data analysis in Weibull++, publish the results as a model, and then use that model to describe the reliability of an item in RCM++.

The Identifiers page of the control panel allows you to enter specific information (e.g., name, part number, etc.) that will be associated with the model when it is published. This will help you to search for the model when you need to use it in another analysis.

To edit the labels for these user-defined identifier fields at the database level, choose **File > Manage Repository > Global Identifiers**.

To edit the labels at the project level, choose **Project > Management > Edit Project Properties** and go to the Global Identifiers tab.

You can enter up to 1,000 characters of text in these fields; if the text is too long to be displayed in the field, you can click the arrow at the bottom of the field to show the remaining text.

Note: If you are working in a secure database, the ability to edit the labels of the user-defined fields is typically restricted to users with the "Manage other repository settings" permission.

3.7.4 Control Panel Notifications Page

The Notifications page is available in the control panel for certain types of analyses. The **Current Status** field provides information on the analysis status of the folio. This information will match the color of the status icon used for the Notification page; a green light indicates that the folio has been analyzed, and a red light indicates that the folio has not been analyzed since changes were last made to it.

The **Latest Notifications** area of the page can display warnings and errors generated during analysis. To clear the display, click the **Clear** icon.

You can copy a warning to the Clipboard by clicking the warning and then clicking the **Copy** icon. The text will then be available to be pasted into other windows or other applications.

You can also export the contents of the **Latest Notifications** area to an Excel file by clicking the **Export to Excel** icon.

3.7.5 Control Panel Utilized Resources Page

In BlockSim only, the diagram control panel contains a Utilized Resources page. This page displays a grouped list of all of the resources that are used directly by blocks in the diagram (i.e., not applied via any other resource or tool). For example, a universal reliability definition (URD) that is assigned to a block will be displayed in this list, but the model assigned to the URD will not be shown, nor will any tasks assigned to the URD. Similarly, if a block belongs to a maintenance group, that maintenance group will be shown in the list. However, if a block has a state change trigger, the maintenance group(s) used for that trigger will not be shown in the list unless some block in the diagram belongs to them. Refer to the "Synthesis Resources" chapter for more information.

This list is not updated automatically; if you have made changes to the diagram, you can click the **Refresh** icon to update the list.

You can double-click a resource in the list to open its properties window for viewing or editing. You can also click the **Resource Manager** icon to view/manage all of the resources available in the project.

3.8 Results Window

The Results window is used in many Synthesis applications to show detailed calculation results. From this window, you can edit the results, copy the results to the Clipboard or print the results.

The toolbar contains the following tools:

Cut cuts the selected text to the Clipboard. Data stored in the Clipboard can be pasted into this and other applications.

Copy copies the selected text to the Clipboard. Data stored in the Clipboard can be pasted into this and other applications.

Paste pastes the contents of the Clipboard into the current control.

Page Setup allows you to specify printing options using the Page Setup window.

Print Preview allows you to preview the current page before it is sent to the printer.

Print sends the current document to the printer.

Send to Excel allows you to save the current selection as a Microsoft Excel (*.xls) file and then opens that file in Excel, if it is installed on your computer.

Note that the Results window may contain multiple sheets, accessible via the page index tabs at the bottom of the window.

3.9 System Panel and Analysis Panel

Some Synthesis applications use two convenient panels to manage all of the analysis information in a particular project (Xfmea, RCM++, RBI and MPC) or in a particular prediction folio within the project (Lambda Predict).

The first panel (called the *System panel* or the *System Hierarchy panel*) allows you to build simple or complex multi-level configurations that contain the items you plan to analyze. The second panel (called the *Analysis panel* or the *Properties panel*) contains all of the properties and analyses for the item that is currently selected.

This section discusses the ways in which you can configure the layout of these two panels to fit your particular workspace preferences.

Change Orientation

You can display the panels side-by-side or with one panel above the other.

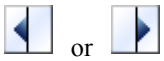
To switch between layouts, choose **View > Workspace Layout > Change Orientation**.

Resize a Panel

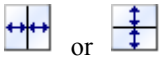
You can resize the panels by dragging and dropping the vertical (or horizontal) separator into the desired position.

Hide a Panel

You can also completely hide either panel so that the other can fill the available space. Choose **View > Workspace Layout > Hide** [**System Panel/System Hierarchy**] or [**Analysis Panel/Properties**].


or

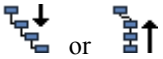
When you wish to return to the two-panel layout, choose the **Split Panels** command.


or

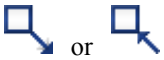
Expand or Collapse Nodes

When the panel presents data in a hierarchical tree configuration (e.g., the system hierarchy, FMEA hierarchy, etc.), the View tab provides several flexible options for expanding and collapsing the nodes (branches) that are currently displayed.

To expand or collapse all branches at the same time, choose **Expand Tree** or **Collapse Tree**.

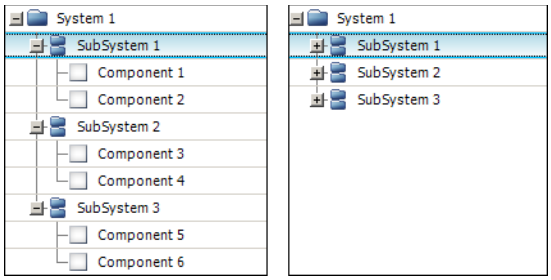


To expand or collapse a specific branch, you can click the + or - icons, or select the item and choose **Expand Node** or **Collapse Node**.



To collapse all branches in the tree to a specific level, select any item at the desired level and choose **Collapse to Level**.

As an example, the following pictures show how the hierarchy collapses to the second (subsystem) level.



Selecting Which Columns to Display

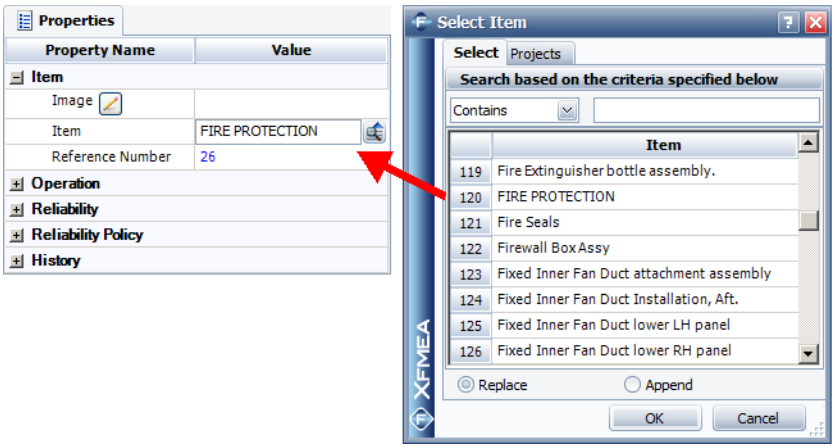
When applicable, you can hide, display or reorder the columns shown in a particular type of hierarchy by right-clicking any column header and choosing **Customize Columns**. These settings are stored per computer/username, and different users may have different display preferences without affecting the stored data. The same preferences can also be managed from the relevant page of the Application Setup.

3.10 Select Existing Window

The Select Existing window provides a list of existing descriptions that might apply to the current text field. It is used extensively in applications such as Xfmea, RCM++, RBI and MPC, which facilitate data management and reporting for analyses that are based on lengthy text descriptions of intended functions, potential failure modes and recommended corrective actions. This feature is also available when applicable in other Synthesis applications.

Whenever the Select Existing window is available for a text field, the icon will be displayed.

The following picture shows an example of how a Select Existing window may be used in Xfmea to populate a text field. The list in the example is generated automatically from text that has been entered in the **Item** field for other projects within the database.



Depending on the type of text field, the Select Existing window may have two or three tabs. The **Select** tab displays a list of existing text descriptions for the selected text field. The **Project** tab and **Phrase Set** tab display a list of all the projects/phrase sets in the current database, allowing you to select the sources for the descriptions that will be displayed in the Select tab.

Filtering the Text Descriptions in the Select Tab

To insert one of the descriptions into the current field, double-click the row or select the row and click **OK**. If the **Replace** option is selected, the description will replace the current contents of the field. If the **Append** option is selected, the description will be added to the end of the current contents of the field.

The descriptions are sorted alphabetically. When the list (table) has focus, you can type a letter to move to the first description in the list that begins with the letter. You can also filter these descriptions by entering one or more keywords into the input box in the filtering area at the top of the window. You can specify how to match the specified keywords using the drop-down list.

- **Contains** returns descriptions that contain the exact string entered. For example, if you type “provides adequate structural” in the input box, then all instances that contain that phrase will be found. This includes descriptions such as “Provides adequate structural support,” but does not include descriptions such as “Provides structural reinforcement,” “Allows adequate air flow,” etc.
- **Contains Any** returns descriptions that contain any of the keywords entered. Keywords should be entered as a comma-delimited list. For example, if you type “provides, adequate, structural” in the input box, then all instances that contain any of those words anywhere within the field will be found. This includes descriptions such as “Provides structural reinforcement,” “Allows adequate air flow,” “Provides adequate structural support,” etc.
- **Contains All** returns descriptions that contain all of the keywords entered. Keywords should be entered as a comma-delimited list. For example, if you type “provides, adequate, structural” in the input box, then all instances that contain all of those words anywhere within the field will be found. This includes descriptions such as “Provides adequate structural support” and “Reinforces structural elements to provide adequate stability,” but does not include descriptions such as “Provides structural reinforcement,” “Allows adequate air flow,” etc.
- **Begins With** returns descriptions where the keyword(s) appear at the start of the field. For example, if you type “provides adequate” in the input box, then only the instances in which that phrase appears at the start of the field will be found. This includes descriptions such as “Provides adequate structural support,” but does not include descriptions such as “Provides support adequate for load of 500 kg,” “Fan provides adequate air flow,” etc.
- **is** returns descriptions that are identical to the exact string entered or matches the whole field. For example, if you type “provides adequate support” in the input box, then any description that contains that phrase, and only that

phrase, will be found. The description “Provides adequate support” would be returned. Descriptions such as “Provides adequate structural support” and “Provides adequate support for load of 500 kg,” would not be included.

As you type in the **Find** field, the list is updated based on the filter criteria that you specified.

Selecting Data Sources in the Projects/Phrase Sets Tab

The Projects tab is available for nearly all types of items or records, while the Phrase Sets tab is available only for FMEA records (Xfmea, RCM++ and RBI). Each time you open a Select Existing window, it will use the same project sources/phrase sets that were selected the last time you used the utility.

In the Project tab, you can select the **Current** option to display the relevant descriptions from the current project only.

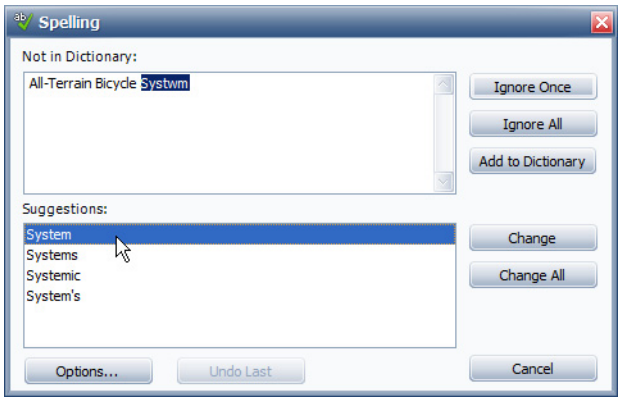
For both the Project and Phrase Set tabs, you can select either the **All** option, which displays the relevant descriptions from all projects/phrase sets in the current database, or the **Selected** option, which allows you to specify which projects/phrase sets to draw the descriptions from. When this option is selected, a list of the projects/phrase sets in the current database will become enabled. If the check box for a project/phrase set is selected, the relevant descriptions from that project will be displayed.

3.11 Check Spelling

Synthesis applications offer the ability to check the spelling of your text. This utility is available in different places within the software. For windows that offer the spell check functionality (e.g., Project Properties window, etc.), the Check Spelling icon will be displayed within the window itself. For data sheets, spreadsheets and system hierarchies that offer the functionality, the Check Spelling icon will be available on the Ribbon and you can access it by choosing **Home > Edit > Spelling**.

Note: In system hierarchies (Xfmea, RCM++, RBI, MPC and Lambda Predict), the utility will check the property fields of the selected item only. This means that sub-items and text in analysis tabs, such as the FMEA tab in Xfmea, are not checked.

Whenever text that is not in the application's dictionary is found, the Check Spelling window displays the text in question and offers a list of suggestions to verify the error or confirm the correction, as shown in the following example.



- **Ignore Once** ignores the current instance of the highlighted word but continues to highlight it if the same word appears again, while **Ignore All** ignores all instances of the word in the current form.
- **Add to Dictionary** adds the highlighted word to the dictionary on your computer so the spell checker will not treat is as a misspelling. The dictionary file is saved in the default Documents folder on your computer (e.g., My Documents\ReliaSoft\Dictionaries).

- **Change** replaces the current instance of the highlighted word with the word that is currently selected in the **Suggestions** area. If no word is selected, the first one will be used. Double-clicking a word in the **Suggestions** area is another way to change the highlighted text.
- **Change All** replaces all instances of the highlighted word with the word that is currently selected in the **Suggestions** area. If no word is selected, the first one will be used.
- **Options** opens the Spelling Options window, which provides additional settings to how the spell checker functions:
  - The **General options** area provides a list of the types of text that can be ignored by the spell checker (e.g., numbers, e-mail addresses, etc.). Select or clear each check box to specify how the spell checker will work on your computer.
  - In the **Edit custom dictionary** area, the **Edit** button opens a window that displays all of the words that you have added to the custom dictionary on your computer. You can edit the custom dictionary by adding or removing words in the list and then clicking **OK**.
  - **International dictionaries** allows you to choose the language of the dictionary that will be used on your computer.
- **Undo Last** reverses the last change that was made.

3.12 Find Tool

Most Synthesis applications offer the ability search for (and in some cases, replace) specific text or values in spreadsheets and in certain analysis folios. When applicable, the Find icon will be available on the Ribbon and you can access it by choosing **Home > Edit > Find** or **Find and Replace**.

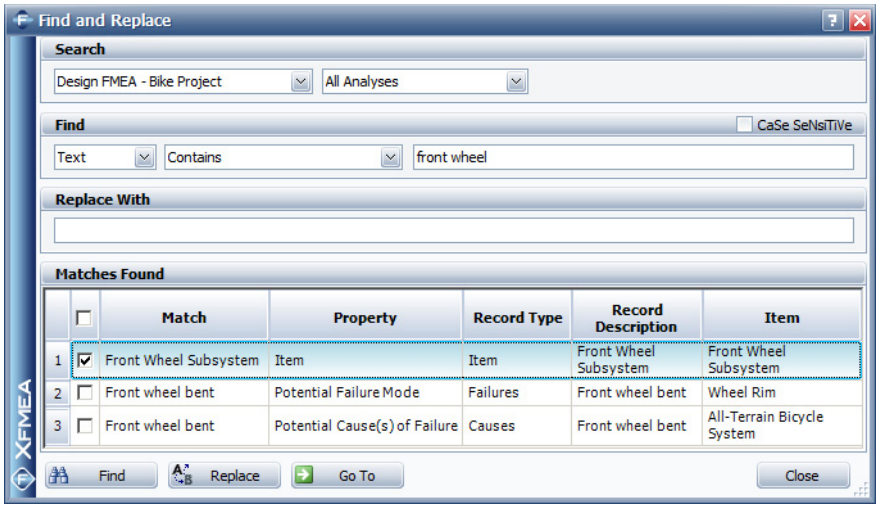
In the Find window, use the search bar to type the text or value you wish to locate. Optional settings that apply to the current sheet may also be displayed and can include any of the following:

- The **Match case** check box limits the search to text that have the same case. For example, the search term “teXt” will not find “Text” because the case does not match.
- The **Match whole field only** check box limits the search to text/numbers that are identical to the search term. For example, the search term “capacitor” will not find “filter capacitor” because the search term makes up only part of the field.
- The **Find entire cells only** check box is similar to the **Match whole fields only** check box, but applies to spreadsheets. In addition, spreadsheets provide the option to specify the order in which to search, either **By Rows** or **By Columns**; and the option to limit the search to within equations (**Formulas**) or within the results of an equation (**Values**).
- The **Replace** button extends the basic Find functionality by allowing you to replace the text or value that matches your search. When you click the button, the **Replace With** field will become visible so that you can type the text or values that will replace the found text/value.
  - To replace all instances without review, click **Replace All**.
  - To review the instances before replacing the material, click **Find Next** to locate the first place the material occurs, then either click **Find Next** to leave the current match as-is and find the next occurrence or click **Replace** to replace the current match and find the next occurrence.

> **Tip:** To ensure that all instances of searched for text are replaced (including multiple instances within the same cell), you may need to click **Replace All** several times.

3.12.1 Searching FMEA Records in Xfmea, RCM++ and RBI

In Xfmea, RCM++ and RBI, the expanded Find utility gives you the flexibility to search for FMEA records within specified projects in the current database. The following picture shows an example of the utility in Xfmea.



The **Search** area contains drop-down lists that allows you to specify which project, analyses or record properties to search. If you want to search all data in the current project, accept the default **All Analyses** option. If you want to search in a specific project and/or in a specific analysis or record property, choose different options from the drop-down lists.

Note: For options in drop-down lists (e.g., the Action Category list) or for options in rating scales (e.g., the Initial Severity for effects) you can search for and replace records in the current project only. For example, this capability might be used if you have created a process FMEA with data transferred from an existing design FMEA and you need to update all of the analysis data in the project to match the new settings. In this case, in the design FMEA's Action Category drop-down list, the actions that were assigned to the “Design” group will be handled in the process FMEA by the “Manufacturing” group. To do so, you can use the Find and Replace window to search for all records that have been assigned to the old category (e.g., “Design”) and replace with the new category (e.g., “Manufacturing”).

The **Find** area allows you to specify the text, date or number that you want to search for. The first drop-down list allows you to choose the type of data (i.e., text, date or number) and the second drop-down list displays the options that will determine a match. When you are searching for text, you can select or clear the **CaSe SeNsiTiVe** check box to determine whether the search will consider the case. For example, if this option is selected, the search term “teXt” will not find “Text” because the case does not match.

Note: You can find fields that have no data in them by choosing **is empty** in the Find area.

The **Replace With** area allows you to specify the text, date or number that can be used to replace the existing data for any matches that you select.

After you have made your selections in the Search and Find areas, click the **Find** button. The **Matches Found** area will display a list of any records that match your search criteria.

If you want to go to a specific record, select the match and click the **Go To** button or double-click the match. The project will open with the matching record selected.

If you want to replace the text, date or number with the data that you have specified in the **Replace With** area, select the match(es) and click the **Replace** button. To select all matches for replacement, click the check box in the table header. To select individual matches, click the check box in each row.

3.13 Quick Parameter Estimator (QPE)

The Quick Parameter Estimator (QPE) allows you to estimate the parameters of a distribution based on information you have about the reliability of a product, the probability of an event occurring or the typical duration of a task. In all Synthesis applications, you can open the QPE from the Model window or Model Wizard by clicking the **QPE** icon.

In Weibull++/ALTA, it's also accessible from the Home tab of the Ribbon and several analysis interfaces (including the Weibull++ standard folio, the Weibull++ Monte Carlo and SimuMatic utilities, and the Expected Failure Times Plot).

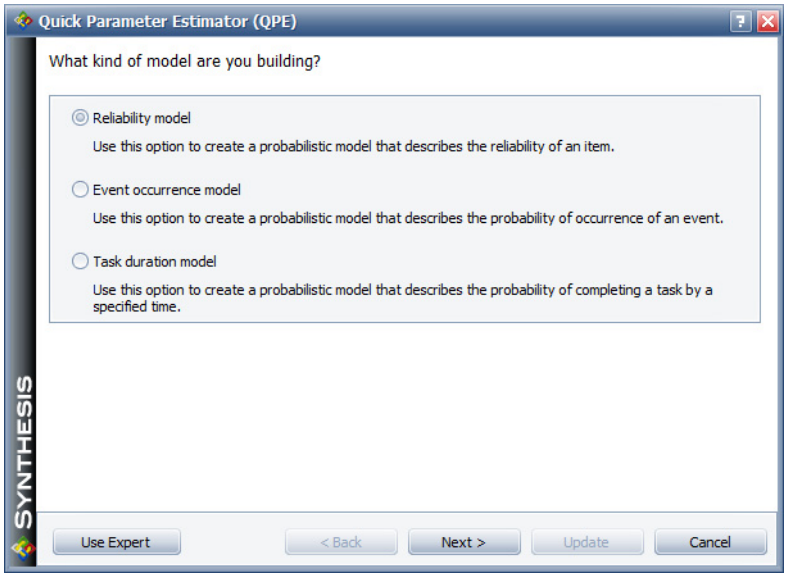
The QPE includes a Wizard view and an Expert view, described below. You can toggle between the different views by clicking the **Use Expert** or **Use Wizard** button at the bottom-left corner of the window.

- The Wizard view automatically selects a distribution and estimates that distribution's parameters based on your responses to a series of questions. The first page of the Wizard view asks you to choose among three different types of models that you can build with the QPE. Subsequent pages will ask more specific questions related to your selected model.
- The Expert view allows you to estimate the parameters of a distribution using either two unreliability values at specified times or one unreliability value and the other parameter(s) of the distribution. Unlike the Wizard view, you must select a distribution to solve the parameter(s) for.

What's Changed? In previous versions of some ReliaSoft applications, the Expert view of the Quick Parameter Estimator was known as the Parameter Experimenter.

3.13.1 Wizard View

To use the Wizard view of the Quick Parameter Estimator (QPE), simply follow the prompts on each page. The first page will present and describe three different types of models that you can build with the QPE.



Below are the three models and the kind of information you'll need to provide in subsequent pages.

To build a **Reliability model** you need to provide information about:

- How age affects the product's reliability
- The product's intended design life and the estimated warranty time
- Best-case, worst-case and most likely unreliability estimates for the product at the end of the design life and warranty time

To build an **Event occurrence model** you need to provide information about:

- Whether age affects the probability of the event's occurrence
- Best-case, worst-case and most likely estimates for how often the event will occur

To build a **Task duration model** you need to provide information about:

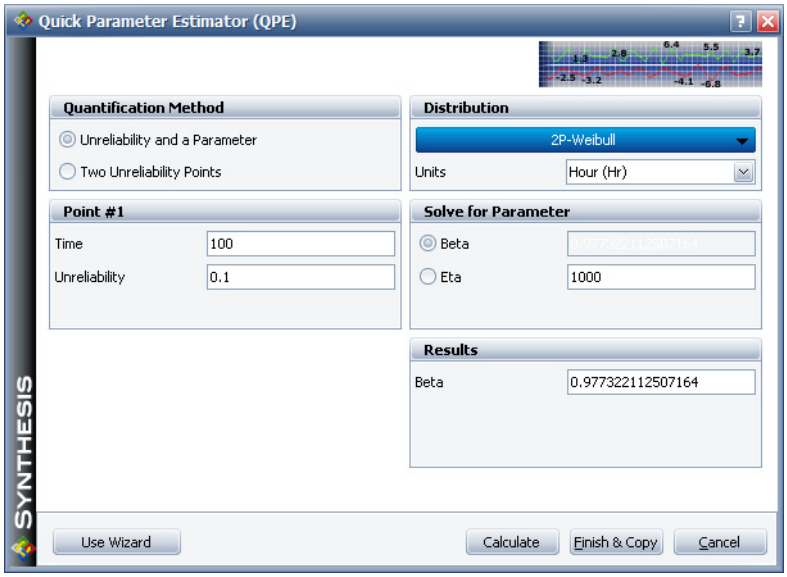
- Best-case, worst-case and most likely estimates for how long it will take to complete the task

The last window will display the tool's selected distribution and the calculated parameters. It will also provide one of the two options described below.

- If you opened the QPE from an analysis folio or utility, the **Update** button will be available in case you wish to update that window using the selected distribution, calculated parameters and selected time units.
- In Weibull++/ALTA, if you opened the QPE from the Ribbon, the **Finish & Copy** button will be available in case you wish to copy the parameters results to the Windows clipboard.

3.13.2 Expert View

To use the Expert view of the Quick Parameter Estimator, click the **Use Expert** button at the bottom-left corner of the window. The window will appear as shown next.



Follow the steps below to use the Expert view:

- Choose a distribution from the **Distribution** drop-down list. This is the distribution that you will solve the parameter(s) for. Then choose the appropriate time units from the **Units** drop-down list.
  - If you are not sure which distribution to select, consider using the Wizard view of the QPE instead.

- Your choice of time units applies to all time inputs and applicable parameters of the distribution (such as the eta parameter when the Weibull distribution is used).

- Select an appropriate option in the **Quantification Method** area. The following two methods are available:
  - The **Unreliability and a Parameter** method solves for one parameter of the distribution. It requires one unreliability point (i.e., an unreliability value at a specified time) and the values of all the parameters of the selected distribution except the parameter you will solve for.
    - If you select this method, you must then enter one unreliability point in the **Point #1** area. For example, if you believe that your product has an unreliability of 10% at 100 hours, then you would enter **100** in the **Time** field and **0.10** in the **Unreliability** field.
    - In the **Solve for Parameter** area, select the parameter that you wish to solve for. The remaining parameters in this area will have input fields enabled. Enter the known values of these parameters.
  - The **Two Unreliability Points** method solves for all the parameters of the distribution. It requires two unreliability points that you will provide in the **Point #1** and **Point #2** areas.

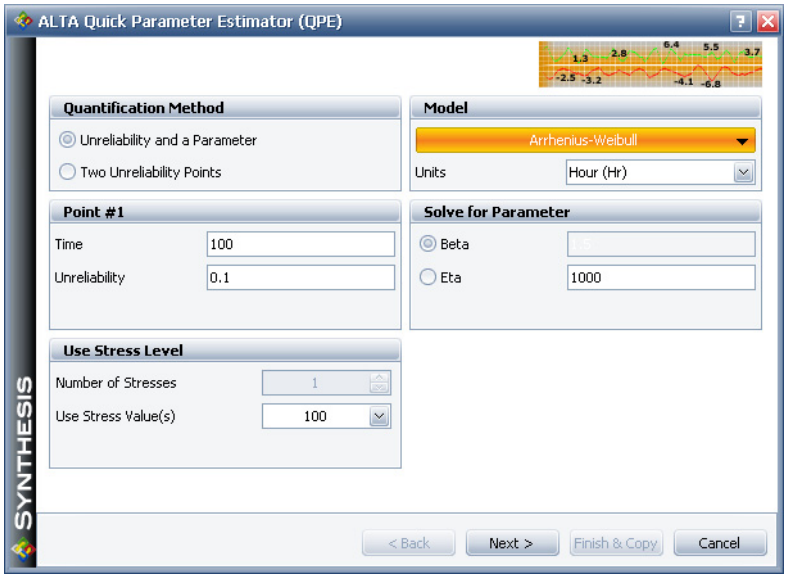
> **Tip:** If you are building an event occurrence or task duration model, you can treat an "unreliability" point as the probability that an event will occur or that a task will be completed by a specified time. For example, if you are modeling the probability of an event's occurrence and you believe that there is a probability of 30% that the event will occur before 150 hours, then you would enter **0.3** in the **Unreliability** field and **150** in the **Time** field.

- Click **Calculate** to solve for the unknown parameter(s).
  - If you opened the QPE from an analysis folio or utility, you may click the **Update** button if you wish to update that window using the selected distribution, calculated parameters and selected time units.
  - In Weibull++/ALTA, if you opened the QPE from the Ribbon, you may click the **Finish & Copy** button if you wish to copy the results to the Windows Clipboard.

3.13.3 ALTA Version of the QPE

In ALTA only, another version of the Quick Parameter Estimator (QPE) allows you to estimate the parameters of a model based on information you have about the reliability of a product at normal and accelerated stress levels. In ALTA, you can open the ALTA QPE from the Ribbon by choosing **Home > Tools > Quick Parameter Estimator (ALTA)**.

The window will appear as shown next.

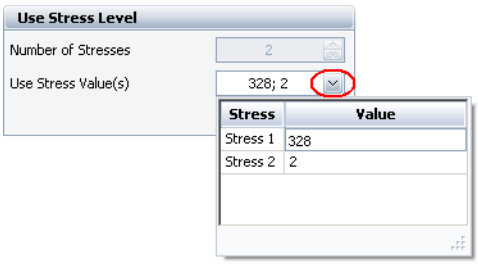


You can also open the ALTA QPE from the ALTA standard folio (by leaving the folio's data sheet empty and clicking **Calculate**), as well as the ALTA Monte Carlo and SimuMatic utilities.

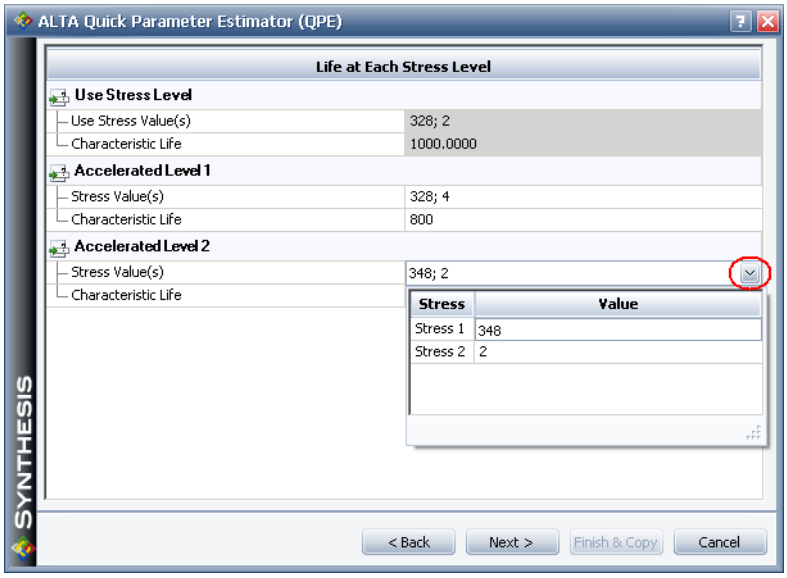
Follow the steps below to use the ALTA QPE:

- The first page of the window will be used to estimate the product's mean life under normal stress conditions.
  - Choose a model from the **Model** drop-down list. This is the model that you will solve the parameter(s) for. Then choose the appropriate time units from the **Units** drop-down list. Your choice of time units applies to all time inputs and applicable parameters of the model (such as the eta parameter when the Weibull distribution is used).
  - Select an appropriate option in the **Quantification Method** area. The following two methods are available:
    - The **Unreliability and a Parameter** method solves for one parameter of the model. It requires one unreliability point (i.e., an unreliability value at a specified time) and the values of all the parameters of the selected model except the parameter you will solve for.
      - If you select this method, you must then enter one unreliability point for normal use conditions in the **Point #1** area. For example, if you believe that your product has an unreliability of 10% at 100 hours under normal use conditions, then you would enter **100** in the **Time** field and **0.10** in the **Unreliability** field.
      - In the **Solve for Parameter** area, select the parameter that you wish to solve for. The remaining parameters in this area will have input fields enabled. Enter the known values of these parameters.
    - The **Two Unreliability Points** method solves for all the parameters of the model. It requires two unreliability points under normal use conditions that you will provide in the **Point #1** and **Point #2** areas.
  - In the **Use Stress Level** area, you must enter the stress level that the product will experience under normal conditions. In multi-stress situations, this stress level will be a combination of stress values for each stress type.
    - If the **Number of Stresses** field is enabled, enter the number of stresses that will be used in your model. (Note that different models have different requirements for the number of stresses that can be used.)

- Click the arrow inside the **Use Stress Value(s)** field. In the table that appears, enter a stress value for each stress type. If there are multiple stress values, they will appear in the field separated by semicolons.



- Click **Next >** to go to the Life at Each Stress Level page. The **Use Stress Level** area of the page will display the product's mean life at the specified use stress level based on your previous inputs. In the **Accelerated Level 1** area, you must enter the estimated mean life for the product at an accelerated stress level.
  - To enter the accelerated stress level, double-click inside the **Stress Value(s)** field. In the table that appears, enter the stress value for each stress.



  - In the **Characteristic Life** field, enter the product's characteristic life at the associated accelerated stress level.
    - For the Weibull distribution, the characteristic life is equal to the value of the eta parameter (i.e., the time at which unreliability = 63.2%).
    - For the lognormal distribution, it is equal to Exp(Log-mean) (i.e., the time at which unreliability = 50%).
    - For the exponential distribution, it is equal to the mean life.

Note: The number of accelerated stress levels you must provide life estimates for will equal the number of stresses that will be used in the selected model. For example, if the model uses two stress types, as shown above, then there will be an **Accelerated Life 2** area in which you must provide a characteristic life estimate for a second accelerated stress level.

- Click **Next >** again to see the calculated parameters.
  - If you opened the QPE from the Monte Carlo or SimuMatic utility, you may click the **Update** button if you wish to update that window using the selected model, calculated parameters and selected time units.

- If you opened the QPE from the Ribbon, you may click the **Finish & Copy** button if you wish to copy the results to the Windows Clipboard.

3.14 History Logs

By default, all Synthesis applications will automatically store and display information about when particular portions of the analysis were created and last updated, and by whom. For example, you can view creation and modification information for system hierarchy items in Xfmea/RCM++/RBI on the Properties tab in the Analysis panel. You can access similar information about folios and diagrams in Weibull++, BlockSim and similar applications by selecting the project item in the current project explorer and choosing **Project > Current Item > View Item History**.

If desired, you also have the option to activate a full history log for any Synthesis project that stores a more detailed history of the specific changes. (In secure databases, you must be the project owner or have the "Manage all projects" permission to do this.) To activate this log for the current project, choose **Project > Management > History Log**.

For any other project, right-click the project in the project list and choose the command on the shortcut menu. In the Project History Log window that appears, click **Activate**.

Note: Keeping a full history log will increase the size of the database and will cause a slight degradation in performance.

Once this log has been activated, there are two interfaces that display information about the changes.

- The **Project History Log** displays changes that were performed throughout the entire project, with any of the Synthesis applications. *See Section 3.14.1 on page 124*.
- The **Record History Log** displays changes that were performed for a specific folio, system hierarchy item, FMEA record, resource, etc. *See Section 3.14.2 on page 125*.

It is also important to note that you have the option to create **Archives** for log entries that occurred before a specific point in time, as a way to make sure that the logs are more manageable to work with. The archived entries can still be viewed in the Project History Log if you specifically select to view them, but they will never be displayed in the Record History Log. *See Section 3.14.3 on page 125*.

You can deactivate the history log at any time from within the Project History Log. When you do this, the recorded data is retained unless you select to clear it in the confirmation message that appears.

3.14.1 Project History Log

To access the Project History Log, choose **Project > Management > History Log**. (In secure databases, you must be the project owner or have the "Manage all projects" permission to do this.)

In addition to allowing you to activate or deactivate the full history log for a particular project, this window also displays all of the changes that were performed throughout the entire project, with any of the Synthesis applications. Use the **Filter Based On** area on the right side of the window to determine which changes will be displayed at any given time. For example, you might choose to display a list of all changes that were made by Joe User in the last week, or a list of all changes that were made by any user in Weibull++ today, and so on.

If desired, you can click **Send to Excel** to export the data that is currently shown in the table to a new *.xlsx file.

3.14.2 Record History Log

Once a history log has been activated for the project, you can access the history log data for any item that is subsequently added or edited. This includes, but is not limited to, project items (e.g., folios, diagrams, MultiPlots, etc.); items shown in the system hierarchy in Xfmea/RCM++/RBI and Lambda Predict; analyses in Xfmea/RCM++/RBI; records for analyses such as FMEAs; and certain resources. Depending on the type of item you are working with, you may access the record history log in various ways.

- For items shown in the current project explorer, select the item and choose **Project > Current Item > View Item History** or right-click the item and choose the command on the shortcut menu. Note that if the history log has not been activated, this command will instead display a message that displays the date and time the item was created and the name of the creator, along with the date and time the item was last modified and the name of the user who made the modifications.

- For analyses in Xfmea/RCM++/RBI, choose the **History Log** command in the Tools group of the Ribbon tab relevant to the analysis or right-click the analysis tab in the Analysis panel and choose the command on the shortcut menu.

- For other records, such as items shown in the system hierarchy, resources, etc., click the **View Item History** icon in the item's properties.

The Record History Log provides information on changes made only to the selected item, and shows only those entries that have not yet been archived. If desired, you can click **Send to Excel** to export the data that is currently shown in the table to a new *.xlsx file.

3.14.3 Creating and Managing Archives

Within the context of the history log, an archive is a subset of the full project history log that contains older entries that are no longer actively of interest. For example, if you want to limit the amount of information that is visible in the log, you might choose to archive all entries that occurred more than one week ago. Entries that have been archived can still be viewed in the Project History Log if you specifically select to view them, but they will no longer be visible in the Record History Log windows.

To create a new archive, specify a time frame in the **Archive Current Entries Through** area and click the Archive button. For example, if you choose to archive entries through **One Week Ago**, all entries in the history log that are dated as of one week ago or earlier will be stored in an archive that is named according to the most recent date in the archive.

To view the entries that have been archived, select the archive of interest from the **Archives** drop-down list in the **Filter Based On** area.

Note: When a restore point is created for a project, all current entries in the history log are automatically archived. *See page 71 in Chapter 2.*

To delete an archive, select the archive from the **Archives** drop-down list in the **Filter Based On** area and click the **Delete** icon that appears in the field.

3.15 Attachments

All Synthesis applications allow you to attach URLs and/or files that were created in other applications. This helps you to keep supporting documentation all together in the same place with your analysis. The locations where you can attach files will vary depending on which Synthesis application you are using. All applications support attachments for the project and for most types of Synthesis resources. Some applications (such as Xfmea, RCM++, RBI, Lambda Predict and MPC) support attachments for individual records in a system hierarchy or analysis hierarchy. BlockSim and RENO support attachments for individual blocks in diagrams and flowcharts.

In all Synthesis applications, attachments can be either linked or embedded.

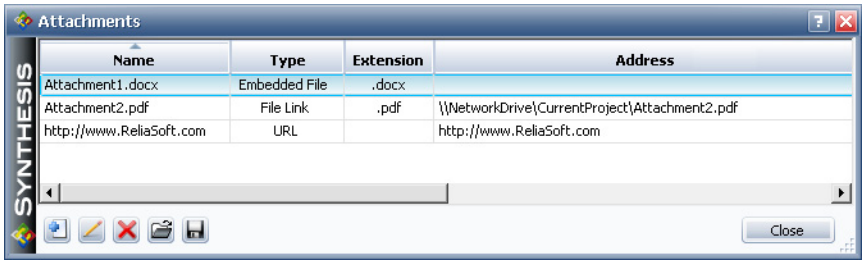
- **Linked** attachments may be URLs or files. For these attachments, the software stores the path to the URL or file. This allows you to open the resource in its original location (e.g., Internet, intranet or network directory), provided that the necessary software is installed on your computer and the link is valid. The files themselves are not stored inside the database.
- **Embedded** attachments are always files (i.e., you cannot embed a URL). For these attachments, the software stores a complete copy of the attached file inside the database. Please note that embedded attachments will increase the file size of the database.

Two windows are used to work with attachments. The Attachments window allows you to manage all of the URLs/files attached to a particular location (e.g., project, resource, hierarchy record or block).

The Add/Edit Attachment window is used to specify the details of an individual file or URL, and will be displayed whenever you choose to create or edit an attachment.

3.15.1 Attachments Window

The Attachments window displays a list of all the URLs/files that are attached to a particular location, as shown next.



Click a column header to sort the attachments by that column; click the column header again to reverse the direction of the sort.

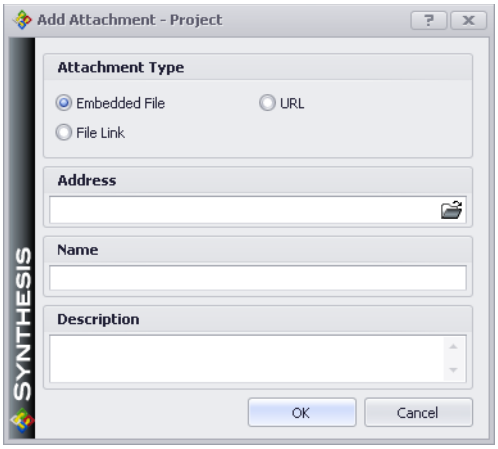
The icons at the bottom of the window allow you to **Add** a new attachment ( ), **Edit** the properties of the selected attachment ( ), **Delete** the selected attachment ( ), **Open** the selected attachment ( ) or **Save** a copy of the selected attachment ( ).

You can also double-click a row to open the selected attachment, or press **DELETE** to remove it.

Note that when you delete an embedded file, the actual file is deleted from the database and this cannot be undone (unless you happen to have a saved backup (*see page 69 in Chapter 2)* or restore point (*see page 71 in Chapter 2)* that you could roll back to). When you delete a file link or URL, this simply removes the link, leaving the original file untouched.

3.15.2 Add/Edit Attachment Window

The Add/Edit Attachment window is displayed whenever you add or edit the properties of an attachment.



Three types of attachments are supported:

- With an **embedded file**, the file is stored directly in the database and it will be accessible to any user who has the required software installed.
- With a **file link**, only the pathname/filename is stored in the database so the file will only be accessible to users who have the required software installed and have access to the specified network folder.
- With a **URL**, you can specify any web-based resource on the Internet or a private intranet (e.g., a web-based document management system, FRACAS incident report, etc.). The application will attempt to open the link in the user's default web browser.

If you are attaching a file, the Address field will display a **Browse** icon ( ) so you can select the file to be attached.

If you are attaching a URL, the **Name** field will be populated automatically as you type the URL into the **Address** field. This is for your convenience only and you can specify a different name if desired.

3.15.3 Attachment Locations

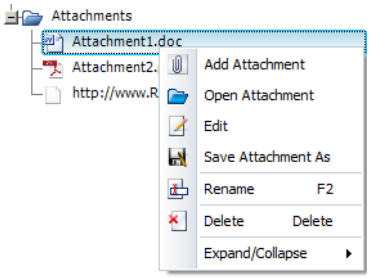
As mentioned above, the locations where you can attach files will vary depending on which Synthesis application you are using.

Tip: In most cases, the caption bar in the Attachments window will provide an indication of which type of attachment you are working with. For example, when you are working with project-level attachments, the caption bar will indicate "Project:" followed by the specific name of the project.

3.15.3.1 Project Attachments

All Synthesis applications support attachments at the project level. In all applications, project attachments can be displayed/managed from the **Attachments** icon in the Project Properties window.

For most applications, project attachments can also be displayed/managed from the Attachments folder in the current project explorer.



In Xfmea, RCM++, RBI and MPC (which do not have a current project explorer), you can also access the project attachments by selecting **Project > Synthesis > Attachments**.

3.15.3.2 Synthesis Resource Attachments

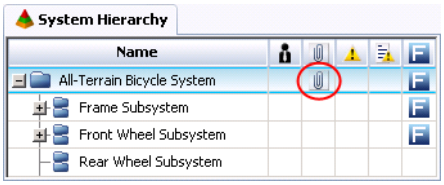
All of the applications that utilize Synthesis resources support attachments at the resource level. For example, you can attach a file to a URD, a model, etc.

To access attachments for an existing resource, you can click the **Attachments** icon in the resource's properties window.

3.15.3.3 Hierarchy Attachments

Some applications, including Xfmea, RCM++, RBI, Lambda Predict and MPC, support attachments for individual items or records in a system hierarchy or analysis hierarchy. For example, in Xfmea, you can create attachments for a particular item, or for a particular record in the FMEA for that item.

To access attachments for items or records in a hierarchy, you can double-click inside the Attachments column. (If this column is not displayed on your computer, right-click inside the column headers and choose **Customize Columns**.) You can also choose **Tools > Attachments** on the Ribbon tab for the hierarchy that you are currently working with.



For records in an analysis hierarchy (such as FMEA or functional failure analysis), the attachment icon can also be accessed from within the record properties window.

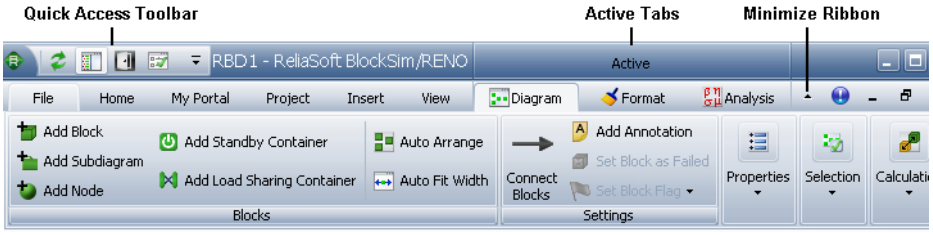
3.15.3.4 Block Attachments

BlockSim supports attachments for individual blocks in a diagram. To access attachments for a block, you can click the **Attachments** icon in the Block Properties window.

Ribbon 4

The application has a user-interface called the *Ribbon*, which replaces the toolbars and menus of previous versions of the software. This interface is designed to improve efficiency by displaying only the commands you need to complete a specific task. The Ribbon is divided into tabs that relate to a task or activity, such as managing the items in the system hierarchy or performing a specific analysis. Each tab contains the related commands, which are intuitively organized into groups. Depending on the activity you are in, some commands may appear dimmed or will not appear at all.

This section is intended to serve as a comprehensive reference to the commands found on the Ribbon. Note that many features in the application also have a shortcut menu (accessed by right-clicking) that gives access to commands that are commonly used for that feature. All commands available in the shortcut menus are also available on the Ribbon. In addition, the Ribbon includes a customizable Quick Access Toolbar that allows you to display frequently used commands. *See Section 4.1 on page 129*.



Tip: To maximize your workspace, you can choose to automatically minimize the Ribbon and display only the tab names. To do this, click the arrow on the right-side of the Ribbon, (labeled "Minimize Ribbon" in the picture below). The Ribbon will be expanded when you click a tab and then minimized after you click a command.

4.1 Quick Access Toolbar

The Quick Access Toolbar is a customizable feature of the Ribbon that allows you to access frequently used commands without changing the tabs.

- To add your favorite commands to the Quick Access Toolbar, right-click the command on the Ribbon and select **Add to Quick Access Toolbar.**
- To remove a command from the toolbar, right-click the command on the Ribbon or on the Quick Access Toolbar and select **Remove from Quick Access Toolbar**.
- Right-clicking anywhere within the Quick Access Toolbar also gives you the following commands:
  - **Show Quick Access Toolbar Below the Ribbon** or **Show Quick Access Toolbar Above the Ribbon**. The command that appears depends on where the toolbar is currently displayed on the Ribbon.

- **Minimize the Ribbon** maximizes your workspace by allowing you to display only the tab names on the Ribbon. The Ribbon will be expanded when you click a tab and then minimized after you click a command. To keep the Ribbon always expanded, clear the check box.

4.2 Backstage View (File Tab)

The Backstage view (File tab) is the first view you will see when you start the application. It allows you to work with databases and application-level settings. In other words, the Backstage view allows you to do things *to* a database, while the commands on the other tabs of the Ribbon allow you to do things *within* the database.

To access the Backstage view from within the application, click the **File** tab on the Ribbon. You can click any Ribbon tab to exit the Backstage view and return to the MDI workspace.

The area on the left side of the Backstage view is called the *Tab Pane*. It contains commands for working with the database and access to the tabs of the Backstage view.

The following commands are available in the Tab Pane:

Save As saves a copy of the standard database to another pathname/filename and/or file type. You can save it as a new standard database (*.rsr9) or a compressed standard database (*.rsgz9).

Tip: Note that Synthesis applications do not provide a **Save** command because changes are saved automatically as you work in all database-driven Synthesis applications. If you used to rely on the ability to close a standalone file without saving recent changes, the **Restore Points** feature provides similar functionality within Synthesis. *Refer to the "Restore Points" section in the "Working in Synthesis" chapter for more information*.

Pack and E-mail compresses a copy of the current standard database to a *.rsgz9 file and attaches it to a new e-mail message. If no e-mail program is installed or no default e-mail program is defined on the computer, a message notifying you of this will be shown. This command is not available if you are working with an enterprise database.

When you open the compressed file, a new database will be created from that file and saved in your local drive (C:\Documents and Settings\username\My Documents\ReliaSoft\Files). Any changes made to the new database will not affect the original copy.

Open Repository opens an existing standard database (*.rsr9) or enterprise database (*.rserp), or a compressed standard database (*.rsgz9). You also can select a database file in the Windows Explorer and double-click or drag it into the MDI to open it. If you want to convert a database that was created with BlockSim 7, BlockSim 6 or from an XML file, choose the appropriate option from the drop-down list in the Open Database window.

Close Repository closes the database. You also can close the database by opening another database or by exiting the application.

- **Recent** shows a list of recently opened database files (*.rsr9 and *.rserp) and a list of recently accessed locations. Clicking a database will open it, and clicking a location will allow you to browse for databases in that location. The number of items in these lists is set in the **Recent Repository List** area of the Synthesis Setup. You can pin items to a list by clicking the pushpin icon associated with the item; when the pin is vertical, the item will not roll off the list.
- **New** allows you to create a new standard database or a new connection file for access to an enterprise database.
- **Launch Application** provides quick launch icons for other Synthesis applications that are installed on your computer. If you are currently connected to a database when you click one of these icons, the same database will automatically be opened in the new application.

- **Manage Repository** contains tools that control various repository-level settings. These include:

Note: Most of these tools are available only when a repository is currently open. If you are working in a secure database, some tools may not appear at all. Your access to these tools is based on the permissions assigned to your user account.

- **Security**

Authorized Users controls the users who can access a secure database. *See page 50 in Chapter 2*.

Prior Logins displays a list of prior logins to the current database and allows you to export this information to Excel. *See page 61 in Chapter 2*.

Reset "In Use" Flags allows you to reset the flags that indicate the "in use" status of projects or items within a project. *See page 61 in Chapter 2*.

- **Notifications**

E-mail Settings controls whether e-mail notifications are enabled and when they are sent. *See page 62 in Chapter 2*.

Notification Groups allows you to define distribution groups to receive e-mail notifications. *See page 63 in Chapter 2*.

- **Repository Settings**

Manage Units allows you to define the time units available for use in any project within the database. *See page 64 in Chapter 2*.

Define Default Names allows you to specify how default names for resources and blocks are created. *See page 65 in Chapter 2*.

Global Identifiers allows you to define the names of the identifiers, which are fields that can be used to search for blocks or resources within the current project. *See page 66 in Chapter 2*.

Project Categories allows you to define the categories that can be used to filter the projects displayed in the project list. *See page 67 in Chapter 2*.

Restore Points allows you to manage restore points, which are exact replicas of the project at a particular point in time (i.e., backups) that can be restored when and if needed. *See page 71 in Chapter 2*.

Compact and Repair helps to reduce the size of the current database to protect against data loss or file corruption. *For details on protecting your data, page 69 in Chapter 2*.

XFRACAS Connection allows you to specify the location of and the connection settings for the XFRACAS tables associated with the current standard database. For enterprise databases, the XFRACAS data stored within the database is accessible automatically. *See page 68 in Chapter 2*.

Dashboard Manager is available only when working with simulation diagrams, and only if you have permission to manage dashboard templates. It opens the Dashboard Manager window, which allows you to see all of the templates that have been defined for use in the Dashboard Viewer in Weibull++, BlockSim and RGA, and to import, rename and/or delete templates. *See page 333 in Chapter 9*.

- **Enterprise**

Repository Creation opens a wizard that leads you through the steps to create a new enterprise repository (SQL Server or Oracle database). To use this option, you must have access to a server with a supported version of SQL Server or Oracle and you must have the appropriate permissions to create a new database on the server.

Note that if no database is currently open, the Repository Creation wizard and the Upgrade Repository wizard are the only options available.

Upgrade Repository allows you transfer all of the existing data from a Version 8 enterprise database to a Version 9 enterprise database. *See page 35 in Chapter 2*.

- The **Help Center** provides a variety of resources to help you use the application more effectively. In addition, the Help Center displays information about the application, including the Compile Release Version (CRV) and License information.

Application Setup opens the Application Setup, which allows you to set your preferences for working with analyses, such as the math precision, default plot settings and the like. *See page 535 in Chapter 22*. The settings apply only to the current computer/username, and they do not affect the results of the analyses.

Synthesis Setup opens the Synthesis Setup, which allows you to set your preferences for the entire Synthesis repository, such as when to backup the database, the color-scheme of the interface and the like. *See page 533 in Chapter 22*. The settings apply only to the current computer/username, and they do not affect the results of the analyses.

4.2.1 ReliaSoft Online Pane

The area on the right side of the Backstage view is called the *ReliaSoft Online Pane*. There are three pages available in this pane, accessed by the icons at the top of the pane:

The **ReliaSoft Online Home** page displays software tips, announcements and upcoming training seminars for this application.

The **Latest News** page displays recent announcements about ReliaSoft's reliability software and services.

The **Synthesis Portal** page displays tips and announcements related to the entire Synthesis Platform.

Note that an active Internet connection is required to see this information.

4.2.2 Help Center (File > Help)

The Help Center, which is part of the Backstage view, is intended to help you use the application more effectively by providing quick access to a variety of support tools. To access the Help Center, choose **File > Help**. For your convenience, the same commands are also available in the Ribbon's Help tab.

Using This Application

Online Help File opens the application's help file. If you have an active Internet connection, the help topics that you see will always be the most up-to-date versions available. If you do not have an active Internet connection, the help topics that you see will be the local copy that was installed on your machine.

User Guide opens the application's *User's Guide* (in PDF format) if you have an active Internet connection.

Online Resources

ReliaSoft.com opens the main page of the ReliaSoft.com website (http://www.ReliaSoft.com).

Weibull.com opens the Weibull.com website, which is devoted entirely to the topic of reliability engineering, reliability theory and reliability data analysis and modeling. (http://www.Weibull.com)

ReliaWiki.org opens the ReliaWiki website, which is both a resource portal and a wiki for professionals in reliability engineering and related fields. (http://www.ReliaWiki.org)

Reliability Discussion Forum opens the Reliability Discussion Forum website, which is designed to facilitate communication within the reliability engineering field. These forums are open to everyone and no password is required. ReliaSoft personnel monitor and participate in the discussions. (http://www.ReliaSoftForums.com)

Examples

Open Examples Folder provides access to a set of example projects that are designed to help you explore various software features.

Quick Start Guide opens the application's *Quick Start Guide* (in PDF format) if you have an active Internet connection. This guide contains step-by-step examples designed to assist you in learning to use the application software.

Technical Support

E-mail Support generates an e-mail to request technical support. The e-mail is pre-populated with information about your license and operating system, which the technical support representative will need for troubleshooting the issue.

Contact ReliaSoft USA provides the contact information you will need to obtain technical support.

Contact Global Support Center helps you find your localized ReliaSoft support center.

Check for Update allows you to download the latest free service release for the software.

- **Licensing** displays your license and registration details. To view or edit additional information, click **License Manger**. The License Manager displays additional information about which products you have currently registered and your contact information on file with ReliaSoft. *See Section 4.2.3 on page 134*.

- **About** displays the application's Compile Release Version (CRV) (sometimes called a "build number"), which allows you to determine whether you have the latest version of the software.

4.2.3 License Manager

The License Manager allows you to view and manage license-related details for the Synthesis applications that are currently activated on your computer. You can use this interface to:

- View and edit the contact information that is on file with ReliaSoft or change the password associated with your ReliaSoft ID.
- Deactivate the current application (if you don't plan to continue to use it on this computer or for this user).
- Check out or check in a CAL (which is applicable only for concurrent network licenses and serves to "reserve a seat" on the license that you share with other users).

To open the License Manager, click the **License Manager** link in the Help Center or choose **Help > Updates & Licensing > License Manager**.

You will be prompted to enter the password that was sent via e-mail when you first registered your ReliaSoft ID. This password is required if you want to modify your contact information. For everything else, you can click **Cancel** to proceed without it. (If you forgot your password, you can request to have the information sent to the e-mail address on file for your registration by clicking **E-mail my password to me**.)

View and Edit Contact Information and Password

The **Contact Information** page displays the name, phone number and other details that are on file with ReliaSoft.

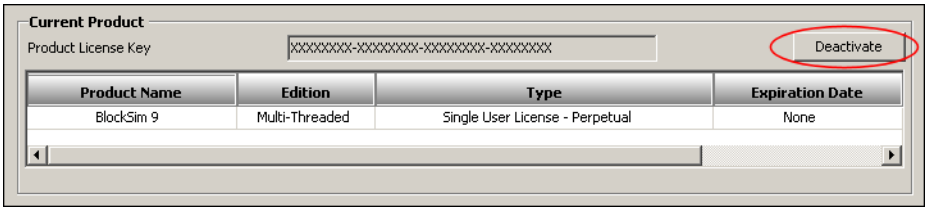
- If you need to download the latest information from the license server, click **Synchronize**.
- If you want to change your password or any of the current contact information, type the new information into the fields on this page and then click **Update Information on Server**.

Both actions require authentication. If you have not already entered the current password that's associated with your ReliaSoft ID, you will be prompted again to enter it.

Deactivate the Current Application

If you don't plan to continue to use the current Synthesis application on a particular computer (or for a particular user), you can use the License Manager to deactivate it. This does not require authentication; you can proceed with the deactivation even if you have not entered your password.

Click the **Deactivate** button in the current product area on the **Products** page.

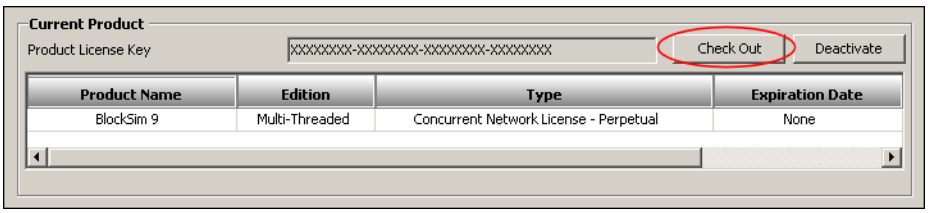


The application will shut down immediately after you deactivate, so you will be prompted to confirm that you're ready to continue. If you later try to access this application again for this computer/user, you will be prompted to repeat the activation process.

Check Out or Check In a CAL (for Concurrent Network Licenses)

If you have a concurrent network license[1], a specified number of *Concurrent Access Licenses* (CALs) are shared by the pool of *Named User Activations*. For example, there could be 30 named users but the application can be active on only 10 computers at the same time.

If you need to be sure that the application is available when you need it, you may have the option to check out (i.e., reserve) a CAL for a specified period of time. For example, you might wish to use this feature if you plan to be working without the Internet access required to obtain a CAL from the license server, or if you need to make sure that all of the CALs will not be taken by other users when you really need to access the software. However, it is important to remember that this reduces the number of CALs available for the rest of the pool of named users until the CAL is checked back in or otherwise released. If your license permits you to check out a CAL, the necessary buttons will be visible in the current product area on the **Products** page.



- If you don't currently have a CAL checked out, the **Check Out** button will be displayed. Click the button to reserve a CAL.
- If you currently have a CAL checked out, the **Check In** button will be displayed instead. Click the button when you're ready to make the CAL available again to the rest of the named user pool.

4.3 Home Tab

The Home tab contains commands related to using the Clipboard and printing diagrams.

Clipboard

Paste

- **Paste** pastes the contents of the Clipboard into the current diagram or sheet.
- **Paste All** pastes the formats, formulas and values of the contents of the Clipboard to the current diagram or sheet.
- **Paste Format** pastes only the formats from the Clipboard to the current diagram or sheet.
- **Paste Formulas** pastes only the formulas from the Clipboard to the current diagram or sheet.
- **Paste Values** pastes only the values from the Clipboard to the current diagram or sheet.
- **Paste With Duplicate Resources** allows you to create an exact duplicate of the pasted block's resources. The duplicate universal reliability definition (URD) of the pasted block will have the same name as the original URD with an increment number added to the end (e.g., URD1_1, URD 1_2, etc.). The duplicate resource will also appear in the Resource Manager window (**Project > Synthesis > Resource Manager**).

Cut cuts the selected text or graphic to the Clipboard. Data stored on the Clipboard can be pasted into this and other applications.

1. For more information on license types, please visit http://www.ReliaSoft.com/license.htm.

Copy copies the selected text or graphic to the Clipboard. Data stored on the Clipboard can be pasted into this and other applications. If you are copying a plot, the way the plot is copied will depend on your selection in the **Plot Copy Type** field in the Synthesis Setup window (**File > Synthesis Setup**).

- **Copy Diagram as Graphic** copies the current diagram to the Clipboard. The diagram can then be pasted into other documents as an enhanced metafile (*.emf) graphic.
- **Copy Block(s) as Graphic** copies the blocks and connectors that are currently selected in the diagram to the Clipboard. The selected material can then be pasted into other documents as an *.emf graphic.

Format Painter allows you to copy the block properties from one block and apply them to another block in the current diagram sheet, phase diagram sheet or maintenance template sheet or to copy the format properties of text in a spreadsheet and apply it to other text.

- To use the Format Painter in a diagram sheet phase diagram sheet or maintenance template sheet, select the block with the properties that are to be copied and then select **Format Painter**. The pointer will turn into a pointing hand. Next, click the block to which the properties are to be applied. The Format Painter Settings window will appear, which allows you to select the properties of the source block that you want to be applied to the destination block.
- To use the Format Painter in an spreadsheet, select the text with the format properties that are to be copied and then select Format Painter. Next, click the text to which the format properties are to be applied.

Edit

Redo reapplies the previously canceled action. You can redo multiple actions by choosing Redo again.

Undo cancels the last editing change you made.

Delete deletes the selected blocks.

Clear deletes content in analysis workbooks.

- **Clear All** deletes the contents and format of the selected cell(s), but does not delete the actual cell(s).
- **Clear Values** deletes only the contents of the selected cell(s). The format will be retained.
- **Clear Formats** deletes only the format of the selected cell(s). The contents will be retained.

Format Selection opens the Format submenu. The commands in the submenu are available only when you are viewing an analysis workbook, except for **Font Color**, **Fill Color** and **Select Font**, which are also available when working with diagrams.

- **Alignment** allows you to set the alignment properties of the selected text.
- **Borders** allows you to customize the borders of selected cells.
- **Font Color** opens the Color window, which allows you to set the color of the selected text.
- **Fill Color** opens the Color window, which allows you to set the color used to fill the selected cell(s) or block.
- **Pattern** opens the Cells Patterns window, which allows you to set the fill color, pattern color and fill pattern properties of the selected cell(s).
- **Select Font** opens the Font window, which allows you to set the font, style and size of the selected text.
- **Custom Number** opens the Format Cells-Number window, which allows you to set the format for how numbers in the selected cell(s) will displayed.

- **Column Width** opens the Column Width window, which allows you to set the column width of the selected cells. Enter the desired width in the input box. You can select the units of measurement (characters, inches or centimeters) from the Units drop-down list. To use the default column width, click **Use Default**. Click **OK** to accept the column width selections and apply them to the current Spreadsheet. Click **Cancel** to cancel any changes made in the Column Width window. You can change the default column width by clicking **Change Default**. This will open the Default Column Width window, which allows you to set a default column width. This window has the same options as the Column Width window.
- **Row Height** opens the Row Height window, which allows you to set the row height of the selected cells. The Row Height window works the same way the Column Width window works, with the additional option of automatically setting the row height based on contents.
- **Freeze Panes** allows you to lock a specific row on the spreadsheet so that the contents of the selected row will always be visible on screen no matter how far you scroll to the right or down. Changes cannot be made to a frozen row.
- **Merge Cells** combines selected cells to form one cell.
- **Unmerge Cells** separates merged cells into the original individual cells.

Select All selects all of the blocks in the diagram.

Find opens the Find window, which allows you to search through the diagram for text and/or numbers.

Spelling activates the Spell Check utility, which allows you to check the spelling within the current sheet.

Print

Print sends the current diagram to the printer.

Print Preview opens the Print Preview window, which allows you to view how the current diagram or sheet will appear on the printed page.

Page Setup opens the Page Setup window, which allows you to specify printing options.

Print Layout displays the page boundary lines on the diagram. Everything within the boundary lines will appear on the printed page and this view allows you to make any necessary adjustments to the final look of the diagram before sending it to the printer.

Launch

Another Synthesis Application provides a drop-down list of the other Synthesis applications that are installed on your computer. When you click an icon, the same database will automatically be opened in the new application.

4.4 My Portal Tab

The My Portal tab contains commands related to using the My Portal feature. *See page 101 in Chapter 3*.

Messages

Create Message opens the Message window, which allows you to compose and send messages within the database.

Edit Message opens the Message window, which allows you to view and edit all of the properties of the message.

Delete Message deletes the current message.

Actions

Create Action opens the Action window and adds an action to the project.

Edit Action opens the Action window, which allows you to view and edit all of the properties of the action record.

Delete Action deletes the current action record.

Add Action to Outlook adds an event, on the Action's due date, to your Outlook calendar.

Users

My Profile opens the User Login and Contact Information window where you change your contact details for your user account in the current database.

Reliability Web Notes

Get Notes for Active Item allows you update the Reliability Web Notes page of the My Portal window with information related to the item that you currently have open. The information includes a brief description of the item, as well as links to examples and theory articles.

4.5 Project Tab

The Project tab contains commands related to creating projects and working with the project properties. Unless otherwise indicated, these commands apply to the project that is currently selected in the project list (*for more information on the project list, see page 97 in Chapter 3*). There are three ways to select a project and apply a command:

- By right-clicking the project in the project list and choosing the command from the shortcut menu.
- By selecting the project in the project list and then clicking the command on the Ribbon.
- By clicking anywhere inside the project that is currently open/visible and then clicking the command on the Ribbon.

Management

Create Project adds a new project to the current database. At a minimum, you must specify the project name.

Open Project opens the selected project.

Close Project closes the project that is currently open/visible and all of the analyses it contains.

Edit Project Properties allows you to view and edit the properties of the selected project. *See page 69 in Chapter 2*.

Duplicate Project creates a copy of the selected project. The new project will have the same name as the original with an increment number added to the end (e.g., Project_1, Project_2, etc.).

Delete Project moves the selected project to the recycle bin. *For more information about using the recycle bin, see page 45 in Chapter 2*.

Manage Projects opens the Manage Projects window, which allows you to edit the security settings and properties of all projects in the database in one location. *See page 69 in Chapter 2*.

Import/Export gives you the option to open either the Import or Export wizard. The Import wizard allows you to import projects, resources or items from an existing database into the current database, while the Export wizard allows you to export to a new or existing database. *See page 72 in Chapter 2*.

Restore Points gives you the option to utilize restore points, which are exact replicas of the project at a particular point in time (i.e., backups). To create a restore point for the selected project, use the **Create Restore Point** command. To restore all data from an existing restore point, use the **Restore Project** command. *See page 71 in Chapter 2*.

Check In/Out allows you to check out the selected project and make it available for editing only to you, while all other users in the database have read-only access to the project. No other user can edit the project unless you check in the project or undo the check out. *See page 42 in Chapter 2*.

Recycle Bin

- **Empty Recycle Bin** permanently deletes all projects under the Recycle Bin heading in the project list. ***There is no undo for emptying the recycle bin***.
- **Restore Project** is available only when you have selected a project under the Recycle Bin heading. The selected project will be recovered from the recycle bin and restored to its original location in the project list.
- **Delete Project** permanently deletes the selected project. ***There is no undo for delete*** unless you have a stored backup (*Section 2.5 on page 69*) or restore point (*Section 2.5.1 on page 71*).

History Log is available only if you are the project owner or have the "Manage all projects" permission. It opens the Project History Log window, which allows you to activate and deactivate the history log for the project and, if the history log is currently activated, to view changes that have been performed throughout the entire project, with any Synthesis application. *See page 124 in Chapter 3*.

E-mail Project

Pack and E-mail compresses a copy of the selected project to a *.rsgz9 file and attaches it to a new e-mail message. If no e-mail program is installed or no default e-mail program is defined on the computer, a message notifying you of this will be shown.

Security

Lock Project moves the project into the Locked heading of the project list. When a project is locked, all database users (including the user who locked the project) will have read-only access to the project. In addition, a locked project cannot be deleted or have its properties and public/private status edited. To unlock a project, choose **Unlock Project**. *See page 42 in Chapter 2*.

Make Private moves the selected project into the Private heading of the project list. To move a private project to the Public heading, choose **Make Public**. *See page 41 in Chapter 2*.

Change Owner allows you to assign a different database user to be the owner for the selected project. *See page 40 in Chapter 2*.

Project Security is available only for public projects in secure databases. It opens the Project Properties window with the Security tab active, where you can specify the user accounts that can view/modify the selected project. *See page 69 in Chapter 2*.

Current Item

The following commands are available when you select a specific item (a block diagram, fault tree diagram, phase diagram sheet, analysis workbook, Word report template or overlay plot sheet) in the current project explorer.

Edit brings the selected item to the front of the windows and activates it as the current control. If an attachment is selected, the command opens the attachment in the appropriate application, if that application is installed on your computer.

Rename allows you to rename the selected item.

Duplicate creates a copy of the selected item within the project. The duplicate will have the same name as the original with an increment number added to the end (e.g., RBD1_1, RBD1_2, etc.).

- **Duplicate With Resources** also creates duplicate resources in the Resource Manager window.

Delete deletes the selected item. ***There is no undo for delete***.

View Item History displays the date and time the item was created and the name of the creator. It also displays the date time the item was last modified and the name of the user who made the modifications.

Item Permissions is available only for secure databases. It opens the Item Permissions window, which allows you to specify the user accounts that can view or modify the item. *See page 69 in Chapter 2*.

Synthesis

Resource Manager opens the Resource Manager window, which allows you to view and edit all of the resources (URDs, models, tasks, etc.) available to the selected project. *See page 175 in Chapter 5*.

Data Management

Item Properties Table opens the Item Properties Table, which allows you to view and/or edit the properties of the blocks in the RBDs and fault trees within the project that is currently open/visible. *See page 246 in Chapter 7*.

Mirror Group Manager opens the Mirror Group Manager window which allows you to add, view and edit groups of mirrored blocks within the project that is currently open/visible. *See page 243 in Chapter 7*.

Maintenance Group Manager opens the Maintenance Group Manager window, which allows you to add, view and edit the maintenance task groups available to the project that is currently open/visible. *See page 279 in Chapter 9*.

4.6 Insert Tab

The Insert tab contains commands related to adding items to a project.

Wizard

Project Item Wizard opens the Project Item Wizard, which provides information about all the analyses that you could add to the project and guides you through any steps required to configure the analysis that you have selected.

Diagrams

Analytical RBD adds an analytical reliability block diagram to the project. *See Chapter 8*.

Analytical Fault Tree adds an analytical fault tree to the project. *See page 228 in Chapter 7*.

Simulation RBD add a simulation reliability block diagram to the project. *See Chapter 9*.

Simulation Fault Tree adds a simulation fault tree to the project. *See page 228 in Chapter 7*.

Phase Diagram adds a phase diagram to the project. *See Chapter 10*.

RENO Flowchart adds a RENO flowchart to the project. *See Chapter 11*.

Tools

Allocation Analysis adds an allocation analysis tool to the project. *See page 435 in Chapter 15*.

Build from Synthesis

Build RBDs and FTs from Synthesis opens a setup window that allows you to create diagrams based on items imported from Xfmea, RCM++ or RBI or on items imported from Lambda Predict. *See page 252 in Chapter 7*.

Build Effect FTs from Synthesis opens a setup window that allows you to create fault trees based on effects. *See page 261 in Chapter 7*.

Reports and Plots

FRED Report adds a FRED report to the project. *See Chapter 20*.

Analysis Workbook opens a wizard to add an analysis workbook to the project. This spreadsheet-type tool allows you to create customized reports that integrate data and/or results from up to four analyses at the same time. Custom plots and charts can be inserted using the Chart Wizard. *See Chapter 18*.

Word Report Template opens wizard to add a Word report template to the project. This tool offers custom reporting functionality that is similar to a Microsoft Word document. You can add formatted text, and insert functions, plots and other graphics, and then generate the report directly in Word. Each report can utilize data and/or results from up to four analyses at the same time. *See Chapter 19*.

Overlay Plot adds an overlay plot to the project that allows you to display in a single plot results from multiple diagrams. This provides an easy visual method to compare analyses. For example, you may wish to show the reliability plots of two product designs in the same plot. *See page 407 in Chapter 12*.

Attachments

Attachment adds linked or attached files to the project. *See page 126 in Chapter 3*.

Simulation Worksheets

Add Simulation Worksheet adds a simulation worksheet to the project that allows you to vary values that are used in simulating a simulation RBD or a RENO flowchart. This allows you to investigate the effect of one or more settings on the simulation results.

4.7 View Tab

The View tab contains commands related to configuring the layout of the application's interface.

Refresh

Refresh refreshes the display in the Project window. If multiple users are accessing the same project simultaneously, this command will refresh your screen with any changes made by other users. When you make a change to the project, your window will be refreshed automatically.

Project Manager

Show Project Manager brings the Project Manager into focus. If the Project Manager is unpinned and hidden, choosing this command will display it. *See page 97 in Chapter 3*.

Tile Project Manager tiles the project list and current project explorer so they are both displayed simultaneously in the Project Manager panel.

Dock Project Manager opens a submenu that allows you to choose the desired position for the Project Manager: Dock Left, Dock Right, Dock Top, Dock Bottom or Floating.

My Portal

Show My Portal opens the My Portal window, which provides information relevant to your work within the database, such as messages from other users, recommended actions for a particular project item, status of other users logged in to the database and other information. *See page 101 in Chapter 3*.

Tile My Portal tiles the Messages, Actions, Users, and Reliability Web Notes pages so they are all displayed simultaneously within the My Portal panel.

Dock My Portal opens a submenu that allows you to choose the desired position for the Portal: Dock Left, Dock Right, Dock Top, Dock Bottom or Floating.

Zoom

Normal Zoom sets the degree of magnification to 100%.

Zoom In increases the degree of magnification.

Zoom Out decreases the degree of magnification.

Custom Zoom allows you to specify the degree of magnification.

Window

Use Tabbed MDI maximizes the windows to fill the full space available in the MDI and shows a tab for each open window. If you clear this command, the open windows will be displayed as separate windows that can be resized and moved around within the MDI.

Window

- **Cascade** cascades all open project windows inside the MDI. This command is available only when you are not using a tabbed MDI view.
- **Tile Horizontally** horizontally tiles all open project windows inside the MDI. This command is available only when you are not using a tabbed MDI view.
- **Tile Vertically** vertically tiles all open project windows inside the MDI. This command is available only when you are not using a tabbed MDI view.
- **Arrange Icons** bottom aligns all icons of the minimized windows inside the MDI. This command is available only when you are not using a tabbed MDI view.
- The **Windows** list displays a list of all project windows currently open inside the MDI. You can make any of the open project windows active by clicking its name in this list.
- **Close All Windows** closes all open project windows inside the MDI, leaving only the Project Manager and My Portal panels open (if they were selected to be shown).

FMRA

Show FMRA opens the FMRA view, which displays the FMRA (failures modes and reliability analysis) hierarchy that is associated with the project. *See page 443 in Chapter 16*.

Disassociate FMRA deletes all of the diagrams and blocks associated with the current FMRA hierarchy and closes the FMRA view.

4.8 Help Tab

The Help tab provides easy access to the same help resources that are available from the Help Center in the Backstage View.

Using This Application

Online Help File opens the application's help file. If you have an active Internet connection, the help topics that you see will always be the most up-to-date versions available. If you do not have an active Internet connection, the help topics that you see will be the local copy that was installed on your machine.

User Guide opens the application's *User's Guide* (in PDF format) if you have an active Internet connection.

Quick Start Guide opens the application's *Quick Start Guide* (in PDF format) if you have an active Internet connection. This guide contains step-by-step examples designed to assist you in learning to use the application software.

Open Examples Folder provides access to a set of example projects that are designed to help you explore various software features.

ReliaSoft.tv opens the ReliaSoft TV website (http://www.ReliaSoft.tv).

Online Resources

ReliaSoft.com opens the main page of the ReliaSoft.com website (http://www.ReliaSoft.com).

Weibull.com opens the Weibull.com website (http://www.Weibull.com).

ReliaWiki.org opens the ReliaWiki website (http://www.ReliaWiki.org).

Reliability Discussion Forum opens the Reliability Discussion Forum website (http://www.ReliaSoftForums.com).

Technical Support

E-mail Support generates an e-mail to request technical support.

Contact ReliaSoft USA provides the contact information you will need to obtain technical support.

Contact Global Support Center helps you find your localized ReliaSoft support center.

Updates & Licensing

Check for Update allows you to download the latest free service release for the software.

License Manager displays your license and registration details. *See Section 4.2.3 on page 134*.

4.9 Active Tabs (Alphabetical)

Active (contextual) tabs contain commands related to a type of analysis. These tabs are displayed only when the related analysis is active. For example, the contextual tabs for an analytical diagram will contain commands related to performing an analytical analysis, while the contextual tabs for a simulation diagram will contain commands related to simulation.

4.9.1 Allocation Tab

The Allocation tab is visible when you are working with the allocation analysis tool. *See page 435 in Chapter 15*.

Analysis

Optimize calculates the reliability allocation and displays the results.

Transfer

Transfer to Spreadsheet save the results in an analysis workbook.

Send to Excel saves the results an Excel file.

4.9.2 Analysis Tab

The Analysis tab is visible when you are working with an analytical diagram. *See Chapter 8*.

Analysis

Analyze analyzes the relationships of the components in the diagram and puts the reliability-wise configuration into a mathematical formula.

Analytical Plot creates a plot based on the analysis results. Clicking **Analytical Plot** before the diagram has been analyzed will automatically perform the analysis and then plot the data.

Analytical QCP opens the analytical Quick Calculation Pad (QCP). The QCP allows you to obtain reliability results based on the algebraic solution for the currently active diagram. Clicking the **Analytical QCP** icon before the diagram has been analyzed will automatically perform the analysis and then open the QCP.

Tools

Show Algebraic Solution generates the algebraic solution for the diagram (i.e., a mathematical representation of the reliability-wise configuration of the diagram) and opens the Equation Viewer. *See page 267 in Chapter 8*.

Show Cut Sets opens the Cut Sets window, which allows you to determine what are all the unique combinations of component failures that can cause system failure. *See page 268 in Chapter 8*.

Share

The **Publish** commands allow you to publish the diagram results as a model for use throughout the project, in any Synthesis application where models are used. **Publish Analytical Model** publishes the analytical solution of the diagram, which is essentially a direct reference to the diagram itself. **Publish Fitted Model** creates a model based on the distribution that has been fitted to the analyzed diagram, which is an approximation of the entire diagram and its subdiagrams. *For more on publishing models, see page 174 in Chapter 5*.

4.9.3 Diagram Tab

The Diagram tab is visible when you are working with an analytical RBD or a simulation RBD. *See page 208 in Chapter 7*.

Blocks

Add Block adds a new standard block to the current diagram. You can define the properties of the new block using the Block Properties window.

Add Subdiagram adds a new subdiagram block to the current diagram. When this menu item is selected, the Select Diagram for the Block window will appear, which allows you to select a diagram or fault tree that exists in the current project to be represented by the subdiagram block.

Add Node adds a new node to the current diagram sheet. You can define the properties of the new node using the Block Properties window.

Add Standby Container adds a new standby container block to the current diagram. You can define the properties of the new standby container block using the Block Properties window.

Add Load Sharing Container adds a new load sharing container block to the current diagram. You can define the properties of the new load sharing container block using the Block Properties window.

Auto Arrange automatically arranges the blocks in the diagram so that all blocks are evenly spaced and centered.

Auto Fit Width automatically arranges the blocks in the diagram to fit the width of the diagram window.

Settings

Connect Blocks allows you to create connectors between blocks in the current diagram. When this command is selected, click the source block, hold down the left mouse button and drag a line from the source block to the destination block. When the crosshairs are located above the destination block, release the mouse button to create a connector. To stop adding connectors and return to the normal mode, right-click the diagram or clear the **Connect Blocks** option (i.e., by choosing the command again).

Add Annotation adds a new annotation to the current diagram. You can edit the text that appears in the annotation by double-clicking it to open the Edit Annotation window.

Set Block as Failed indicates that the block is "off" and the block will be considered to be failed, inactive or absent from the system for reliability calculations. An X will appear on the block to indicate that the block is failed. If de-selected, the block will be considered active for reliability calculations.

Set Block Flag allows you to display or remove a red, yellow or green flag above the selected block. These flags have no effect on the analysis or simulation so you can use them for any purpose of interest to you. For example, each flag color could represent a level of analysis that has been completed.

- **Remove Flag** removes the flag from the selected block.
- **Red Flag** displays a red flag for the selected block.
- **Yellow Flag** displays a yellow flag for the selected block.
- **Green Flag** displays a green flag for the selected block.

Properties

Block Properties opens the Block Properties window, which allows you to define the failure, maintenance, optimization, throughput and other characteristics of the selected block (*see page 194 in Chapter 6*). If an annotation is selected, this command opens the Edit Annotation window, which allows you to edit the text for the annotation.

Line Bend alters the bend style of the selected connectors. The options are: Default, Straight Line, Angle, Right Angle and Custom. The following options are available only if the style is **Custom**: Remove Bend Point and Remove All Bend Points.

Standby Settings apply only to blocks in Standby containers (*see page 220 in Chapter 7*).

- **Set Block to Active (Not Standby)** changes the status of the selected block in a standby container in a diagram from standby to active. Clear this option to change the status of the selected block to standby. This option is only available when a block in a standby container is selected or when a dependent event block (i.e., an event block that is connected to a standby container gate block) is selected.
- **Set Standby Order** opens the Standby Blocks Order window where you specify the order in which the standby blocks are used.

Selection

Diagram Actions

- **Refresh Diagram** updates the diagram to show any changes.
- **Change Diagram Type** opens the Change Diagram Type window, which allows you to change the current diagram to the opposite diagram type (i.e., you can change the current analytical diagram to a simulation

diagram, or the current simulation diagram to an analytical diagram). In this window, you have the following options:

- Select **Duplicate using the new type** option to create a new diagram using the new type. The new diagram will be named "Copy of [original diagram name]." If you do not select this option, the original diagram itself will be changed to the new type.
- Select **Perform the same change to any subdiagrams** to change any subdiagrams to the new diagram type. Note that if you select both this and the **Duplicate using the new type** check box, new subdiagrams will be created along with the new main diagram.

Note: If you create an analytical diagram from a simulation diagram, by either duplication or conversion, the maintenance and throughput characteristics defined in the simulation diagram will not be displayed in the analytical diagram, as they are not relevant to the analysis. However, the properties are not lost, and if you then create a simulation diagram from that analytical diagram, all original properties will be retained.

- **Show Block Details** opens the Block Details window, which displays the details of the blocks and resources in the current diagram.
- **Diagram Style** opens the Diagram Style window, which allows you to edit the appearance of the diagram.
- **Open Subdiagram** opens the diagram that the selected subdiagram block represents. This option is only available when a subdiagram block is selected. *For more on subdiagrams, see page 218 in Chapter 7*.
- **Open Parent Diagram** opens a sub-menu that allows you to select a parent diagram (i.e. a diagram that contains a subdiagram block representing the current diagram) to open.
- **Export Graphic** saves the diagram as a graphic in one of the following formats: *.wmf, *.png, *.gif or *.jpg. You will be able to use the exported graphic in any application, provided that the application supports the file format.

Cascade Select selects all the blocks that are connected from the selected block, including the selected block itself. This allows you to easily select all blocks that are connected to a block.

Trace Path traces the path of the selected block. Click another block or the diagram to remove the traced path. The color of the traced path is the same as the color for a selected connector and can be set in the Diagram Style window.

Calculation

Optimum Replacement opens the Optimum Replacement window, which allows you to define the cost of planned and unplanned replacements for selected block(s) and calculate the optimum replacement time, as well as the cost per unit time at that replacement time. *See page 440 in Chapter 15*.

4.9.4 Fault Tree Tab

The Diagram tab is visible when you are working with an analytical fault tree or a simulation fault tree. *See page 228 in Chapter 7*.

Gates and Events

Add Gate allows you to select a new gate block to add to the current fault tree. *See page 230 in Chapter 7*. When adding blocks to a fault tree, they will automatically be connected to the currently selected block, if any. For all gates, you set the gate properties in the Block Properties window (*see page 194 in Chapter 6*).

- **Add AND Gate** adds a new AND gate block to the current fault tree.
- **Add OR Gate** adds a new OR gate block to the current fault tree.
- **Add NOT Gate** add a new NOT gate to the current fault tree.
- **Add NAND Gate** add a new NAND gate to the current fault tree.

- **Add NOR Gate** add a new NOR gate to the current fault tree.
- **Add Voting Gate** adds a new voting gate block to the current fault tree.
- **Add Inhibit Gate** adds a new inhibit gate block to the current fault tree.
- **Add Standby Gate** adds a new standby container gate block to the current fault tree.
- **Add Load Sharing Gate** adds a new load sharing container gate block to the current fault tree.
- **Add PAND Gate** adds a new PAND (priority AND) gate to the current fault tree.
- **Add SEQ Gate** adds a new SEQ gate to the current fault tree.

Add Event allows you to select a new event block to add to the current fault tree. When adding blocks to a fault tree, they will automatically be connected to the currently selected block, if any. For all events, you set the event properties in the Block Properties window.

- **Basic Event** adds a basic event block to the current fault tree.
- **Undeveloped Event** adds an undeveloped event block to the current fault tree
- **Trigger Event** adds a trigger event block to the current fault tree
- **Resultant Event** adds a resultant event block to the current fault tree
- **Conditional Event** adds a conditional event block to the current fault tree

Add Subdiagram adds a new subdiagram block to the current fault tree. When this menu item is selected, the Select Diagram for the Block window will appear, which allows you to select a diagram sheet or fault tree sheet that exists in the current project to be represented by the subdiagram block.

Auto Arrange automatically arranges the events and gates in the fault tree so that all blocks are evenly spaced and centered.

Settings

Connect Blocks allows you to create connectors between blocks in the current diagram. When this command is selected, click the source block, hold down the left mouse button and drag a line from the source block to the destination block. When the crosshairs are located above the destination block, release the mouse button to create a connector. To stop adding connectors and return to the normal mode, right-click the diagram or clear the **Connect Blocks** option (i.e., by choosing the command again).

Add Annotation adds a new annotation to the current diagram. You can edit the text that appears in the annotation by double-clicking it to open the Edit Annotation window.

Set Block as Failed indicates that the block is “off” and the block will be considered to be failed, inactive or absent from the system for reliability calculations. An X will appear on the block to indicate that the block is failed. If de-selected, the block will be considered active for reliability calculations.

Set Block Flag allows you to display or remove a red, yellow or green flag above the selected block. These flags have no effect on the analysis or simulation so you can use them for any purpose of interest to you. For example, each flag color could represent a level of analysis that has been completed.

- **Remove Flag** removes the flag from the selected block.
- **Red Flag** displays a red flag for the selected block.
- **Yellow Flag** displays a yellow flag for the selected block.
- **Green Flag** displays a green flag for the selected block.

Conversion

Convert Block To allows you to convert the selected block in a fault tree to another event or gate.

Create RBD creates a reliability block diagram that is equivalent to the current fault tree.

Create Transfer Gate creates a subdiagram that contains the selected gate and all of the gates/event under it, and then replaces the selected gate with a subdiagram block.

Properties

Block Properties opens the Block Properties window, which allows you to define the failure, maintenance, optimization, throughput and other characteristics of the selected block (*see page 194 in Chapter 6*). If an annotation is selected, this command opens the Edit Annotation window, which allows you to edit the text for the annotation.

Line Bend alters the bend style of the selected connectors. The options are: Default, Straight Line, Angle, Right Angle and Custom. The following options are available only if the style is Custom: Remove Bend Point and Remove All Bend Points.

Standby Settings apply only to blocks in Standby containers (*see page 220 in Chapter 7*).

- **Set Block to Active (Not Standby)** changes the status of the selected dependent event block from standby to active. De-select this option to change the status of the selected block to standby. This option is only available when a block in a standby container is selected or when a dependent event block (i.e., an event block that is connected to a standby container gate block) is selected.
- **Set Standby Order** opens the Standby Blocks Order window where you specify the order in which the standby events are used.

Selection

Diagram Actions

- **Refresh Diagram** updates the diagram to show any changes.
- **Change Diagram Type** opens the Change Diagram Type window, which allows you to change the current diagram to the opposite diagram type (i.e., you can change the current analytical diagram to a simulation diagram, or the current simulation diagram to an analytical diagram). In this window, you have the following options:
  - Select **Duplicate using the new type** option to create a new diagram using the new type. The new diagram will be named "Copy of [original diagram name]." If you do not select this option, the original diagram itself will be changed to the new type.
  - Select **Perform the same change to any subdiagrams** to change any subdiagrams to the new diagram type. Note that if you select both this and the **Duplicate using the new type** check box, new subdiagrams will be created along with the new main diagram.

Note: If you create an analytical diagram from a simulation diagram, by either duplication or conversion, the maintenance and throughput characteristics defined in the simulation diagram will not be displayed in the analytical diagram, as they are not relevant to the analysis. However, the properties are not lost, and if you then create a simulation diagram from that analytical diagram, all original properties will be retained.

- **Show Block Details** opens the Block Details window, which displays the details of the blocks and resources in the current diagram.
- **Diagram Style** opens the Diagram Style window, which allows you to edit the appearance of the diagram.
- **Open Subdiagram** opens the diagram that the selected subdiagram block represents. This option is only available when a subdiagram block is selected. *For more on subdiagrams, see page 218 in Chapter 7*.
- **Open Parent Diagram** opens a sub-menu that allows you to select a parent diagram (i.e. a diagram that contains a subdiagram block representing the current diagram) to open.
- **Export Graphic** saves the diagram as a graphic in one of the following formats: *.wmf, *.png, *.gif or *.jpg. You will be able to use the exported graphic in any application, provided that the application supports the file format.

Cascade Select selects all the blocks that are connected from the selected block, including the selected block itself. This allows you to easily select all blocks that are connected to a block.

Trace Path traces the path of the selected block. Click another block or the diagram to remove the traced path. The color of the traced path is the same as the color for a selected connector and can be set in the Diagram Style window.

Calculation

Optimum Replacement opens the Optimum Replacement window, which allows you to define the cost of planned and unplanned replacements for selected block(s) and calculate the optimum replacement time, as well as the cost per unit time at that replacement time. *See page 440 in Chapter 15*.

4.9.5 Flowchart Tab

The Flowchart tab is visible when you are working with a RENO flowchart. *See Chapter 11*.

Blocks

For all blocks, you set the block properties in the Block Properties window (*see page 194 in Chapter 6*).

Add Standard Block adds a new standard block to the current flowchart.

Add Result Storage adds a new result storage block to the current flowchart.

Add Conditional Block adds a new conditional block to the current flowchart.

Add Binary Node adds a new binary node to the current flowchart.

Add Logic Gate adds a new logic gate to the current flowchart.

Add Branch Gate adds a new branch gate to the current flowchart.

Add Summing Gate adds a new summing gate to the current flowchart.

Add Flag Marker adds a new flag marker to the current flowchart.

Add GoTo Flag adds a new go-to-flag block to the current flowchart.

Add Counter Block adds a new counter block to the current flowchart.

Add Reset Block adds a new reset block to the current flowchart.

Add Stop Flag adds a new stop flag to the current flowchart.

Add UI Block adds a new UI block to the current flowchart.

Add Subchart adds a new subchart block to the current flowchart. When this menu item is selected, the Select Diagram for the Block window will appear, which allows you to select a flowchart that exists in the current project to be represented by the subchart block.

Auto Arrange automatically arranges the blocks in the diagram so that all blocks are evenly spaced and centered.

Auto Fit Width automatically arranges the blocks in the diagram to fit the width of the diagram window.

Settings

Connect Blocks allows you to create connectors between blocks in the current diagram. When this command is selected, click the source block, hold down the left mouse button and drag a line from the source block to the destination block. When the crosshairs are located above the destination block, release the mouse button to create a connector. To stop adding connectors and return to the normal mode, right-click the diagram or clear the **Connect Blocks** option (i.e., by choosing the command again).

Add Annotation adds a new annotation to the current diagram. You can edit the text that appears in the annotation by double-clicking it to open the Edit Annotation window.

Block Properties opens the Block Properties window, which allows you to define the failure, maintenance, optimization, throughput and other characteristics of the selected block (*see page 194 in Chapter 6*). If an annotation is selected, this command opens the Edit Annotation window, which allows you to edit the text for the annotation.

Set Block Flag allows you to display or remove a red, yellow or green flag above the selected block. These flags have no effect on the analysis or simulation so you can use them for any purpose of interest to you. For example, each flag color could represent a level of analysis that has been completed.

- **Remove Flag** removes the flag from the selected block.
- R**ed Flag** displays a red flag for the selected block.
- **Yellow Flag** displays a yellow flag for the selected block.
- **Green Flag** displays a green flag for the selected block.

Selection

Diagram Actions

- **Refresh Diagram** updates the diagram to show any changes.

- **Diagram Style** opens the Diagram Style window, which allows you to edit the appearance of the diagram.
- **Open Flowchart** opens the flowchart that the selected subchart block represents. This option is only available when a subchart block is selected.
- **Open Parent Diagram** opens a sub-menu that allows you to select a parent flowchart (i.e., a flowchart that contains a subchart block representing the current flowchart) to open.
- **Export Graphic** saves the diagram as a graphic in one of the following formats: *.wmf, *.png, *.gif or *.jpg. You will be able to use the exported graphic in any application, provided that the application supports the file format.

Cascade Select selects all the blocks that are connected from the selected block, including the selected block itself. This allows you to easily select all blocks that are connected to a block.

Trace Path traces the path of the selected block. Click another block or the diagram to remove the traced path. The color of the traced path is the same as the color for a selected connector and can be set in the Diagram Style window (**Format > Styles > Diagram Style**).

Connector Priority allows you to change the order of execution of outgoing paths from a block. Note that you can display connector priorities by selecting the **Show Connection Priorities** check box in the diagram's control panel.

4.9.6 FMRA Tab

The FMRA tab is available when working with a Failure Modes and Reliability analysis (FMRA) diagram. *See Chapter 16*.

Refresh

Synchronize FMRA allows you to copy any changes you make in the BlockSim FMRA records to the Xfmea/RCM++/RBI analysis.

Add Record

Add System adds a new top level (system) item to the FMRA hierarchy. The new item will be added to the bottom of the hierarchy list.

Add Next Level Item adds a new item to the next level below the item that is currently selected in the FMRA hierarchy, and adds a block to the corresponding diagram. The new item will be added to the bottom of the hierarchy list for that level.

Add Same Level Item adds a new item to the same level as the item that is currently selected in the FMRA hierarchy, and adds a block to the corresponding diagram. The new item will be added to the bottom of the hierarchy list for that level.

Add Function adds a function to a selected record and a block to the corresponding diagram. The new function will be added to the bottom of the hierarchy list.

Add Failure adds a failure mode to a selected function and a block to the corresponding diagram. The new failure will be added to the bottom of the hierarchy list for that level.

Add Cause adds a cause to a selected failure and a block to the corresponding diagram. The new cause will be added to the bottom of the hierarchy list for that level.

Diagram

Connect Blocks allows you to create connectors between blocks in the current diagram. When this command is selected, click the source block, hold down the left mouse button and drag a line from the source block to the destination block. When the crosshairs are located above the destination block, release the mouse button to create a connector. To stop adding connectors and return to the normal mode, right-click the diagram or clear the **Connect Blocks** option (i.e., by choosing the command again).

Add Node adds a new node to the current diagram sheet.

Auto Arrange automatically arranges the blocks in the diagram so that all blocks are evenly spaced and centered.

Block Properties opens the Block Properties window, which allows you to define the failure, maintenance, optimization, throughput and other characteristics of the selected block (*see page 194 in Chapter 6*). If an annotation is selected, this command opens the Edit Annotation window, which allows you to edit the text for the annotation.

Line Bend alters the bend style of the selected connectors. The options are: Default, Straight Line, Angle, Right Angle and Custom. The following options are available only if the style is Custom: Remove Bend Point and Remove All Bend Points.

Diagram Actions

- **Refresh Diagram** updates the diagram to show any changes.
- **Show Block Details** opens the Block Details window, which displays the details of the blocks and resources in the current diagram.
- **Diagram Style** opens the Diagram Style window, which allows you to edit the appearance of the diagram.
- **Open Subdiagram** opens the diagram that the selected subdiagram block represents. This option is only available when a subdiagram block is selected.
- **Open Parent Diagram** opens a sub-menu that allows you to select a parent diagram (i.e. a diagram that contains a subdiagram block representing the current diagram) to open.
- **Export Graphic** saves the diagram as a graphic in one of the following formats: *.wmf, *.png, *.gif or *.jpg. You will be able to use the exported graphic in any application, provided that the application supports the file format.

Inheritance

If the **Inherit** command is unavailable, then this indicates that the reliability definition of the selected block is based on the reliability of its lower-level records.

If the **Define at this Level** command is unavailable, then this indicates that the reliability definition of the selected block is based on that block's universal reliability definition (URD).

System Settings opens the System Settings window, which allows you to edit the item name and operation time of the system.

Toggle Diagram Type allows you to select the opposite diagram type (i.e., you can select to change the current FMRA analytical diagrams to FMRA simulation diagrams, or the current FMRA simulation diagrams to FMRA analytical diagrams). Note that the change affects all of the diagrams in the FMRA.

Calculations

Calculate Tree calculates the reliability of each record in the FMRA hierarchy. This command is available only when working with FMRA analytical diagrams. The reliability for each record will appear in the Reliability column of the FMRA (if it is shown). If the Reliability column is not visible in the FMRA, right-click the column headers, then click **Customize Columns** to open a window and select the **Reliability** check box.

Calculate Branch calculates the reliability of each record for items within the selected system only. This command is available only when working with FMRA analytical diagrams. The reliability for each record will appear in the Reliability column of the FMRA (if it is shown). If the Reliability column is not visible in the FMRA, right-click the column headers, then click **Customize Columns** to open a window and select the **Reliability** check box.

Simulate Tree Elements opens the Batch Mode Simulation window, which allows you to select the systems to include in the simulation run. *See page 307 in Chapter 9*. This command is available only when working with FMRA simulation diagrams. If the simulation settings for the selected system(s) have not yet been defined, you will be given the option to configure the settings before starting the simulation.

Highlight applies a color-coding gradient to the results shown in each column, from green (least "desirable" result) to red (most "desirable" result). This command toggles highlighting on and off.

Mirror Groups

Mirrors gives you the option to add the selected cause to a mirror group or remove it from the group.

4.9.7 Format Tab

The Format tab is visible when you are working with an analytical diagram, simulation diagram or fault tree.

Format

Select Font opens the font window which allows you to set the font properties for the selected blocks.

Font Color opens the Color Palette, which allows you to set the font color for the selected text.

Fill Color opens the Color Palette, which allows you to set the fill color for the selected text.

Styles

Diagram Style opens the Diagram Style window, which allows you to customize the appearance of the current diagram and set the default properties for connectors and new blocks that are added to the diagram. *See page 197 in Chapter 6*.

Block Style opens the Block Style window, which allows you to customize the appearance of the selected block or annotation. *See page 204 in Chapter 6*.

Connector Style opens the Connector Style window where you can customize the appearance of the selected connectors. *See page 205 in Chapter 6*.

Alignment and Spacing

Align Blocks allows you to specify how you want the selected blocks to be aligned.

- **Align Tops** aligns the selected blocks to the top of the first selected block.
- **Align Middles** aligns the selected blocks to the vertical center of the first selected block.
- **Align Bottoms** aligns the selected blocks to the bottom of the first selected block.
- **Align Lefts** aligns the selected blocks to the left of the first selected block.
- **Align Centers** aligns the selected blocks to the horizontal center of the first selected block.
- **Align Rights** aligns the selected blocks to the right of the first selected block.
- **Align to Grid** aligns the selected blocks to the grid.

Make Spacing Equal allows you to specify how you want the blocks to be equally spaced.

- **Horizontal** equally spaces the selected blocks according to the horizontal spacing of the first two selected blocks.
- **Vertical** equally spaces the selected blocks according to the vertical spacing of the first two selected blocks.
- **Both** equally spaces the selected blocks according to the horizontal and vertical spacing of the first two selected blocks.

Make Same Size allows you to specify how you want the blocks to be resized.

- **Width** resizes the selected blocks to be the same width as the first selected block.
- **Height** resizes the selected blocks to be the same height as the first selected block.
- **Both** resizes the selected blocks to be the same width and height as the first selected block.

Bring to Front moves the currently selected blocks in front of other objects on the diagram.

Send to Back moves the currently selected blocks behind other objects on the diagram.

4.9.8 FRED Tab

The FRED tab is visible when you are working with a FRED report. *See Chapter 20*.

Levels

Append Level Below adds a row below the selected item that shows the blocks in the subdiagram.

Remove Level(s) Below removes any rows that have been appended to the selected block.

Restart FRED Report clears all levels of the FRED report except the original diagram so that you can restart the report creation.

Auto Arrange automatically arranges the blocks in the report so that all blocks are evenly spaced and centered.

Auto Expand expands the FRED report to show all subdiagrams and blocks.

Options

Diagram Style opens the Diagram Style window, which allows you to edit the appearance of the FRED report.

Change Block Color opens the Color Palette, which allows you to select a different color for the selected block.

Set Color Limits opens the Set Color Limits window, which allows you to set the color limits for the blocks in FRED Reports.

Export Graphic allows you to save the FRED Report diagram in one of the following formats: *.wmf, *.png, *.gif or *.jpg. You will be able to use the exported graphic in any application, provided that the application supports the file format.

Cascade Select selects all the blocks that are connected from the selected block, including the selected block itself. This allows you to easily select all blocks that are connected reliability-wise to a block.

Trace Path traces the path of the selected block. Click another block or the FRED Report to remove the traced path. The color of the traced path is the same as the color for a selected connector and can be set in the FRED Report Options window.

Export Data to Workbook opens the Select Workbook window, which allows you to select a workbook in the current project or to create a new workbook to export the FRED data to.

4.9.9 Phase Tab

The Phase tab is visible when you are working with a phase diagram. *See Chapter 10.*

Phase

Add Operational Phase adds a new operational phase to the current Phase diagram sheet. You can define the properties of the new operational phase using the Phase Properties window.

Add Maintenance Phase adds a new maintenance phase to the current Phase diagram sheet. You can define the properties of the new maintenance phase using the Phase Properties window.

Add Node adds a new node to the current phase diagram sheet. You can define the properties of the new node using the Block Properties window (*see page 194 in Chapter 6*).

Add Stop Block to indicate that the simulation of the mission ends. A new simulation may then begin, if applicable.

Add Subdiagram Phase adds a new subdiagram block to the current phase diagram. When this menu item is selected, the Select Diagram for the Block window will appear, which allows you to select a phase diagram sheet that exists in the current project to be represented by the subdiagram block.

Auto Arrange automatically arranges the phases in the Phase diagram so that all phases are evenly spaced and centered.

Auto Fit Width automatically arranges the blocks in the diagram to fit the width of the diagram window.

Settings

Connect Blocks allows you to create connectors between blocks in the current phase diagram. When this command is selected, click the source block, hold down the left mouse button and drag a line from the source block to the destination block. When the crosshairs are located above the destination block, release the mouse button to create a connector. To stop adding connectors and return to the normal mode, right-click the diagram or clear the **Connect Blocks** option (i.e., by choosing the command again).

Add Annotation adds a new annotation to the current phase diagram. You can edit the text that appears in the annotation by double-clicking it to open the Edit Annotation window.

Block Properties opens the Block Properties window, which allows you to define the failure, maintenance, optimization, throughput and other characteristics of the selected block (*see page 194 in Chapter 6*). If an annotation is selected, this command opens the Edit Annotation window, which allows you to edit the text for the annotation.

Set Block Flag allows you to display or remove a red, yellow or green flag above the selected block. These flags have no effect on the analysis or simulation so you can use them for any purpose of interest to you. For example, each flag color could represent a level of analysis that has been completed.

- **Remove Flag** removes the flag from the selected block.
- **Red Flag** displays a red flag for the selected block.
- **Yellow Flag** displays a yellow flag for the selected block.
- **Green Flag** displays a green flag for the selected block.

Selection

Diagram Actions

- **Refresh Diagram** updates the phase diagram to show any changes.
- **Show Block Details** opens the Block Details window, which displays the details of the blocks and resources in the current phase diagram.
- **Diagram Style** opens the Diagram Style window, which allows you to edit the appearance of the phase diagram. *See page 197 in Chapter 6*.
- **Open Diagram** opens the diagram that the selected subdiagram block represents. This option is only available when a subdiagram block is selected.
- **Open Parent Diagram** opens a sub-menu that allows you to select a parent diagram (i.e., a diagram that contains a subdiagram block representing the current diagram) to open.
- **Export Graphic** saves the diagram as a graphic in one of the following formats: *.wmf, *.png, *.gif or *.jpg. You will be able to use the exported graphic in any application, provided that the application supports the file format.

Cascade Select selects all the blocks that are connected from the selected block, including the selected block itself. This allows you to easily select all blocks that are connected to a block.

Trace Path traces the path of the selected block. Click another block or the diagram to remove the traced path. The color of the traced path is the same as the color for a selected connector and can be set in the Diagram Style window (**Format > Styles > Diagram Style**).

4.9.10 Plot Tab

The Plot tab is visible when you are working with a plot sheet.

Plot

Redraw Plot updates the plot to reflect the changes that have been made.

Analytical QCP opens the analytical Quick Calculation Pad (QCP). The QCP allows you to obtain reliability results based on the algebraic solution for the currently active diagram. *This is displayed only for analytical diagrams*.

Simulation QCP opens the simulation Quick Calculation Pad (QCP). The QCP allows you to obtain reliability, maintainability and availability metrics based on the simulation results. *This is displayed only for simulation diagrams*.

Actions

Copy Plot Graphic allows you to copy the plot to the Clipboard as a graphic. You must select an image type using one of the following options.

- **Select As Bitmap** when you need a bitmap version to be used in an external application.
- **Select As Metafile Optimized for Synthesis Spreadsheet** when you will be copying and pasting the image into Synthesis spreadsheets (e.g., analysis workbooks).
- **Select As Metafile Optimized for External Use** when you need a metafile that will be used by an external application.

You can specify which image type to use by default whenever you copy a plot to the Clipboard by selecting the option in the Synthesis Setup window (**File > Synthesis Setup**).

Plot Setup opens the Plot Setup window, which allows you to customize most aspects of the plot appearance including the titles, axis numbers and line styles. *See Chapter 13*.

RS Draw launches ReliaSoft Draw, which is a graphics editor that allows you to annotate your plot and view your plot in greater detail. With RS Draw, you can insert text, draw an object, mark the coordinates of a particular point or paste another picture into your plot. You can also re-arrange the objects in your plot by selecting and moving them to the positions you desire. In addition, you can save the annotated plot in any of the following formats: draw canvas (*.rdc), Windows metafile (*.wmf), portable network graphics (*.png), GIF or JPG. *See Chapter 14 for information on the features available in ReliaSoft Draw.*

Export Plot Graphic saves the plot as a graphic in one of the following formats: *.wmf, *.png, *.gif or *.jpg. You will be able to use the exported graphic in any application, provided that the application supports the file format.

Show/Hide Plot Items opens the Show/Hide Plot Items window, which allows you to select the data you want to display in the plot.

Confidence Bounds

Confidence bounds are available only for analytical diagrams. They can be shown for the Unreliability vs. Time, Reliability vs. Time and Failure Rate vs. Time plots in cases where all blocks that can fail in the diagram use models that have Fisher matrix bounds. To have Fisher matrix bounds, a model must have been created by publishing from a data source, such as a Weibull++ or ALTA data sheet, that was calculated using Fisher matrix bounds. Additionally, bounds are not available if the diagram has containers, subdiagrams or mirrored blocks.

Confidence Bounds opens the Confidence Bounds Setup window, which allows you to define the properties of the confidence bounds lines to be displayed on the plot.

Hide Confidence Bounds removes the line(s) depicting the confidence bounds from the current plot.

4.9.11 Sheet Tab

The Sheet tab is visible when you are working with an analysis workbook. *See Chapter 18*.

Format and View

Select Font opens the Font window, which allows you to set the font, style and size of the selected text.

Font Color opens the Colors window, which allows you to set the color of the selected text.

Fill Color opens the Colors window, which allows you to set the color used to fill the selected cell(s).

Fill Pattern opens the Cells Patterns window, which allows you to set the fill color, pattern color and fill pattern properties of the selected cell(s).

Alignment allows you to select the alignment properties of the selected text.

Borders allows you to customize the borders of selected cells.

Column Width opens the Column Width window, which allows you to specify the width of the selected column. Enter the desired width in the input box. You can select the units of measurement (characters, inches or centimeters) from the Units drop-down list. To use the default column width, click **Use Default**. You can change the default column width by clicking **Change Default**. This will open the Default Column Width window, which allows you to set a default column width.

Custom Number opens the Format Cells-Number window, which allows you to set the format for how numbers in the selected cell(s) will displayed.

Merge Cells combines selected cells to form one cell.

Unmerge Cells separates merged cells into the original individual cells.

More Settings

- **Goto** opens the Goto window, which allows you to automatically move the cursor to a particular cell location. Uppercase or lowercase may be used.
- **Freeze Panes** allows you to lock a specific row on the spreadsheet so that the contents of the selected row will always be visible on screen no matter how far you scroll to the right or down. You can also freeze

several consecutive rows at once by highlighting a cell on each row to be locked and then clicking **Freeze Panes**. Changes cannot be made to a frozen row(s).

- **Sheet Options** opens the Sheet Options window, which allows you to specify what elements you want to be displayed on the spreadsheet, such as gridlines and row/column headings, and what editing actions can be performed on the spreadsheet, such as the ability to resize rows/columns and select individual cells.
- **Calculation Options** opens the Calculation Options window, which allows you to change formula recalculation, iteration or precision.
- **Recalculate Formulas** forces a recalculation of all formulas in the spreadsheet.
- **Cell Protection** opens the Format Cells window, which allows you to select whether the selected cell(s) will be locked or hidden when you select the **Enable Protection** command. Cells that are locked cannot be edited when the sheet protection command is enabled.

  The Cell Protection setting is especially useful when your report contains a mix of data and formula cells where a user may inadvertently overwrite a cell containing a formula.
- **Enable Protection** see Cell Protection.
- **Defined Names** opens the Defined Names window, which allows you to create and manage variable names that can be referenced in any function. You can define a name for any spreadsheet cell or data source.

Sheet Actions

Function Wizard opens the Function Wizard, which helps you to build functions in analysis workbooks. These functions can return results ranging from mathematical values (e.g., sine, pi or averages) to reliability metrics based on associated data sheets (e.g., reliability, probability of failure, expected number of failures, etc.). *See page 477 in Chapter 18*.

Chart Wizard allows you to create new plots and charts based on the current spreadsheet. To use, select a range of spreadsheet cells for which you would like to create a chart for, then click **Chart Wizard** and select the area you want to place the chart on.

AutoSum allows you to compute the sum of the values in a range of selected cells. If a cell range is not selected, the command computes the sum of the values in the cells directly above the current cursor location.

Sort allows you to reorder the selected data either in ascending or descending order. Note that you may need to be careful when sorting certain types of data in order prevent errors in analysis. Note the following:

- If your analysis takes into account the order position of a data point (1st, 2nd, 3rd, etc. position). Sorting may result to an inaccurate analysis.
- If the range of cells that you wish to sort contain blank cells, the blank cells may be put in the first position (ascending order) or the last position (descending order).
- If the values in two or more columns are related, sorting only one column may result in errors. In that situation, use the Custom Sort command.

Rows/Columns

Delete deletes the selected cells. Note that when the cells are deleted, the information inside the cells is also deleted.

- **Shift Cells Left** shifts the cells to the left to accommodate the deleted cells.
- **Shift Cells Up** shifts the cells up to accommodate the deleted cells.
- **Entire Row** deletes the row(s) containing the selected cell(s). All subsequent rows are shifted up to accommodate the deleted cells.

- **Entire Column** deletes the column(s) containing the selected cell(s). All subsequent columns are shifted left to accommodate the deleted cells.

Insert allows you to insert cells into the currently active sheet at the current cursor location. If a range of cells is selected in the sheet, then the same number of cells will be inserted.

- **Shift Cells Right** shifts the cells to the right to accommodate the inserted cells.
- **Shift Cells Down** shifts the cells down to accommodate the inserted cells.
- **Entire Row** inserts a row of empty cells for each row containing a selected cell. All subsequent rows are shifted down to accommodate the inserted cells.
- **Entire Column** inserts a column of empty cells for each column containing a selected cell. All subsequent columns are shifted right to accommodate the inserted cells.

Transfer Data

Send to Excel exports all sheets to an Excel file.

Send to Word exports the current sheet to a Word file. A Word document opens with the data from the current sheet displayed as a Word table.

4.9.12 Simulation Tab

The Simulation tab is visible when you are working with a simulation diagram (*see Chapter 9*) or a RENO flowchart (*see Chapter 11*).

Simulation

Simulate opens the Maintainability/Availability Simulation window, for simulation diagrams (*see page 302 in Chapter 9*), or the RENO Simulation window, for flowcharts (*see page 399 in Chapter 11*). These windows allow you to specify simulation settings for the current diagram and run the simulation.

Simulation Plot creates a plot based on the simulation results. Clicking **Simulation Plot** before the diagram has been simulated will automatically perform the simulation and then plot the data.

Simulation QCP opens the simulation Quick Calculation Pad (QCP). The QCP allows you to obtain reliability, maintainability and availability metrics based on the simulation results. Clicking the QCP icon before the diagram has been simulated will automatically perform the simulation and then open the QCP. This command is available only when working with simulation RBDs and fault trees.

Debug is available only when working with RENO flowcharts. It opens the RENO Simulation Debugger, which allows you to trace and examine the output of a flowchart by running a step-by-step simulation through each block. *See page 396 in Chapter 11*.

Tools

Show Simulation Details displays the most recently generated simulation results in the Simulation Results Explorer. *See page 308 in Chapter 9*.

Check Analysis allows you to check for inconsistencies in the simulation diagram, such as a block with a corrective task but no failure model, or a block with scheduled tasks but no corrective task. This command is available only when working with simulation diagrams.

Batch Mode is available only when working with simulation diagrams. It opens the Batch Mode Simulation window, which allows you to simulate multiple diagrams consecutively, without user input in between simulation. *See page 307 in Chapter 9*.

Share

Publish Fitted Model allows you to publish the distribution that has been fitted to the simulated diagram as a model for use throughout the project, in any Synthesis application where models are used. *For more on publishing, see page 174 in Chapter 5*.

Dashboard

Dashboard Designer opens the Dashboard Designer window, which allows you to create templates that specify the contents of the Dashboard Viewer. This command is available only when working with simulation diagrams, and only if you have permission to create dashboard templates.

Dashboard Viewer opens the Dashboard Viewer, which allows you to view Synthesis data in a variety of graphical presentations. This command is available only when working with simulation diagrams.

Dashboard Manager opens the Dashboard Manager window, which allows you to see all of the dashboard templates that have been defined, and to import, rename and/or delete templates. This command is available only when working with simulation diagrams, and only if you have permission to create dashboard templates.

4.9.13 Workbook Tab

The Workbook tab is visible when you are working with an analysis workbook. *See Chapter 19*.

Worksheet

Insert Sheet adds a new sheet to the workbook.

Delete Sheet deletes the current sheet. ***There is no undo for delete***.

Select Sheet opens the Select Sheet window where you can select which sheet to work with.

Move/Copy Sheet opens the Move or Copy Sheet window, which allows you to move or copy any of the sheets within the current workbook. Select the sheet that you want to move the current sheet in front of or select (move to end) to move the current sheet to the end of all the sheets in the workbook. Select **Create a copy** to create a copy of the current sheet.

Actions

Function Wizard opens the Function Wizard, which helps you to build functions in analysis workbooks. These functions can return results ranging from mathematical values (e.g., sine, pi or averages) to reliability metrics based on associated data sheets (e.g., reliability, probability of failure, expected number of failures, etc.). *See page 477 in Chapter 18*.

Spreadsheet View sets the analysis workbook to Spreadsheet View, which displays the grid and the column/row headings.

Report View sets the analysis workbook to Report View, which hides the grid and the column/row headings, and displays the analysis workbook in the format in which it will be printed.

Recalculate recalculates formulas when relevant values change in any of the referenced recalculated data sources.

Defined Names opens the Defined Names window, which allows you to create and manage variable names that can be referenced in any function. You can define a name for any spreadsheet cell or data source.

Associate Data Sources allows you to define or change up to four default data sources that can be referenced by functions in the analysis workbook.

Save Workbook as Template allows you to save the current report as a template so that you can reuse its settings.

Synthesis Resources 5

In Synthesis applications, *resources* contain various types of information that can be shared between analyses. Resources may be created in one application and then shared with other Synthesis applications where they may be relevant. When the resource is updated with new information, the change is reflected in all analyses that rely upon it.

An example of a Synthesis resource is a model that represents a product's probability of operating successfully over time. The model may be defined manually or published from an application such as Weibull++/ALTA, RGA or Lambda Predict. Then it can be used in applications such as BlockSim, Xfmea, RCM++ or RBI as part of the universal reliability definition (URD) of a particular component/assembly.

The following sections discuss:

- A summary of all Synthesis resource types - Section 5.1 (p. 165)
- Local vs. global resources - Section 5.1.1 (p. 167)
- How to use wizards to create, select or edit resources - Section 5.2 (p. 168)
- The two most fundamental resource types:
  - Universal Reliability Definitions (URDs) - Section 5.3 (p. 170)
  - Models - Section 5.4 (p. 172)
- How to use the Resource Manager - Section 5.5 (p. 175)
- How to use the Dependency Viewer - Section 5.6 (p. 181)

5.1 Types of Resources

This table describes the types of resources that you can use in different Synthesis applications. These resources can be created while you're performing an analysis and also from the Resource Manager.

Note: The Resource Manager in each Synthesis application displays only the resources that may be relevant for that application. For a summary of which applications can create and/or use different types of resources, see: http://www.ReliaSoft.com/synthesis/integration.htm.

Resource	Used In
Universal reliability definitions (**URDs**) describe a set of reliability and maintenance characteristics for a particular component/assembly. *See Section 5.3 on page 170*.	BlockSim, RCM++, Xfmea, RBI
Models can represent probabilities, durations or costs, either fixed or time-dependent. They are used by other resources to represent the reliability of a component, the duration of a task, the expected cost of a repair, etc. *See Section 5.4 on page 172*.	Can be published from analyses in Weibull++, ALTA, BlockSim, DOE++, RGA and Lambda Predict, or created manually. Used in BlockSim, RENO, RCM++, RBI and Xfmea. (*For details on how to publish models, see Section 5.4.1 on page 174*.)
Tasks represent maintenance activities: • Corrective tasks are unplanned maintenance performed when a failure occurs. • Scheduled tasks can include preventive maintenance, inspections and on condition maintenance. *For more information on tasks, see page 281 in Chapter 9.*	BlockSim, RCM++, RBI
Task packages represent groups of tasks that are performed together at scheduled intervals. *See page 297 in Chapter 9*.	BlockSim, RCM++, RBI
Crews represent the personnel who will perform a maintenance task. *See page 292 in Chapter 9*.	BlockSim, RCM++, RBI
Spare part pools determine whether a spare part will be available, how long it will take to obtain and how much it will cost. *See page 294 in Chapter 9*.	BlockSim, RCM++, RBI
Maintenance groups are used to model situations in which some event within the group can trigger maintenance or state changes for other components/assemblies. *See page 279 in Chapter 9*.	BlockSim, RCM++, RBI
Switches describe how the activity transfers between active and standby blocks in BlockSim's standby containers/gates. *See page 224 in Chapter 7*.	BlockSim
Mirror groups are used to represent the exact same component/event in more than one location within your analysis. *See page 243 in Chapter 7*.	BlockSim, RCM++, Xfmea, RBI
Maintenance templates define the activities that will be performed during a maintenance phase in a phase diagram. *See page 344 in Chapter 10*.	BlockSim
Actions describe specific assignments that need to be performed. *See page 103 in Chapter 3*.	All applications via My Portal; RCM++, RBI and Xfmea for analyses

Resource	Used In
Controls may be used in FMEAs and/or control plans to represent methods to reduce or eliminate the risk associated with potential failures.	RCM++, Xfmea, RBI
Variables store numerical values that can be programmatically varied during simulation. *See page 381 in Chapter 11*.	BlockSim, RENO, DOE++
RENO functions store equations that are evaluated based on input values passed to the function during simulation.*See page 383 in Chapter 11*.	RENO
RENO static functions store equations that are evaluated before simulation begins.*See page 383 in Chapter 11*.	RENO
RENO simulation definitions are used to trigger simulation of a BlockSim diagram from within a RENO flowchart, allowing you to use one or more results from the simulation in the flowchart.*See page 386 in Chapter 11*.	RENO
RENO tables store arrays of values in rows and columns.*See page 389 in Chapter 11*.	RENO

5.1.1 Local and Global Resources

By default, resources are *local*, which means they can be used only within the project in which they were created. For certain types of resources, however, you can set them to be *global resources*, which are available for any project within the database. This can be done for resources of the following types:

- URDs
- Models
- Corrective tasks and scheduled tasks
- Task packages
- Crews
- Pools
- Maintenance groups
- Variables

If you make a resource global, any resources assigned to it will become global as well (e.g., if a URD has an assigned model, making that URD global will also make the model global). Similarly, you cannot assign a local resource to a global one.

To make a resource global, select it in the Resource Manager and choose **Home > Actions > Make Global**.

Icons are used in the Resource Manager and in resource wizards to indicate which resources are local and which are global.

Local Resource

Global Resource

Note: When you create a restore point, any global resources used in the project are converted to local resources and stored with the backup.

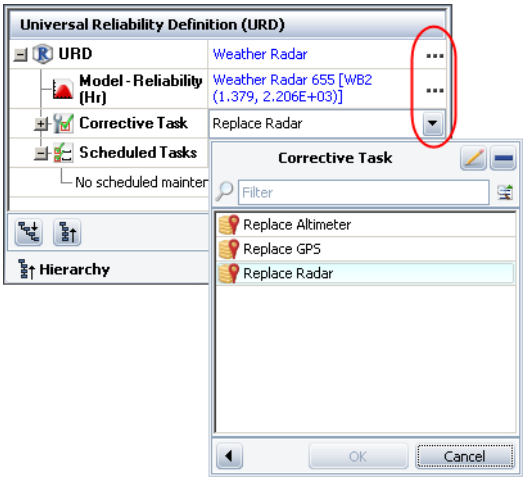
5.2 Creating and Selecting Resources

This topic describes how to use resource wizards to create and edit Synthesis resources while you're performing an analysis or creating a URD. In addition, you may also need to:

- Use the Resource Manager (*Section 5.5 on page 175*) to see all of the resources that are available to use in the current project. In addition to creating and editing resources, the manager also allows you to:
  - Delete resources that are no longer needed.
  - Create global resources that are available to all projects in the database.
- Publish a model that is linked to an existing analysis. *See Section 5.4.1 on page 174*.

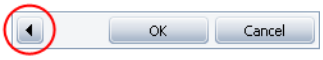
Opening a Resource Wizard

Throughout all Synthesis applications, the availability of a resource wizard is usually indicated by an ellipsis (...) or an arrow in the resource field. Simply click twice in the field to open the wizard.



Back Arrow and Main Page

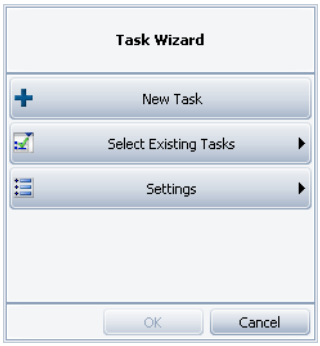
Each wizard contains multiple pages with different options on each page. If the option you need is not immediately visible, you can click the **Back** arrow to see the Main page, which shows all the options that are applicable for the current situation.



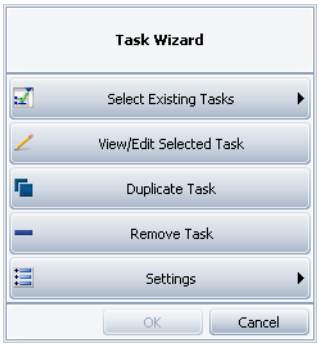
As an example, the following pictures show some of the different options that will be available on the Main page for specific situations and resource types.

If a model hasn't been defined...

If a task hasn't been defined...



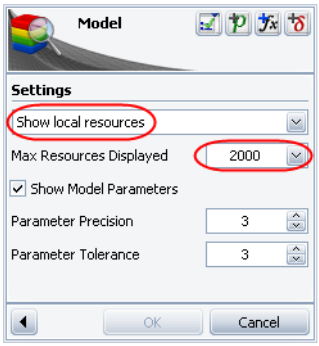
If a task has already been defined...

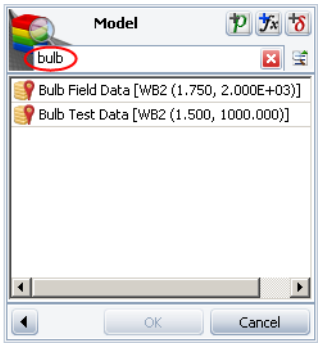


Selecting an Existing Resource

The Select page of the wizard displays a list of existing resources that meet the criteria specified on the Settings page. You can then further limit the list by typing inside the text box.

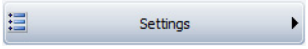
As an example, the following pictures show a situation in which the settings are configured to display up to 2,000 models (more resources take longer to load) from the current project only (called local resources), and the user has selected to see only the ones in which the name includes “bulb.”





In addition, for model resources only, you must specify whether you want to display the model parameters in the list (which also takes longer to load); and if so, the number of decimal places (called *Parameter Precision*) and the point at which the software will switch to scientific notation (called *Parameter Tolerance*).

To change the initial criteria for the list, click the **Back** arrow and then click the **Settings** button on the main page.



Alternatively, if you need more information about the available resources and/or a wider range of filtering tools, you can click the icon to open the Select Resource window.

Creating a Resource

In most wizards, you can click the **Create New** button or icon to create a new resource.

In the model wizard, click the button or icon for the type of model you want to create:

New Constant

New Distribution

New Dynamic

Editing a Resource

If a resource has already been assigned, you can view and/or edit its properties by clicking the **View/Edit** button or icon.

If you have the permissions needed to edit the resource, any changes you make will apply everywhere that the resource is used. This includes analyses in other Synthesis applications.

Removing a Resource

If a resource has already been assigned, you can remove it from this location by clicking the **Remove** button or icon.

The resource will remain in the database for use in other locations. If you want to completely remove a resource from the database, you'll need to use the Resource Manager or Select Resource window.

5.3 Universal Reliability Definitions (URDs)

A universal reliability definition (URD) describes a set of reliability and maintenance characteristics for a particular component/assembly. Like any other Synthesis resource, you can create or edit URDs while you're performing a relevant analysis, and from the Resource Manager.

Note: Depending on where you are using a URD, only those properties that are relevant are applied. For example, if you apply a URD to a block in an analytical diagram in BlockSim, only the failure model associated with the URD will be applied to the block.

Naming the Resource

For a new resource, a name will be proposed automatically based on the default naming criteria established for the current database in the Define Default Name Formats window. You can replace this with your own name of up to 150 characters, if desired. Remember that the name and identifiers are the primary way in which your team will be able to find the Synthesis resources you need for your analyses.

Hierarchy Tab

Use the Hierarchy tab to view and/or edit the URD's properties. You can choose or create:

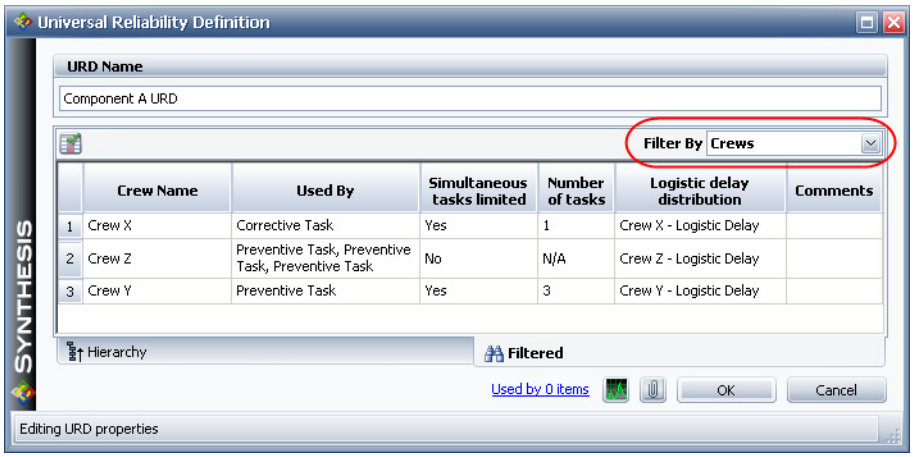
- A **model** to describe the behavior associated with the URD. This can be a reliability model, a probability of failure model or an event occurrence model.
- A **corrective task** that describes the maintenance action taken to restore a failed component to operational status.
- Zero, one or many **scheduled tasks** that describe the preventive maintenance, inspections and/or on condition maintenance tasks. The tasks are displayed in the order in which they will be performed; you can use the up and down arrows in the cells in this column to move the task up and down the priority list.

Additionally, the following are shown:

- **Identifiers** allows you to enter additional identifying information that can be used to search for this resource.
- **History** provides information about the creation and last modification of the resource. If the history log has been activated at the project level, you can click the **View Item History** icon to open the Record History Log for the resource. *For more information, refer to the "History Logs" section in the "Working in Synthesis" chapter.*

Filtered Tab

Use the Filtered tab to see the resources that are shown in the Hierarchy tab, filtered by resource type. For example, the following picture shows the crews used in this URD.



Double-click any row to view or edit the properties of the resource.

You can change the columns shown in this view by clicking the **Column Chooser** icon.

Trace Usage

For an existing resource, a link at the bottom of the properties window indicates how many times it is currently being used. If you need more information, click the link or the **Trace Usage** icon.

This opens the Dependency Viewer, which allows you to explore both where the resource is being used and what other resources are associated with it. *See Section 5.6 on page 181.*

5.4 Models

In Synthesis applications, *models* can represent probabilities, durations or costs, either fixed or time-dependent. They are used by other resources to represent the reliability of a component, the duration of a task, the expected cost of a repair and many other characteristics.

Like any other Synthesis resource, you can create or edit models while you're performing a relevant analysis, and from the Resource Manager (*Section 5.5 on page 175*). In addition, you can publish models that are based on an existing analysis (e.g., a life data analysis in Weibull++, a diagram in BlockSim, etc.). *For more information on publishing models, see Section 5.4.1 on page 174.*

This topic describes the properties for both types of models. The interface will vary based on the particular situation.

Naming the Resource

For a new resource, a name will be proposed automatically based on the default naming criteria established for the current database in the Define Default Name Formats window. You can replace this with your own name of up to 150 characters, if desired. Remember that the name and identifiers are the primary way in which your team will be able to find the Synthesis resources you need for your analyses.

Model Basis and Status

In published models only, the **Model Basis and Status** area identifies the analysis that the model is based on. If the published model reflects the latest analysis results, the status is "Synchronized." If the analysis has been modified since the model was last published, the status is "Out of Sync."

Tip: If you have the ability to open the original analysis (i.e., if the required Synthesis application is activated on your computer and your user account has permissions to access the analysis), the **Source** will be configured as a link and there will also be a **Data Source** button at the bottom of the window.

Model Category

The category determines where and how a model can be used. If you are using a resource wizard to create a new model, it will be assigned automatically based on what's relevant for the current field. If you are publishing a model or creating it from the Resource Manager, you must select the appropriate option. Once a model has been created, you cannot change its category.

- **Reliability**, **Probability of Failure** and **Event Occurrence** models represent a likelihood of occurrence.
- **Duration** models represent a length of time.
- **Cost per Unit Time** models are used for costs that accrue over time (e.g., the crew charges $50 per hour).
- **Cost** models are used for costs that don't depend on time (e.g., the part costs $100 or the crew charges $50 per service call in addition to their hourly rate).

Model Type

- A **distribution** model represents behavior that varies based on factors such as time and/or applied stress. For a published model, the inputs and parameters will be specified automatically based on the associated analysis. For manually created models, you can:
  - Select a distribution from the drop-down list and then enter the required parameter(s).
  - Use the Quick Parameter Estimator (QPE) to estimate the parameters of a distribution based on what you know about the behavior.

> **Tip:** If you have the ability to open the original analysis (i.e., if the required Synthesis application is activated on your computer and your user account has permissions to access the analysis), the **Source** will be configured as a link and there will also be a **Data Source** button at the bottom of the window.

- A **constant** model represents a fixed probability (e.g., 0.9), duration (e.g., 2 hours) or cost (e.g., $10 or $10 per hour).
- A **dynamic** model represents a fixed probability, duration or cost, based on a specified variable. The variable can then be programmatically varied during simulation in one of two ways, thereby changing the value of the model for successive simulation runs.
  - For RENO flowcharts, the variable can be varied using RENO's sensitivity analysis/multiple analyses features.
  - When simulating a simulation worksheet, the values used by the variable will be specified in the worksheet.

  When dynamic models are used outside of these circumstances, they are treated as constant models using the defined initial value of the variable.

Identifiers and History

For manually created models, the Identifiers tab of the properties window shows the following:

- **Identifiers** allows you to enter additional identifying information that can be used to search for this resource.
- **History** provides information about the creation and last modification of the resource. If the history log has been activated at the project level, you can click the **View Item History** icon to open the Record History Log for the resource. *For more information, refer to the "History Logs" section in the "Working in Synthesis" chapter.*

For published models, identifiers are shown on the Identifiers tab of the control panel for the associated analysis.

Trace Usage

For an existing resource, a link at the bottom of the properties window indicates how many times it is currently being used. If you need more information, click the link or the **Trace Usage** icon.

This opens the Dependency Viewer, which allows you to explore both where the resource is being used and what other resources are associated with it. *See Section 5.6 on page 181.*

5.4.1 Publishing Models

This section provides general information about how to create and work with *published models*, which are model resources that have been published from and continue to be associated with an existing analysis. For information about additional options and requirements in specific applications, see:

- "Publishing Fitted and Analytical Models" in the BlockSim documentation.
- "Publishing Models from R-DOE Results" in the DOE++ documentation.
- "Publishing from Failure Rate Predictions" in the Lambda Predict documentation.
- "Publishing Models from Analysis Results" in the RGA documentation.

The Synthesis Platform does not automatically update the published model when something changes in the original analysis, but users have the option to republish the model at any time. For any other analysis that uses the model, the change will be reflected the next time that other analysis is recalculated/resimulated.

Publishing Tools

For most Synthesis applications, the following model publishing tools are found on the Publishing page of the control panel. In Lambda Predict, they are located on the Model tab of the Properties panel.

Publish Model publishes the analysis to a model, making it accessible to all Synthesis applications.

- If the analysis has never been published, this will create a new model and you will be prompted to specify the name and category.
- If the analysis has already been published, this will update the existing model and change the model's status from "Out of Sync" to "Synchronized."

Publish to Existing Model allows you to select an existing model (using the Select Resource window) and replace it with an association to the current analysis.

Trace Usage opens the Dependency Viewer (*Section 5.6 on page 181*), which allows you to see where the model is used.

View Model displays the properties for the published model. This information will be available to any Synthesis user who is considering whether to use the model in a particular analysis.

Remove Association removes the link between the current analysis and the published model. The model will continue to be a resource in the database but it can no longer be synchronized with the analysis.

Published Model Summary

After the model has been published, the summary table in the control panel (or Model tab in Lambda Predict) will display the model's name, along with the following details:

- When the **Status** changes from "Unpublished" to "Published," the label becomes a link that opens the model's properties window.
- **Linked?** indicates whether the published model has been used. If the model is in use, you can click the link in this field to open the Dependency Viewer.
- **Synchronized?** displays "Synchronized" if the published model reflects the latest results from the associated analysis. If that analysis has been modified since the model was last published (e.g., if more data has been added, an analysis setting has changed, etc.), the status will display as "Out of Sync."

- The **Created By** and **Modified By** fields display the names of the users who created and last updated the model. Click either of the links to see the dates and times.

Model Basis and Status

When a model has been published from an existing analysis, its properties window will include the following additional information:

Model Basis & Status	
Source	Folio 1/Data 1
Application	Weibull++
Type	Standard Folio
Status	Synchronized

- **Source** is the Synthesis analysis that the model was published from.
- **Application** is the Synthesis application that is required to view/edit the original analysis.
- **Type** is the kind of folio or diagram that the model is based on. For example, this might be a standard folio in Weibull++, an analytical diagram in BlockSim, a failure rate prediction in Lambda Predict, etc.
- **Status** displays "Synchronized" if the published model reflects the latest results from the associated analysis. If that analysis has been modified since the model was last published (e.g., if more data has been added, an analysis setting has changed, etc.), the status will display as "Out of Sync."

Tip: If you have the ability to open the original analysis (i.e., if the required Synthesis application is activated on your computer and your user account has permissions to access the analysis), the **Source** will be configured as a link and there will also be a **Data Source** button at the bottom of the window.

5.5 Resource Manager

The Resource Manager allows you to create, view, edit and delete Synthesis resources. It also helps you to trace where each resource is used, and to create global resources (*Section 5.1.1 on page 167*) that are available to all projects in the database.

To open the Resource Manager, choose **Project > Synthesis > Resource Manager**.

To use the utility, choose the type of resource that you want to work with in the navigation panel on the left side of the window. A table that displays the resources of the selected type will be shown on the right side of the window. For information on configuring the table and limiting the resources shown, refer to Choosing and Configuring Columns (*Section 5.5.1 on page 177*) and Finding and Filtering Records (*Section 5.5.2 on page 177*).

Note: The Resource Manager in each Synthesis application displays only the resources that may be relevant for that application. For a summary of which applications can create and/or use different types of resources, see: http://www.ReliaSoft.com/synthesis/integration.htm.

Creating, Viewing, Editing and Deleting Resources

You can:

- Create a new resource of the selected type by choosing **Home > Edit > New**.

To add multiple resources at the same time, specify the quantity in the **Home > Batch Add > Number to Add** field and then choose **Home > Batch Add > Add Items**. You will specify the properties for one of the new resources, and the additional resources will be duplicates of that. You can then edit each resource individually, or in BlockSim you can go to the Item Properties Table to edit them in a spreadsheet format.

- Duplicate an existing resource by selecting the row in the table and choosing **Home > Edit > Duplicate**.

- View or edit an existing resource by double-clicking the row in the table or by selecting the row and choosing **Home > Edit > View**.

- Delete an existing resource by selecting the row in the table and choosing **Home > Edit > Delete**.

  You cannot delete a resource that is currently being used in any other resource or analysis.

Note: A published model cannot be edited or deleted from the Resource Manager. If changes are needed, you must update the original analysis and then republish the model. If you wish to delete the model, you must first remove the association with the original analysis and also make sure the model is not being used. *See Section 5.4.1 on page 174.*

Tracing Where Resources Are Used

With a single resource selected, you can choose **Home > Actions > Trace Usage** to open the Dependency Viewer, which provides information on where the resource is used and any additional resources that the currently selected resource itself uses. *See Section 5.6 on page 181.*

If you need to know which resources are not currently being used (e.g., because you want to delete obsolete records), choose **Administration > Selection > Show Only Unused**.

Local and Global Resources

As explained in Section 5.1.1 on page 167, local resources are available only in the current project, while global resources are available for use in any project in the database. To make a local resource global, select the row in the table and choose **Home > Actions > Make Global**.

Any associated resources will also be made global. For example, if you make a URD global, any associated models and tasks will also be made global. Similarly, you cannot assign a local resource to a global one.

The far left column in the Resource Manager displays an icon that indicates whether each resource is local or global.

Local Resource

Global Resource

You can use the **Home > Filter > [Show Local Resources/Show Global Resources/Show All Resources]** command to show only local resources, only global resources or all resources.

Exporting Information from the Resource Manager

To print the current view of the Resource Manager, choose **Home > Edit > Print**.

To send the contents of the Resource Manager to Excel, or to a built-in analysis workbook or spreadsheet if that's available in the current application, choose **Administration > Actions > Transfer Report**.

In the Transfer Report window that appears, you can select to export either the current item (i.e., the table for the currently displayed page) or the full report (i.e., the tables for all pages in the Resource Manager).

5.5.1 Choosing and Configuring Columns

In the Resource Manager and in BlockSim's Item Properties Table, you can configure the columns to suit your needs. These settings are stored per computer/username for each application, and different users may have different display preferences without affecting the stored data.

- To rearrange the columns, select a column header and drag it to the place in the grid where you want the column to appear. The column positions that you specify will be “remembered” and used each time you use the application.
- To resize the columns, you can do any of the following:
  - Drag the edge of the column header to the desired position.
  - To automatically resize one column to fit the data, double-click the edge of the column header or right-click the column header and choose **Best Fit** on the shortcut menu.
  - To resize all columns to fit the data, choose **Best Fit (all columns)** on the shortcut menu.
- To sort the table by a given column, click the column header. You can reverse the direction of the sort by clicking the column header again. You can also right-click the column header and choose **Sort Ascending** or **Sort Descending** on the shortcut menu, or choose **Clear Sorting** to stop sorting by the selected column.

You can specify which columns will appear in the table in two ways:

- Click the **Columns** icon at the bottom of the navigation panel to display a list of available columns in the navigation panel.

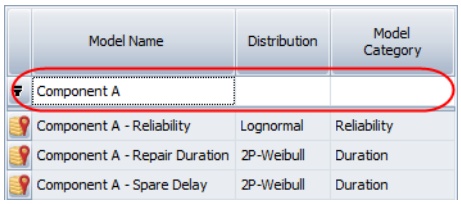
- Choose **Home > Columns > Column Chooser** or right-click any column header and choose **Column Chooser** to open the Select Columns window, which offers a sortable list of the available columns for the current type of item. The order that you sort by in this window does not affect the column positions in the Resource Manager or Item Properties Table.

5.5.2 Finding and Filtering Records

In table-based interfaces such as the Resource Manager, BlockSim's Item Properties Table and the Reliability Data Warehouse interfaces, there are several ways to find and filter data. Each of these tools can be used separately, or in conjunction with the others.

5.5.2.1 Auto Filter Row

The Auto Filter row appears by default immediately below the column headers. Enter text in any cell in this row to display only those items that start with matching text in that field.

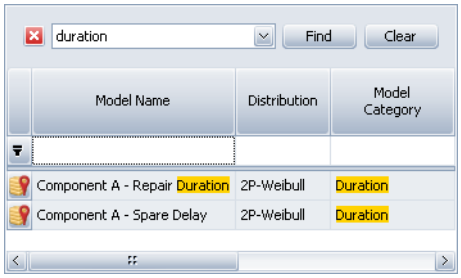


You can hide or show the Auto Filter row by choosing **Home > Filter > Display Auto Filter Row** or by right-clicking any column header and choosing **[Hide/Show] Auto Filter Row**.

5.5.2.2 Find Panel

The Find panel, which is hidden by default, allows you to search for specific text in any record property. Right-click any column header and choose **Show Find Panel** to display the panel above the column headers.

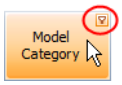
Enter text in the field and click **Find** to filter the items displayed in the table and highlight the matching text. You can click **Clear** to remove the filter. To close the panel, click the **X** or right-click any column header and choose **Hide Find Panel**.



5.5.2.3 Advanced Filters

The Advanced Filters functionality is always enabled in the XFRACAS to RDW interface and in the RDW to Folio window. In the Resource Manager and the Item Properties Table, you can enable it by choosing **Home > Filter > Enable Advanced Filters**.

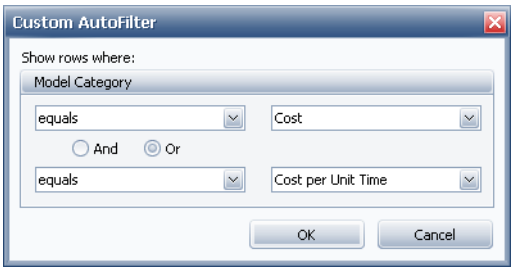
When advanced filters are enabled, a **Filter** icon will appear when you point to any column header in the table.



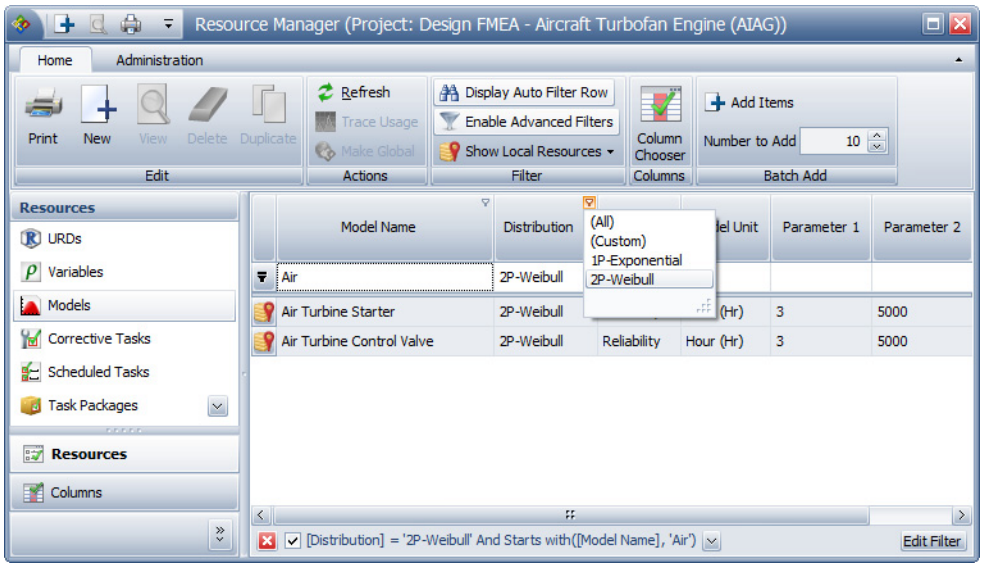
Click this icon to see a drop-down list of filters that you can apply for the currently selected column. These filters are automatically generated based on the values that currently exist in the column. For example, if you have reliability, duration and cost models in the project, those categories will automatically be available from the Model Category drop-down list.

If no automatically generated filters are available or if you want to specify more than one possible value, you can choose **(Custom)** to open the Custom AutoFilter window, which allows you to specify two filter criteria for the

property, in either an “and” or an “or” relationship. For instance, the window shown next will filter the available models to show only the cost and cost per unit time models.



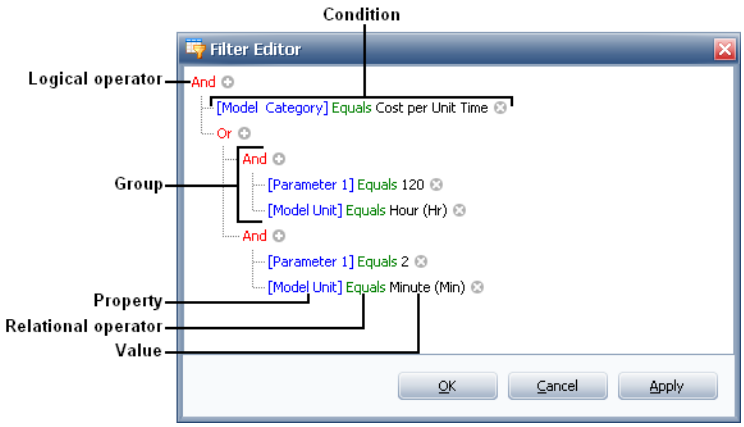
You can apply multiple filters in these ways to narrow the resources in the table as needed. Each column that has a filter applied will continue to display the Filter icon in the column header. The filters that you apply will appear at the bottom of the table, as shown next.



Click the check box to apply or remove the set of filters. Click the **X** to delete the set of filters entirely. To delete a single filter from the set, right-click the relevant column header and choose *Clear Filter*.

5.5.2.4 Filter Editor

When you have Advanced Filters enabled, you can edit the filters by clicking the **Edit Filter** button in the filter display or by right-clicking any column header and choosing **Filter Editor**. Note that this window also includes filters applied in the Auto Filter row.



Click any colored element in the window to view a drop-down list of available values.

- Click a red label if you want to:
  - change the logical operator (And, Or, Not And, Not Or)
  - remove the group
  - add a sub-group
  - clear the entire filter
- Use the + and x icons to add or remove conditions within an existing group.
- Click a blue label if you want to change the property (e.g., Model Name, Created By, etc.).
- Click a green label if you want to change the relational operator (e.g., Equals, Contains, Begins with, etc.).

For example, the filter shown above will display only cost per unit time models in which the rate equals either $120 per hour or $2 per minute.

5.5.3 Removing Unused and Duplicate Resources

The Resource Manager provides several tools to make it easy to delete unused resources and merge duplicate resources.

Finding and Removing Unused Resources

To find unused resources of the selected type, choose **Administration > Selection > Show Only Unused**.

Once you have found the unused resources, you can decide which ones to delete.

You can return to viewing all resources of the selected type by choosing the **Show All** command.

Finding Duplicate Resources

You may sometimes have duplicate resources within a project (i.e., several models that all represent the identical distribution, several tasks that are identical, etc.) within a project. For example, this might happen if you have imported diagrams that all reference the same model, but the imports happened at separate times and therefore the model had to be imported again each time.

To find duplicate resources, choose **Administration > Selection > Show Only Duplicates**.

In the window that appears, select the properties that must match in order for the resources to be considered duplicates. For example, if you are looking for duplicate URDs, you might select to compare the model, corrective task and scheduled tasks, but not the URD name. Once you have selected the columns to compare, click **OK** to show the duplicate resources.

You can merge multiple resources of the same type into one resource, either automatically or manually. Note that models must be of the same category in order to be merged, and you cannot merge published models.

IMPORTANT: Merging cannot be undone, and only the information from the “target” resource (i.e., the resource that you merge the others into) will be retained.

Automatically Merging Duplicates

To merge resources automatically, choose **Administration > Actions > Merge All Duplicates**.

This will find all sets of duplicates within the selected resource type (e.g., three identical Weibull models and 2 identical lognormal models); and for each set, it will merge the duplicates into the first instance found. After you confirm that this is what you want to do, you will be prompted to select the properties that must match in order for the resources to be considered duplicates. The process begins as soon as you select at least one column and click **OK**. It cannot be undone.

Selecting Resources to Merge

To merge resources manually, select the resources and choose **Administration > Actions > Merge & Delete**.

The window shows a list of the resources you selected to merge and allows you to choose which one will be retained (i.e., the one that the other selected resources will be merged into). After you click **OK**, this resource will be used in all places where the other selected resources were previously used.

5.5.4 Select Resource Window

The Select Resource window allows you to choose an existing resource to use at your current location. You can access it by clicking the **Select Existing** icon in any resource wizard.

The Select Resource window is similar to the Resource Manager, with the following differences:

- It shows only the resource type that you are currently working with.
- Each row in the table contains a radio button that allows you to select the resource for use.
- The window contains **OK** and **Cancel** buttons that allow you to apply the selected resource or close the window without applying a resource.

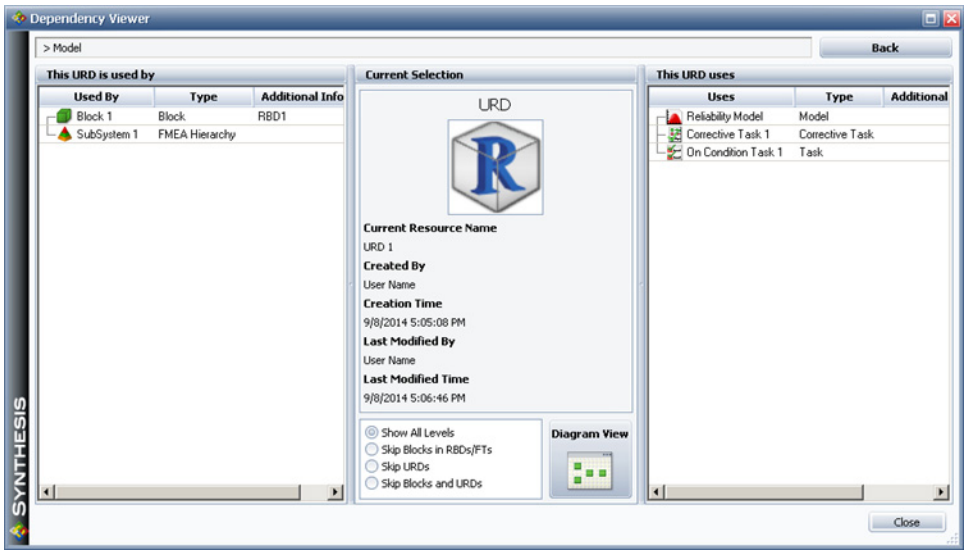
5.6 Trace Usage: Dependency Viewer

Because resources can be used multiple times, it is important to know where a given resource is used before making changes to it. The Dependency Viewer, which is accessed by clicking the **Trace Usage** icon in each resource properties window or in the Resource Manager, provides information on where the resource is used and what additional resources, if any, it uses.

The selected resource is shown in the **Current Selection** area in the middle of the Dependency Viewer.

The items that use the selected resource, if any, will be displayed in the **This [resource] is used by** area on the left. If it uses any resources, those resources will be displayed in the **This [resource] uses** area on the right. For example, in

the picture shown next, the selected resource is a URD. It is used by Block 1 in a BlockSim diagram and by SubSystem 1 in an FMEA analysis. The URD uses an assigned model and two tasks.



You can edit any resource shown in the Dependency Viewer by right-clicking it and choosing **Edit Item** on the shortcut menu. The resource's properties window will open.

You can double-click an item in either area to make it the current selection in the window. For example, you could double-click a task used by the URD to see the model(s), crew(s) and/or spare part pool(s) that the task uses.

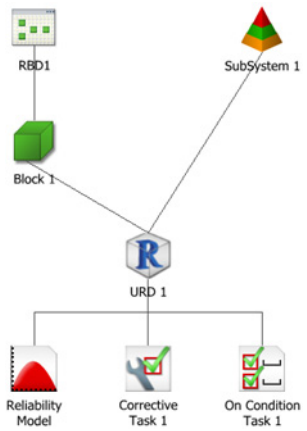
As you change selected items, each previous item that you selected will be added to the "path" of recent selections at the top of the window; click **Back** to go back through the path to the previously selected items. For example, the following picture shows a path in which the user started with the URD, then proceeded to its corrective maintenance task, the crew for that task and the logistic delay for that crew (not shown in the path because it is the current selection). The button now provides a quick way to go back through those same resources in reverse order if needed.

You can click the **Diagram View** button to open the diagram view of the Dependency Viewer (described in the next section). The currently selected item will be the main block in the diagram that is created.

Choosing What You See

In the **Current Selection** area, you can choose what is shown in the **This [resource] is used by** area. You have the option to skip the URD and/or the block that the resource is used by. For example, the image below shows the connections among analyses and resources that we have been discussing. If you have selected to show all levels, you

can trace from Reliability Model to URD 1 to Block 1 to RBD 1. But if you have selected **Skip Blocks and RBDs**, it just shows that Reliability Model is used by RBD 1.

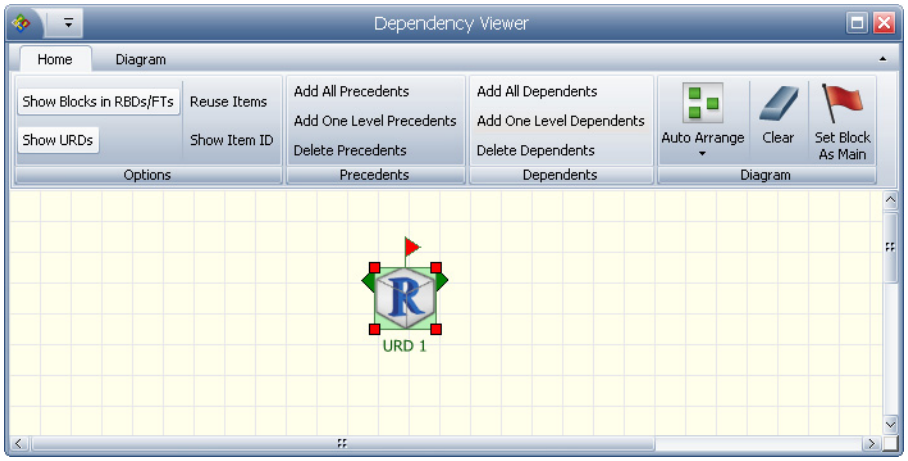


Note that blocks are always skipped in RENO flowcharts (i.e., only the flowchart will be shown as the precedent for a resource used by any block within the flowchart).

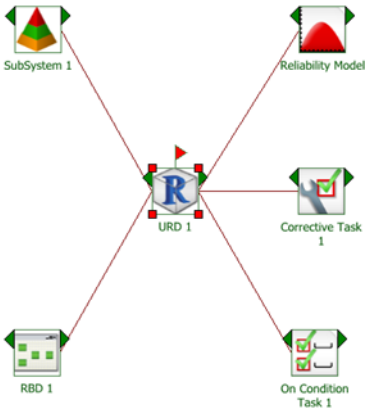
5.6.1 Dependency Viewer - Diagram View

The diagram view of the Dependency Viewer, which is accessed by clicking the **Diagram View** button, offers a diagram-based alternative to help you understand the connections among resources in your project.

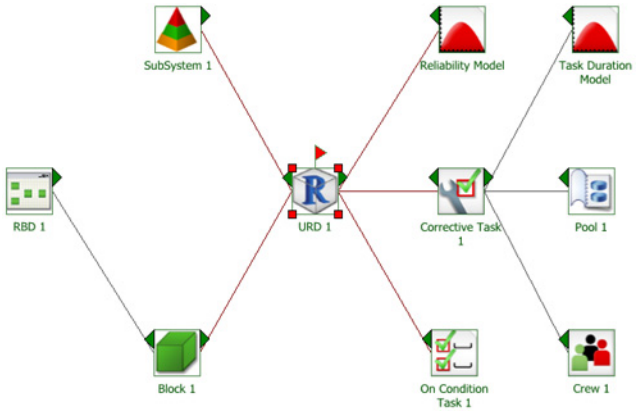
To understand how to use this view, consider the same example shown in the previous section. Clicking the **Diagram View** button for that example will yield a diagram view that shows only the main item (in this case, the URD), which is marked with a flag:



To view a graphical representation of the information that was shown in the Dependency Viewer, choose **Add One Level Precedents** and **Add One Level Dependents**. This will show the items that use the resource (i.e., its precedents), and the resources that it uses, if any (i.e., its dependents).



If you want to see all levels, choose **Add All Precedents** and **Add All Dependents**:

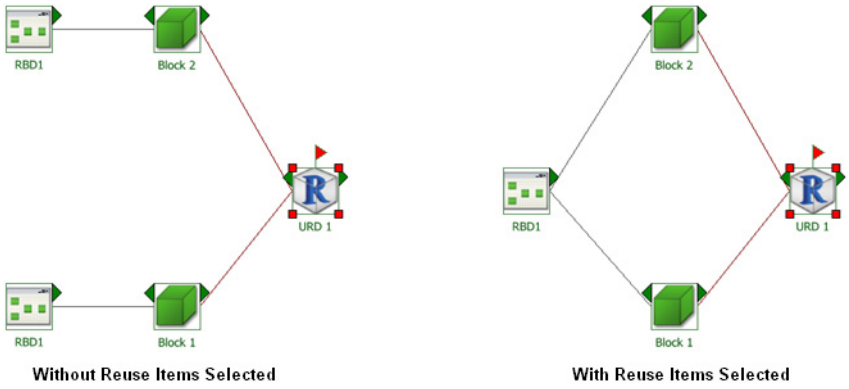


If you want to add or remove one branch at a time, click the green arrows on the sides of the blocks. Alternatively, if you want to remove one branch, select the block and choose **Delete Precedents** or **Delete Dependents**.

To set a different item as the main item in the diagram, select it and choose **Set Block as Main**. This serves the same purpose as changing the selected block in the Dependency Viewer.

Use the commands in the Options group of the Ribbon's Home tab to specify how the diagram will be constructed:

- Select **Reuse Items** to show each item only once in the diagram. For example, if you are viewing a URD that is used by two blocks in the same diagram and you choose **Add All Precedents**, the Reuse Items command will affect the display as shown next:



  This command must be selected before the diagram is created, or you must clear the current diagram to apply it.

- Use the **Show Blocks in RBDs/FTs** and **Show URDs** commands to determine whether blocks and/or URDs are shown as intermediate steps between resources and the analyses in which they are used.
- Select **Show Item ID** to display the internal ID assigned to each resource, block and/or diagram. These IDs can be used to differentiate among items that share the same name.

The Diagram tab of the Ribbon offers zoom options and printing options, as well as the ability to copy the diagram or to export it as a graphics file.

Building and Customizing Diagrams

6

BlockSim and RENO both use diagrams of various types. These diagrams differ in terms of their purposes and capabilities: BlockSim's reliability block diagrams (RBDs) and fault trees are symbolic logic techniques that can be applied to analyze system reliability and related characteristics, and RENO's flowcharts are used to build and simulate models of probabilistic or deterministic problems, processes or scenarios in order to help you understand and/or quantify them. However, all of the diagram types have underlying similarities in how they are created and configured.

This chapter provides general information about the diagrams and the tools available for creating them, including:

- Techniques for building diagrams - Section 6.1 (p. 188)
- Using the Block Properties window - Section 6.2 (p. 194)
- Tools for configuring diagrams to look the way you want them to - Section 6.3 (p. 197)

More specific information about the individual diagram types is available in other sections of the User's Guide.

- If you're interested in using reliability block diagrams and/or fault trees:
  - The BlockSim RBDs and Fault Trees chapter discusses the tools that are common for both analytical and simulation diagrams (such as block types, configuration types, tools for managing the properties for all blocks within a particular diagram, etc.). *See Chapter 7*.
  - The BlockSim Analytical Diagrams chapter discusses the analysis results and plots that are available for analytical RBDs or fault trees. *See Chapter 8*.
  - The BlockSim Simulation Diagrams chapter discusses the analysis results and plots that are available for simulation RBDs or fault trees. *See Chapter 9*.
- If you're interested in phase diagrams, see BlockSim Phase Diagrams. *See Chapter 10*.
- If you're interested in using flowcharts for probabilistic event and risk analysis, see RENO Flowcharts. *See Chapter 11*.

6.1 Techniques for Building Diagrams

To add a diagram to a project, choose **Insert > Diagrams > [Diagram Type]**.

To add blocks of any type (e.g., standard blocks, node blocks, fault tree gates, etc.) to a diagram, choose the applicable **Add** command on the [Diagram/Fault Tree/Phase/Flowchart] tab of the Ribbon. For example, to add a standard block to a reliability block diagram, choose **Diagram > Blocks > Add Block**.

To add an AND gate to a fault tree, choose **Fault Tree > Gates and Events > Add Gate > Add AND Gate**.

Note: If a gate is currently selected in the fault tree when you add an item, the new item will be added below the selected gate and connected to it. If an event is currently selected in the fault tree when you add a gate, the new gate will be added above the selected event and connected either to the event or, if there is a gate above the event, to that gate. If an event is currently selected in the fault tree when you add another event, the new event will not be connected to the original selected event.

You also can cut or copy existing blocks (in the current diagram or in any other diagram of the same type) and paste them into the diagram using the **Cut**, **Copy** and **Paste** commands on the Home tab of the Ribbon. Blocks created in this manner will possess the same style and properties as the original block.

To select an individual block, click it. There are several ways to select multiple blocks:

- Hold down the **CTRL** key and click the blocks.
- Click in the diagram and drag the selection area to include the blocks you want to select.
- Choose **Home > Edit > Select All** to select all blocks in the current diagram.

- Click a block and then choose **[Diagram/Fault Tree/Phase/Flowchart] > Selection > Cascade Select** to select all blocks that are connected to that block, either before it or after it and including the selected block itself.

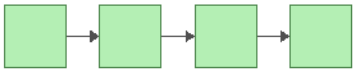
What's Changed? In previous versions of BlockSim and RENO, Templates were used to store reusable blocks that could be dragged into diagrams. In the Synthesis version, the reusability of resources makes this functionality unnecessary.

6.1.1 Arranging Blocks

Align

To align blocks in a diagram, first select a block and then hold down the **CTRL** key while clicking each of the other blocks. Choose **Format > Alignment and Spacing > Align Blocks > [Alignment Style]**.

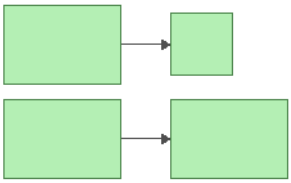
The selected blocks will be aligned with the first block selected, according to the specified alignment style. For example, if you choose **Format > Alignment and Spacing > Align Blocks > Align Middles**, the blocks will be aligned to the vertical center of the first selected block, as shown next.



Resize

To resize blocks in a diagram, select the block that has the desired dimension(s) and then hold down the **CTRL** key while clicking each of the blocks that you want to resize. Choose **Format > Alignment and Spacing > Make Same Size > [Dimension]**.

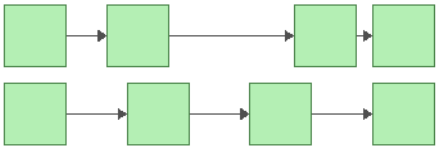
The selected blocks will be resized to match the width and/or height of the first block selected. For example, if you select choose **Format > Alignment and Spacing > Make Same Size > Height**, the blocks will be resized to the same height as the first selected block.



Adjust Spacing

To adjust the spacing of blocks in a diagram, select the blocks that will be affected. Choose **Format > Alignment and Spacing > Make Spacing Equal > [Direction]**.

The selected blocks will be moved so that they are equally spaced over the original amount of horizontal and/or vertical area.



6.1.2 Connecting Blocks

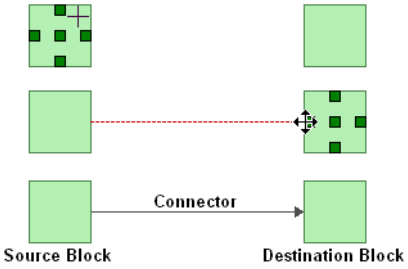
Connectors represent the flow of the process or system being diagrammed, and the lines contain information about the relationship between the specific source and destination blocks that they connect.

To create a connector between blocks, you can:

- Choose **[Diagram/Fault Tree/Phase/Flowchart] > Settings > Connect Blocks**.

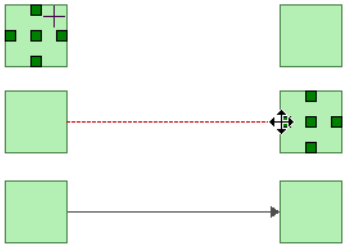
- Hold down the **ALT** key while selecting the source block and then drag a line to the destination block.

In both methods, the pointer will change to display small crosshairs. Click the source block, hold down the left mouse button and drag a line from the source block to the destination block. When the crosshairs are located above the destination block, release the mouse button to create a connector.



The location where the connector attaches to each block is determined by your selection in the **Automatic Connection Preference** field on the Connector Handles and Indicators page of the Diagram Style window (**Format > Styles > Diagram Style**). If the field is set to **Left-Right**, the connector will attach to the sides of the blocks; if it is set to **Top-Bottom**, the connector will attach to the top and bottom of the blocks. If **No Preference** is chosen, the connector will attach to the closest edges.

If you have selected **Custom** in the **Bend Style** field on the Connector page of the Diagram Style window, then when you are connecting blocks, five boxes will appear on each block that you point to. These boxes are connection points. You can click a connection point to use it as the location where the connector connects to the block. This allows you to draw connectors to and from any side of a block. The center connection point places the line using the default routing scheme, as defined in the **Automatic Connection Preference** field on the Connector Handles and Indicators page of the Diagram Style window.



Note that once you have added a connector that uses the Custom bend style, you can click the connector and drag the selected point to create a bend in the line. In RBDs and fault trees, these custom bends can be removed by clicking the connector and choosing **[Diagram/Fault Tree] > Properties > Line Bend > Remove All Bend Points** or, to remove a single bend point, by clicking the bend point and choosing [**Diagram/Fault Tree] > Properties > Line Bend > Remove Bend Point**. In all diagram types, you can right-click the connector and choose **Remove Bend Point** or **Remove all Bend Points**.

To stop adding relationships and return the pointer to its normal mode, release the **ALT** key, click inside the diagram or clear the **Connect Blocks** option by choosing the command again.

Tip: If you do not right-click the diagram or clear the **Connect Blocks** option to return the pointer to its normal mode, you will not be able to perform certain other activities, such as moving or deleting blocks. If you are experiencing difficulties with the program, make sure that the pointer is in its normal mode.

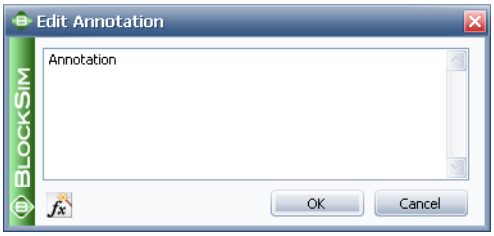
6.1.3 Annotations

Annotations are blocks that can be added to a diagram and can contain notes, comments or any other information that you want to appear with your diagram. Annotations are not taken into consideration when the diagram is analyzed but will be saved with the project.

To add an annotation to a diagram, choose **Add Annotation** on the [Diagram/Fault Tree/Phase/Flowchart] tab of the Ribbon (e.g., **Diagram > Settings > Add Annotation**).

The annotation will appear in the diagram.

By default, the text "Annotation" will appear in the block. You can change this text in the Edit Annotation window, accessed by double-clicking the annotation block or by choosing **[Diagram/Fault Tree/Phase/Flowchart] > Properties > Block Properties**.



In BlockSim diagrams, you can click the **Function Wizard** icon to insert a function that provides information about the diagram.

If you are working with an analytical diagram, you can insert:

- General Functions, which return basic information about the database and project, such as the project name or the current date.
- Data Source Functions, which return information based on the model used with the specified data source, such as the acceleration factor or the reliability at given time.

If you are working with a simulation diagram, you can insert:

- General Functions, which return basic information about the database and project, such as the project name or the current date.
- Simulation Functions, which return information about the simulation results and settings.

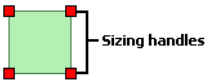
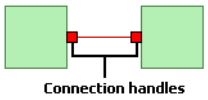
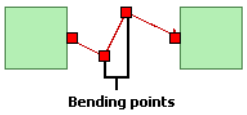
6.1.4 Keyboard and Mouse Combinations

This section describes the various keyboard and mouse key combinations that you can use while working in BlockSim and RENO.

Note the following:

- A unit is equal to either 1 millimeter or 0.1 inches, depending on the system setting.
- When selecting multiple objects, you can select either blocks or connectors. A selection cannot include both blocks and connectors. Note that while you can include Standby Containers and Load Sharing Containers in a selection, any changes you make, such as resizing the containers, do not apply to the blocks or connectors in the containers. If you start a selection inside a container, that selection can include only those blocks and connectors within that container.

If you want to do this	Do this
Selecting Objects	
Select a block or connector	Click the block or connector.
Select multiple blocks or connectors	Press **CTRL** and click the blocks or connectors.
Select a group of blocks	Click within the diagram and drag the selection to include the desired blocks.
Add additional blocks or connectors to the currently selected group	With a group of blocks or connectors selected, press **CTRL** and then make another selection. The new selection will be merged with the existing selection.
Select all blocks in the diagram	Press **CTRL+A**.
Clear all selections in the diagram	Press **ESC**.
Switch the status of a block or connector from selected to not selected, or from not selected to selected.	Press **SHIFT** and click the block or connector.
Moving Objects	
Move a block	Click the block and drag to the desired location.
Move multiple blocks	Select the blocks then click any block within the selection and drag the selection to the desired location.
Move block(s) by 1-unit increments	Select the block(s) then press the desired arrow key.
Move block(s) the distance of 1 grid line.	Select the block(s) then press **CTRL** + the desired arrow key. If only one block is selected, this shortcut will first align the block to the grid and then each subsequent use of the shortcut will move the block by one grid line.

If you want to do this	Do this
Resizing Objects	
Resize a block	 Select the block then click one of the sizing handles and drag it to the desired size.
Resize block(s) by 1-unit increments	Select the block(s) then press **SHIFT** and the desired arrow key. The up and down keys move the bottom edge of each selected block by a single unit; the right and left arrow keys move the right edge of each selected block by a single unit.
Resize a block(s) by 5-unit increments	Select the block(s) then press **CTRL** + **SHIFT** + the desired arrow key. The up and down arrow keys move the bottom edge of each selected block by 5 units; the right and left arrow keys move the right edge of each selected block by 5 units.
Connectors	
Add a new connector	Press **ALT** and drag the connector from the source block to the destination block.
Change a connector's source or destination	 Select the connector then drag the appropriate connection handle to the new source or destination block.
Add a bending point to a connector. Note that the connector must use the Custom style. *To set the style, see Section 6.3.1.5 on page 200.*	 Click the connector and drag the bending point to the desired location.
Scrolling	
Scroll vertically	Rotate the mouse wheel.
Scroll horizontally	Press **SHIFT** and rotate the mouse wheel.
Scroll in any direction	With nothing selected, press the desired arrow key.
Scroll horizontally to the far left	Press **HOME**.
Scroll horizontally to the far right	Press **END**.
Scroll to the upper left corner of the diagram.	Press **CTRL+ HOME** or **CTRL+ PAGE UP**
Scroll to the lower right corner of the diagram	Press **CTRL+END** or **CTRL+PAGE DOWN**.

If you want to do this	Do this
Scroll by the size of the visible area of the diagram	With nothing selected, press **CTRL** + the desired arrow key. For vertical scrolling, you can also press **PAGE UP** and **PAGE DOWN**.
Move the diagram in any direction	Press **SHIFT**, click the diagram and drag. This is like dragging the diagram sheet itself (e.g. to see objects lower in the diagram, you would drag the diagram upward).
Other	
Zoom in or out	Press **CTRL** and rotate the mouse wheel.
Exit the Format Painter or Connect Blocks modes	Press **ESC**.
Create a copy of a block and move the copy to the desired location.	Press **CTRL+ SHIFT** and click the block then drag to the desired location.
Add blocks to the mirror group currently shown in the Mirror Group Manager window.	Press **CTRL + SHIFT** and drag the block from the diagram into the Mirrored Blocks area of the Mirror Group Manager.

6.2 Block Properties

The Block Properties window allows you to configure the functional properties of any block selected in a diagram. Double-click the block or choose **[Diagram/Fault Tree] > Properties > Block Properties** or [**Phase/Flowchart] > Settings > Block Properties**, depending on the type of diagram you are working with, to open the Block Properties window.

You can also define the properties for multiple blocks simultaneously by selecting the blocks and then opening the Block Properties window. If different types of blocks are selected (e.g., standard and node), only the properties that apply to all selected blocks will be available in the Block Properties window. You cannot open the Block Properties window if there is an ambiguous selection (e.g., if you selected both standard blocks and containers).

Note: Double-clicking a subdiagram or subchart block will open the subdiagram that the block is based on. Open the Block Properties window for a subdiagram or subchart block by selecting the block and choosing the command on the Ribbon or pressing **CTRL+E**.

Regardless of the type of block you are working with, the Block Properties window always has the following elements:

- **Name and Description**: Each block is given a default name that is determined based on the block type's default name format. You can type new text either instead of or in addition to the default name. The **Name** field must be populated. Description text is optional.

  Note that in BlockSim diagrams an asterisk is used (*) to represent default block names. The block name that is displayed in the diagram will replace the asterisk with the block's default name; this allows the block name to be updated dynamically. For example, a node's default name indicates its *k-out-of-n* value. If you change the value in the node's **Number of Paths Required** field, the node's name will automatically update when you close the Block Properties window.

Display properties for the block name and description may be set in the Block Style window for the individual block or in the Diagram Style window for all blocks added to the diagram.

For standard blocks (including contained blocks) in RBDs and for events in fault trees, this area contains an **Optimum Replacement** icon. Click this to open the Optimum Replacement window for the current block; this window allows you to determine the most cost-effective time to replace the component based on costs for planned (i.e., preventive) and unplanned (i.e., corrective) replacement.

- **Identifiers**: These fields allow you to enter additional information about the block, such as a part number, a revision number, keywords that apply to the block, etc.
- **Active Block**: This drop-down list at the bottom of the Block Properties window allows you to specify another block in the diagram that you want to set the block properties for. This allows you to set the properties for multiple blocks without having to open and close the Block Properties window for each block. If you have made changes to the current block's properties, when you change the active block, a window may appear asking if you want to apply the changes you made to the active block. This window will appear only if the **Ask to apply changes when selecting different block in Properties window** option is selected on the Other page of the Application Setup (**File > Application Setup**).
  - Click **Yes** to apply the changes to the current active block and then change the active block.
  - Click **No** to discard the changes to the current active block and then change the active block.
  - Click **Cancel** to leave the current block active. Your changes will be neither applied nor discarded at this time.
- **Style** opens the Block Style window, which allows you to customize the appearance of the active block.
- Attachments allows you to attach related files to the block. Clicking this icon opens the Attachments window.

The other properties available will vary depending on the type of block you are working with. Consequently, these properties are documented in the chapters relating to the block types.

For RBDs, the block types include:

- Standard blocks represent some functional portion of the system, such as a component, subsystem, assembly or failure mode.
- Node blocks allow you to construct *k-out-of-n* configurations. In these configurations, a specified number of paths must reach the node without failing in order for the system to be successful.
- Subdiagram blocks represent existing diagrams. For example, you might have an RBD that models an assembly. You can use a subdiagram block to include that assembly in an RBD that represents a larger system.
- Standby containers hold blocks that operate in a standby configuration, where one or more of the blocks in the container are active while one or more blocks are dormant (i.e., standby or quiescent). The dormant blocks are available to become active under specified circumstances. The blocks inside the container describe the characteristics of the items that operate together in the standby configuration.
- Load sharing containers hold blocks that operate in a load sharing configuration. This indicates that two or more blocks in the container share the work. The blocks inside the container describe the characteristics of the items that operate together in the load sharing configuration. For example, you might have two blocks in a load sharing container in a configuration where the container must provide an output of 8 volts. If both blocks are operational and each is set to provide 50% of the load, then each block will provide an output of 4 volts. If one of the blocks fails, however, then the other block will have to provide the entire 8 volts of output.

For fault trees, the block types include:

- AND gates: The output event occurs if all input events occur.
- OR gates: The output event occurs if at least one of the input events occurs.
- Voting OR gates: The output event occurs if some quantity (*k*) or more of the (*n*) input events occur.
- NOT gates: The output event occurs if the input event does not occur, and vice versa. This gate performs inversion.
- NAND gates: The output event occurs if any one of the input events does not occur.
- NOR gates: The output event occurs only if none of the input events occur.
- Inhibit gates: The output event occurs if all input events occur and an additional conditional event occurs.
- Standby gates identify blocks that operate in a standby configuration. Events that are below a standby gate are dependent events, which describe the characteristics of the items that operate together in the standby configuration.
- Load sharing gates identify blocks that operate in a load sharing configuration. Events that are below a load sharing gate are dependent events, which describe the characteristics of the items that operate together in the load sharing configuration.
- Priority AND gates: The output event occurs if all input events occur in a specific sequence.
- Sequence enforcing gates: Events are constrained to occur in a specific sequence and the output event occurs if all input events occur in that specified sequence.
- Events are simply representations of some occurrence, typically a failure or error.
- Subdiagram blocks represent existing diagrams.

For RENO flowcharts, the block types include:

- Flowchart standard blocks evaluate a mathematical expression, and then pass the result (output value) of the expression to the next block(s) in the flowchart.
- Result storage blocks store numerical values passed to them during simulations, and then compute or hold a result.
- Conditional blocks function as “if” statements. They check the incoming value against a conditional expression, with possible outcomes of true and false.
- Binary nodes take an incoming value and multiply it by a specified value. The resulting value is passed down the “true” path. The value passed down the “false” path is the incoming value multiplied by (1 minus the specified value). Binary nodes are primarily intended to facilitate building decision trees.
- Logic gates check multiple incoming values against a conditional expression, with possible outcomes of true and false. The overall block outcome is determined based on the gate type, which specifies how many of the incoming values must meet the condition in order for the outcome to be true.
- Branch gates function as “switch” statements. They check the incoming value against a number of cases and pass different outputs depending on which case or *branch* evaluates to true.
- Summing gates perform a simple mathematical operation (addition, subtraction, multiplication or division) on all incoming values and return a single value to all outgoing paths.
- Flag markers and go-to-flag blocks are used to build subroutines and loops that repeat steps in a flowchart until a certain condition is reached. The flag marker marks a spot in the path of the flowchart, and the go-to-flag transfers the flow of execution to that spot.

- Stop flags halt the simulation process when they are encountered. No further simulations, runs or analyses will be performed until you next simulate the flowchart.
- Counter blocks record the number of times the simulation has passed through them, and then pass the recorded value to the next block in the flowchart.
- Reset blocks force RENO to generate new values for all static functions in the flowchart while simulation is in progress.
- UI blocks pause flowchart simulation until the user provides some form of input.
- Subchart blocks represent other flowcharts within the project. Subchart blocks are generally used to break down a larger flowchart into simpler steps to reduce complexity and improve traceability.

6.3 Diagram Settings

BlockSim/RENO makes it easy to modify the appearance of your diagrams. You can specify settings for the diagram itself and for the items within the diagram both as a group and individually. The following sections describe how to:

- Use the Diagram Style window to customize the properties of the diagram and to specify default settings for the blocks and connectors within the diagram - Section 6.3.1 (p. 197)
- Use diagram skins to save your settings in the Diagram Style window and reuse those settings quickly and easily - Section 6.3.2 (p. 203)
- Use the Block Style window to customize the properties for selected blocks (either singly or multiply selected) - Section 6.3.3 (p. 204)
- Use the Connector Style window to customize the properties for selected connectors - Section 6.3.4 (p. 205)

6.3.1 Diagram Style Window

The Diagram Style window allows you to customize the appearance of the current diagram and all components within it and to set the default properties for new items added to the diagram. These default settings are used for all new blocks added to the diagram; in addition, if you change the block and/or connector settings in the Diagram Style window, the changes will be applied to all blocks and/or connectors that are using the diagram defaults (i.e., all blocks and/or connectors that do not have customized settings).

For selected blocks (either singly or multiply selected), you can use the Block Style window to customize one or more properties. For selected connectors, you can use the Connector Style window to customize one or more properties.

To access the Diagram Style window, choose **Format > Styles > Diagram Style**.

The Diagram Style window consists of multiple pages that are accessible by clicking the page names in the navigation panel on the left side of the window; the number and type of pages depend on the type of diagram that you are working with.

- The Diagram pages allow you to customize the appearance of:
  - The diagram sheet (i.e. the background upon which blocks and other elements are placed).
  - Aspects of the appearance of diagram components (i.e., blocks and connectors) that cannot be configured at the individual component level.
- The Defaults pages allow you to customize the default appearance settings for diagram components (i.e., connectors, blocks and annotations). These settings are used for all new components added to the diagram and for all existing components that are set to use default settings in the Block Style window or the Connector Style

window. There is a single page of default settings for connectors (although, in the case of phase diagrams, there are three types of connectors — success, failure and other, which is used for connectors from nodes to phases — each of which has a page of settings). For each block type, there is a General page, a Description page and an Image page. For annotations, there is a General page and an Image page.

You can save your settings in the Diagram Style window as a skin so that you can reuse them quickly and easily.

6.3.1.1 Background & Grid Page

The Background and Grid page of the Diagram Style window allows you to customize the appearance of the diagram. You can specify:

- Whether to display grid lines and, if so, the style of the lines (e.g. solid, dash, etc.).
- The size of the grid squares, in inches.
- The size and color of the shadow "cast" by blocks in the diagram. Note that this applies to all block types, but the shadow will be shown only if the **Show Shadow** check box is selected on the Defaults > General page for that block type.
- The colors used for:
  - The background.
  - The grid lines.
  - The page separator lines, which appear in the Print Layout view and indicate the boundaries of each printed page. You can display this view by choosing **Home > Print > Print Layout**.

- The horizontal and vertical spacing, in inches, that will be used if you choose **[Diagram/Fault Tree/Phase/Flowchart] > Blocks > Auto Arrange**.

  For fault trees, you can also select the **Always Auto Arrange** check box to turn on Auto Arrange by default.
- Whether to display the block ID, which is an internal ID number assigned to each block when it is created. You can choose not to display the ID or to display it in decimal or hexadecimal format. If it is displayed, it will appear directly below the block name (i.e., its location, alignment and font will be dictated by the properties that are specified for the block caption).

6.3.1.2 Connector Handles and Indicators Page

The Connector Handles and Indicators page of the Diagram Style window allows you to customize the following properties of the connectors in the diagram:

- If you select **Show Connectors Over Blocks**, any connectors that cross blocks will be shown over (i.e., in front of) those blocks. Otherwise, the blocks will be displayed on top.
- Where connectors attach to blocks if the blocks are not in direct horizontal or vertical alignment. If the **Automatic Connection Preference** field is set to **Left-Right**, the connector will attach to the sides of the blocks; if it is set to **Top-Bottom**, the connector will attach to the top and bottom of the blocks. If **No Preference** is chosen, the connector will attach to the closest edges.
- The colors used for:
  - All connector handles.
  - Selected connectors.

- Connectors to/from highlighted blocks (i.e., blocks that are part of a multiple selection).
- Connectors leading to or from a selected block. This can be helpful in visualizing how a block is connected within a complex diagram.

- The size of the arrow head, in inches.

6.3.1.3 Block Handles and General Indicators Page

The Block Handles and General Indicators page of the Diagram Style window allows you to customize the following properties of the blocks in the diagram:

- The colors used for the handles that appear when you are:
- Resizing a block.
- Connecting blocks.
- The size of block flags. Block flags, which are added by selecting one or more blocks and choosing **[Diagram/Fault Tree/Phase/Flowchart] > Settings > Set Block Flag**, can be used to indicate anything you want.

For example, each flag could represent a level of analysis that has been completed. These flags are not used by BlockSim for analyses or simulations.

- The style (e.g., solid, dash, etc.), color and thickness of the border for block flags.
- The properties for a highlighted block, including:
  - The colors used for the caption, border and fill (i.e., the area background).
  - The fill style (i.e., how the color is applied to the area).
- The style, color and thickness of the line that is used to cross out blocks that are set as failed. In BlockSim diagrams, blocks can be set as failed by selecting the **Set block** as failed option in the Block Properties window. You can also set this option by selecting the block in the diagram and choosing **[Diagram/Fault Tree] > Settings > Set Block as Failed**.

6.3.1.4 Block Corner Indicator Pages

The Block Corner Indicators pages of the Diagram Style window allow you to customize the appearance of status indicators that are displayed at the corners of blocks. Such indicators include the colored boxes that are used to indicate certain block and text labels that may appear independently or in conjunction with a box. The available Block Corner Indicators pages will vary depending on the type of diagram that you are working with.

- In RBDs, you can modify the following indicators:
  - Block belongs to a maintenance group: Indicated by a marker at the upper left corner of the block. Clicking the indicator opens the Maintenance Group Manager.
  - Mirrored block: Indicated by a marker at the lower left corner of the block. A label indicating the name of the mirror group that the block belongs to is also displayed at the lower left corner. Clicking the indicator opens the Mirror Group Manager.
  - Block with state change triggers (SCT) enabled: Indicated by a marker at the lower right corner of the block.
- In fault trees, you can modify the following indicators:
  - Block belongs to a maintenance group: Indicated by a marker at the upper left corner of the block. Clicking the indicator opens the Maintenance Group Manager.

  - Mirrored block: Indicated by a marker at the lower left corner of the block. A label indicating the name of the mirror group that the block belongs to is also displayed at the lower left corner. Clicking the indicator opens the Mirror Group Manager.
  - Gate/Event result: Reliability or unreliability results at the block level can be displayed at the upper right of each block in an analytical fault tree by selecting the **Show Results** option in the control panel.
  - Block with state change triggers (SCT) enabled: Indicated by a marker at the lower right corner of the block.
- In phase diagrams, you can modify the following indicator:
  - Maintenance phase: Indicated by a marker at the upper left corner of the block.
- In flowcharts, you can modify the following indicator:
  - Conditional formatting: Indicated by a marker at the upper left corner of the block.

For each type of indicator marker, the following properties are customizable:

- The marker size, in inches. This property is managed via the Diagram Style window for all indicator marker types. Note that for conditional formatting, this is the only property that can be customized.
- The background color, fill color and fill style. The background color is the underlying color applied to the marker; the fill color is applied over the background color in the pattern specified by the fill style. These properties are managed via the Diagram Style window for SCT markers and for maintenance phase markers. For maintenance group markers, the settings are specific to the maintenance group and therefore are managed by clicking the **Indicator Style** icon in the Maintenance Group Manager; similarly, for mirror group markers, they are managed via the Mirror Group Manager.
- The style (e.g., solid, dash, etc.), color and thickness of the marker border. These properties are managed via the Diagram Style window for SCT markers and for maintenance phase markers. For maintenance group markers, the settings are specific to the maintenance group and therefore are managed by clicking the **Indicator Style** icon in the Maintenance Group Manager; similarly, for mirror group markers, they are managed via the Mirror Group Manager.

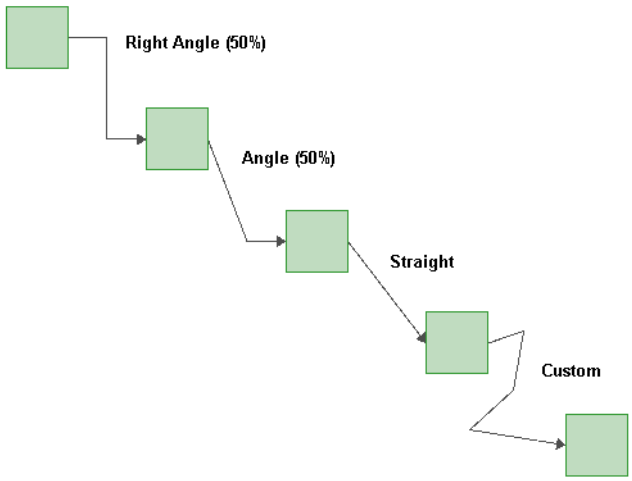
For the mirror group label and the gate result, you can customize the color and font used for the text.

6.3.1.5 Connector Page

The Connector pages of the Diagram Style window allows you to specify the following default settings for the connectors in the diagram:

- Line style (e.g., solid, dash, etc.), thickness and color.

- The kind of bend in the lines. If you select **Angle** or **Right Angle**, the **Line Bend %** field will be enabled. This field allows you to specify the point, in percent, at which the line will bend. For example, if you enter **50%**, an angle (bend) will appear in the line at the horizontal halfway point. The available styles are shown next.



  If you select **Custom**, you can click the connector and drag the selected point to create a bend in the line. In RBDs and fault trees, these custom bends can be removed by clicking the connector and choosing **[Diagram/Fault Tree] > Properties > Line Bend > Remove All Bend Points** or, to remove a single bend point, by clicking the bend point and choosing **[Diagram/Fault Tree] > Properties > Line Bend > Remove Bend Point**. In all diagram types, you can right-click the connector and choose **Remove Bend Point or Remove all Bend Points**.

- The location, text color and font used for captions on connectors. Captions are entered for each connector in the Connector Style window, which is accessed by selecting the connector and choosing **Format > Styles > Connector Style** or by double-clicking the connector. Possible locations include:
  - Auto: The software will place the caption where it fits best.
  - Middle: The caption will be as close to the middle of the connector as possible, depending on bends in the line.
  - Source: The caption will be displayed where the connector meets the source block.
  - Destination: The caption will be displayed where the connector meets the destination block.
  - Hidden: The caption will not be displayed.
- Whether or not the arrow head is displayed.

6.3.1.6 General Page

The General pages in the Diagram Style window allow you to customize the following properties for the block type:

- The shape, background color, fill color and fill style used for the block type. The background color is the underlying color applied to the block; the fill color is applied over the background color in the pattern specified by the fill style. The options available in the **Shape** drop-down list depend on the block type. You can change the shape of standard blocks in RBDs and of events in fault trees; all other block types have permanently assigned shapes.
- The style (e.g., solid, dash, etc.), color and thickness of the block border.
- How the block caption appears, including:
  - **Location** and **Horizontal Alignment** allow you to specify where the caption will be displayed (e.g., under the block, inside the block, etc.) and whether it will be left aligned, center aligned or right aligned. If you choose to place the caption to the left, right or inside the block, you can also specify the **Vertical Alignment** (i.e., at

the top, center or bottom of the block). If you choose to hide the caption, the Alignment fields will not be available.

- The width and height of the area used for the text. You can enter a fixed value for either or both of these. If **Block Width** is selected in the **Width** field, the caption text will wrap, if necessary, up to the vertical limit set in the **Height** field. If **Auto Height** is selected in the **Height** field, no limit is placed on the height of the text area.
- The color and font used for the text.

- Whether or not a shadow is shown for the block. If a shadow is shown, the size and color of the shadow are determined by the settings in the **Shadow** area on the Diagram > Background and Grid page.
- The size of new blocks. Unlike the other options on this page, changes to this setting will apply only to blocks subsequently added to the diagram.

6.3.1.7 Description Page

The Description pages in the Diagram Style window allow you to customize the appearance of the description area for the block. This area displays the text entered in the **Description** field in the Block Properties window. You can customize the following properties:

- The background color, fill color and fill style (i.e., how the color is applied to the area). The background color is the underlying color applied to the block; the fill color is applied over the background color in the pattern specified by the fill style.
- The style (e.g., solid, dash, etc.), color and thickness of the description area border.
- How the description area appears, including:
  - **Location** and **Horizontal Alignment** allow you to specify where the description area will be displayed (i.e., above the block, inside the block or hidden) and whether it will be left aligned, center aligned or right aligned. If you choose to place the description inside the block, you can also specify the **Vertical Alignment** (i.e., at the top, center or bottom of the block). If you choose to hide the description area, the Alignment fields will not be available. If you place the description inside the block, only the description text will be shown. The settings for the surrounding area (i.e., the size, background color, fill color, fill style and border properties) will not be used. The properties for the background color, fill color and fill style of the block will be used instead.
  - The color and font used for the text.
- The width and height of the area used for the description. You can enter a fixed value for either or both of these, or you can set them to be the same as the settings used for the caption area. If **Block Width** is selected in the **Width** field, the caption text will wrap, if necessary, up to the vertical limit set in the **Height** field. If **Auto Height** is selected in the Height field, no limit is placed on the height of the text area.

6.3.1.8 Image Page

The Image pages in the Diagram Style window allow you to specify the image that is displayed on the block, if any.

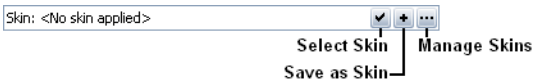
Click the **Load** button to browse for the desired image file. Use the **Position** drop-down list to specify how the image will be placed on the block (e.g., stretched to fit inside the block's current dimensions, aligned from the top left corner, etc.). Selecting the **Transparency Mask Color** check box allows you to choose a color that will be shown as transparent when the image is displayed on the block.

> **Note:** If you choose to use an image that is larger than 800x600 pixels, you will be given the option to downsample the image in order to reduce its size and alleviate possible delays related to database size.

Click the **Clear** button to remove the loaded image and use no image at all.

6.3.2 Diagram Skins

You can save your settings in the Diagram Style window as a skin so that you can reuse them quickly and easily.

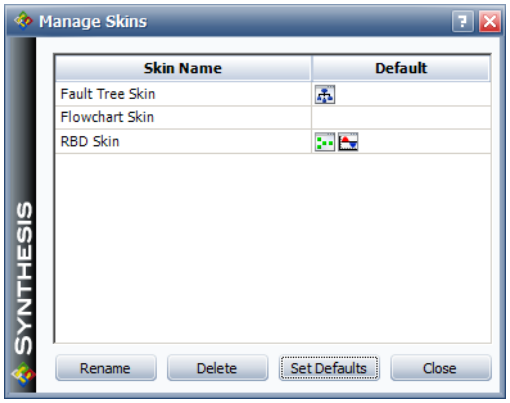


To save a skin, once you have specified the desired settings on all pages of the window, click the **Save as Skin** icon in the **Skin** field at the bottom of the window. Enter a name for the skin and click **OK**.

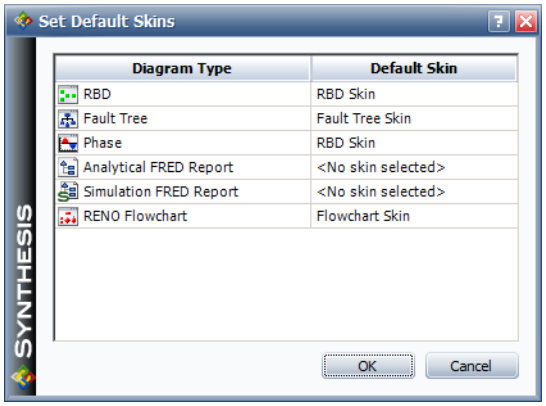
Note: Skins are stored on your computer at C:\Users\Public\Documents\ReliaSoft\Skins.

To apply a skin to a diagram, click the **Select Skin** icon, choose the skin from the list in the Select Skin window and click **OK**. The name of the selected skin will be displayed in the lower left corner of the Diagram Style window. When you click **OK** to close the Diagram Style window, the skin will be applied to the diagram.

To view all of the skins (for all types of diagrams) that are available on this computer, click the **Manage Skins** icon.



In this window, you can rename or delete existing skins. You can also choose which skin will be used by default for each new diagram you create. To set the defaults, click the **Set Defaults** button.



For each diagram or report type, click inside the Default Skin column and choose a skin from the drop-down list. When you set a skin as the default for an item type, an icon representing that item type will appear in the Default column for the skin in the Skin Manager.

The Skin Manager can also be accessed from the Display page of the Application Setup window (**File > Application Setup**).

Note: The skins managed here are diagram skins, which control the default appearance of new diagrams. The software also offers skins that control the overall appearance of the interface. These skins can be managed via the Synthesis Setup (**File > Synthesis Setup**).

6.3.3 Block Style Window

You can customize the appearance of a block or annotation using the Block Style window. Changes made in this window will apply only to the current block; the Diagram Style window allows you to make changes to all new blocks added to the diagram as well as to all current blocks in the diagram that are using default settings.

To open the Block Style window, select a block or annotation and choose **Format > Styles > Block Style**.

You can also activate the Block Style window from the Block Properties window by clicking the **Style** button at the bottom of the window.

6.3.3.1 Block Tab

The Block tab of the Block Style window allows you to specify the appearance of the block itself and of the caption associated with the block. You can customize the following:

- The shape and size of the block, as well as whether or not a shadow is shown for the block. The options available in the **Shape** drop-down list depend on the block type. You can change the shape of standard blocks in RBDs and of events in fault trees; all other block types have permanently assigned shapes. If **Diagram Default** is selected for the shape and/or whether the shadow is shown, the setting selected in the Diagram Style window will be used.
- The background color, fill color and fill style used for the block as well as the style (e.g., solid, dash, etc.), color and thickness of the block border. (The background color is the underlying color applied to the block; the fill color is applied over the background color in the pattern specified by the fill style.) To modify these properties, you must clear the **Use Default Interior and Border Settings** check box. If this check box is selected, these properties use the settings defined in the Diagram Style window on the Defaults > General page for the current block type.
- How the block caption appears, including:
  - **Location** and **Horizontal Alignment** allow you to specify where the caption will be displayed (e.g., under the block, inside the block, etc.) and whether it will be left aligned, center aligned or right aligned. If you choose to place the caption to the left, right or inside the block, you can also specify the **Vertical Alignment** (i.e., at the top, center or bottom of the block). If you choose to hide the caption, the Alignment fields will not be available. If **Diagram Default** is selected, the setting selected in the Diagram Style window will be used.
  - The width and height of the area used for the text. You can enter a fixed value for either or both of these. If **Block Width** is selected in the **Width** field, the caption text will wrap, if necessary, up to the vertical limit set in the **Height** field. If **Auto Height** is selected in the **Height** field, no limit is placed on the height of the text area. To use the default settings for these properties, as specified in the Diagram Style window on the Defaults > General page for the current block type, choose the **Diagram Default** option.
  - The color and font used for the text. To modify these properties, you must clear the **Use Default Color and Font** check box. If this check box is selected, these properties use the settings defined in the Diagram Style window on the Defaults > General page for the current block type.

6.3.3.2 Description Tab

The Description tab of the Block Style window allows you to specify the appearance of the description area for the block. This area displays the text entered in the Description field on the Name & Description page of the Block Properties window. You can customize the following properties:

- The width and height of the area used for the description. You can enter a fixed value for either or both of these, or you can set them to be the same as the settings used for the caption area. If **Block Width** is selected in the **Width** field, the caption text will wrap, if necessary, up to the vertical limit set in the **Height** field. If **Auto Height** is selected in the **Height** field, no limit is placed on the height of the text area. If **Diagram Default** is selected, the setting selected in the Diagram Style window will be used.
- The background color, fill color and fill style used for the description area, as well as the style (e.g., solid, dash, etc.), color and thickness of the description area border. (The background color is the underlying color applied to the block; the fill color is applied over the background color in the pattern specified by the fill style.) To modify these properties, you must clear the Use **Default Description Settings** check box. If this check box is selected, these properties use the settings defined in the Diagram Style window on the Defaults > Description page for the current block type.
- How the block description appears, including:
  - **Location** and **Horizontal Alignment** allow you to specify where the description area will be displayed (i.e., above the block, inside the block or hidden) and whether it will be left aligned, center aligned or right aligned. If you choose to place the caption inside the block, you can also specify the **Vertical Alignment** (i.e., at the top, center or bottom of the block). If you choose to hide the description area, the Alignment fields will not be available. If **Diagram Default** is selected, the setting selected in the Diagram Style window will be used.

    If you place the description inside the block, only the description text will be shown. The settings for the surrounding area (i.e., the size, background color, fill color, fill style and border properties) will not be used. The properties for the background color, fill color and fill style of the block will be used instead.
  - The color and font used for the text. To modify these properties, you must clear the **Use Default Color and Font** check box. If this check box is selected, these properties use the settings defined in the Diagram Style window on the Defaults > Description page for the current block type.

6.3.3.3 Image Tab

The Image tab of the Block Style window allows you to specify the image that is displayed on the block, if any.

To change the image used, clear the **Use Default Image** check box. You can then click the **Load** button to browse for the desired image file. Use the **Position** drop-down list to specify how the image will be placed on the block. Selecting the **Transparency Mask Color** check box allows you to choose a color that will be shown as transparent when the image is displayed on the block.

Note: If you choose to use an image that is larger than 800x600 pixels, you will be given the option to downsample the image in order to reduce its size and alleviate possible delays related to database size.

Click the **Clear** button to remove the loaded image and use no image at all.

6.3.4 Connector Style Window

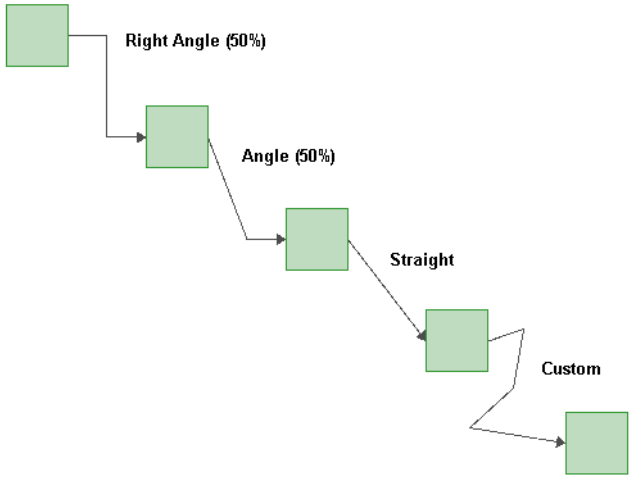
You can customize the appearance of a connector using the Connector Style window. Changes made in this window will apply only to the current connector; the Diagram Style window allows you to make changes to all new connectors added to the diagram as well as to all current connectors in the diagram that are using default settings.

To open the Connector Style window, select a connector and choose **Format > Styles > Connector Style** or double-click the connector.

For all properties on this page, the default settings, if any, are specified in the Diagram Style window on the Defaults > Connector page. You can:

- Enter a caption for the connector and specify the location, text color and font used for the caption. Possible locations include:
  - Auto: The software will place the caption where it fits best.
  - Middle: The caption will be as close to the middle of the connector as possible, depending on bends in the line.
  - Source: The caption will be displayed where the connector meets the source block.
  - Destination: The caption will be displayed where the connector meets the destination block.
  - Hidden: The caption will not be displayed.

  To use the default location, choose the **Diagram Default** option. To modify the text color and font, you must clear the **Use Default Color and Font** check box. If this check box is selected, these properties use the default settings.
- Specify the connector's line style (e.g., solid, dash, etc.), thickness and color. To modify these properties, you must clear the **Use Default Line Settings** check box. If this check box is selected, these properties use the default settings.
- Specify the kind of bend in the lines. If you select **Angle** or **Right Angle**, the **Line Bend %** field will be enabled. This field allows you to specify the point, in percent, at which the line will bend. For example, if you enter **50%**, an angle (bend) will appear in the line at the horizontal halfway point. The available styles are shown next.



  If you select **Custom**, you can click the connector and drag the selected point to create a bend in the line. These custom bends can be removed by clicking the connector and choosing **[Diagram/Fault Tree] > Properties > Line Bend > Remove All Bend Points** or, to remove a single bend point, by clicking the bend point and choosing **[Diagram/Fault Tree] > Properties > Line Bend > Remove Bend Point**.

  To use the default setting, choose the **Diagram Default** option.
- Specify whether or not the arrow head is displayed. To use the default setting, choose the **Diagram Default** option.

Building RBDs and Fault Trees 7

Analyses in BlockSim are based on two basic diagram types: reliability block diagrams (RBDs) and fault trees. These are both symbolic logic diagrams that can be applied to analyze system reliability and related characteristics. Either type can be configured as an analytical diagram (which uses the exact algebraic equation for the system model but imposes limitations on what can be modeled and what results can be obtained) or a simulation diagram (which provides more modeling options and results but must be analyzed with discrete event simulation).

- Configuring and using blocks in RBDs, including:
  - A visual guide to the blocks available for use in RBDs- Section 7.1.1 (p. 208)
  - Diagram configurations and constraints - Section 7.1.2 (p. 210)
  - Configuring and using blocks in RBDs, including:
    - Standard blocks - Section 7.1.3 (p. 211)
    - State change triggers - Section 7.1.4 (p. 216)
    - Node blocks - Section 7.1.5 (p. 217)
    - Subdiagrams - Section 7.1.6 (p. 218)
    - Standby containers - Section 7.1.7 (p. 220)
    - Load sharing containers - Section 7.1.8 (p. 226)
- Configuring and using blocks in fault trees, including:
  - The relationship between RBDs and fault trees - Section 7.2 (p. 228)
  - A visual guide to the blocks available for use in fault trees - Section 7.2.1 (p. 230)
  - Diagram analysis constraints - Section 7.2.2 (p. 232)
  - Configuring and using blocks in fault trees, including:
    - AND gates - Section 7.2.3.1 (p. 232)
    - OR gates - Section 7.2.3.2 (p. 233)
    - NOT gates - Section 7.2.3.3 (p. 233)
    - NAND gates - Section 7.2.3.4 (p. 234)
    - NOR gates - Section 7.2.3.5 (p. 235)
    - Voting gates - Section 7.2.3.6 (p. 235)
    - Inhibit gates - Section 7.2.3.7 (p. 236)
    - Standby gates - Section 7.2.3.8 (p. 237)
    - Load sharing gates - Section 7.2.3.9 (p. 238)
    - Priority AND gates - Section 7.2.3.10 (p. 239)

  - Sequence enforcing gates - Section 7.2.3.11 (p. 240)
  - Fault tree events and dependent events- Section 7.2.4 (p. 242)
- Using blocks in more than one location through mirroring - Section 7.3 (p. 243)
- Viewing the details of the blocks and resources in the current diagram in the Block Details window - Section 7.4 (p. 246)
- Using the Item Properties Table to view and/or edit the properties of the blocks in the diagrams and the resources within the current project - Section 7.5 (p. 246)
- Performing calculations via the Quick Calculation Pad (QCP) - Section 7.6 (p. 249)
- Publishing fitted and analytical models, if desired - Section 7.7 (p. 251)
- Building diagrams based on information from other Synthesis applications - Section 7.8 (p. 252)

7.1 Reliability Block Diagrams

A reliability block diagram (RBD) is a graphical representation of how the components of a system are reliability-wise connected. Building RBDs in BlockSim involves adding blocks to a diagram and then arranging and connecting the blocks so that they represent the reliability-wise configuration of a system or process. Each block represents a component of the overall system or process that is represented by the RBD. You can define individual blocks with the reliability and other characteristics of the components they represent, and also customize the appearance of each block to enhance the presentation of your diagrams.

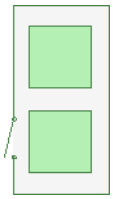
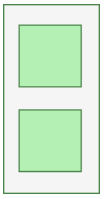
Note: An RBD is used to analyze the reliability of a system with a fixed configuration. In some cases, however, a single, fixed configuration does not accurately represent the system's performance over the course of a mission. Aspects of the system may change over time, including the system's reliability configuration; the resources available to the system; or the failure, maintenance and/or throughput properties of its individual components. A phase diagram can be used to represent/analyze such a system *(see Chapter 10)*.

7.1.1 RBD Block Types

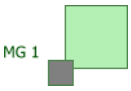
In order to provide the flexibility required to create RBDs that model actual systems as realistically as possible, there are several different types of blocks that you can use in your diagrams. For the different types of blocks, the appearance and options available in the Block Properties window will vary. This section is intended to help you identify the basic types of blocks found in RBDs.

Note: The shapes, colors and locations shown here represent the default settings. These settings may be changed via the Block Style window and/or the Diagram Style window (**Format > Styles > [Block Style/Diagram Style]**).

Visual Indicator	Block Type	Description
	Standard Block	Standard blocks represent some functional portion of the system, such as a component, subsystem, assembly or failure mode.
	Node Block	A block that is circular in shape is a node block. Node blocks allow you to construct *k-out-of-n* configurations. In these configurations, a specified number of paths must reach the node without failing in order for the system to be successful.

Visual Indicator	Block Type	Description
	Subdiagram Block	A block that is in the shape of a folder indicates that the block represents an existing diagram.
	Standby Container/ Contained Standby Blocks	A block that is gray and has an opening in its border is a standby container, which allows you to identify blocks that operate in a standby configuration. The blocks inside the container describe the characteristics of the items that operate together in the standby configuration.
	Load Sharing Container/ Contained Load Sharing Block	A block that is gray and has a solid border is a load sharing container, which allows you to identify blocks that operate in a load sharing configuration. The blocks inside the container describe the characteristics of the items that operate together in the load sharing configuration.

For standard blocks, there are several status indicators that may appear on the block. Although each status is presented individually for demonstration purposes, a single block can be defined with more than one type of status.

Visual Indicator	Block Type	Description
	Inactive Block	An X through the block indicates that the block is inactive for reliability calculations (i.e., set to "off"). This is done by selecting the **Set block as failed** option in the Block Properties window or by selecting the block in the diagram and choosing **[Diagram/Fault Tree] > Settings > Set Block as Failed**. This option can be used for "what-if" analyses to investigate the impact of a block on system metrics such as reliability, availability, throughput, etc.
	Block Belongs to a Maintenance Group (*For more about maintenance groups, see page 279 in Chapter 9.*)	A red circle in the upper left corner indicates that the block belongs to a maintenance group. A maintenance group is a set of blocks where some event within the group can trigger either maintenance or state changes for one or more blocks, either within the group or outside of it. Clicking the indicator opens the Maintenance Group Manager.
	Mirrored Block	A gray box in the lower left corner indicates that the block is a mirrored block. The mirror group to which it belongs is displayed beside the box. All blocks in the mirror group represent the same component in more than one location. Clicking the indicator opens the Mirror Group Manager.
	Block Has State Change Triggers Enabled	A blue diamond in the lower right corner indicates that the block has state change triggers enabled. State change triggers allow you to specify events that will activate and/or deactivate the block during simulation.

Visual Indicator	Block Type	Description
	Multi Block	A block that appears to be a stack of blocks indicates that the block represents more than one block with the same properties. These blocks can be configured in series or parallel. The settings are specified by selecting the **Represents multiple blocks** option in the Block Properties window.

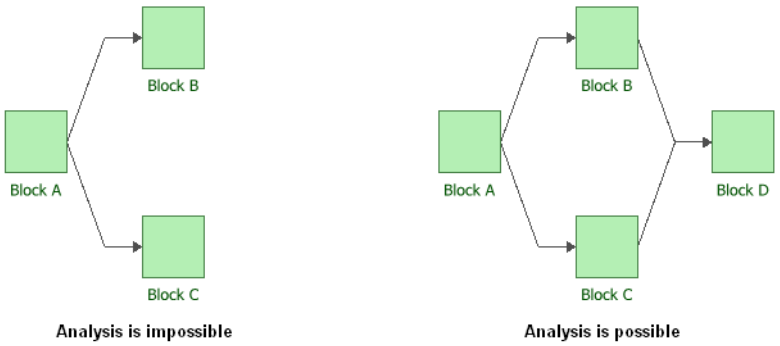
7.1.2 RBD Configurations and Constraints

BlockSim allows you to create an extensive array of RBD configurations. These can be as simple as units arranged in a pure series or parallel configuration. There can also be systems of combined series/parallel configurations or "complex" systems that cannot be decomposed into groups of series and parallel configurations. The supported configurations include:

- Series configurations, parallel configurations, combined (series and parallel) configurations and complex configurations - Section 7.1.3 (p. 211)
- *k-out-of-n* parallel configurations - Section 7.1.5 (p. 217)
- Configurations with inherited subdiagrams - Section 7.1.6 (p. 218)
- Standby redundancy configurations - Section 7.1.7 (p. 220)
- Load sharing configurations - Section 7.1.8 (p. 226)
- Configurations with mirrored blocks - Section 7.3 (p. 243)

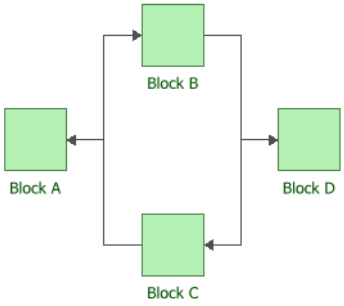
In order to perform analysis, all RBDs created in BlockSim must conform to the following constraints:

- A diagram must contain at least one block.
- If a diagram contains more than one block, all blocks in the diagram must be joined with connectors.
- Diagrams must contain a single starting block and a single ending block, as shown next.



If a diagram consists of only one block, that block represents both the starting and ending block. Starting and/or ending blocks that possess no reliability or maintenance characteristics can be defined by making sure no URD is assigned in the Block Properties window. This indicates that the block cannot fail and that the reliability of the component at all times is considered to be 1 or 100%.

- Circular references, as shown below, are not allowed. There must be a clear path towards the end of the RBD and the RBD cannot loop back on itself.



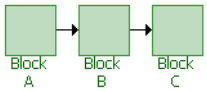
Analysis is impossible

- A standby container must contain at least one active block and one standby block.
- A load sharing container must contain at least one block.

7.1.3 Standard Blocks and Basic Configurations

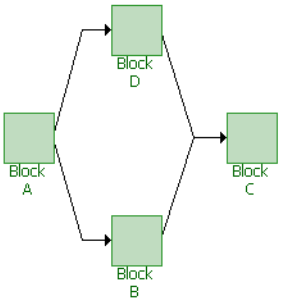
Standard blocks represent some functional portion of the system, such as a component, subsystem, assembly or failure mode. Using only standard blocks, you can define series configurations, parallel configurations and combinations of the two, as well as complex configurations.

In a ***series configuration***, all items must work for the system to work. In the RBD shown next, the system fails if either Block A, Block B or Block C fails.



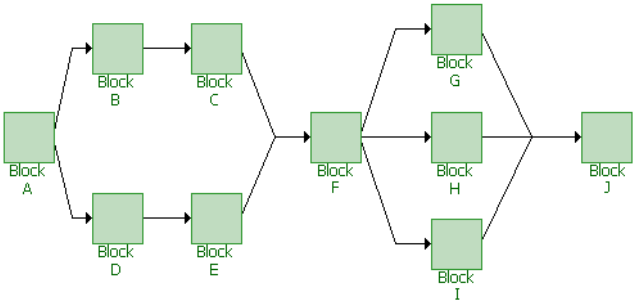
The ReliaWiki resource portal has more information on series configurations at: http://www.ReliaWiki.org/index.php/RBDs_and_Analytical_System_Reliability.

In a ***parallel configuration***, items are considered to be redundant. In the RBD shown next, the system will continue to function if either Block D or Block B fails.

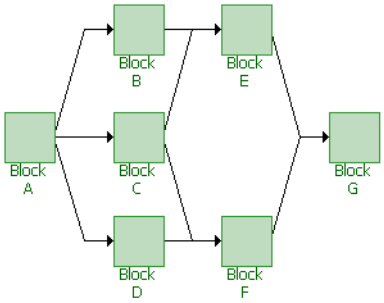


The ReliaWiki resource portal has more information on parallel configurations at: http://www.ReliaWiki.org/index.php/RBDs_and_Analytical_System_Reliability.

You can combine series and parallel configurations in a single RBD, as shown next.



You can also use BlockSim to create "complex" block diagrams. ***Complex configurations*** cannot be expressed as a simple combination of series and/or parallel configurations and thus require a more advanced analytical treatment. A complex reliability block diagram is shown next.



Multiple analytical methods exist for obtaining the reliability of a complex system, such as:

- Bayes' theorem method
- Boolean truth table method
- Probability maps method
- Logic diagrams method
- Decomposition method
- Event space method
- Path-tracing method

Solution using these methods is cumbersome and time consuming. BlockSim uses a proprietary method that employs a combination of several of these methods, saving the user time and effort.

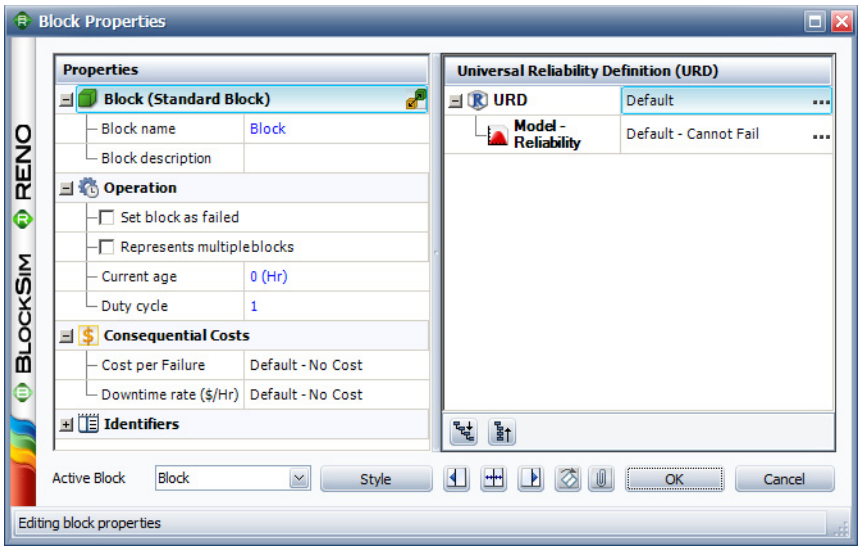
The ReliaWiki resource portal has more information on complex configurations at: http://www.ReliaWiki.org/index.php/RBDs_and_Analytical_System_Reliability.

The properties available for configuring a standard block will vary depending on whether it is in an analytical RBD or a simulation RBD. In addition to the common block properties (*described on page 194 in Chapter 6*), you will need to describe how the block behaves.

An analytical diagram does not take into account any repair or maintenance information, so you can configure a standard block in this type of diagram by defining only the reliability characteristics and basic operational properties. In a simulation diagram, you can also define maintenance activities (i.e., a corrective task and/or scheduled tasks), a maintenance group and throughput information. A maintenance group is a set of blocks that can all be maintained at the same time. The throughput properties define the amount of output that can be processed by the block in a given period of time. *For more information on analytical diagrams, see Chapter 8; for simulation diagrams, see Chapter 9*.

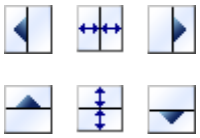
Properties of a Standard Block

The Block Properties window for a standard block is divided into two areas: the **Universal Reliability Definition** area allows you to define the block's reliability and maintenance characteristics, while the **Properties** area allows you to define all remaining block properties. By default, the **Properties** area is on the left side of the window and the **Universal Reliability Definition** area is on the right side, as shown in the following example.



You can click the **Change Orientation** icon to toggle the orientation of the areas between horizontal and vertical.

You can also display only one of the areas or both of the areas by using the following icons to move the splitter all the way to the left/top, to the center, or all the way to the right/bottom.



Reliability and maintenance characteristics are defined by assigning a universal reliability definition (URD) in the **Universal Reliability Definition** area. URDs are resources that can be shared among analyses. Each one includes a model (either a reliability model, a probability of failure model or an event occurrence model) and may also include a corrective task and/or scheduled tasks. When you assign a URD to a block, only those characteristics that are relevant to the block are used. Thus, you could use a URD that includes tasks for a block in either an analytical or a simulation diagram. For the block in the analytical diagram, only the model would be relevant, and all maintenance information would be ignored during analysis. For the block in the simulation diagram, the model and the tasks would be relevant, and all properties would be considered during simulation.

What's Changed? In previous versions, reliability and maintenance characteristics were defined directly in the Block Properties window for each block. Using the URD to define reliability and maintenance characteristics makes it easy to reuse this information wherever it may be required.

To assign a URD to a block, you can either select an existing URD or create a new one. Click the **URD** field and then click the arrow to display the URD wizard. In the wizard, you can click **Select Existing URD** to display a list of the available URDs. You can also click **Create New URD** to open the Universal Reliability Definition window; in this window, specify the failure model and, if you are working with a simulation diagram, any associated tasks, then click **OK** to create the new URD and apply it to the block.

Once a URD has been assigned to the block, you can view and/or edit its properties from the URD Wizard by clicking the **View/Edit** icon.

You can remove the URD from the block by clicking the **Remove** icon in the URD Wizard.

You can also add, change or edit the components of the URD (i.e., the model, corrective task and/or scheduled tasks) directly from the **URD** area. Be aware that any changes you make here will apply everywhere that the URD is used.

If no URD is assigned to the block, you can create a new model for the block by clicking the **Model** field and then clicking the arrow. This opens the Model Wizard, which allows you to define a model for the block; this has the effect of automatically creating a new URD that uses the new model and assigning it to the block.

The additional options available to configure the block, located in the **Properties** area, include:

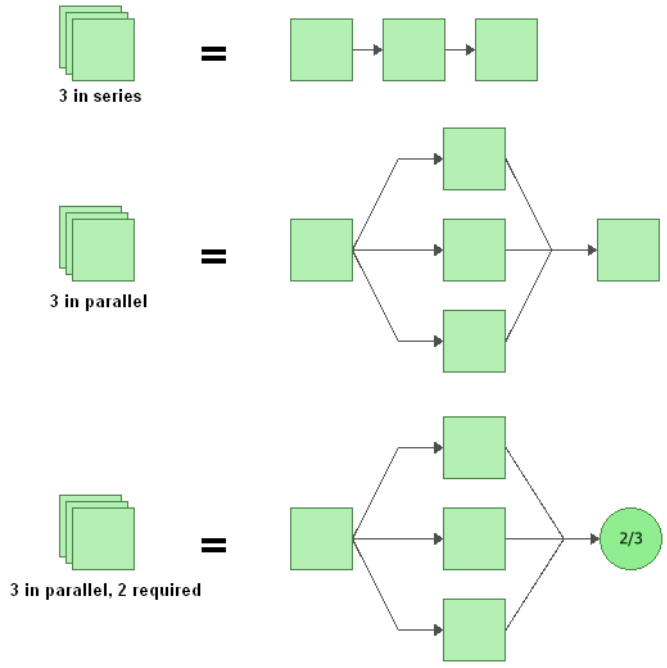
- **Set block as failed** if selected, indicates that the block is "off" or absent from the system. An X will be displayed on the block to indicate that it is failed. The block will be considered to be failed throughout the entire analysis/simulation and no maintenance actions will be performed (i.e., any failure and maintenance properties will be ignored). This option can be used for "what-if" analyses to investigate the impact of a block on system metrics such as reliability, availability, throughput, etc. You can also set this option by selecting the block in the diagram and choosing **Diagram > Settings > Set Block as Failed**.

  When this option is selected, no other properties will be available for the block; note, however, that any properties you have already specified are simply hidden because they are not relevant. The settings will reappear if you clear the **Set block as failed** option.

- **Represents multiple blocks** if selected, indicates that the block represents more than one block with the same properties. This can save time in defining and updating block properties, and also save space within the diagram.

  When this option is selected, you can enter the number of components represented by the block and specify whether they are connected in series or in parallel. If they are connected in parallel, you can also specify how many of the items are required to be functioning in order for the multi block to be functioning. The images shown

next provide the equivalent configurations for multi blocks representing three blocks in series, three blocks in parallel and three blocks where at least two of the three must be functioning.



IMPORTANT: When a block represents multiple identical blocks, although the components represented by the multi block have the same reliability, maintenance and other characteristics, they are considered separate blocks in analytical diagrams. They will behave comparably over the course of the simulation of a simulation diagram, but not identically at any given time during simulation. For instance, if one of the components fails, the others may continue to function, which can affect when maintenance is performed, costs associated with the block, etc. This is in contrast to the functionality of mirrored blocks. Blocks that belong to a mirror group are multiple instances of a single block. Mathematically, they are considered to be a single block in analytical diagrams. When a simulation diagram with mirrored blocks is simulated, every event associated with one block in a mirror group will also apply to all other blocks in the mirror group. *For details about mirrored blocks, see Section 7.3 on page 243*.

- **Current Age** defines the age of the block at the start of the analysis. This allows you to define blocks that have already accumulated age (i.e., "used" components). This field requires that you specify a time unit.
- **Duty Cycle** allows you to model situations where the actual usage of a block during system operation is not identical to the usage for which you have data (either from testing or from the field). This can include situations where the item:
  - Does not operate continuously (e.g., a DVD drive that was tested in continuous operation, but in actual use within a computer accumulates only 18 minutes of usage for every hour the computer operates).
  - Is subjected to loads that are greater than or less than the rated loads (e.g., a motor that is rated to operate at 1,000 rpm but is being used at 800 rpm).
  - Is affected by changes in environmental stress (e.g., a laptop computer that is typically used indoors at room temperature, but is being used outdoors in tropical conditions).

  In these cases, continuous operation at the rated load is considered to be a duty cycle of 1. Any other level of usage is expressed as a percentage of the rated load value or operating time. For example, consider the DVD drive

mentioned above; its duty cycle value would be 18 min / 60 min = 0.3. A duty cycle value higher than 1 indicates a load in excess of the rated value.

The ReliaWiki resource portal has more information on duty cycles at: http://www.ReliaWiki.org/index.php/Time-Dependent_System_Reliability_(Analytical).

- **Consequential Costs** allows you to choose or create models to represent costs that are always associated with the block. **Cost per Failure** uses a cost model, and **Downtime Rate** uses a cost per unit time model. If no models are assigned, it is assumed that there are no additional costs.

The following additional options are available only when you are working with a simulation diagram:

- **Operates even if system is down** indicates that the block continues to operate even if the system is down (and will continue to accumulate age).
- **Belongs to a maintenance group** if selected, you can specify the maintenance group that the block belongs to.
- **Enable state change triggers (SCT)** allows you to specify the starting state of the block (i.e., off or on) and its state upon repair, then specify events that will activate and/or deactivate the block during simulation. You can choose to activate or deactivate the block when items in specified maintenance groups go down or are restored. The current block does not need to be part of the specified maintenance group to use this functionality. This allows you to model a cold standby configuration (i.e., one where the component cannot fail when in standby) without using a standby container, which may be useful if you are using a parallel or complex configuration, as blocks can be connected only in series in standby containers.

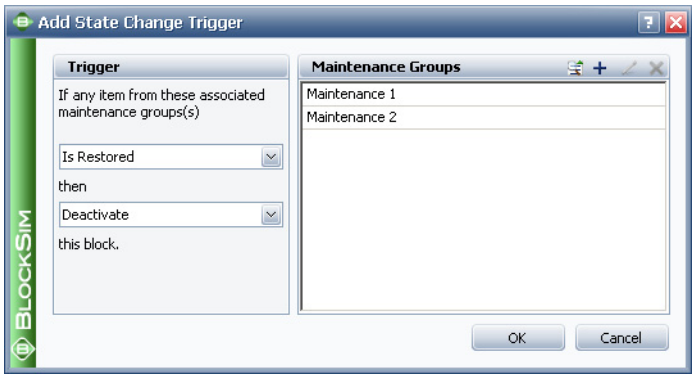
  To add a state change trigger, click the **Add** icon in the **Add a State Change Trigger** field. The State Change Trigger window will appear, allowing you to define the trigger. Each trigger that you add will then be displayed in the Block Properties window. To edit an existing state change trigger, click the **Edit** icon for the state change trigger to open the State Change Trigger window.
- **Throughput** properties define the amount of output that can be processed by the block in a given period of time. *For more information about throughput analysis, see page 277 in Chapter 9*.

Once the block is configured, you can click the **Optimum Replacement** icon in the **Properties** area to open the Optimum Replacement window for the current block; this window allows you to determine the most cost-effective time to replace the component based on costs for planned (i.e., preventive) and unplanned (i.e., corrective) replacement.

7.1.4 State Change Triggers

State change triggers (SCT) allow you to specify the starting state of a block (i.e., off or on) and its state upon repair, then specify events that will activate and/or deactivate the block during simulation. This allows you to model a cold standby configuration (i.e., one where the component cannot fail when in standby) without using a standby container, which may be useful if you are using a parallel or complex configuration, as blocks can be connected only in series in standby containers. State change triggers are available for standard blocks in simulation RBDs and for events in simulation fault trees.

The State Change Trigger window is where you specify the events that will activate and/or deactivate the block. It is accessed from the Block Properties window by clicking the **Add** icon in the **Add a State Change Trigger** field or the **Edit** icon for an existing state change trigger.



The state change trigger can either activate or deactivate the block when items in specified maintenance groups go down or are restored. To define the state change trigger, specify the triggering event (i.e., an item goes down or an item is restored), the state change (i.e., the block is activated or deactivated) and the maintenance group(s) in which the triggering event must happen in order to trigger the state change.

The current block does not need to be part of the specified maintenance group(s) to use this functionality.

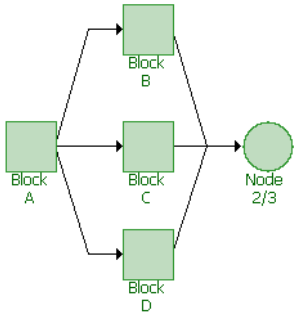
The ReliaWiki resource portal has more information on state change triggers at: http://www.ReliaWiki.org/index.php/State_Change_Triggers.

For an example using state change triggers, please read "Advanced RBD Modeling Using State Change Triggers" at: http://www.weibull.com/hotwire/issue134/hottopics134.htm.

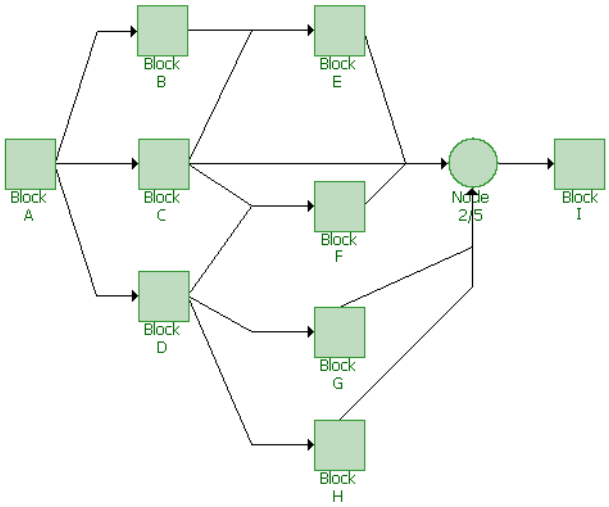
7.1.5 Node Blocks and k-out-of-n Configurations

Node blocks allow you to specify a form of redundancy known as *k-out-of-n* redundancy. A *k-out-of-n* node can have *n* paths leading into it and requires that *k* out of *n* paths must function for the system to function.

The configuration in the figure shown next displays a *k-out-of-n* configuration where either Block B, Block C or Block D can fail without causing system failure, but if any two of the blocks fail, the system will fail.



The figure shown next displays another *k-out-of-n* node, this time used in a complex configuration. If more than three of the five blocks (blocks C, E, F, G and H) leading to the node fail, the system will fail.



The ReliaWiki resource portal has more information on *k-out-of-n* configurations at: http://www.ReliaWiki.org/index.php/RBDs_and_Analytical_System_Reliability.

Properties of a Node Block

To configure a node block, in addition to the common block properties (*described on page 194 in Chapter 6*), you will need to specify the number of paths that must reach the node in the **Number of Paths Required** field. If the required number of paths does not succeed, then the node is considered to be failed.

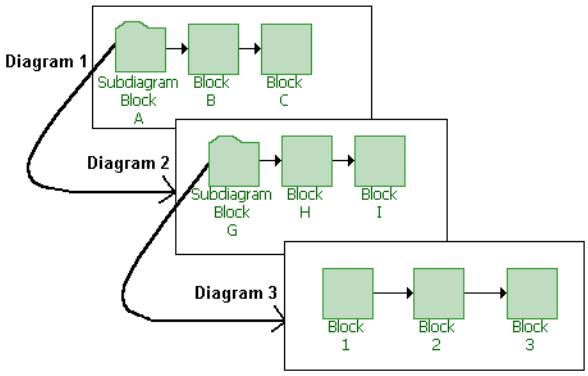
For node blocks in simulation diagrams, you can also specify throughput properties, which define the amount of output that can be processed by the block in a given period of time. *For more information on throughput analysis, see page 277 in Chapter 9*.

7.1.6 Subdiagrams

Subdiagram blocks represent other diagrams within the project. Using subdiagram blocks allows you to maintain separate diagrams for portions of a system and to incorporate those diagrams as components of another diagram. This also allows you to generate and analyze extremely complex diagrams representing many subsystems, subsubsystems, etc.

In the figure shown next, Subdiagram Block A in Diagram 1 represents the series configuration of the subsystem reflected in Diagram 2, while Subdiagram Block G in Diagram 2 represents the series configuration of the

subsubsystem in Diagram 3. Note that although RBDs are shown in this example, subdiagrams in fault trees work in a similar way.

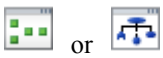


Tip: It is appropriate to use a subdiagram block when you want to take into consideration all properties of all individual blocks in the subdiagram. There may be times, however, when you are interested only in the outcome of the subdiagram as a whole (i.e., its fitted distribution and/or system reliability equation). In such cases, you can create a URD that uses the subdiagram's published results as its failure model; you can then represent the subdiagram within the current diagram by creating a standard block using that URD. You might do this:

- When the subdiagram is a lowest replaceable unit (LRU). That is, when the subdiagram fails, the entire assembly that it represents will be replaced (e.g., a motherboard in a computer).
- In order to represent an analytical subdiagram within a simulation diagram or to represent a simulation subdiagram within an analytical diagram.
- In order to use complex diagrams inside container blocks.
- If your diagram is extremely complex, in order to streamline simulation so that it takes less time.
- It is important to remember that in all of these cases, the subdiagram will be treated as an LRU and no results will be available for anything within it.

Properties of a Subdiagram Block

To configure a subdiagram block, in addition to the common block properties (*described on page 194 in Chapter 6*), you will need to specify the diagram that the subdiagram block represents. You will be automatically prompted for this information when you add the subdiagram block. If you are working with an analytical diagram, only analytical diagrams will be shown in this list; if you are working with a simulation diagram, only simulation diagrams will be available. If the diagram that the subdiagram block will represent has not yet been created, you can click the **Create New RBD** icon or the **Create New Fault Tree** icon.



A new diagram of the same type as the current diagram will be added to the project.

Double-clicking a subdiagram block will open the subdiagram that the block is based on. You can open the Block Properties window for a subdiagram block by selecting the block and choosing the command on the Ribbon or pressing **CTRL+E**.

When you are viewing or editing the subdiagram properties in the Block Properties window, you can change the diagram that it represents by choosing an existing diagram from the **Based on Diagram** drop-down list. As when you

are adding the subdiagram block, only diagrams of the same type (analytical or simulation) as the current diagram will be available in this list.

What's Changed? In BlockSim 7, you had the option to use reliability/maintainability characteristics from the blocks in the linked subdiagram or to manually define reliability/maintainability characteristics for the subdiagram block and use an approximate failure distribution. In the Synthesis version, the subdiagram block always uses the characteristics from the blocks in the linked subdiagram. If you want to consider the subdiagram's failure distribution only at a diagram level, you can analyze/simulate the subdiagram, fit a model and publish it, then represent the subdiagram using a standard block that uses the published model, as noted in the tip above.

Starting in version 9.0.10, you can assign a **Duty Cycle** to a subdiagram block. Duty cycles allow you to model situations where the actual usage of the assembly represented by the subdiagram during system operation is not identical to the usage for which you have data (either from testing or from the field). This can include situations where the item:

- Does not operate continuously (e.g., a DVD drive that was tested in continuous operation, but in actual use within a computer accumulates only 18 minutes of usage for every hour the computer operates).
- Is subjected to loads that are greater than or less than the rated loads (e.g., a motor that is rated to operate at 1,000 rpm but is being used at 800 rpm).
- Is affected by changes in environmental stress (e.g., a laptop computer that is typically used indoors at room temperature, but is being used outdoors in tropical conditions).

In these cases, continuous operation at the rated load is considered to be a duty cycle of 1. Any other level of usage is expressed as a percentage of the rated load value or operating time. For example, consider the DVD drive mentioned above; its duty cycle value would be 18 min / 60 min = 0.3. A duty cycle value higher than 1 indicates a load in excess of the rated value.

If a duty cycle is specified for the subdiagram block and there are also duty cycles specified for blocks within the diagram that it represents, their effects are compounded. For instance, if the subdiagram block has a duty cycle of 1.5 and one of the blocks in the diagram that it represents has a duty cycle of 0.75, the effective duty cycle for that block within the subdiagram would be 1.5 x 0.75 = 1.125.

The ReliaWiki resource portal has more information on duty cycles at:
http://www.ReliaWiki.org/index.php/Time-Dependent_System_Reliability_(Analytical).

For subdiagram blocks in simulation diagrams, you can also define a maintenance group and throughput information. A maintenance group is a set of blocks where some event within the group can trigger either maintenance or state changes for one or more blocks, either within the group or outside of it. Throughput properties define the amount of output that can be processed by the block in a given period of time.

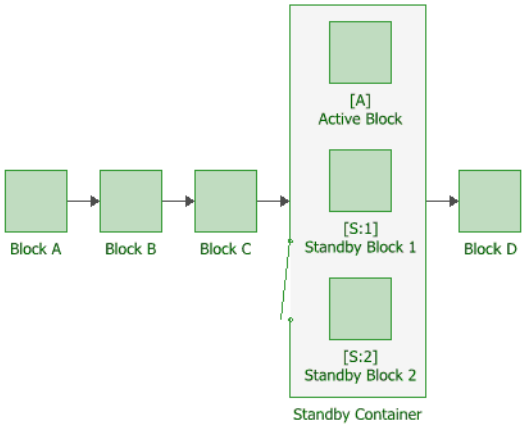
7.1.7 Standby Containers

Standby containers allow you to identify blocks or paths that operate in a standby redundancy configuration. Each path in the container consists of one or more blocks (connected in series if there is more than one block in the path); one or more of the paths in the container are active while one or more paths are dormant (i.e., standby or quiescent). The dormant paths are available to become active under specified circumstances. In BlockSim, the standby container can also be configured to represent a switch that must be triggered in order to transfer the activity to the standby path (and then back to the original active path, if the standby path later fails or if the original active path is set to be reactivated after repair). Therefore, the standby container can also have an assigned switch that describes the reliability characteristics for the switch (i.e., the probability that the switch will function properly when it is needed) and any maintenance activities that apply to the switch.

For example, suppose that a system contains two batteries but requires only one functioning battery in order for the system to succeed. At any given time, one battery operates while the other waits in a standby state. In addition, the success of the system depends on the successful switch from the active battery to the standby battery, when required. If the active battery fails and there is a successful switch from one battery to the other, then the standby battery becomes active and the system continues to operate. If the switch fails, then the system fails. The system will also fail if the switch is successful but the standby battery has failed while in a standby state.

Note: When analyzing diagrams with containers, the analysis and throughput results are presented at the container level. In addition, optimization via allocation analysis is performed at the container level.

The figure shown next includes a standby container with three items in standby redundancy configuration where one component is active and the other two components are on standby. In this configuration, one block in the container must be operating or, because the container is part of a series configuration, the system will fail.



Properties of a Standby Container

The container has properties that describe the way the container behaves. A block inside a container, a contained block, also has its own block properties.

To configure a standby container, in addition to the common block properties (*described on page 194 in Chapter 6*), you will need to specify the **Number of Active Paths Required**, which is the number of active paths within the container that must succeed in order for the container to succeed. If the required number of paths does not succeed, then the container is considered to be failed.

Additional options are available for the standby container, including:

- **Set block as failed** if selected, indicates that the container is failed (i.e., the block is "off" or absent from the system). An X will be displayed on the container to indicate that it is failed. The block will be considered to be failed throughout the entire analysis/simulation and no maintenance actions will be performed (i.e., any failure and maintenance properties will be ignored). This option can be used for "what-if" analyses to investigate the impact of a block on system metrics such as reliability, availability, throughput, etc. You can also set this option by selecting the block in the diagram and choosing **Diagram > Settings > Set Block as Failed**.

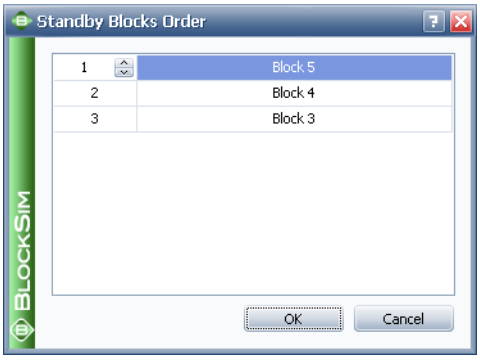
  When this option is selected, no other properties will be available for the block; note, however, that any properties you have already specified are simply hidden because they are not relevant. The settings will reappear if you clear the Set block as failed option.

- **Adjust Numerical Convergence Settings** opens the Algorithm Setup window, which allows you to modify the parameters of the numerical integration that is used in analyzing the container, as well as to specify a level of accuracy that should be reached during calculations. This option is available only for analytical diagrams. *For more information about the container algorithm setup, see Chapter 23*.
- The **Switch** field allows you to create or select a switch resource (*described in Section 7.1.7.2 on page 224*), which defines the standby container as a switch that is necessary to transfer the activity from active to standby blocks. The switch resource includes information on the reliability of the switch itself and of the switch action, any restarts and any maintenance tasks that apply to the switch.

For standby containers in simulation diagrams, you can also specify throughput properties, which define the amount of output that can be processed by the block in a given period of time.

If a standby container has more than one contained block in a standby state, you can specify the order in which the standby blocks are used. To do this, select the standby container and choose **Diagram > Properties > Standby Settings > Set Standby Orders** or click the **Set Standby Orders** icon in the Block Properties window for the standby container.

In the Standby Blocks Order window that appears, the standby blocks are displayed in the order in which they will be used. Select a block and use the up and down arrows that appear in its order number cell to move the block up and down the list. For example, in the window shown next, you would click the down arrow once to move Block 5 down one position; the result would be that Block 4 would be used first, then Block 5, then Block 3.



The ReliaWiki resource portal has more information on standby configurations at: http://www.ReliaWiki.org/index.php/Time-Dependent_System_Reliability_(Analytical).

7.1.7.1 Contained Standby Blocks

Add a contained block to a container by selecting the container and then choosing **Diagram > Blocks > Add Block**.

The properties available for configuring a contained standby block will vary depending on whether it is in an analytical RBD or a simulation RBD. In addition to the common block properties (*described on page 194 in Chapter 6*), you will need to describe how the block behaves.

The first step is to assign a URD for the block when it is in an active state and one for when it is in a standby state (i.e., quiescent or dormant). As explained in the section on standard blocks (*Section 7.1.3 on page 211*), each URD

uses a failure model to describe the block's reliability characteristics; for blocks in simulation diagrams, the corrective task and/or scheduled tasks associated with the URD are also applied to the block.

Note: In the case that both quiescent and active failure models are the same, the units are in a simple parallel configuration (also called a *hot standby* configuration). When the probability of failure of the standby component is less in the quiescent state than in the active state, that is called a *warm standby* configuration. Lastly, when the probability of failure of the standby component is zero in the quiescent state (i.e., the component cannot fail when in standby), that is called a *cold standby* configuration.

The following options may be available for contained standby blocks in both analytical and simulation diagrams:

- **Duty Cycle** allows you to model situations where the actual usage of a block during system operation is not identical to the usage for which you have data (either from testing or from the field). This can include situations where the item:
  - Does not operate continuously (e.g., a DVD drive that was tested in continuous operation, but in actual use within a computer accumulates only 18 minutes of usage for every hour the computer operates).
  - Is subjected to loads that are greater than or less than the rated loads (e.g., a motor that is rated to operate at 1,000 rpm but is being used at 800 rpm).
  - Is affected by changes in environmental stress (e.g., a laptop computer that is typically used indoors at room temperature, but is being used outdoors in tropical conditions).

  In these cases, continuous operation at the rated load is considered to be a duty cycle of 1. Any other level of usage is expressed as a percentage of the rated load value or operating time. For example, consider the DVD drive mentioned above; its duty cycle value would be 18 min / 60 min = 0.3. A duty cycle value higher than 1 indicates a load in excess of the rated value.

  The ReliaWiki resource portal has more information on duty cycles at: http://www.ReliaWiki.org/index.php/Time-Dependent_System_Reliability_(Analytical).
- **Active/Standby** allows you to specify whether the block is in an active or standby state at the beginning of the simulation.

  Tip: You can quickly change a block's state by selecting or clearing the **Diagram > Properties > Standby Settings > Set Block to Active (Not Standby)** command.
- **Consequential Costs** allows you to choose or create models to represent costs that are always associated with the block. **Cost per Failure** uses a cost model, and **Downtime Rate** uses a cost per unit time model. If no models are assigned, it is assumed that there are no additional costs.

The following options are available only for contained standby blocks in simulation diagrams:

- **Reactivate After Repair** if selected, the block will return to the active state immediately after it has been repaired. Otherwise, the block will remain in the standby state after if has been repaired and will be switched back to the active state only if and when the contained standby block that was activated in the initial switch fails. This option is available only if the active state is selected.
- **Operates even if system is down** if selected, indicates that the block continues to operate even if the system is not operating. If it is not selected, the block does not continue to accumulate age when the system is down.

In a simulation diagram, you can also assign a maintenance group for the contained standby block. A maintenance group is a set of blocks where some event within the group can trigger either maintenance or state changes for one or more blocks, either within the group or outside of it.

Once the block is configured, you can click the **Optimum Replacement** icon in the **Properties** area to open the Optimum Replacement window for the current block; this window allows you to determine the most cost-effective time to replace the component based on costs for planned (i.e., preventive) and unplanned (i.e., corrective) replacement.

7.1.7.2 Switches

A standby container, priority AND gate or sequence enforcing gate can be configured as a switch that is necessary to transfer the activity from the active block to a standby block (and then back to the original active block, if the standby block later fails or if the original active block is set to be reactivated after repair). To do this, assign a switch resource to the container or gate. Switches are resources that can be shared among analyses and can be managed via the Resource Manager.

To assign a switch to a container or gate, you can either select an existing switch or create a new one. Click the **Switch** field and then click the arrow to display the Quick Switch Action Wizard. In the wizard, you can click **Select Existing Switch** to display a list of the available switches. You can also click **Create New Switch** to open the Switch window; in this window, define the properties of the switch, then click **OK** to create the new switch and apply it to the block.

Once a switch has been assigned to the block, you can view and/or edit its properties from the Switch Action Wizard by clicking the **View/Edit** icon.

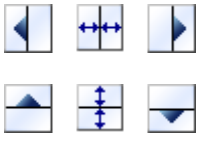
You can remove the switch from the block by clicking the **Remove** icon in the Switch Action Wizard.

You can also add, change or edit the components of the switch (i.e., the model, corrective task, scheduled tasks and/or maintenance group) directly from the **Switch** field's subfields. Be aware that any changes you make here will apply everywhere that the switch is used.

Switch Window

The Switch window is divided into two areas: the **Resources** area allows you to define the switch's reliability and maintenance characteristics, while the **Properties** area allows you to define all remaining switch properties. By default, the **Properties** area is on the left side of the window and the **Resources** area is on the right side. You can click the **Change Orientation** icon to toggle the orientation of the areas between horizontal and vertical.

You can also display only one of the areas or both of the areas by using the following icons to move the splitter all the way to the left/top, to the center, or all the way to the right/bottom.



For a new resource, a name will be proposed automatically based on the default naming criteria established for the current database (in the Define Default Name Formats window). You can replace this with your own name of up to 150 characters, if desired. Remember that the name and identifiers are the primary way in which your team will be able to find the Synthesis resources you need for your analyses.

In the **Properties** area, the following properties are available to configure the switch:

- **Switch Action**
  - **Probability per Request** allows you to specify the probability that the switch action will succeed the first time it is needed. This also applies to the switch back to the original active block, if any.
  - **Delay Time** allows you to specify the amount of time it takes to perform the switch each time it happens. This includes the time for the first switch and any restarts, if applicable. You must specify the units used.
  - **Allow Restarts** if selected, indicates that if the switch fails on the first start, there will be attempts to restart the switch. If this option is selected, you must specify the **Number of Restarts** allowed. If the switch is not successful by the last specified restart, then it is considered to be failed. In the **Switch Restart Probability** field, specify the likelihood that the switch will succeed when you attempt to restart after an initial failure. This value is specified as a decimal, where 1 indicates a 100% probability of success.
  - **Switch operates even if the system is down** if selected, indicates that the switch continues to operate even if the system is not operating. If not selected, indicates that the switch will not continue to accumulate age.
- **Identifiers** allows you to enter additional identifying information that can be used to search for this resource.
- **History** provides information about the creation and last modification of the resource. If the history log has been activated at the project level (**Project > Management > History Log**), you can click the **View Item History** icon to open the Record History Log for the resource.

The **Resources** area allows you to specify the reliability and maintenance properties of the switch, including the model that represents the reliability of the switch itself (either a reliability model, a probability of failure model or an event occurrence model), any tasks that apply to the switch and the maintenance group that it is associated with, if any. The maintenance information in this area is applicable only when the switch is used in a simulation diagram. If the Switch window is accessed from within an analytical diagram, the maintenance-related fields (i.e., those pertaining to tasks and to maintenance groups) will not appear even if the switch has maintenance properties.

For an existing resource, a link at the bottom of the properties window indicates how many times it is currently being used. If you need more information, click the link or the **Trace Usage** icon.

This opens the Dependency Viewer, which allows you to explore both where the resource is being used and what other resources are associated with it.

7.1.7.3 Standby Without Containers

If you need to use a parallel or complex configuration as a path within a container, you can do the following: Create a diagram that models the required path configuration and analyze or simulate the diagram, then publish the results as a model. Create a URD that uses the model. In the container, add a contained block and assign the URD that you just created to it. Note that when you do this, results for the components within the path will not be available in your final results; only the fitted model is used.

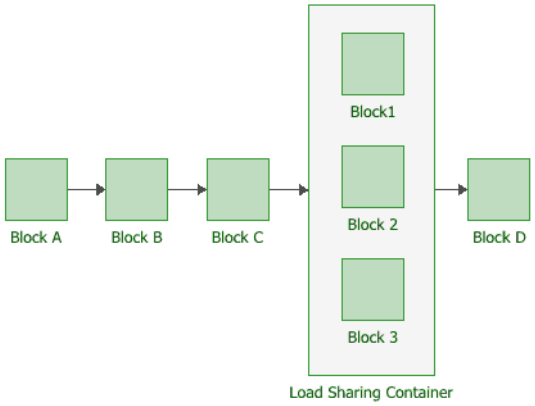
You can also model a cold standby configuration (i.e., one where the component cannot fail when in standby) without using a standby container by using the **Enable State Change Triggers** option in the Block Properties window. This option allows you to specify the starting state of the block (i.e., off or on) and then specify events that will activate and/or deactivate the block during simulation. You can choose to activate or deactivate the block when items in specified maintenance groups go down or are restored. The current block does not need to be part of the specified maintenance group to use this functionality.

7.1.8 Load Sharing Containers

Load sharing containers allow you to identify blocks that are in load sharing redundancy. This indicates that two or more blocks in the container share the responsibility for keeping the system running properly; if one of them fails, the other(s) can take on an increased "load" so the system can continue to operate. Therefore, load sharing components exhibit different failure characteristics when one or more fail.

For example, suppose that two heat dissipation devices operate in a load sharing configuration and together they are expected to provide dissipation for a temperature difference of 100 K of heat. When both units are operating, they share the load and each dissipates 50% of the heat. If one unit fails, then the load on the other unit is increased to 100% of the heat.

In the figure shown next, Blocks 1, 2 and 3 are in a load sharing container and have their own failure characteristics. If the container's properties specify that only one contained block must succeed in order for the container to succeed, then all three contained blocks must fail for the container to fail. However, as individual contained blocks fail, the failure characteristics of the remaining contained blocks change because they now have to carry a higher load to compensate for the failed ones.



Properties of a Load Sharing Container

The container has properties that describe the way the container behaves. A block inside a container, a contained block, also has its own block properties.

To configure a load sharing container, in addition to the common block properties (*described on page 194 in Chapter 6*), you will need to specify the **Number of Paths Required**, which is the number of active blocks/paths within the container that must succeed in order for the container to succeed. If the required number of blocks/paths does not succeed, then the container is considered to be failed. The contained blocks must be within the load sharing container, but they do not need to be connected to each other.

Additional options are available for the load sharing container, including:

- **Set block as failed** if selected, indicates that the container is failed (i.e., the block is "off" or absent from the system). An X will be displayed on the container to indicate that it is failed. The block will be considered to be failed throughout the entire analysis/simulation and no maintenance actions will be performed (i.e., any failure and maintenance properties will be ignored). This option can be used for "what-if" analyses to investigate the

impact of a block on system metrics such as reliability, availability, throughput, etc. You can also set this option by selecting the block in the diagram and choosing **Diagram > Settings > Set Block as Failed**.

When this option is selected, no other properties will be available for the block; note, however, that any properties you have already specified are simply hidden because they are not relevant. The settings will reappear if you clear the **Set block as failed** option.

- **Adjust Numerical Convergence Settings** opens the Algorithm Setup window, which allows you to modify the parameters of the numerical integration that is used in analyzing the container, as well as to specify a level of accuracy that should be reached during calculations. This option is available only for analytical diagrams. *For more information about the container algorithm setup, see Chapter 23*.

For load sharing containers in simulation diagrams, you can also specify throughput properties, which define the amount of output that can be processed by the block in a given period of time.

What's Changed? The load on contained load sharing blocks is calculated differently in the Synthesis version of BlockSim. Version 7 required a life-stress relationship for such configurations and based the re-calculation of load after block failure on that relationship; the Synthesis version calculates load using the weight proportionality factor as a multiplier. Because of this, if you convert a diagram that uses load sharing containers, you will need to manually configure the contained load sharing blocks after conversion.

The ReliaWiki resource portal has more information on load sharing configurations at: http://www.ReliaWiki.org/index.php/Time-Dependent_System_Reliability_(Analytical).

7.1.8.1 Contained Load Sharing Blocks

Add a contained block to a container by selecting the container and then choosing **Diagram > Blocks > Add Block**.

The properties available for configuring a contained load sharing block will vary depending on whether it is in an analytical RBD or a simulation RBD. In addition to the common block properties (*described on page 194 in Chapter 6*), you will need to describe how the block behaves.

The first step is to assign a URD for the block. As explained in the section on standard blocks (*Section 7.1.3 on page 211*), the URD uses a failure model to describe the block's reliability characteristics; for blocks in simulation diagrams, the corrective task and/or scheduled tasks associated with the URD are also applied to the block.

What's Changed? The load on contained load sharing blocks is calculated differently in the Synthesis version of BlockSim. Version 7 required a life-stress relationship for such configurations and based the re-calculation of load after block failure on that relationship; the Synthesis version calculates load using the weight proportionality factor as a multiplier. Because of this, if you convert a diagram that uses load sharing containers, you will need to manually configure the contained load sharing blocks after conversion.

The following options may be available for contained load sharing blocks in both analytical and simulation diagrams:

- **Weight Proportionality Factor** is compared against the factors in the other blocks in the load sharing container to determine how the load will be shared. For example, if the container holds two blocks and one has a factor of 3 while the other has a factor of 9, then the first block will receive 25% of the load (3/12) and the second block will receive 75% of the load (9/12). Alternatively, if the factor for both blocks is set to 9, then both blocks will receive 50% of the load (9/18).

- **Duty Cycle** allows you to model situations where the actual usage of a block during system operation is not identical to the usage for which you have data (either from testing or from the field). This can include situations where the item:
  - Does not operate continuously (e.g., a DVD drive that was tested in continuous operation, but in actual use within a computer accumulates only 18 minutes of usage for every hour the computer operates).
  - Is subjected to loads that are greater than or less than the rated loads (e.g., a motor that is rated to operate at 1,000 rpm but is being used at 800 rpm).
  - Is affected by changes in environmental stress (e.g., a laptop computer that is typically used indoors at room temperature, but is being used outdoors in tropical conditions).

  In these cases, continuous operation at the rated load is considered to be a duty cycle of 1. Any other level of usage is expressed as a percentage of the rated load value or operating time. For example, consider the DVD drive mentioned above; its duty cycle value would be 18 min / 60 min = 0.3. A duty cycle value higher than 1 indicates a load in excess of the rated value.

  The ReliaWiki resource portal has more information on duty cycles at: http://www.ReliaWiki.org/index.php/Time-Dependent_System_Reliability_(Analytical).
- **Consequential Costs** allows you to choose or create models to represent costs that are always associated with the block. **Cost per Failure** uses a cost model, and **Downtime Rate** uses a cost per unit time model. If no models are assigned, it is assumed that there are no additional costs.

When you are working with a simulation diagram, the following options are also available:

- **Operates even if system is down** indicates that the block continues to operate even if the system is not operating. If it is not selected, the block does not continue to accumulate age when the system is down.
- **Belongs to a maintenance group** if selected, you can specify the maintenance group that the block belongs to.

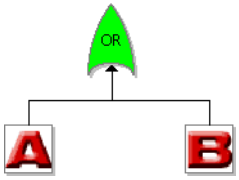
Once the block is configured, you can click the **Optimum Replacement** icon in the Properties area to open the Optimum Replacement window for the current block; this window allows you to determine the most cost-effective time to replace the component based on costs for planned (i.e., preventive) and unplanned (i.e., corrective) replacement.

7.2 Fault Trees

A fault tree diagram follows a top-down structure and represents a graphical model of the pathways within a system that can lead to a foreseeable, undesirable loss event (or a failure). The pathways connect contributory events and conditions using standard logic symbols.

Fault tree diagrams consist of gates and events connected with lines. The AND and OR gates are the two most commonly used gates in a fault tree. To illustrate the use of these gates, consider two events (called input events) that can lead to another event (called the output event). If the occurrence of either input event causes the output event to occur, then these input events are connected using an OR gate. Alternatively, if both input events must occur in order

for the output event to occur, then they are connected by an AND gate. The next figure shows a simple fault tree diagram in which either A or B must occur in order for the output event to occur.



In this diagram, the two events are connected to an OR gate. If the output event is system failure and the two input events are component failures, then this fault tree indicates that the failure of A or B causes the system to fail. The RBD equivalent for this configuration is a simple series system with two blocks, as shown next.

In fault tree analysis, a fault tree diagram is always drawn top-down with the lowest item being an event.

Relationship Between RBDs and Fault Trees

The most fundamental difference between fault tree diagrams and RBDs is that you work in the "success space" in an RBD while you work in the "failure space" in a fault tree. In other words, the RBD looks at success combinations while the fault tree looks at failure combinations. In addition, fault trees have traditionally been used to analyze fixed probabilities (i.e., each event that composes the tree has a fixed probability of occurring) while RBDs may include time-varying distributions for the blocks' success or failure, as well as for other properties such as repair/restoration distributions.

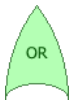
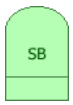
Although the symbols and structures of the two diagram types differ, most of the logical constructions in a fault tree diagram can also be modeled with an RBD. In general, a fault tree can be easily converted to an RBD. It is generally more difficult to convert an RBD into a fault tree, especially if one allows for highly complex configurations.

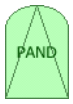
In BlockSim, you can choose **Fault Tree > Conversion > Create RBD** to create a reliability block diagram that is equivalent to the current fault tree. In addition, you can you can create hybrid analyses by linking fault trees as subdiagrams to RBDs and vice versa.

7.2.1 Fault Tree Block Types

This section is intended to help you identify the basic types of blocks found in fault trees.

Note: The shapes, colors and locations shown here represent the default settings. These settings may be changed via the Diagram Style window and/or the Block Style window (**Format > Styles > [Diagram Style/Block Style]**).

Visual Indicator	Block Type	Description
	AND Gate	In an AND gate, the output event occurs if all input events occur.
	OR Gate	In an OR gate, the output event occurs if at least one of the input events occurs.
	NOT Gate	In a NOT gate, the output event occurs if the input event does not occur, and vice versa. This gate performs inversion. NOT gates are available only in analytical fault trees.
	NAND Gate	In a NAND gate, the output event occurs if any one of the input events does not occur. NAND gates are available only in analytical fault trees.
	NOR Gate	In a NOR gate, the output event occurs only if none of the input events occur. NOR gates are available only in analytical fault trees.
	Voting Gate	In a Voting OR gate, the output event occurs if some quantity (k) or more of the (n) input events occur. The voting gate is visually distinguished from the OR gate by the k/n value displayed on it
	Inhibit Gate	In an Inhibit gate, the output event occurs if all input events occur and an additional conditional event occurs.
	Standby Gate	A standby gate allows you to identify events that operate in a standby configuration. Events that are below a standby gate are dependent events, which describe the characteristics of the items that operate together in the standby configuration.

Visual Indicator	Block Type	Description
	Load Sharing Gate	A load sharing gate allows you to identify events that operate in a load sharing configuration. Events that are below a load sharing gate are dependent events, which describe the characteristics of the items that operate together in the load sharing configuration.
	Priority AND Gate	In a priority AND (PAND) gate, the output event occurs if all input events occur in a specific sequence.
	Sequence Enforcing Gate	In a sequence enforcing (SEQ) gate, events are constrained to occur in a specific sequence and the output event occurs if all input events occur in that specified sequence.
	Basic Event	Events are simply representations of some occurrence, typically a failure or error. Different shapes are used to represent different types of events. These shapes convey additional visual information for the analysis but all event types are handled in the same way during analysis or simulation. *For information on how event types are often used, see Section 7.2.4 on page 242*.
	Undeveloped Event	
	Trigger Event	
	Resultant Event	
	Conditional Event	
	Subdiagram	A block that is in the shape of a folder indicates that the block represents an existing diagram. Subdiagrams can be used in fault trees in the same way they are used in RBDs.

7.2.2 Fault Tree Analysis Constraints

In order to permit analysis, all fault tree diagrams created in BlockSim must conform to the following constraints:

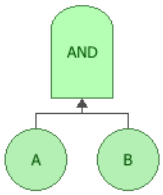
- A diagram must contain a top gate and at least one event.
- All blocks in the diagram must be connected.
- An event can have only one output event, and no input events or gates.
- Additional gates are not allowed below a load sharing gate or standby gate.
- The inputs to negative gates (i.e., NOT, NAND and NOR gates) must be fixed probabilities, or evaluate to fixed probabilities; this is checked at analysis, and any input that evaluates to a time-varying model will result in an error.

7.2.3 Fault Tree Gates

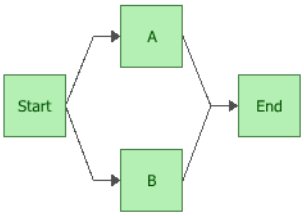
7.2.3.1 AND Gates

AND gates allow you to represent a scenario in which the output event occurs if all input events occur. In system reliability, this implies that both components must fail (input) in order for the system to fail (output). In a reliability block diagram, this is equivalent to a simple parallel configuration.

The next two figures show both the fault tree diagram and reliability block diagram representation of a system with two components, A and B, in which the system fails if both A and B fail. In a fault tree, this is modeled using the AND gate.



In an RBD, it is a parallel configuration.

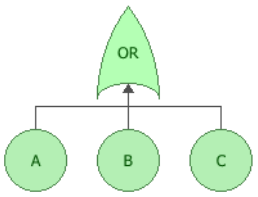


To configure an AND gate, only the common block properties are available. No further configuration of the gate is necessary.

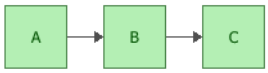
7.2.3.2 OR Gates

OR gates allow you to represent a scenario in which the output event occurs if at least one of the input events occurs. In system reliability, this implies that if any component fails (input event), the system will fail (output event). In a reliability block diagram, this is equivalent to a series configuration.

The next two figures show both the fault tree diagram and reliability block diagram representation of a system with three components, A, B and C, in which the system fails if A, B or C fails. In a fault tree, this is modeled using the OR gate.



In an RBD, it is a series configuration.



To configure an OR gate, only the common block properties are available. No further configuration of the gate is necessary.

7.2.3.3 NOT Gates

The NOT gate functions as an inverter; the output event is the inverse of the input event. Thus, in cases where you are dealing with true/false values, a false input event will lead to a true output event, and vice versa (i.e., the output occurs when the input event does not occur, and vice versa).

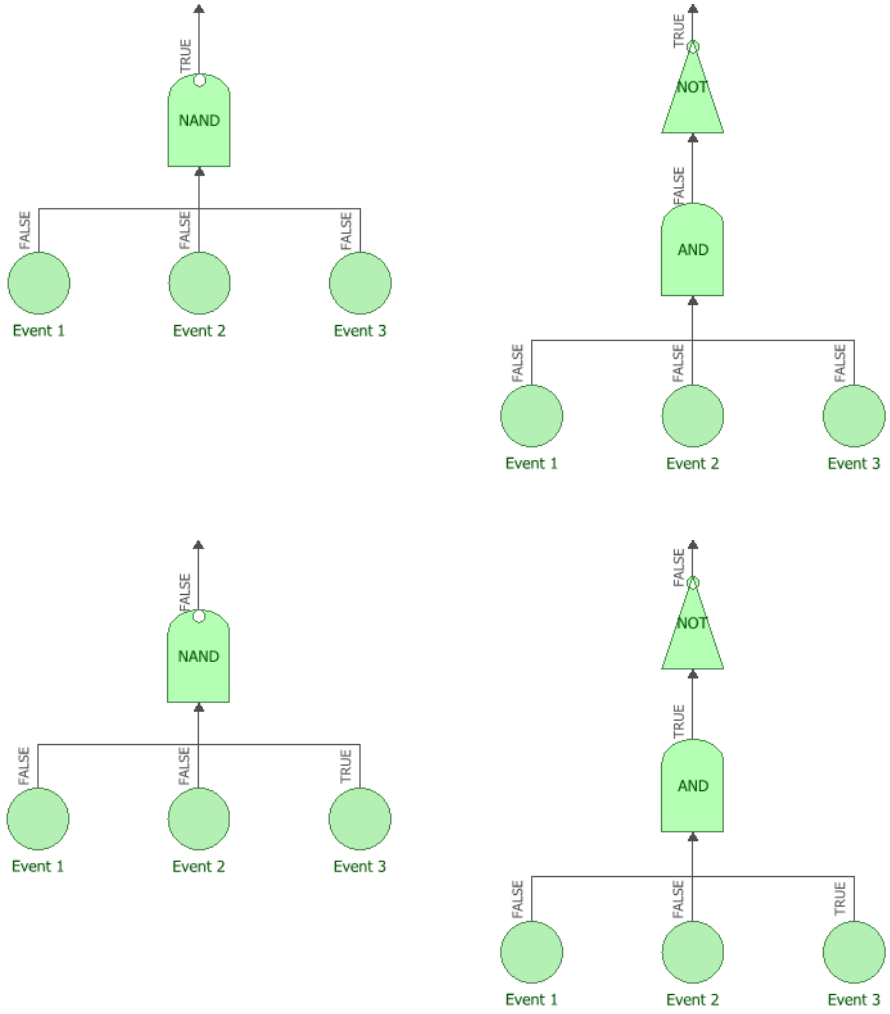
In cases dealing with probabilities and/or reliability calculations, the output is equal to 1-input.

NOT gates are available only in analytical fault trees. There is no equivalent configuration in reliability block diagrams. The inputs to a NOT gate must be fixed probabilities, or evaluate to fixed probabilities; this is checked at analysis, and any input that evaluates to a time-varying model will result in an error.

To configure a NOT gate, only the common block properties are available. No further configuration of the gate is necessary.

7.2.3.4 NAND Gates

A NAND gate has a true output only if all input events are false; it has a false output event when any input event is true. In other words, if any of the input events occurs, then the output event will not occur. This is logically equivalent to using an AND gate followed by a NOT gate.

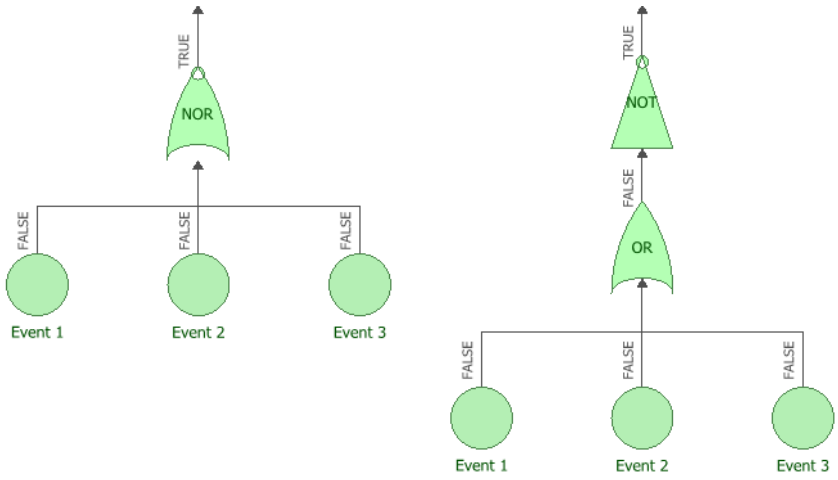


NAND gates are available only in analytical fault trees. There is no equivalent configuration in reliability block diagrams. The inputs to a NOT gate must be fixed probabilities, or evaluate to fixed probabilities; this is checked at analysis, and any input that evaluates to a time-varying model will result in an error.

To configure a NAND gate, only the common block properties are available. No further configuration of the gate is necessary.

7.2.3.5 NOR Gates

A NOR gate has a true output event when all input events are false. In other words, if none of the input events occur, then the output event will occur. This is logically equivalent to using an OR gate followed by a NOT gate.



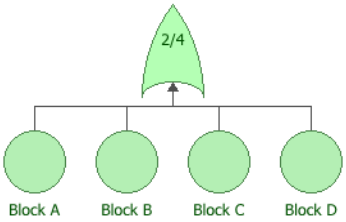
NOR gates are available only in analytical fault trees. There is no equivalent configuration in reliability block diagrams. The inputs to a NOT gate must be fixed probabilities, or evaluate to fixed probabilities; this is checked at analysis, and any input that evaluates to a time-varying model will result in an error.

To configure a NOR gate, only the common block properties are available. No further configuration of the gate is necessary.

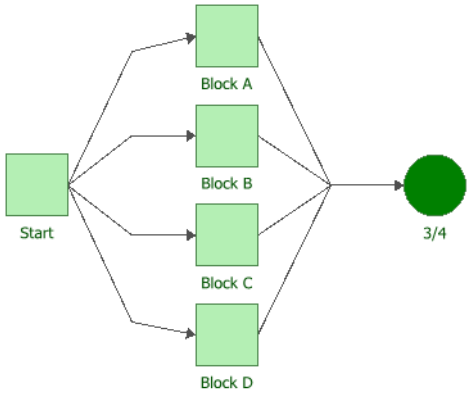
7.2.3.6 Voting Gates

Voting gates allow you to represent a scenario in which the output event occurs if some quantity (k) or more of the input events occur. In system reliability, this implies that if any *k-out-of-n* components fail (input events), the system will fail (output event). In a reliability block diagram, this is similar to a *k-out-of-n* parallel configuration. That is, a fault tree looks at the *k-out-of-n* failures for the system failure while an RBD looks at the *k-out-of-n* successes for system success.

The next two figures show both the fault tree diagram and reliability block diagram representation of a system with four components, A, B, C and D, in which the system fails if any two of the components fail. In a fault tree, this is modeled using a voting gate with a vote number of 2.



This is represented by the RBD shown next. In this configuration, the system will not fail if three out of four components are operating, but will fail if more than one fails.

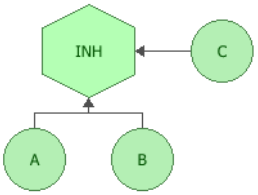


To configure a voting gate, in addition to the common block properties, you will need to use the **Vote number** field to specify the number of input events required to trigger the output event.

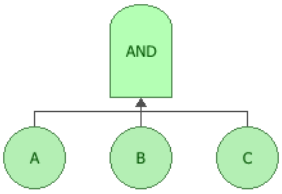
7.2.3.7 Inhibit Gates

Inhibit gates allow you to represent a scenario in which the output event occurs if all input events occur and an additional conditional event (typically an event external to the configuration represented by the fault tree) also occurs. An inhibit gate is basically an AND gate with an additional event. An inhibit gate does not really provide any additional modeling capabilities, but is used to illustrate the fact that an additional event must also occur.

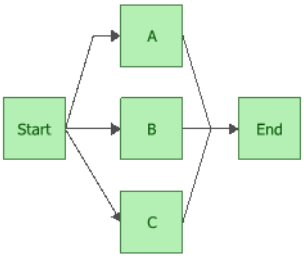
The next figures show fault tree diagram representations of a system in which events A and B must occur as well as a third event, C (the "conditional event"), in order for the system to fail. Traditional fault tree diagrams have the conditional event drawn to the side and the AND gate drawn as a hexagon, as shown next.



The following figure uses an AND gate with three events, A, B and C. Note that analytically, this representation does not differ from the first figure.



The next figure shows the reliability block diagram representation of the previous fault tree diagrams.

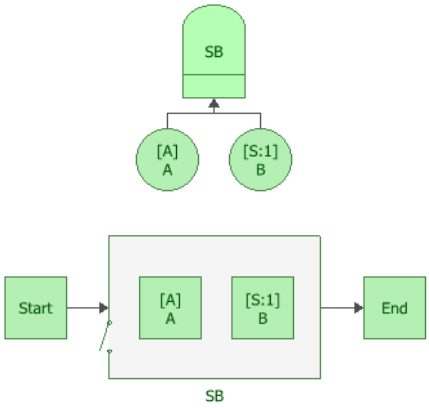


To configure an inhibit gate, in addition to the common block properties, you will need to specify the conditional event. The **Conditional event** field provides a drop-down list that contains all of the events that are connected to the inhibit gate. You can select which of the events is the additional conditional event (typically an event external to the configuration represented by the fault tree). When you specify the conditional event, it will automatically be moved to the side of the inhibit gate, as shown in the first configuration depicted above.

7.2.3.8 Standby Gates

Standby gates in fault trees behave just like standby containers in RBDs. The blocks connected to a standby gate are called dependent events and are similar to contained standby blocks in an RBD in that one or more of the dependent events are active while one or more are dormant (i.e., standby or quiescent). Also, just like standby containers, standby gates act as switches that can fail and define the number of active blocks whose failures would cause system failure (or the active vote number required). Additional gates are not allowed below a standby gate.

The next two figures show both the fault tree diagram and reliability block diagram representation of a system with two units, A and B, in a standby configuration where unit A is active and unit B is in a standby state.



To configure a standby gate, in addition to the common block properties, you will need to specify the **Active Vote Number**, which is the number of events leading to the gate that must succeed in order for the gate to succeed. If the required number of events does not succeed, then the gate is considered to be failed.

Additional options are available for the standby gate, including:

- **Set block as failed** if selected, indicates that the gate is failed (i.e., the block is "off" or absent from the system). An X will be displayed on the block to indicate that it is failed. The block will be considered to be failed throughout the entire analysis/simulation and no maintenance actions will be performed (i.e., any failure and maintenance properties will be ignored). This option can be used for "what-if" analyses to investigate the impact of a block on system metrics such as reliability, availability, throughput, etc. You can also set this option by selecting the block in the diagram and choosing **Fault Tree > Settings > Set Block as Failed**.

  When this option is selected, no other properties will be available for the block; note, however, that any properties you have already specified are simply hidden because they are not relevant. The settings will reappear if you clear the **Set block as failed** option.

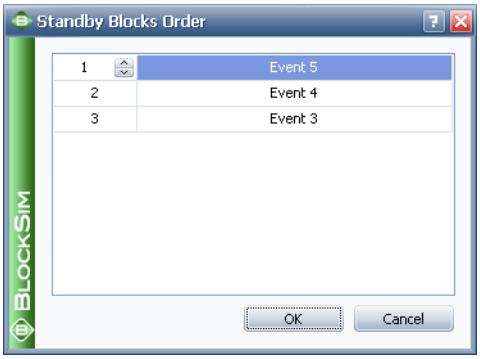
- **Adjust numerical convergence settings** opens the Algorithm Setup window, which allows you to modify the parameters of the numerical integration that is used in analyzing the gate, as well as to specify a level of accuracy that should be reached during calculations. This option is available only for analytical diagrams. *For more information about the container algorithm setup, see Chapter 23*.

- The **Switch** field allows you to create or select a switch resource (*described in Section 7.1.7.2 on page 224*), which defines the standby gate as a switch that is necessary to transfer the activity from active to standby blocks.

The switch resource includes information on the reliability of the switch itself and of the switch action, any restarts and any maintenance tasks that apply to the switch.

If a standby gate has more than one dependent event in a standby state, you can specify the order in which the standby events are used. To do this, select the standby gate and choose **Fault Tree > Properties > Standby Settings > Set Standby Orders** or click the **Set Standby Orders** icon in the Block Properties window for the standby gate.

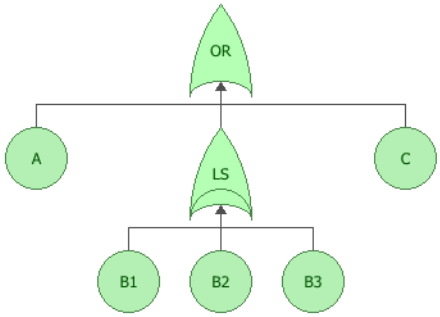
In the Standby Blocks Order window that appears, the standby events are displayed in the order in which they will be used. Select an event and use the up and down arrows that appear in its order number cell to move the event up and down the list. For example, in the window shown next, you would click the down arrow once to move Event 5 down one position; the result would be that Event 4 would be used first, then Event 5, then Event 3.



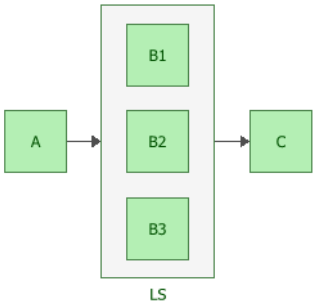
7.2.3.9 Load Sharing Gates

Load sharing gates behave just like load sharing containers in that they define the load and the number of paths required (*k-out-of-n* vote). The blocks connected to a load sharing gate are called dependent events and are similar to contained load sharing blocks in an RBD. Additional gates are not allowed below a load sharing gate.

The next two figures show a fault tree diagram using a load sharing gate and the equivalent reliability block diagram. To understand these diagrams, remember that the fault tree looks at failure combinations while the RBD looks at success combinations. Thus, the fault tree shows a system that will fail if component A fails, component C fails or three out of the three B components fail.



In the case of the RBD, the system will succeed if component A functions, at least one of the three B components functions and component C functions.



To configure a load sharing gate, in addition to the common block properties, you will need to specify the **Vote number**, which is the number of events that lead to the gate that must succeed in order for the gate to succeed. If the number of events does not succeed, then the gate is considered to be failed.

Additional options are available for the load sharing container, including:

- **Set block as failed** if selected, indicates that the gate is failed (i.e., the block is "off" or absent from the system). An X will be displayed on the block to indicate that it is failed. The block will be considered to be failed throughout the entire analysis/simulation and no maintenance actions will be performed (i.e., any failure and maintenance properties will be ignored). This option can be used for "what-if" analyses to investigate the impact of a block on system metrics such as reliability, availability, throughput, etc. You can also set this option by selecting the block in the diagram and choosing **Fault Tree > Settings > Set Block as Failed**.

  When this option is selected, no other properties will be available for the block; note, however, that any properties you have already specified are simply hidden because they are not relevant. The settings will reappear if you clear the **Set block as failed** option.

- **Adjust numerical convergence settings** opens the Algorithm Setup window, which allows you to modify the parameters of the numerical integration that is used in analyzing the gate, as well as to specify a level of accuracy that should be reached during calculations. This option is available only for analytical diagrams. *For more information about the container algorithm setup, see Chapter 23*.

7.2.3.10 Priority AND (PAND) Gates

With a priority AND (PAND) gate, the output event occurs if all input events occur in a specific sequence. This differs from the sequence enforcing gate in the fact that the input events can occur out of sequence (i.e., they are not constrained to occur in a specific sequence), but the output event only occurs if the sequence is followed. This would be equivalent to an RBD configuration using a standby container where the standby units have a quiescent failure distribution (most often the same as the active failure distribution) and/or there is a probability of failure for the switch. Note that it is most common for the switch to be considered 100% reliable, which is the default configuration in BlockSim.

To illustrate this, consider the case of two motors in a standby configuration with motor A being the primary motor and motor B in standby. If motor A fails, then the switch (which can also fail) activates motor B. Thus, the system will fail if motor A fails and the switch fails to switch, or if the switch succeeds but motor B fails subsequent to the switching action. In this scenario, the events must occur in the order noted; however, it is possible for the switch or motor B to fail (in a quiescent mode) without causing a system failure, if A never fails.

The blocks connected to a priority AND gate are called dependent events and are similar to contained standby blocks in an RBD in that one or more of the dependent events are active while one or more are dormant (i.e., standby or quiescent). Also, just like standby containers, priority AND gates act as switches that can fail and define the number

of active blocks whose failures would cause system failure (or the active vote number required). Additional gates are not allowed below a sequence enforcing gate.

To configure a priority AND gate, in addition to the common block properties, you will need to specify the **Active vote number**, which is the number of events leading to the gate that must succeed in order for the gate to succeed. If the required number of events does not succeed, then the gate is considered to be failed.

Additional options are available for the priority AND gate, including:

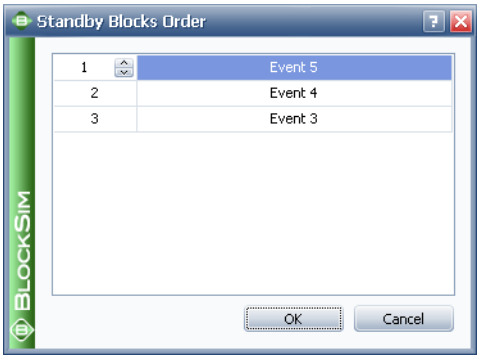
- **Set block as failed** if selected, indicates that the gate is failed (i.e., the block is "off" or absent from the system). An X will be displayed on the block to indicate that it is failed. The block will be considered to be failed throughout the entire analysis/simulation and no maintenance actions will be performed (i.e., any failure and maintenance properties will be ignored). This option can be used for "what-if" analyses to investigate the impact of a block on system metrics such as reliability, availability, throughput, etc. You can also set this option by selecting the block in the diagram and choosing **Fault Tree > Settings > Set Block as Failed**.

  When this option is selected, no other properties will be available for the block; note, however, that any properties you have already specified are simply hidden because they are not relevant. The settings will reappear if you clear the **Set block as failed** option.

- The **Switch** field allows you to create or select a switch resource (*described in Section 7.1.7.2 on page 224*), which defines the standby gate as a switch that is necessary to transfer the activity from active to standby blocks. The switch resource includes information on the reliability of the switch itself and of the switch action, any restarts and any maintenance tasks that apply to the switch.

If a priority AND gate has more than one dependent event in a standby state, you can specify the order in which the standby events are used. To do this, select the gate and choose **Fault Tree > Properties > Standby Settings > Set Standby Orders** or click the **Set Standby Orders** icon in the Block Properties window for the gate.

In the Standby Blocks Order window that appears, the standby events are displayed in the order in which they will be used. Select an event and use the up and down arrows that appear in its order number cell to move the event up and down the list. For example, in the window shown next, you would click the down arrow once to move Event 5 down one position; the result would be that Event 4 would be used first, then Event 5, then Event 3.



7.2.3.11 Sequence Enforcing (SEQ) Gates

A sequence enforcing (SEQ) gate indicates that component failures can occur only in the order specified by its input events. It is a variation of an AND gate in which each item must happen in sequence. In other words, events are constrained to occur in a specific sequence and the output event occurs if all input events occur in that specified

sequence. This is identical to a cold standby configuration (i.e., k units in standby with no quiescent failure distribution and no switch failure probability).

The blocks connected to a sequence enforcing gate are called dependent events and are similar to contained standby blocks in an RBD in that one or more of the dependent events are active while one or more are dormant (i.e., standby or quiescent). Also, just like standby containers, sequence enforcing gates act as switches that can fail and define the number of active blocks whose failures would cause system failure (or the active vote number required). Additional gates are not allowed below a sequence enforcing gate.

To configure a sequence enforcing gate, in addition to the common block properties, you will need to specify the **Active vote number**, which is the number of events leading to the gate that must succeed in order for the gate to succeed. If the required number of events does not succeed, then the gate is considered to be failed.

Additional options are available for the sequence enforcing gate, including:

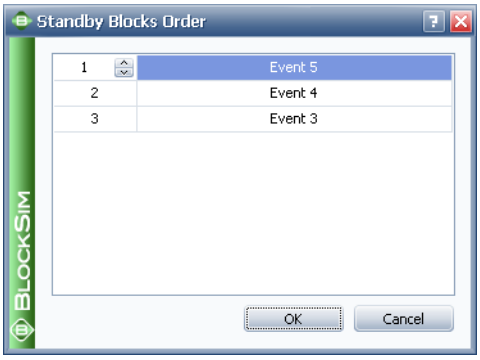
- **Set block as failed** if selected, indicates that the gate is failed (i.e., the block is "off" or absent from the system). An X will be displayed on the block to indicate that it is failed. The block will be considered to be failed throughout the entire analysis/simulation and no maintenance actions will be performed (i.e., any failure and maintenance properties will be ignored). This option can be used for "what-if" analyses to investigate the impact of a block on system metrics such as reliability, availability, throughput, etc. You can also set this option by selecting the block in the diagram and choosing **Fault Tree > Settings > Set Block as Failed**.

  When this option is selected, no other properties will be available for the block; note, however, that any properties you have already specified are simply hidden because they are not relevant. The settings will reappear if you clear the **Set block as failed** option.

- The **Switch** field allows you to create or select a switch resource (*described in Section 7.1.7.2 on page 224*), which defines the standby gate as a switch that is necessary to transfer the activity from active to standby blocks. The switch resource includes information on the reliability of the switch itself and of the switch action, any restarts and any maintenance tasks that apply to the switch.

If a sequence enforcing gate has more than one dependent event in a standby state, you can specify the order in which the standby events are used. To do this, select the gate and choose **Fault Tree > Properties > Standby Settings > Set Standby Orders** or click the **Set Standby Orders** icon in the Block Properties window for the gate.

In the Standby Blocks Order window that appears, the standby events are displayed in the order in which they will be used. Select an event and use the up and down arrows that appear in its order number cell to move the event up and down the list. For example, in the window shown next, you would click the down arrow once to move Event 5 down one position; the result would be that Event 4 would be used first, then Event 5, then Event 3.



7.2.4 Fault Tree Events

In traditional fault trees, different shapes are used to represent different types of events. These shapes convey additional visual information for the analysis. However, unlike gates, each type of event in a fault tree is treated the same way analytically. Thus, BlockSim does not distinguish between event types during analysis or simulation; the types are purely informational.

In BlockSim, all events can be assigned properties via URDs, just like blocks in reliability block diagrams. This is an enhancement to traditional fault tree events, which generally include only a fixed probability of occurrence and/or a constant failure rate.

BlockSim provides the following event blocks to fit the conventions that are followed in most traditional fault trees. It should be noted that the analysis method is the same for all events; therefore, if you wish to include additional event shapes that are not available, you can use the Block Style window to change the appearance of a block to illustrate a different type of event (**Format > Styles > Block Style**).

Visual Indicator	Event Type	Description
	Basic Event	A basic event (or failure event) is identical to a standard block in a reliability block diagram. Traditionally, a basic event is represented by a circle.
	Undeveloped Event	The diamond representation indicates that the event could have been developed further (i.e., expanded into a separate fault tree), but that it was chosen not to do so for the analysis.
	Trigger Event	A trigger event is an event that can be set to occur or not occur (i.e., it usually has a fixed probability of 0 or 1). It is usually used to turn paths on or off or to make paths of a tree functional or non-functional. Furthermore, the terms failed house and working house have been used to signify probabilities of 0 and 1, respectively. In BlockSim, a trigger event has the same properties as a basic event. To use the block to turn the path off, select the **Set block as failed** option in the Block Properties window. You can also set this option by selecting the block in the diagram and choosing **Fault Tree > Settings > Set Block as Failed**. To use the block to turn the path on, make sure no URD is assigned; this means that the block cannot fail.
	Resultant Event	A resultant event is represented by a square and specifies a result in a fault tree diagram.
	Conditional Event	A conditional event is represented by a rounded square and specifies a condition in a fault tree diagram (i.e., an event, possibly external to the configuration represented by the fault tree, that must take place for the gate to succeed). It can be applied to any gate. For example, this symbol would be appropriate to use for the event that is specified in the **Conditional Event** drop-down list in the Block Properties window for an inhibit gate.

All events have the same properties available for configuration, with the exception of dependent events. In addition to the common block properties, you will need to describe how the event behaves. These properties are the same as those available for configuring standard blocks in RBDs except that you cannot define throughput properties, as throughput is not calculated in fault trees.

An event can have only one output event, and no input events or gates.

Dependent Events

A dependent event is any event that leads to a standby, load sharing, priority AND or sequence enforcing gate. A dependent event has the appearance of the original event. For example, if you connect a conditional event to a load sharing gate, it will have the appearance of a conditional event. The only change will be the properties that are available in the Block Properties window.

For a dependent event leading to a standby, priority AND or sequence enforcing gate, the reliability/maintenance characteristics correspond to those of the contained standby block in an RBD. For a dependent event leading to a load sharing gate, the reliability/maintenance characteristics correspond to those of the contained load sharing block in an RBD.

7.3 Mirroring (Using Blocks in Multiple Locations)

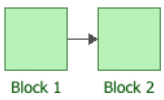
Mirrored blocks allow you to place the same block in more than one location. In other words, mirrored blocks allow you to represent a single component more than once in a diagram or in multiple diagrams within a project. This is useful, for example, when simulating bi-directional paths within the diagram or if the same item performs more than one function and each function has a different reliability-wise impact on the system. The ability to mirror across diagrams allows you to represent situations where a single component's failure affects multiple subsystems that are represented in separate subdiagrams.

Note: For information about using mirror groups for common cause failures in FMRAs, see page 452 in Chapter 16.

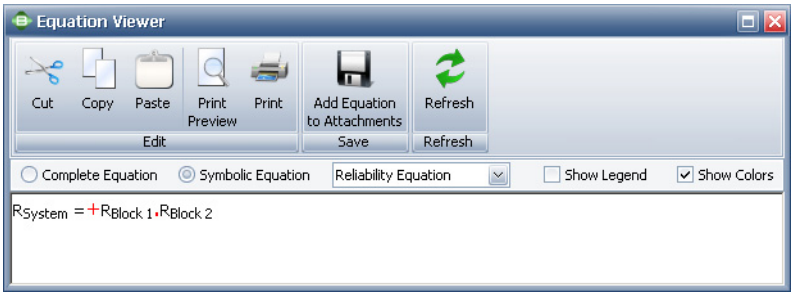
Mirroring is accomplished by adding blocks to a mirror group. Mirror groups are resources that can be shared among analyses and can be managed via the Resource Manager. The mirrored blocks are treated as fully equivalent multiple instances of a single block, rather than considered as an original and copies.

What's Changed? In BlockSim 7, mirroring was accomplished by choosing one block as the original (or "target") and pointing all of the copies (or "mirrored") blocks to that original block. The mirror groups used in the Synthesis version simplify management of mirrored blocks by allowing you to see all instances of the block in the Mirror Group Manager.

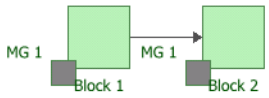
Any changes made to the properties of a block in a mirror group will apply to all other blocks in the mirror group. For analytical diagrams, mirrored blocks have a significant effect on the system reliability equation. Consider a simple RBD like the one shown next.

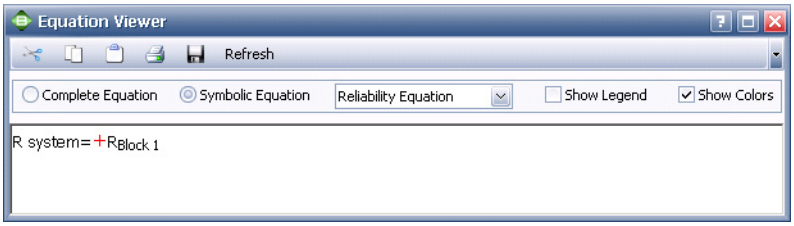


The system reliability equation for this diagram is shown next.



If, however, the two blocks in the RBD belong to the same mirror group, they are actually a single block. The diagram and system reliability equation are shown next.





When a diagram with mirrored blocks is simulated, every event associated with one block in a mirror group will also apply to all other blocks in the mirror group.

The Mirror Group window allows you to create, view and edit mirror groups. It can be accessed from the Mirror Groups page of the Resource Manager by choosing **Home > Edit > New**, by selecting a mirror group and choosing **Home > Edit > View** or by double-clicking a mirror group.

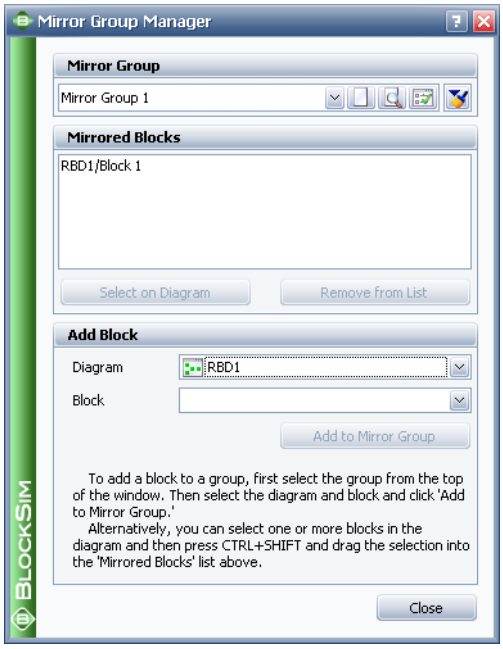
At the top of the window, you can specify the mirror group name. You can replace the default name with your own name of up to 150 characters, if desired. Users with the "Manage other repository settings" permission can change the default names by choosing **File > Manage Repository > Define Default Names**.

You can also enter comments about the mirror group in this window.

For an existing resource, a link at the bottom of the properties window indicates how many times it is currently being used. If you need more information, click the link or the **Trace Usage** icon.

This opens the Dependency Viewer, which allows you to explore both where the resource is being used and what other resources are associated with it.
To add blocks to a mirror group, choose **Project > Data Management > Mirror Group Manager**.

The Mirror Group Manager will appear, as shown next.



The **Mirror Group** area allows you to choose an existing mirror group from the drop-down list. In addition, you can click the **Select Existing** icon to open the Select Resource window. If there is not an existing mirror group that meets your needs, you can create a new one by clicking the **Create New** icon to open a window that allows you to specify a name for the new mirror group, as well as any comments. Click the **View** icon to view the characteristics of the selected mirror group.

You can also click the **Indicator Style** icon to modify the appearance of the corner indicator that will appear on each block that belongs to the mirror group, including:

- The indicator's background color, fill color and fill style. The background color is the underlying color applied to the indicator; the fill color is applied over the background color in the pattern specified by the fill style.
- The style (e.g., solid, dash, etc.), color and thickness of the indicator border.

The **Mirrored Blocks** area displays all of the blocks in the mirror group, in the format [Diagram Name]\[Block Name].

- To add a block to the mirror group, you can select the diagram and block in the **Add Block** area and click the **Add to Mirror Group** button. You also can select the block(s) in the diagram, press **CTRL+SHIFT** and drag the block to the list in the **Mirrored Blocks** area of the Mirror Group Manager.
- To view a mirrored block, select the block in the list and click the **Select on Diagram** button. The diagram will be displayed with the block selected.
- To remove a block from the mirror group, select the block in the list and click the **Remove from List** button. The block's properties will not change, but it will no longer be part of the mirror group. Subsequent changes to, or simulation of, other blocks in the mirror group will not affect it.

Note: Adding blocks to or removing blocks from a mirror group changes the configuration of the diagram(s) in which the blocks are located, necessitating reanalysis or resimulation.

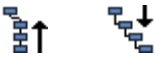
By default, a mirrored block in a diagram is identified by a gray box at the lower left corner of the block, and the name of the mirror group is displayed by the box. These settings may be changed via the Block Corner Indicators pages of the Diagram Style window (**Format > Styles > Diagram Style**).

7.4 Block Details

The Block Details window displays the details of the blocks and resources in the current RBD, fault tree or phase diagram. To access the Block Details window from within a diagram, choose **[Diagram/Fault Tree/Phase] > Selection > Diagram Actions > Show Block Details**.

Click an item in the navigation panel on the left to display a complete hierarchical list of its properties in the area on the right. When a resource is shown in the area on the right (i.e., when a resource is part of the properties of the item that is currently selected in the navigation panel), you can click the + beside it to view its properties.

You can collapse or expand all nodes in the properties hierarchy using the **Collapse All** and **Expand All** icons.



You can select the **Show resources only** check box at the bottom of the window to hide all properties except for the resources. In the case of blocks, this hides all block properties except for the resources that the block uses; for resources, it hides the identifier fields.

You can click the **Send to Workbook** icon to add a new analysis workbook to the project and save the results in it.

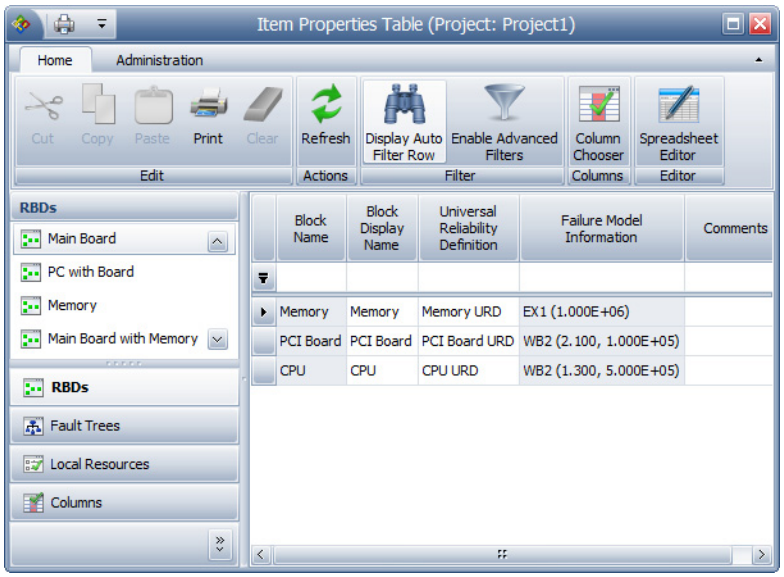
You can click the **Send to Excel** icon to save the results as an Excel file.

7.5 Item Properties Table

The Item Properties Table, accessed by choosing **Project > Data Management > Item Properties Table**, is designed to support batch editing for blocks and resources used in or available to BlockSim diagrams within the current project.

This provides you with another way of viewing and editing properties without having to open the properties windows for each item. Any changes made to properties in the Item Properties Table will be applied to the corresponding

properties window and any changes made to properties in a properties window will be applied to the Item Properties Table. The following picture shows an example of the Item Properties for a particular project.



The Item Properties Table consists of a navigation panel on the left side of the window and the table area on the right side of the window. The navigation panel contains pages that allow access to all of the RBDs, fault trees and phase diagrams in the project, as well as the following local resource types:

- URDs
- Variables
- Models
- Tasks
- Task packages
- Crews
- Spare part pools
- Maintenance groups
- Switches
- Mirror groups
- Maintenance templates

Note: Global resources are not shown in the Item Properties Table.

For information on configuring the table and limiting the resources shown, see pages 177 and 177 in Chapter 5.

The columns in the table will vary depending on the type of item you are working with. For instance, the table for a reliability block diagram shows the Block Name, Block Display Name, Universal Reliability Definition, Failure Model Information and Comments columns by default. The table for universal reliability definitions shows the URD Name, Failure Model, Failure Model Information, Corrective Task, Scheduled Tasks and Comments columns. However, you can specify which columns will appear in the table by using the **Column Chooser** command on the Ribbon.

To edit the information for the blocks or resources, click inside a cell and enter the desired information. To accept your input, click outside the cell. You can also press **TAB** to accept the input and move the focus to the next cell to the right or **ENTER** to accept the input and move the focus to the next cell down.

You can also use standard cut, copy, paste and clear commands within the table, including when you have multiple cells selected.

Some columns, such as Start Age and Duty Cycle, require numerical inputs; if the input must be within a certain range, the software will check that your input meets the requirement and, if it does not, will show a message explaining the requirement. Other columns require specialized types of input; for example, the URD column for blocks requires that you specify a URD that exists in the project, while the Item Operates Even if System Down column requires a yes/no answer. These columns provide a drop-down list of valid choices. To edit the Comments column, double-click inside the cell to open an input box.

If the property does not apply to the corresponding block or resource, "N/A" will appear in the column and the cell will have a gray background. Properties that cannot be edited via the Item Properties Table, such as the block name, will also have a gray background.

IMPORTANT: If you change the information in one column, it may affect the information required in another column. For example, if you are editing a model and you change the Model Type column from "Weibull-2" to "Weibull-3," then the Parameter 3 column will change from "N/A" to requiring a numerical input for the third parameter for the Weibull distribution (i.e., gamma).

Models created by publishing results from an analysis in a Synthesis application cannot be edited in the Item Properties Table. To edit a model that was published from an analysis, you must return to the original data source, make the necessary changes, recalculate and republish the model.

Since the required parameters for a model will vary depending on the model type, the way you use the parameter columns in the Item Properties Table will vary. The parameter columns in the Item Properties Table for each model type are presented in the table shown next.

Model Type	Parameter 1	Parameter 2	Parameter 3	Parameter 4	Parameter 5	Parameter 6	Parameter 7	Parameter 8	Parameter 9	Parameter 10	Parameter 11	Parameter 12
Fixed Reliability	Reliability											
Fixed Unreliability	Unreliability											
Fixed Duration	Duration											
Weibull-2	Beta	Eta										
Weibull-3	Beta	Eta	Gamma									
Mixed Weibull-2	Population 1 Beta	Population 1 Eta	Population 1 Portion	Population 2 Beta	Population 2 Eta	Population 2 Portion						
Mixed Weibull-3	Population 1 Beta	Population 1 Eta	Population 1 Portion	Population 2 Beta	Population 2 Eta	Population 2 Portion	Population 3 Beta	Population 3 Eta	Population 3 Portion			
Mixed Weibull-4	Population 1 Beta	Population 1 Eta	Population 1 Portion	Population 2 Beta	Population 2 Eta	Population 2 Portion	Population 3 Beta	Population 3 Eta	Population 3 Portion	Population 4 Beta	Population 4 Eta	Population 4 Portion
Normal	Mean	Std										
Lognormal	Log-Mean	Log-Std										
Exponential-1	Mean Time or Lambda											
Exponential-2	Mean Time or Lambda	Gamma										
Generalized Gamma	Mu	Sigma	Lambda									
Gamma	Mu	K										
Logistic	Mu	Sigma										
Loglogistic	Mu	Sigma										
Gumbel	Mu	Sigma										
Arrhenius-Weibull	Beta	B	C									
Arrhenius-Lognormal	Log-Std	B	C									
Arrhenius-Exponential	B	C										
Eyring-Weibull	Beta	A	B									
Eyring-Lognormal	Log-Std	A	B									
Eyring-Exponential	A	B										
IPL-Weibull	Beta	K	n									
IPL-Lognormal	Log-Std	K	n									
IPL-Exponential	K	n										

The changes you make in the Item Properties Table will be saved when you close the window. You can also choose **Home > Actions > Refresh** to commit your changes immediately and refresh the Item Properties Table to display the most current information, including recent changes by other users.

Editing Multiple Items

You can choose the **Spreadsheet Editor** command on the Ribbon to open the Spreadsheet Editor, which displays the blocks or resources in a spreadsheet-based view. This enables you to use the capabilities inherent in a spreadsheet (e.g., batch copying, formulas, etc.) to make editing multiple items easier. When you are finished editing, choose **Close > OK** to save your changes and close the Spreadsheet Editor or **Close > Cancel** to close the Spreadsheet Editor without saving your changes.

Exporting Information from the Item Properties Table

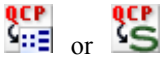
To print the contents of the Item Properties Table, choose the **Print** command on the Ribbon.

To send the contents of the Item Properties Table to an analysis workbook or to an Excel worksheet, click the **Administration** tab on the Ribbon and choose the **Transfer Report** command.

In the Transfer Report window that appears, you can select to send either the current item (i.e., the table for the currently displayed page) or the full report (i.e., the tables for all pages in the Item Properties Table) to an analysis workbook or to Excel. If you transfer the full report, each table will appear on a new tab in the analysis workbook or the Excel workbook.

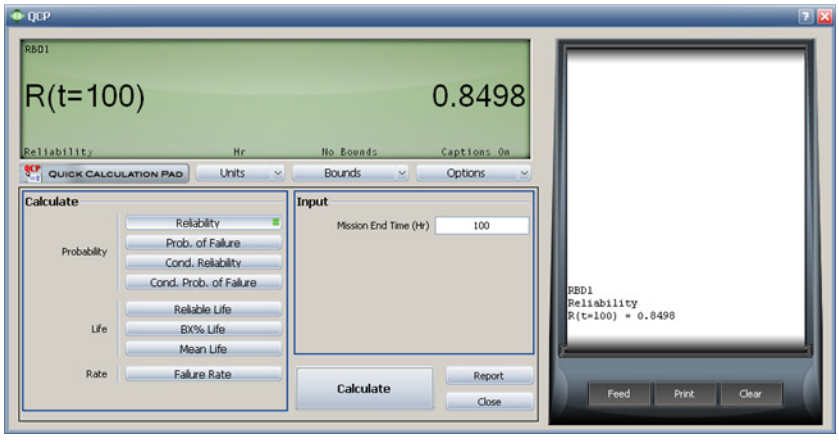
7.6 Quick Calculation Pad (QCP)

The Quick Calculation Pad (QCP) provides a convenient way of calculating a variety of useful metrics. You can access the tool by clicking the QCP icon on the Analytical or Simulation page of the diagram control panel.



or

The following example shows the QCP for analytical diagrams.



This section provides a general description of the QCP and how to use it in most analyses. For more focused information about the types of calculations that can be performed for an analytical diagram, see page 270 in Chapter 8. For information about the types of calculations available for simulation diagrams and phase diagrams, see page 326 in Chapter 9.

How to use the QCP

To use the QCP, do the following:

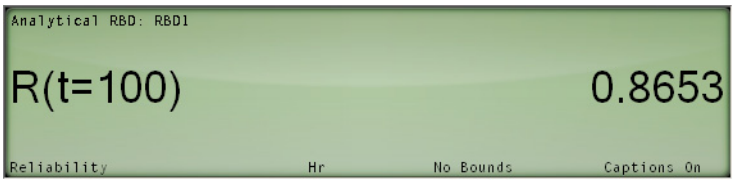
- Choose a metric in the **Calculate** area.
- If applicable, use the **Units** drop-down list to specify the units for time values that are entered as inputs and/or displayed as results. The units in the QCP can be different from the units that were used for the diagram. For example, you could enter the age of the block in hours but then calculate the reliability for 1 year of operation — the application will convert the times automatically based on the conversion factors specified for the database.
- For analytical diagrams, you can use the **Bounds** drop-down list to specify what type of confidence bounds to calculate. The following graphics illustrate the types. The ReliaWiki resource portal provides more information on the background theory of confidence bounds at: http://www.ReliaWiki.org/index.php/Confidence Bounds.

If you select **Both One-Sided**, the QCP will calculate both the lower one-sided bound and the upper one-sided bound.

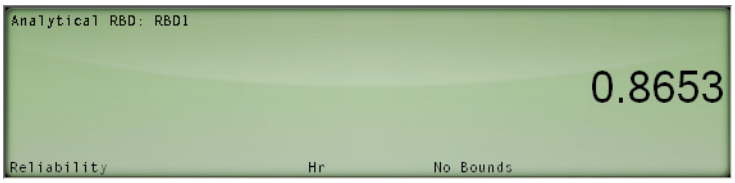
> **Note:** Confidence bounds are available only in cases where all blocks that can fail in the analytical diagram use models that have Fisher matrix bounds. To have Fisher matrix bounds, a model must have been created by publishing from a data source, such as a Weibull++ or ALTA data sheet, that was calculated using Fisher matrix bounds. Additionally, bounds are not available if the diagram has containers, subdiagrams or mirrored blocks, or if any block uses the cumulative damage model.

- Make any required inputs in the **Input** area.
- Click **Calculate**. The calculated value(s) will always be displayed in the results area at the top of the window, and clicking the **Report** button displays a summary of the current calculation input/output in the Results window. You can also use the **Options** drop-down list to configure other settings, including:
  - **Precision** sets the number of decimal places displayed in the results.
  - **Scientific Notation** sets the point at which numbers will be converted to normalized scientific notation. For example, setting this to 3 means that all numbers with a value of 1,000 or more will be converted to normalized scientific notation (e.g., 1.0E+3).
  - **Set Display Font** allows you to change the font style and size that is used to display the calculated value(s) in the results.
  - Select **Captions** if you want additional information to be displayed in the results area along with the values (a green light in the button indicates that this option is selected). For example, if you have selected to calculate

the reliability for a given period of time, the first picture shows the display with captions and the second picture shows without.



QCP with captions on.



QCP with captions off.

- Select **Show Calculation Log** if you want to display a log on the right side of the window that records some or all of the calculations performed during this QCP session (a green light in the button indicates that this option is selected).
  - If **Auto Print Results** is selected under **Options**, all results will be automatically printed to the log each time you perform a calculation.
  - Otherwise, you can choose which results to add to the log by using the **Print** button at the bottom of the log display.

  Under the log display, you can also click **Feed** to "advance the tape" or **Clear** to delete all data from the current log. If you click inside the log, you can copy some or all of the text to the Clipboard.
- Select **Non-Modal QCP** to lock the QCP in a top window position so it can remain open while you have access to all diagrams. The calculations performed in the QCP will be based on the currently active diagram. If this option is not selected, you will need to close the QCP to access any diagram. This setting can also be changed using the **While QCP is open, have access to all folios** option on the Other page of the Application Setup (**File > Application Setup**).

7.7 Publishing Fitted and Analytical Models

To publish the results of a simulation diagram as a model, you must first fit a distribution to the simulation results. The published model will always be based on the fitted distribution. For an analytical diagram, however, you can publish models based on the analytical solution and/or the fitted distribution.

- An *analytical model* is based on the analytical solution. This is essentially a single reliability model object that references all items in the diagram, as well as their dependents. Each reference to a model like this is really a reference to the diagram and its subdiagrams, and uses resources accordingly. The benefit to this approach is that no approximation is needed and any changes made to the models or dependencies of the blocks within it will be automatically incorporated in the model. The drawback is that such models are complex and consume computer memory and resources.
- A *fitted model* is based on the fitted distribution, and yields an approximation of the entire diagram (and its subdiagrams). This results in a significant reduction of complexity, as the entire diagram and its dependents no longer need to be used for any calculations. The drawback is that any changes that are subsequently made to the diagram and/or its dependents are not reflected unless the distribution is refitted and then republished.

Each type of model has its own area, with all associated tools for publishing the model and viewing its properties, on the Publishing page of the analytical diagram control panel. (*For more information on publishing a model, see page 174 in Chapter 5*.)

Removing Associations for Analytical Models

For all published models, you can remove the association between the current analysis and the published model by clicking the **Remove Association** icon.

When you do this, the model will continue to be a resource in the database but it can no longer be synchronized with the original analysis. However, in the case of a published analytical model, removing the association with the diagram would actually remove the model itself, because it is a direct reference to the diagram. Therefore, to remove the association for published analytical models, you must specify a replacement model. If a fitted model exists for the diagram, it will be used as the replacement. If no fitted model exists, you will be prompted to specify the properties for a replacement model.

Similarly, if you delete an analytical diagram with a published analytical model, you must specify a replacement model. In this case, a fitted model must not only exist but also have been published in order to be used as the replacement. If no fitted model has been published, you will be prompted to specify the properties for a replacement model.

7.8 Creating Diagrams from Synthesis Data

The integration of the Synthesis applications makes it easy for you to import data from your analyses in the Synthesis versions of Xfmea, RCM++, RBI or Lambda Predict and create RBDs and/or fault trees that allow you to perform more in-depth calculations. Two possibilities exist:

- You can import items from analyses in other Synthesis applications in the current project, along with their reliability data, and create diagrams that reflect the items' reliability-wise relationships.
  - When importing from Xfmea/RCM++/RBI, you can create either fault trees or RBDs - Section 7.8.1 (p. 252)
  - When importing from Lambda Predict, you can create RBDs - Section 7.8.2 (p. 258)
- You can import records to create a fault tree based on the effects in analyses performed in other Synthesis applications in the current project - Section 7.8.3 (p. 261)

IMPORTANT: These tools are intended to provide a quick and easy way to bring data from other Synthesis applications into BlockSim, and to create diagrams that can be used as starting points for analyses within BlockSim. BlockSim is capable of modeling very complex configurations, and you may wish to make changes to the diagrams before analyzing or simulating them in BlockSim so that your results more accurately represent the real-world configuration of your systems. For example, the FMRA in Xfmea, RCM++ and RBI assumes that all items are in series configurations, so all diagrams created using these tools represent series configurations unless you subsequently modify them.

Note: For blocks and diagrams representing FMEA records (i.e., functions, failures or causes), the names/descriptions are based on the Short Description field for the record.

7.8.1 Building Diagrams from Xfmea/RCM++/RBI

You can create RBDs that represent items from analyses performed in the current project in Xfmea/RCM++/RBI in order to perform system maintainability and availability analysis for these items. To do this, choose **Insert > Build from Synthesis > Build RBDs and FTs from Synthesis**.

The Build RBDs and FTs from Synthesis window consists of two steps. When the window opens, you will see the Options step. Specify that you want to use RCM++/Xfmea/RBI as the data source in the **Synthesis Element to Build From** area. (You can use data from the Synthesis versions of RCM++, Xfmea or RBI.)

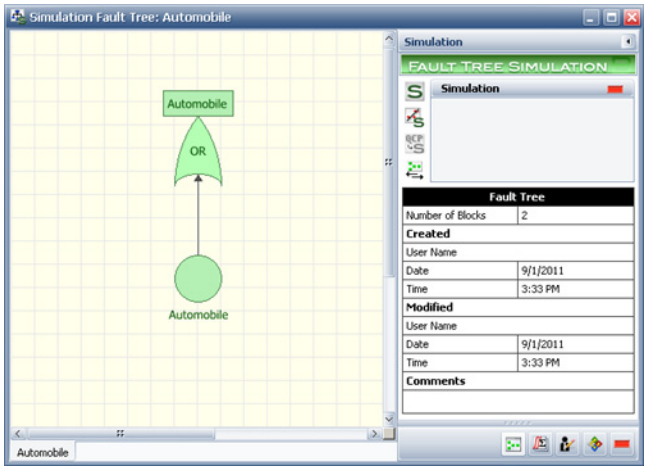
In the **Diagram Type** area:

- Specify whether you want to represent the imported items' reliability-wise relationships using fault trees, RBDs or a hybrid of the two. If you choose the **Hybrid** option, all systems will be imported as RBDs, while FMEA information will be imported as fault trees.
- Choose whether you want the new diagrams to be analytical diagrams or simulation diagrams.
- If you select the **Create folders** check box in this area, each diagram will be added to the current project explorer within its own folder under the Fault Trees or RBDs folder. The new folders will be arranged hierarchically in the same way as the information in the original analysis, thus making their hierarchical relationship visually apparent within the current project explorer.

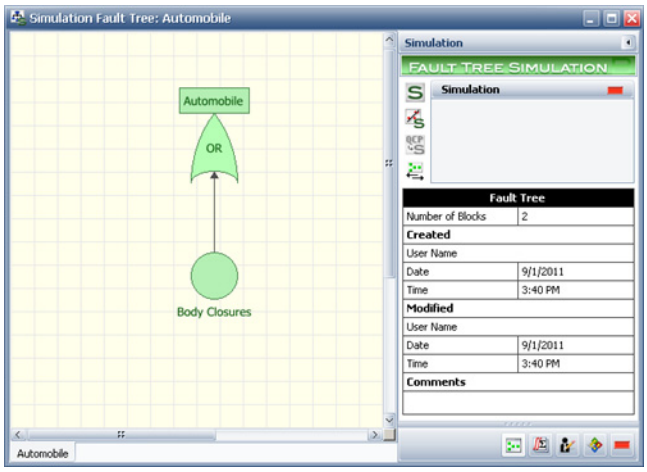
The reliability information for each branch of the FMRA hierarchy is taken from the highest-level item for which reliability is specifically defined, rather than inherited. For example, imagine that you are importing from the following hierarchy and creating fault trees from the information:

Name	RecordType
Automobile	**Item**
Body Closures	Item
Front Door L.H.	Item
Maintain integrity of inner door panel.	Function
Discoloration	Failure
Upper edge of protective wax application specified for inner door panels is too low	Cause
Insufficient wax thickness specified	Cause
Inappropriate wax formulation specified	Cause
Corner design prevents spray equip from reaching all areas	Cause
Insufficient room between panels for spray head access	Cause

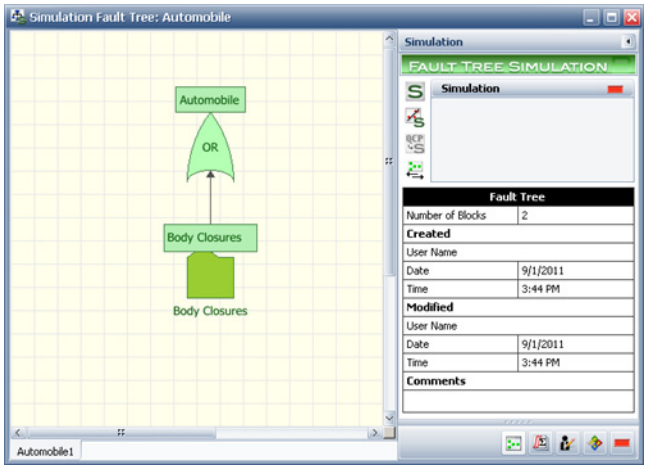
If a URD is assigned to the Automobile item, then only the following diagram will be created:

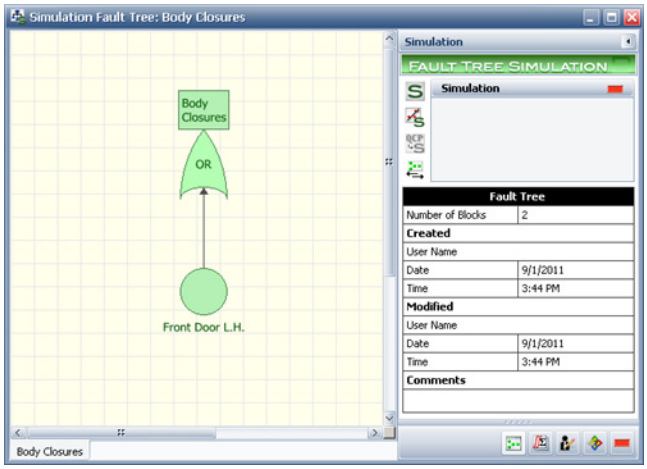


If a URD is assigned to the Body Closures item and the Automobile item is set to inherit its reliability characteristics from that, then the following diagram will be created:

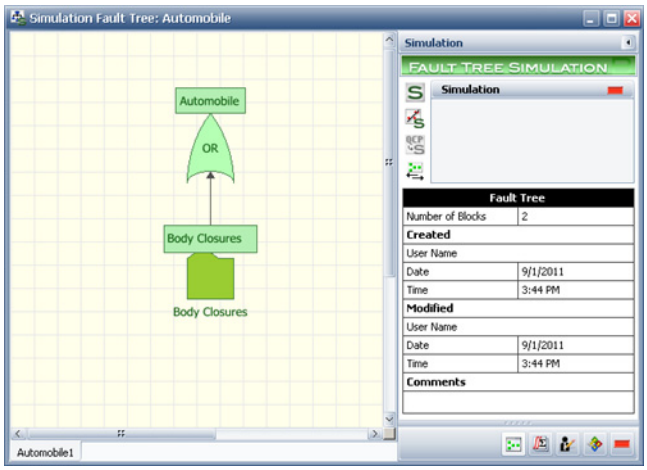


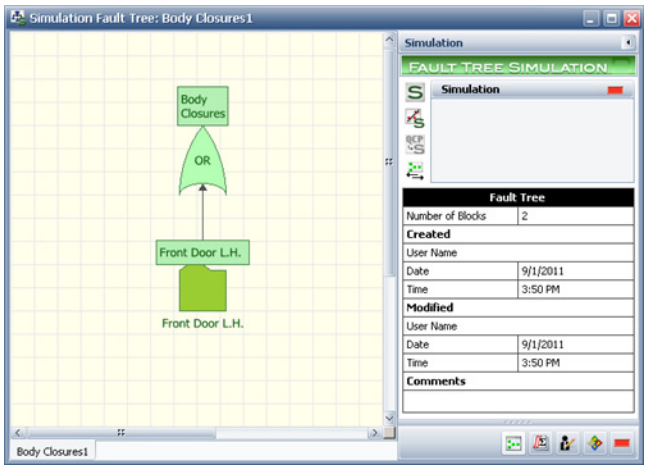
If a URD is assigned to the Front Door L.H. item and the Automobile and Body Closures items are set to inherit their reliability characteristics, then the following diagrams will be created:

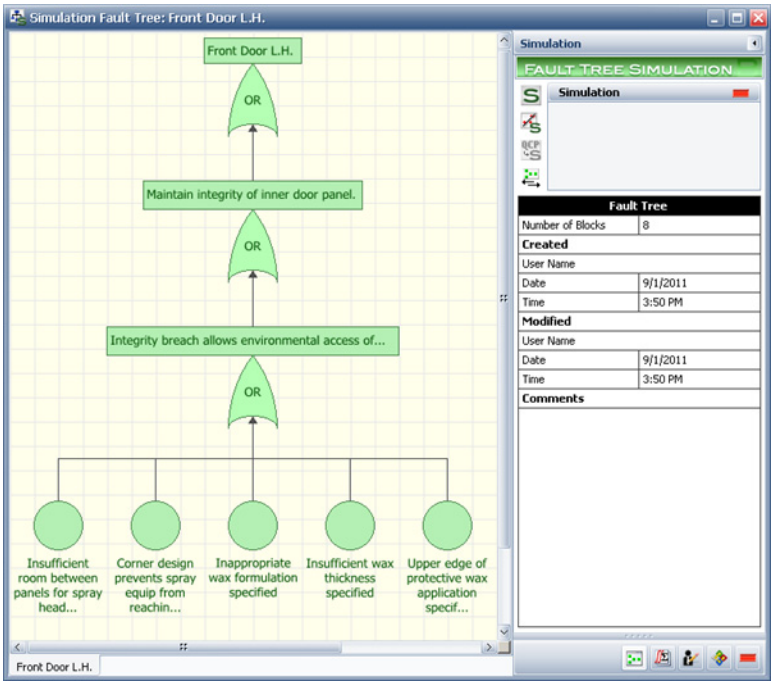




Finally, if all items are set to inherit their reliability characteristics and URDs are assigned only to the causes in the FMEA for the Front Door L.H. item, then the following diagrams will be created:







If you are creating RBDs or a hybrid of RBDs and fault trees, the **Subdiagram Grouping** area will be available. The options in this area allow you to control which levels of analysis will be represented by subdiagrams. For example, consider the final case above, where all system hierarchy items are set to inherit their reliability characteristics and URDs are assigned only to the causes in the FMEA for the Front Door L.H. item.

- If you are creating RBDs, your choices in the **Subdiagram Grouping** area will determine how the system and reliability information is represented across diagrams, as follows:
  - If you select **Create system-level diagrams**, then a diagram will be created for each item in the system hierarchy. The top level (i.e., Automobile) diagram will contain a subdiagram block representing the next level item (i.e., Body Closures) and so on. The diagram representing the lowest level in the system hierarchy (i.e., Front Door L.H.) will contain the FMEA information. This may be shown as causes or additional levels of subdiagrams may be included to represent the function and/or failure, depending on your choices for the next two options.
  - If you select **Create subdiagrams for functions**, then the function level of the FMEA will be represented by a diagram. Note that if you select this check box but do not choose to create subdiagrams for items in the system hierarchy, then the function diagram will be the top level diagram.
  - If you select **Create subdiagrams for failures**, then the failure level of the FMEA will be represented by a diagram. Note that if you select this check box but do not choose to create subdiagrams for items in the system hierarchy, then the failure diagram will be the top level diagram.

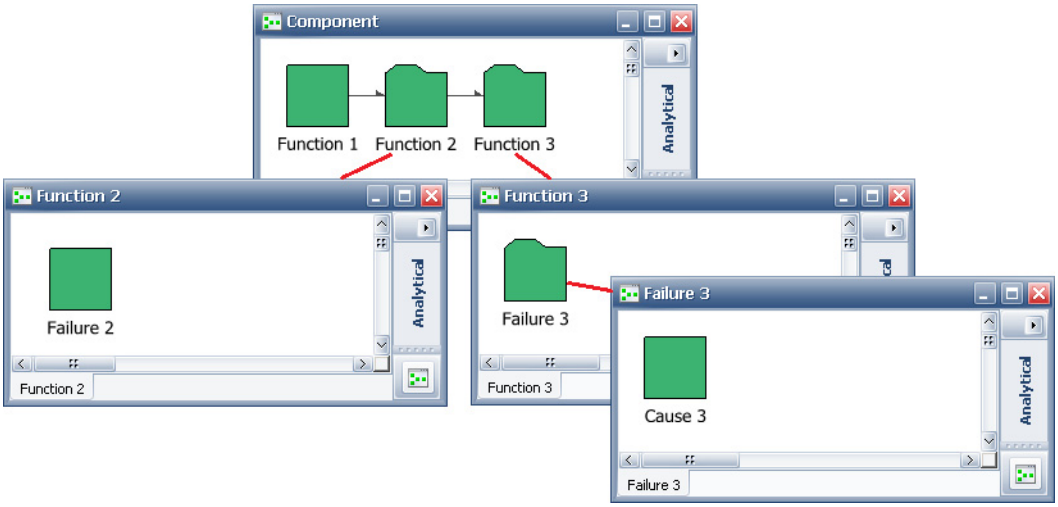
  This is true not only for system hierarchy items, but also for FMEA records. For example, if reliability is defined for a function, that function will be shown as a block in the resulting RBD and no subdiagram will be created for it (i.e., any failures and causes assigned to the functions will not be imported, as they are not relevant to the reliability).

  The **Create subdiagrams for functions** and **Create subdiagrams for failures** check boxes control only how information is organized, not what information is imported. All items with reliability information defined will be

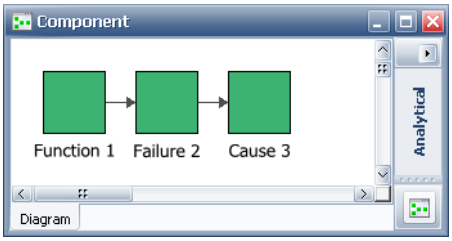
imported, regardless of your selections for these check boxes. For example, consider a component with an FMEA that includes three functions, each of which has a failure and a cause defined. Its FMRA is shown next.

Name	Record Type	Reliability Policy
Component	Item	Inherit
Function 1	Function	Function 1_URD
Function 2	Function	Inherit
Failure 2	Failure	Failure 2_URD
Function 3	Function	Inherit
Failure 3	Failure	Inherit
Cause 3	Cause	Cause 3_URD

As you can see, the reliability is defined at the function level for Function 1, at the failure level for Failure 2 and at the cause level for Cause 3. If you imported this configuration with both the **Create subdiagrams for functions** and **Create subdiagrams for failures** check boxes selected, you would get the diagrams shown next.



On the other hand, if you imported the same configuration with neither of the two check boxes selected, you would get the following diagram:



- If you are creating a hybrid of RBDs and fault trees, only the **Create subdiagrams for functions** check box will be available. In this case, system-level diagrams are always created. Your selection in this area determines whether the fault trees are created at the function level or at the failure level.

When creating either fault trees or RBDs, you can select the **Import items even though overridden by higher level definition** check box to allow you to import lower-level items even if a higher-level item has defined reliability characteristics. Selecting this option causes the reliability of the higher-level item(s) to be set to inherit from the lower-level item(s) for the purposes of the import. The settings will not be changed in the original analysis.

Click the **Next** button to view the Hierarchical Representation step, which displays a hierarchical view that corresponds to the diagram(s) that will be created. You can select or clear items in this tree to specify whether they will be included in the diagram(s). If the tree does not correspond to the diagram(s) you have in mind, you can click the **Back** button to return to the Options step and try again. Otherwise, click **OK** to create the diagram(s).

7.8.2 Building Diagrams from Lambda Predict

You can use the system configuration and failure rate predictions from Lambda Predict as a starting point for creating RBDs in BlockSim. As more information becomes available, you can then replace the predicted failure rate models with new models that are based on Weibull++ analyses and other sources.

Importing the system configuration data from Lambda Predict is a two-step process:

1. First, you must publish the failure rates calculated in Lambda Predict as reliability models. To do this, open the project in Lambda Predict. In the prediction folio, select the standard item (e.g., G, ML, BL, TL, F or MC) or assembly (e.g., or) that you want to create the RBD for, and then choose **Prediction Tools > Share > Publish Branch**.

   Once the models are published, you can close the project in Lambda Predict if desired. (For more general information on publishing failure rate models in Lambda Predict, see "Publishing Models from Failure Rate Predictions" in the Lambda Predict User's Guide.)

2. Open the project in BlockSim, and then choose **Insert > Build from Synthesis > Build RBDs and FTs from Synthesis**.

   When the window opens, specify that you want to use Lambda Predict as the data source in the **Synthesis Element to Build From** area. The utility will provide the following options:

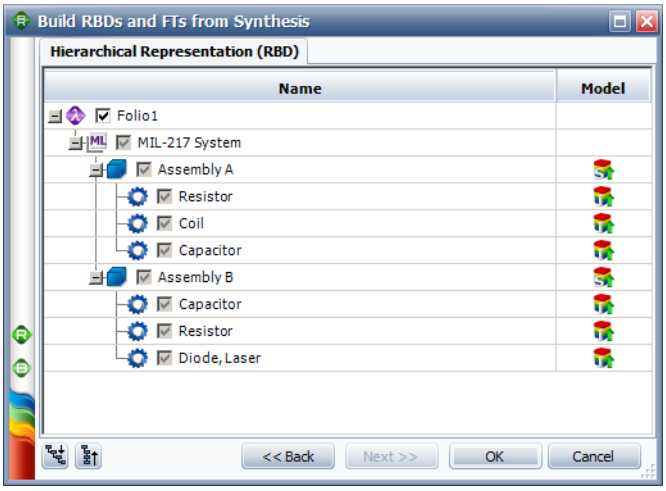
   - **Simulation** or **Analytical** - use the drop-down list to specify whether the new RBD(s) will be analytical diagrams or simulation diagrams.
   - **Create Folders** - select this check box if you want the new diagram(s) to be organized into a separate folder in BlockSim's current project explorer. (Note that selecting this option can make it easier to delete and recreate the diagrams if you need to. You can delete the entire folder instead of deleting each diagram individually.)
   - **Create subdiagrams for blocks** - in this context, the term "blocks" refers to the assemblies in Lambda Predict. For each block in the prediction folio, you can:
     - Use a single RBD block to represent the reliability of the entire assembly. This treats the assembly as a "black box" in the RBD, using the item model that was published for the block.
     - Use a subdiagram in the RBD to represent the assembly. This allows you to consider the separate failure rates for all of the components that make up the assembly.

Note: For many of the reliability prediction standards available in Lambda Predict, a portion of the assembly's failure rate may be calculated at the block level. In Lambda Predict, this portion is called the block's supplemental model. For example, in MIL-217 analyses, the failure rate for a printed circuit board (PCB) assembly is a summation of the individual failure rates of its components plus the failure rate due to the physical connection of the components to the board. If you have chosen to create subdiagrams for such assemblies, the diagrams will contain the reliability models of all the components in the assembly plus the supplemental model that represents the failure rate due to the connections.

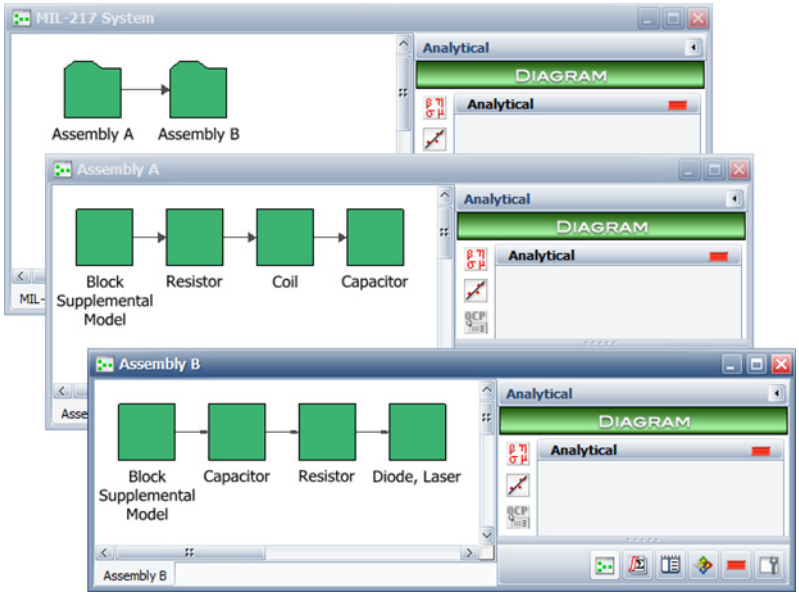
After you have specified your preferences, click **Next** to see the failure rate models that were published from Lambda Predict. As an example, the following pictures show two ways that models published from the same MIL-217 prediction might be displayed, depending on whether the **Create subdiagrams for blocks** option was selected.

In the first scenario, the **Create subdiagrams for blocks** option was selected; therefore, the utility displays all the assemblies and components in the system configuration. The window also displays the supplemental

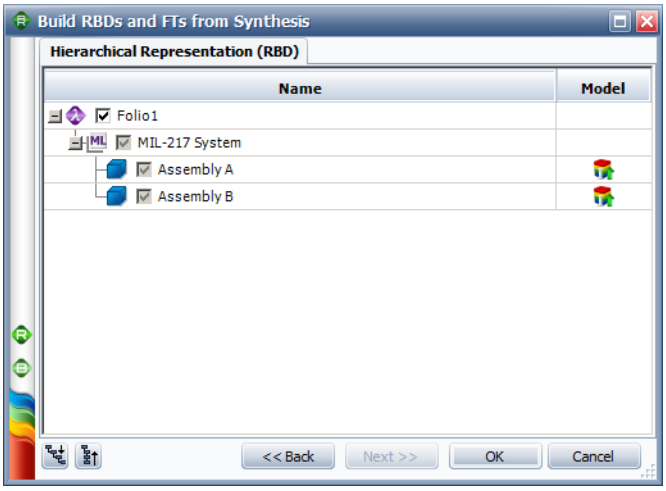
model for each assembly and the item model for each component, as shown next. (Note that the supplemental model icon will be displayed only if it's applicable for the particular assembly and the model has been published from Lambda Predict.)



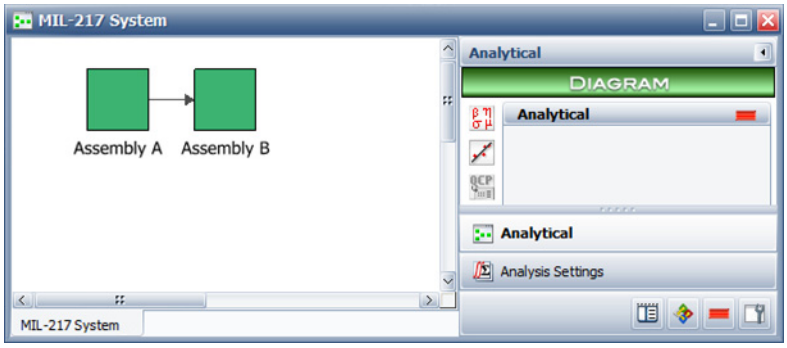
If you select all of the items shown in this hierarchy and click **OK**, BlockSim will create a diagram for the "MIL-217 System" that links to one subdiagram for Assembly A and another for Assembly B, as shown next.



In the second scenario, the **Create subdiagrams for blocks** option was not selected; therefore, the utility displays only the assemblies. In this case, the item model that represents the failure rate calculated for each assembly will be used.



If you select all of the items shown in this hierarchy and click **OK**, BlockSim will create one diagram for the "MIL-217 System" that includes a single block for each assembly, as shown next.



Note the following:

- If you have made changes to the analysis in Lambda Predict since the last time you published the models, it is recommended to use the Publish Branch command to publish them again. If an item does not have a published model, it will not be included in the BlockSim diagram.
- Components with a value greater than 1 in their quantity field in Lambda Predict are represented by multi blocks. *For more information about multi blocks, see page214*.
- Parallel configurations at the assembly level, which Lambda Predict models using redundancy, are modeled in BlockSim using nodes.
- Component adjustment factors defined in Lambda Predict will be imported as block duty cycles. Adjustment factors for assemblies are not imported.

7.8.3 Building Effect Fault Trees from Synthesis

To create a fault tree based on one or more effects in analyses performed in other Synthesis applications in the current project, choose **Insert > Build from Synthesis > Build Effect FTs from Synthesis**.

In the window that appears, you can search for and select the effects that you want to use as the top level in the fault tree.

Specify the data source that you want to use in the **Synthesis Element to Build From** area. You can use data from the Synthesis versions of Xfmea/RCM++/RBI or XFRACAS.

In the **Criteria to Filter Effects** area, choose the text field that you want to search for each effect record. Supply the desired text that you are searching for in the text box, and specify the desired type of match in the middle drop-down list. For example:

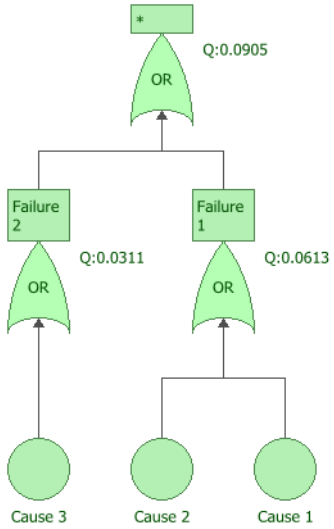
- If you specify that the **Short Description** field **contains** the term **fire**, the search will return all effects whose short description contains the string "fire" in any FMEA within the current project. This is a string search, rather than a complete word search, so the search will also return effects containing words such as "fired" and "misfire."
- If you specify that the **Short Description** field **contains any** of the terms **fire, spark**, the search will return all effects whose short description contains the string "fire" or the string "spark."
- If you specify that the **Short Description** field **contains all** of the terms **fire, spark**, the search will return all effects whose short description contains both the string "fire" and the string "spark."
- If you specify that the **Short Description** field **begins with** the string **corro**, the search will return all effects whose short description begins with that string. This search would return effects such as "Corroded panel" or "Corrosion of panel" but not "Panel is corroded."
- If you specify that the **Short Description** field **is** the term **endangers operator**, the search will return all effects whose short description contains only that exact string.

Note that none of these searches is case sensitive.

The **Select Effects** table updates dynamically to show only the effects that meet the search criteria you have specified. Select the check box for each effect that you would like to include as part of the top level item in the fault tree. You can select or clear the check box in the header area to select or clear the check boxes for all effects in the table.

Click the **Next** button to view a hierarchical representation of the fault tree, which displays the group of effects selected for inclusion with their associated failures and the causes associated with each failure. You can select or clear items in this tree to specify whether they will be included in the fault tree. If the tree does not correspond to the fault tree you have in mind, you can click the **Back** button to return to the Select Effects table and try again. Otherwise, click **OK** to create the fault tree.

The fault tree shows the causes as events, with associated reliability information if it was specified in the original analysis. The cause events are connected to OR gates that represent the failures associated with the causes; these OR gates are connected to the top OR gate, which represents the selected effect(s).



Note: Reliability information for fault trees created in this way is based on the reliability defined at the cause level. Any reliability information defined for items, functions or failures is ignored.

BlockSim Analytical Diagrams

8

BlockSim's reliability block diagrams and fault trees can be configured as analytical diagrams (which use the exact algebraic equation for the model but impose limitations on what can be modeled and what results can be obtained) or as simulation diagrams (which provide more modeling options and results but must be analyzed with discrete event simulation).

In the analytical (or algebraic analysis) approach, the system's *pdf* is obtained analytically from each component's failure distribution using probability theory. In other words, the analytical approach involves the determination of a mathematical expression that describes the reliability of the system in terms of the reliabilities of its components. Once the system's reliability function has been determined, other calculations can then be performed to obtain metrics of interest for the system. There are several kinds of results available for analytical diagrams:

- The algebraic solution for the diagram (i.e., a mathematical representation of the reliability-wise configuration of the diagram) can be viewed in the Equation Viewer - Section 8.2 (p. 267)
- The minimal cut sets, or the unique combinations of component failures that can cause system failure, can be viewed in the Cut Sets window - Section 8.3 (p. 268)
- The analytical Quick Calculation Pad (QCP) provides quick access to some of the most frequently requested reliability results. These include standard and conditional reliability/probability of failure, failure rate, warranty time, BX information and mean time to failure - Section 8.4 (p. 270)
- A wide variety of plots help you to visualize the analytical results. These include plots pertaining to the system and to the blocks, offering information about reliability/unreliability, failure rate, pdf and reliability importance - Section 8.5 (p. 272)

In addition, allocation analysis can be performed based on the results from analytical diagrams. Allocation analysis helps to decide how the system's reliability can best be improved, based on the reliability importance of the blocks selected for consideration. *For more information on allocation analysis, see page 435 in Chapter 15*.

Analytical diagrams are appropriate when one is performing "reliability analysis." In the context of BlockSim, we use the term "reliability analysis" to refer to all analyses that do not include repairs or restorations of the component. When considering only the failure characteristics of the components, the analytical approach can be used. However, when both failure and maintenance characteristics need to be considered, the simulation method must be used to take into account the additional events.

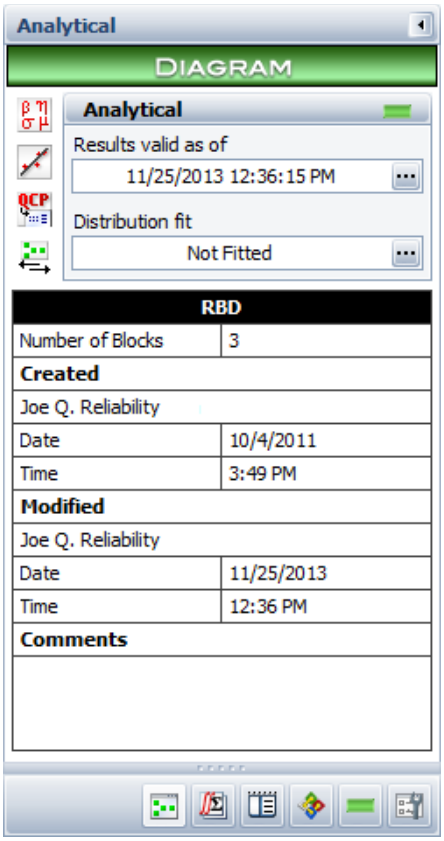
Watch a video example of working with an analytical RBD at: http://www.ReliaSoft.tv/blocksim/appexamples/blocksim_app_ex_1.html.

8.1 Analytical Diagram Control Panel

The control panel associated with BlockSim analytical diagrams and fault trees consists of multiple pages, each containing options for performing particular tasks. This section focuses on the Analytical page and Analysis Settings page of the analytical diagram and fault tree control panels, which contain most of the tools you will need to analyze a diagram or fault tree. (*For more information about the control panel in general, see page 109 in Chapter 3*.)

8.1.1 Control Panel Analytical Page

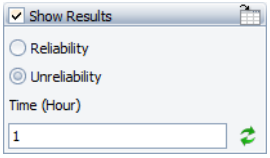
The Analytical page of the control panel for analytical diagrams and fault trees is shown next.



The Analytical page of the control panel contains the following options:

- The **Analytical** area displays the status and results of the analysis.
  - **Status** If the light is green, then the diagram has been analyzed. If the light is red, then the diagram has not been analyzed. An analyzed diagram will remain in the analyzed state unless a change is made to the structure of the diagram (e.g., a block is added, a connector is removed, a model is changed, etc.).
  - **Show Algebraic Solution (...)** displays the algebraic solution for the diagram in the Equation Viewer.
  - **Distribution fit** displays the distribution that has been fitted to the analytical results. Click the button (...) to open the Distribution Estimator to fit a distribution and/or for information on the fitted distribution parameters. The fitted distribution can be used to describe the overall behavior of the system and can be published for use as a model elsewhere in the project.
  - **Show Results** if selected, results at the event or gate level will be displayed at the upper right of each block in the diagram. This option is available only for fault trees. Select to display the results as reliability or unreliability values and specify the time at which the results should be calculated. The units for the time are the default units specified for the database. (To determine which time unit is defined as the default, you can

open the Define Units window by choosing **File > Manage Repository > Manage Units**. The default unit is the one that is selected in the Use as Default column.)



Click the **Update** icon to update the results shown in the diagram without reanalyzing the diagram.

You can click the **Spreadsheet View** icon to view the results for all gates and events in a spreadsheet-style report in the Results window.

- The **Information** area displays information about the diagram or the currently selected block. If no block is selected in the diagram, the **Information** area displays the number of blocks in the diagram and information about the diagram creation and last modification; in addition, if the diagram results have been published as a model, the name of the model will appear as a link allowing you to view the model in the Synthesis Viewer window. If a block is selected in the diagram, it displays a subset of the properties found in the Block Properties window. Items in the table that are resources (e.g., models, switches, etc.) are links; you can click such items to view and/or edit their properties. In both cases, this area also offers a **Comments** field; click the field heading to enter comments. For blocks, these comments are also accessible in the Identifiers properties in the Block Properties window.
- The **Tools** area on the left side of the control panel gives you quick access to the tools you will need to analyze the diagram and generate additional results.

  Analyze analyzes the relationships of the components in the diagram and puts the reliability-wise configuration into a mathematical formula.

  Plot creates a plot based on the analysis results. Clicking the **Plot** icon before the diagram has been analyzed will automatically perform the analysis and then plot the data.

  QCP opens the analytical Quick Calculation Pad (QCP). The QCP allows you to obtain reliability results based on the algebraic solution for the currently active diagram. Clicking the QCP icon before the diagram has been analyzed will automatically perform the analysis and then open the QCP.

  Change Diagram Type changes the current diagram to the opposite diagram type (i.e., changes the current analytical diagram to a simulation diagram, or the current simulation diagram to an analytical diagram). The subdiagram setting used in the change will be the one you specified the last time you used the Change Diagram Type window (**Diagram > Diagram Actions > Change Diagram Type**). In this window, you have the following options:

  - Select **Duplicate using the new type** to create a new diagram using the new type. The new diagram will be named "Copy of [original diagram name]." If you do not select this option, the original diagram itself will be changed to the new type.
  - Select **Perform the same change to any subdiagrams** to change any subdiagrams to the new diagram type. Note that if you select both this and **Duplicate using the new type**, new subdiagrams will be created along with the new main diagram.

Note that if you create an analytical diagram from a simulation diagram, by either duplication or conversion, the maintenance and throughput characteristics defined in the simulation diagram will not be displayed in the analytical diagram, as they are not relevant to the analysis. However, the properties are not lost, and if you then create a simulation diagram from that analytical diagram, all original properties will be retained.

8.1.2 Control Panel Analysis Settings Page

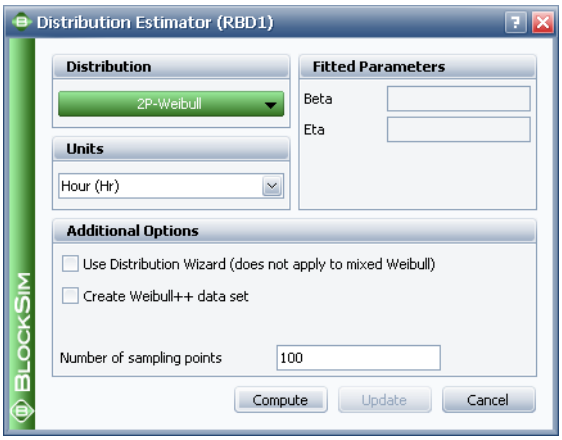
The Analysis Settings page of the control panel for analytical diagrams and fault trees contains the following options:

- **Use IBS**: if selected, the software looks for blocks with identical failure characteristics and attempts to simplify the equation, if possible, when analyzing the diagram (i.e., identical block simplification). This is useful when analyzing large diagrams that contain a lot of the same blocks. However, using this option means that more types of changes to the diagram will invalidate the mathematical equation that describes the reliability-wise configuration of the diagram and will require that the diagram be reanalyzed. If you select or clear this check box after you have already analyzed the diagram, you will have to reanalyze it.
- **Report Settings: Report folio model** allows you to specify the results model that will be used when the diagram is used as the default data source for an analysis workbook or a Word report. You can choose to use either the analytical results or the fitted model.

8.1.3 Distribution Estimator

Once a diagram has been analyzed or simulated, the results can be fitted to a distribution, which can be used to describe the overall behavior of the system. The fitted distribution may then be published for use as a model elsewhere in the project. The distribution is based on the results shown on the System Point Results report in the Simulation Results Explorer.

The Distribution Estimator allows you to specify the distribution used and to view the parameters of the fitted distribution. It is accessed by clicking the (**...**) button in the **Distribution Fit** field in the control panel. When the window is opened, the current distribution, if any, is displayed in the **Distribution** area and the calculated parameters are displayed in the **Fitted Parameters** area.



To calculate the parameters for a different distribution, choose the distribution in the **Distribution** area and specify the time units in the **Units** area, then click **Compute**. BlockSim will use the Weibull++ life data analysis engine to calculate the distribution parameters based on the diagram.

If you select the **Use Distribution Wizard** check box and click **Compute**, BlockSim uses Weibull++'s Distribution Wizard to select the distribution that best fits your data and then calculates the fitted parameters.

If you select the **Create Weibull++ data set** check box and click **Compute**, a Weibull++ standard folio containing a free-from data sheet will be created in the project and the data points used in fitting the distribution will be entered into the data sheet. If Weibull++ is installed on your computer, it will automatically be opened.

For analytical diagrams, the **Number of sampling points** field allows you to specify how many data points are generated in order to fit the distribution to the diagram results. (For simulation diagrams, the simulation results are

used for this purpose.) While a higher number of sampling points will result in a more accurate fitted distribution, it can also slow down the calculation.

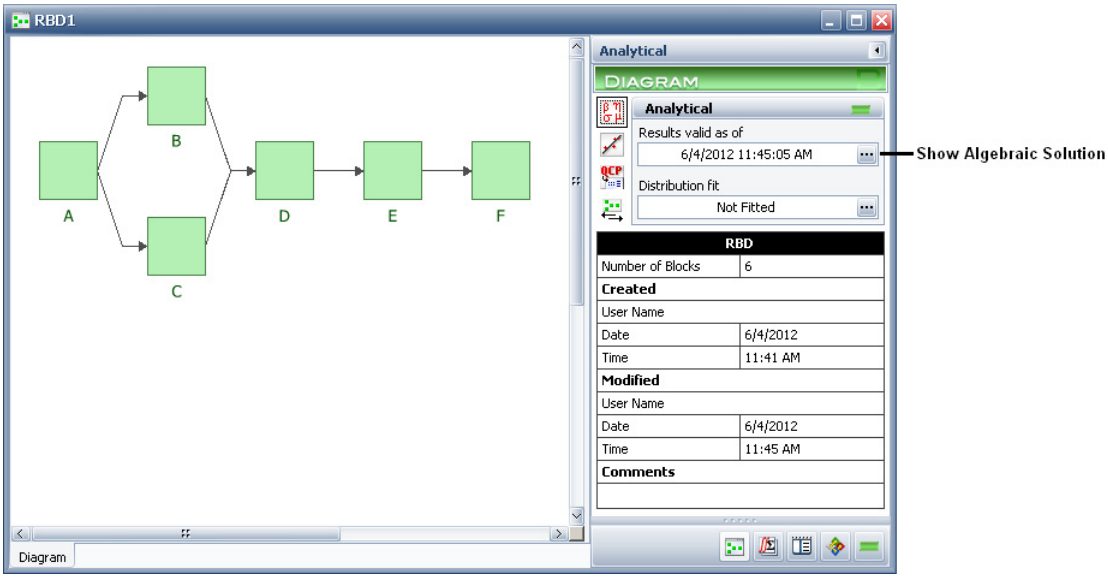
To apply the specified distribution and parameters to the diagram and close the Distribution Estimator, click **Update**. To close the Distribution Estimator without applying changes, click **Cancel**.

8.2 Equation Viewer

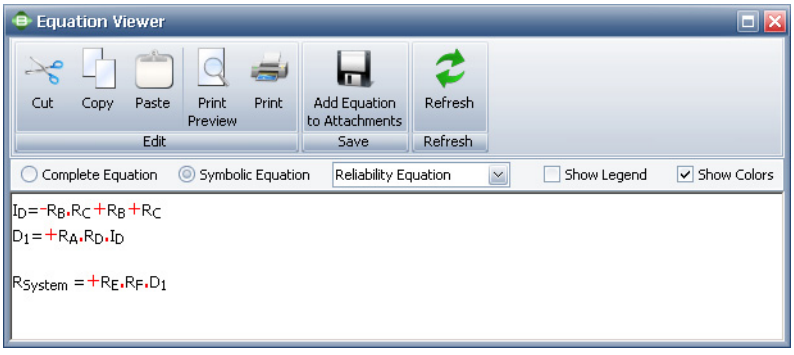
The Equation Viewer shows the algebraic equation for the system reliability, pdf and failure rate as a function of the component reliabilities. To open the Equation Viewer, choose **Analysis > Tools > Show Algebraic Solution**.

or click the **Show Algebraic Solution** icon (**...**) on the control panel.

There are two ways to view the system reliability equation: as a symbolic equation or as the complete equation. When you are using the symbolic equation view, you can choose to view the reliability equation, the pdf equation or the failure rate equation for the system. The symbolic equation uses tokens to represent portions of the equation. If you use the symbolic equation view, you should read the solution from the bottom up, replacing any occurrences of a particular token with its definition. For example, consider the RBD shown next.



When this diagram is analyzed (without using identical block simplification), the symbolic reliability equation is:



This would be read as follows:

$$I_D = -R_B \cdot R_C + R_B + R_C \qquad (1)$$

$$D_1 = +R_A \cdot R_D \cdot I_D \qquad (2)$$

$$R_{System} = +R_E \cdot R_F \cdot D_1 \qquad (3)$$

We begin with Eqn. (3). This equation contains the token D1, which is defined in Eqn. (2). This is then substituted into Eqn. (3), yielding:

$$R_{System} = +R_E \cdot R_F \cdot (+R_A \cdot R_D \cdot I_D)$$

Now Eqn. (4) contains the token ID, which is defined in Eqn. (1). Substituting this into Eqn. (4) yields:

$$R_{System} = +R_E \cdot R_F \cdot (+R_A \cdot R_D \cdot (-R_B \cdot R_C + R_B + R_C))$$

When you use the complete equation view, BlockSim automatically performs all token substitutions. In this view, you can view only the system reliability equation; the pdf equation and the failure rate equation are not available. You should be aware that in some cases, BlockSim may not be able to render the complete equation in the Equation Viewer even though it has internally computed it. This could happen if the memory available on your computer is insufficient and is therefore unable to construct the equation algebraically. This usually does not happen unless there are over one million terms.

Note: Due to the complexity of equations representing standby and load sharing configurations, BlockSim returns a single token for the reliability or failure rate of blocks in these configurations regardless of whether you are using the symbolic equation view or the complete equation view.

Select **Show Legend** to display the reliability characteristics for each block within the analyzed diagram. Select **Show Colors** to use color-coding in the window to make the equations easier to view. If you have made changes to these settings, click **Refresh** to update the contents of the window.

You can click **Add Equation to Attachments** to save the equation to the project attachments as an embedded *.rtf file.

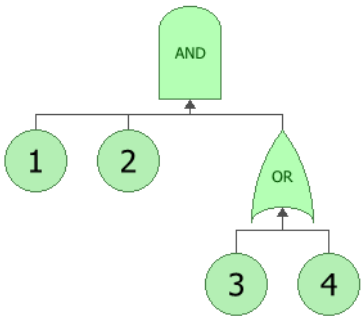
8.3 Cut Sets

Minimal cut sets are all the unique combinations of component failures that can cause system failure. Specifically, a cut set is said to be a minimal cut set if, when any basic event is removed from the set, the remaining events collectively are no longer a cut set.[1]

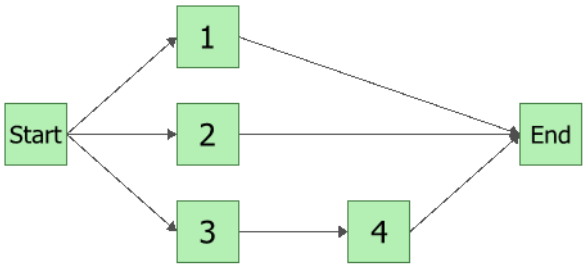
Minimal cut sets have traditionally been used to obtain an estimate of reliability for complex RBDs or fault trees that cannot be simplified by a combination of the simple constructs (parallel, series, *k-out-of-n*). BlockSim has the capability to derive an exact analytical solution to complex diagrams and therefore does not utilize the cut sets methodology. Nevertheless, cut sets are provided in BlockSim for interested users.

Minimal cut sets can be used to understand the structural vulnerability of a system. The longer a minimal cut set is, the less vulnerable the system is to that combination of events. Also, numerous cut sets indicate higher vulnerability. Cut sets can also be used to discover single point failures (one independent element of a system that causes an immediate hazard to occur and/or causes the whole system to fail).

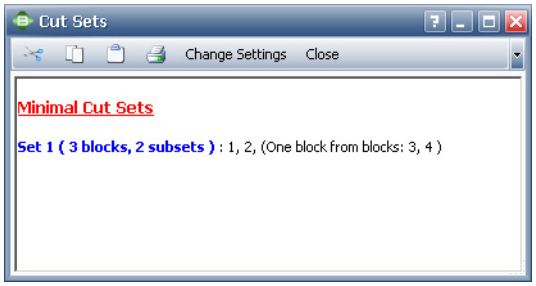
As an example, consider the fault tree shown next.



The system will fail if {1, 2, 3 and 4 fail} or {1, 2 and 3 fail} or {1, 2 and 4 fail}. All of these are cut sets. However, the one including all components is not a minimal cut set because, if 3 or 4 is removed, the remaining events are also a cut set. Therefore, the minimal cut sets for this configuration are {1, 2, 3} or {1, 2, 4}. This may be more evident by examining the RBD equivalent of the fault tree, as shown next.



The minimal cut sets are shown next.

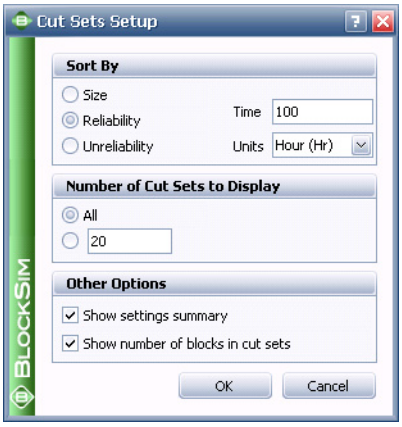


As you can see, BlockSim considers this to be a single set with two subsets, each of which contains three blocks.

You can view the cut sets for an analytical diagram by choosing **Analysis > Tools > Show Cut Sets**.

1234
147

The Cut Sets Setup window will appear, as shown next.



In the **Sort By** area, select the criterion you want to sort the cut sets by. You can select:

- **Size**: Cut sets will be shown in ascending order based on the number of blocks they contain.
- **Reliability** or **Unreliability**: Cut sets will be shown in ascending order of reliability or unreliability. Reliability is the probability that at least one block in the cut set survives. Unreliability is the probability that all blocks in the cut set fail. If you select either of these options, you must specify a time at which the reliability or unreliability is calculated and the units used for the time.

In the **Number of Cut Sets to Display** area, you can select to display all cut sets or specify the number of cut sets that will be shown.

In the **Other Options** area, you can select the additional information that will appear in the minimal cut sets display, including:

- **Show settings summary**: If selected, the diagram name and a summary of the settings chosen in the Cut Set Setup window will appear at the top of the cut sets display.
- **Show number of blocks in cut sets**: If selected, the number of blocks in each cut set will be shown in parentheses after the set number.

Click **OK** to accept your selections and display the minimal cut sets in the Cut Sets window.

In addition to standard commands for cut, copy, paste and print, the toolbar in the Cut Sets window also provides the **Change Settings** button, which reopens the setup window so you can change the settings and regenerate the display.

For more information about cut sets and their uses, please read “Minimal Cut Sets” at: http://www.Weibull.com/hotwire/issue63/relbasics63.htm.

8.4 Analytical Quick Calculation Pad

BlockSim includes a Quick Calculation Pad (QCP) for computing useful metrics. You can access the QCP by choosing **Analysis > Analysis > Analytical QCP** or by clicking the icon on the Analytical page of the diagram control panel.

To perform a calculation, select the appropriate option and enter any required inputs in the **Inputs** area, then click **Calculate**. *For more detailed information on how to use the QCP in general, see page 249 in Chapter 7.*

The following calculations are available for analytical diagrams:

Probability

- **Reliability**

  Calculates the probability that a new system will operate without failure for a given period of time. Enter the time at which you wish to calculate the reliability in the **Mission End Time** field. The mission is assumed to start at time = 0.

  For example, a reliability of 90% for a mission end time of 3 years means that if 100 identical systems are fielded, then 90 of them will still be operating at the end of 3 years.

> **Note:** Reliability calculations are available even when the linked RBD or fault tree contains static blocks. If the diagram contains one or more static blocks (i.e., blocks that use a constant model), the mission end time for the static blocks is assumed to be equal to the time for which the fixed reliability value has been defined.

- **Probability of Failure**

  Calculates the probability that a new system will be failed in a given period of time. Enter the time at which you wish to calculate the probability of failure in the **Mission End Time** field. The mission is assumed to start at time = 0.

  Probability of failure is also known as unreliability and it is the inverse of the reliability. For example, a probability of failure of 10% for a mission end time of 3 years is equivalent to a 90% reliability.

- **Conditional Reliability**

  Calculates the probability that a system will successfully operate at a specific time interval given that it has operated successfully up to a specified time. Enter the start time of the interval in the **Mission Start Time** field and enter the duration of the interval in the **Mission Additional Time** field.

  For example, a system may have a reliability of 90% for 3 years. If the system has operated for 2 years without failure, the conditional reliability for an additional year (for a total of 3 years of operation) may be 95%.

> **Note:** This assumes that all components start with the same accumulated age; that is, all components start life at 0 and age to the **Mission Start Time** value. It is possible to consider an individual component that has already accumulated some age (i.e., a used component) in the same formulation by changing the block's **Current Age** value in the Block Properties window.

- **Conditional Probability of Failure**

  Calculates the probability that a system will be failed at a specific time interval given that it has not failed up to a specified time. Enter the start time of the interval in the **Mission Start Time** field and enter the duration of the interval in the **Mission Additional Time** field.

  For example, a product may have a 10% probability of failure for 3 years. If the product has operated for 2 years without failure, the conditional probability of failure for an additional year (for a total of 3 years of operation) may be 5%.

Life

- **Reliable Life**

  Calculates the estimated time at which a specified reliability value will be achieved. Enter the reliability goal in the **Required Reliability** field. For example, a goal of 90% reliability with a reliable life of 4 years means that if 100 identical systems are fielded, then 90 of them will be still be operating at the end of 4 years.

What's Changed? In previous versions of BlockSim, this calculation was called "Warranty Time."

- **BX% Life**

  Calculates the estimated time at which a specified probability of failure will be achieved. Enter the probability of failure in the **BX% Information At** field. For example, a B10 life of 4 years means that the system has a 10% probability of failure in 4 years of operation (note that this is equivalent to a 90% reliability with a reliable life of 4 years).

Note: In the early days of reliability engineering, bearing manufacturers used the term "B10 life" to refer to the time by which 10% of the components would fail. Keeping with tradition, ReliaSoft retained this nomenclature but replaced "10" with "X," since the software allows you to get this information at any percentage point and not just at 10% (e.g., B1, B5, etc.).

- **Mean Life**

  Calculates the mean time to failure for the non-repairable system in the diagram.

Note: The mean life is a performance index and does not provide any information about the behavior of the failure distribution of the system.

Rate

- **Failure Rate**

  Calculates the instantaneous number of failures per unit time that can be expected at a certain time given that the system survives to that age. Enter the time at which you wish to calculate the failure rate in the **Time** field.

  For example, a failure rate of 0.01 at 100 hours means that a system that survives to 100 hours has approximately a 1% probability of failure in the next hour.

8.5 Analytical Plots

Plots can be created and viewed by clicking the **Plot** icon located on the Analytical page of the control panel for each diagram.

If you click the **Plot** icon before the diagram has been analyzed, it will automatically be analyzed and then the data will be plotted in the plot sheet. (Note that the **Plot** icon on the Plot sheet control panel refreshes the plot. If data or analysis settings have been changed since the diagram was last analyzed, you must reanalyze the data before refreshing the plot.)

You can add additional plot sheets to the diagram by right-clicking the sheet index tabs at the bottom of the window and choosing **Insert Analytical Plot** on the shortcut menu. You can delete a plot sheet from the diagram by selecting the plot sheet, right-clicking and choosing **Delete Plot**.

The plots available for analytical diagrams fall into three types:

System Plots

System plots display information about the entire system.

- **Unreliability vs. Time** shows the unreliability of the system over time. The unreliability value provides the probability that the event (failure) will occur by a specific time.
- **Reliability vs. Time** shows the reliability of the system over time.
- ***pdf* Plot** shows the system's probability density function, which illustrates the behavior of the life distribution function. Direct probability or reliability values cannot be read from this plot. The area below the plotted line, starting at time zero and ending at the value on the x-axis, is the probability of failure by that time.
- **Failure Rate vs. Time** shows the system's failure rate over time. The failure rate is the number of failures occurring per unit time.

Metrics Plots

Metrics plots display information about the reliability importance of the blocks in the system.

- **RI vs. Time** shows the reliability importance of the blocks in the system over time. Reliability importance is a measure of the relative importance of each component in a system with respect to the overall reliability of the system (*see page 439 in Chapter 15*).
- **Static RI** shows the static reliability importance of a specified number of blocks in the system at a specific time. To view this plot, you must specify the time at which the reliability importance is calculated and the number of blocks to display. The plot will show the blocks with the highest reliability importance (e.g., if you specify to show five blocks, the plot will show the five blocks with the highest reliability importance). Point to a bar to view the block's name, reliability and calculated importance. Blocks with the highest reliability are shown in green, blocks with medium reliability are shown in yellow and blocks with the lowest reliability are shown in red.
- Static RI Tableau shows the static reliability importance of the blocks in the system at a specific time in tableau format. To view this plot, you must specify the time at which the reliability importance is calculated. The block name appears on each area of the plot. Point to a plot area to view the block's name, reliability and calculated importance. Blocks with the highest reliability are shown in green, blocks with medium reliability are shown in yellow and blocks with the lowest reliability are shown in red.

Blocks Plots

Blocks plots display information about the individual blocks that make up the system.

For blocks plots, when you first select the plot, you must specify which blocks to include by using the **Available Blocks** area of the control panel to select which blocks to view. You can also view the corresponding system values by selecting the **Show System** check box.

- **Block Unreliability** shows the unreliability of the selected blocks plotted against time. The unreliability value provides the probability that the event (failure) will occur by a specific time.
- **Block Reliability** shows the reliability of the selected blocks plotted against time.
- **Block *pdf*** shows the probability density function of the selected blocks plotted against time.
- **Block Failure Rate** shows the failure rate over time of the selected blocks plotted against time. The failure rate is the number of failures occurring per unit time.

8.5.1 Setting Confidence Bounds

Where applicable, you can add confidence bounds to a plot by choosing **Plot > Confidence Bounds > Confidence Bounds**. The Confidence Bounds Setup window will appear, allowing you to define the properties of the confidence bounds lines to be displayed on the plot.

Note: Confidence bounds are available for the Unreliability vs. Time, Reliability vs. Time and Failure Rate vs. Time plots in cases where all blocks that can fail in the diagram use models that have Fisher matrix bounds. To have Fisher matrix bounds, a model must have been created by publishing from a data source, such as a Weibull++ or ALTA data sheet, that was calculated using Fisher matrix bounds. Additionally, bounds are not available if the diagram has containers, subdiagrams or mirrored blocks, or if any block uses the cumulative damage model.

- **Sides** allows you to specify which lines will be displayed on the plot.
  - **None** if selected, confidence bounds will not be displayed on the plot. Note that you can also remove confidence bounds from your plot by choosing **Plot > Confidence Bounds > Hide Confidence Bounds**.
  - **Two-Sided** displays two-sided confidence bounds on the plot.
  - **One-Sided** displays one-sided confidence bounds on the plot. You can select one type or both types.
    - **Show Top** displays the top line for one-sided confidence bounds on the plot.
    - **Show Bottom** displays the bottom line for one-sided confidence bounds on the plot.
- **Type** allows you to specify which types of confidence bounds you want displayed on the plot. You can select one type or both types. This area is not available for Failure Rate vs. Time plots.
  - **Time (Type I)** displays the confidence bounds on time.
  - **Reliability (Type II)** displays the confidence bounds on:
    - Reliability for Reliability vs. Time plots
    - Unreliability for Unreliability vs. Time plots
  - **Both** displays both types.
- **Confidence Level %** allows you to set the percentage for the confidence bounds.
- **Resolution** allows you to specify the resolution of the confidence bounds lines on the plot by dragging the slider or, for fine adjustments, clicking the slider and then using the arrow keys. The number of points plotted to create the lines is displayed beside the field name.

BlockSim Simulation Diagrams 9

In contrast to analytical diagrams, simulation diagrams can take into account repair and restoration actions, including behaviors of crews, spare part pools, throughput, etc. These actions mean that the age of system components is no longer uniform nor is the operation time of the system continuous. If one includes information on the repair and maintenance characteristics of the components and resources available in the system, other information can also be analyzed/obtained, such as system availability, throughput, spare parts utilization, life cycle costs etc. This can be accomplished through discrete event simulation.

In simulation, random failure times are generated from each component's failure distribution. These failure times are then considered in accordance with the way the components are reliability-wise arranged within the system. The overall results are analyzed in order to determine the behavior of the entire system.

The advantages of the simulation approach are:

- It can be used for highly complex scenarios involving a multitude of probabilistic events, such as corrective maintenance, preventive maintenance, inspections, imperfect repairs, crew response times, spare part availability, etc. When events such as these are considered, analytical solutions may become impossible when dealing with complex systems.
- Discrete event simulation also has the capability of:
  - Examining resource utilization, efficiency and costs.
  - Optimizing procedures and resource allocation.
  - Analyzing relationships between systems and components.
  - Maximizing throughput.
  - Minimizing work downtimes.

The disadvantages of the simulation approach are:

- It can be time-consuming.
- The results are dependent on the number of simulations.
- There is a lack of repeatability in the results due to the random nature of data generation.

For these reasons, if your scenario and/or results of interest do not require you to use a simulation diagram, it is recommended that you use an analytical diagram.

9.1 Available Analyses for Simulation Diagrams

Repairable system analysis via simulation focuses on maintainability/availability analysis. Additional related analyses made possible by using simulation include throughput analysis and life cycle cost analysis.

Results relevant to these analyses are available in several different locations:

- The Simulation Results Explorer provides a comprehensive set of metrics on the system, blocks, crews and spare part pools, including results relevant to maintainability/availability, costs and throughput - Section 9.10 (p. 308)
- The simulation Quick Calculation Pad (QCP) provides quick access to some of the most frequently requested reliability results. These include mean and point availability/unavailability, point reliability/probability of failure, mean time to first failure and time at which a certain availability or reliability will be shown - Section 9.11 (p. 326)
- A wide variety of plots help you to visualize the simulation results. These include plots pertaining to the system, blocks, crews and spare part pools, offering information about maintainability/availability, costs, throughput and resource utilization - Section 9.12 (p. 328)

Maintainability/Availability Analysis

Maintainability is defined as the probability of performing a successful repair action within a given time. In other words, maintainability measures the ease and speed with which a system can be restored to operational status after a failure occurs. For example, if it is said that a particular component has a 90% maintainability in one hour, this means that there is a 90% probability that the component will be repaired within an hour. In maintainability, the random variable is time-to-repair, in the same manner as time-to-failure is the random variable in reliability.

Availability is a performance criterion for repairable systems that accounts for both the reliability and maintainability properties of a component or system. It is defined as the probability that the system is operating properly when it is requested for use. That is, availability is the probability that a system is not failed or undergoing a repair action when it needs to be used.

The ReliaWiki resource portal has more information on maintainability/availability analysis at: http://www.ReliaWiki.org/index.php/Introduction_to_Repairable_Systems.

Throughput Analysis

Throughput is the amount of output that can be processed in a given period of time. For information about throughput analysis, please see the next section on page 277.

Life Cycle Cost Analysis

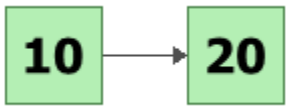
A life cycle cost analysis examines the costs incurred by a system or a component over its entire life span. Typical costs for a system may include:

- Acquisition costs (or design and development costs).
- Operating costs:
  - Cost of failures (e.g., contractually-specified penalties, risk of damage to equipment or people, etc.)
  - Cost of repairs.
  - Cost for spares.
  - Downtime costs (e.g., contractually-specified restitution).
  - Loss of production.
- Disposal costs.

A complete life cycle cost (LCC) analysis may also include other costs (such as the initial costs of land, equipment and/or installation), as well as other accounting/financial elements (such as discount rates, interest rates, depreciation, present value of money, etc.). With respect to the cost inputs for such an analysis, the costs involved are either deterministic (such as acquisition costs, disposal costs, etc.) or probabilistic (such as cost of failures, repairs, spares, downtime, etc.). Most of the probabilistic costs are directly related to the reliability and maintainability characteristics of the system. To perform life cycle cost analysis in BlockSim, you will need to provide cost information in all relevant fields. Additional inputs relating to downtime may also be required, depending on your system. The ReliaWiki resource portal has more information on life cycle cost analysis at: http://www.ReliaWiki.org/index.php/Additional_Analyses#Life_Cycle_Cost_Analysis.

9.1.1 Throughput Analysis

Throughput is the amount of output that can be processed in a given period of time. To understand system throughput, consider this example: Assume that each component in the system processes (or makes) something while operating. The RBD shown next contains two components in series, each processing/producing 10 and 20 items per unit time (for explanation purposes, we will call that "ipu") respectively.



In this case, the system configuration is not only the system's reliability-wise configuration but also its production/ processing sequence. In other words, the first component processes/produces 10 ipu and the second component can process/produce up to 20 ipu. However, a block can only process/produce items it receives from the blocks before it. Therefore, the second component in this case is only receiving 10 ipu from the block before it. If we assume that neither component can fail, then the maximum items processed from this configuration would be 10 ipu. If the system were to operate for 100 time units, then the throughput of this system would be 1,000 items, or (100 x 10).

Throughput analysis can be performed for simulation RBDs. Results are available at the block and system levels, taking into account the reliability and maintenance properties of the blocks and the system. In order to perform throughput analysis, you must specify throughput properties in the Block Properties window for the relevant blocks and select the **Run Throughput Simulation** option on the Analysis Settings page of the diagram's control panel.

Note: While throughput analysis is not available for fault trees, fault trees may be used as subdiagrams for simulation RBDs. To perform throughput analysis in this case, you would define throughput properties for the subdiagram in the Block Properties window for the subdiagram block. In simulation, the fault tree is simulated and determined to be up or down. If it is down, no throughput is passed to the next block in the RBD; if it is up, then the throughput properties defined for the subdiagram block determine the amount of throughput passed to the next block.

In many real-world cases, throughput can change over time (i.e., throughput through a component is not a constant, but a function of time). You can model variable throughput using phase diagrams (*see page 351 in Chapter 10*).

The ReliaWiki resource portal has more information on throughput analysis at: http://www.ReliaWiki.org/index.php/Additional_Analyses#Throughput_Analysis.

Block Throughput Properties

The Throughput properties are available in the Block Properties window for all block types in simulation RBDs except contained blocks. These fields allow you to define the characteristics of the block that pertain to analyzing the throughput, which is the amount of output the block can process in a given period of time.

The **Throughput** field allows you to enter the quantity of output and the units used for time. In addition, the **Allocation** field allows you to specify how that output is routed if multiple outgoing paths from the block exist:

- If **Weighted allocation across paths** is selected, the allocation will be weighted according to the capacity of each block if there is more than one block that can receive the units. The software calculates the total capacity of all possible paths and then determines the percentage of the total capacity represented by each block. It then sends that percentage of the "stream" to each block. For example, if 3 blocks can receive 9 units and Block A can process 5 units, Block B can process 5 units and Block C can process 10 units, then Block A receives 25% (5 + 5 + 10 = 20 and 5/20 = 0.25) of the stream (or 9 * 0.25 = 2.25), Block B receives 25% (5/20 = 0.25) of the stream (2.25) and Block C receives 50% (10/20 = 0.5) of the stream (4.5).
- If **Allocate equal share to all paths** is selected, an equal share of units will be allocated to each of the blocks if there is more than one block that can receive the units. In the example described above, each of the 3 blocks would receive 1/3 of the stream, or 3 units. This option may result in a backlog if any blocks cannot process their entire allocation.

If there is more than one block that can receive units and one of those blocks has failed, the units will form a backlog for the failed block if the **Send units to failed blocks** option is selected. If it is not selected, the failed block will not receive any of the units and the entire stream will be distributed among the unfailed blocks. For example, in the first scenario, if Block B fails and the user has not selected to send units to failed blocks, then Blocks A and C will both receive 4.5. If this option is selected, then all three blocks will receive 3 units and a backlog will begin to build up for Block B.

Units that a block cannot process, either because it is failed or because the units are in excess of its processing capacity, are kept in backlog. The **Process/ignore backlog** field allows you to specify whether backlogged units, if any, are processed or ignored. The **Ignore backlog** option is appropriate for modeling a configuration where backlog cannot accumulate, whereas the **Process backlog** option models a configuration where backlog may accumulate.

The use of these options interacts with certain other settings. Specifically:

- If the block is set to process backlog, then backlog is accumulated in two cases:
  1. All outgoing blocks are failed and the block itself does not have the **Send units to failed blocks** option selected.
  2. The block receives more units than it can process, either because it receives more than its capacity or because it is failed and the previous block sends throughput to failed blocks.
- If the block is set to ignore backlog, and one of the two cases above is true, then the throughput will still technically be stored as backlog but will never be processed. In practical terms this is equivalent to ignoring backlog.

If you have selected to process backlog, you can select the **Limited backlog** option to place a limit on the number of units that can be stored in the backlog. Once the limit that you specify is reached, no additional units are added to the backlog.

For subdiagram blocks, the **Throughput properties** field allows you to specify how you want throughput for the block determined in simulation. You can choose to use the throughput properties defined in the subdiagram that the block represents or to use throughput properties defined on this page. This option is available only if the subdiagram is an RBD, because fault trees do not have throughput properties. If you select the **Use throughput properties as defined at this level** option, the **Throughput, Allocation** and **Send units to failed blocks** fields will appear.

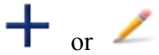
9.2 Maintenance Groups

Maintenance groups are resources that are available for use throughout the project and can be managed via the Resource Manager. A maintenance group is a set of blocks where some event within the group can trigger either maintenance or state changes for one or more blocks, either within the group or outside of it. You can use a maintenance group:

- To turn a block on or off. State change triggers are used to activate or deactivate a block when items in one or more specified maintenance groups go down or are restored. The block whose state is being changed may or may not belong to any of the maintenance groups. This allows you to model a cold standby configuration (i.e., one where the component cannot fail when in standby) without using a standby container, which may be useful if you are using a parallel or complex configuration, as blocks can be connected only in series in standby containers. *For more information on state change triggers, see page 216 in Chapter 7*.
- To trigger a scheduled task (i.e., a preventive task, inspection or on condition task). You can set the task to be performed based on events in one or more maintenance groups. Triggering events within the maintenance group can include block failure; start of corrective, preventive or inspection tasks; and/or block restoration. The block(s) affected by the task may or may not belong to any of the maintenance groups. For more information on how tasks can be scheduled, see *Section 9.4.5 on page 290*.

What's Changed? In previous versions, item group numbers offered a limited version of this functionality, where assigning blocks to the same maintenance group caused their maintenance to be performed together. The maintenance groups used in the Synthesis version allow a great deal more modeling flexibility.

The Maintenance Group window allows you to create, view and edit maintenance groups. It can be accessed by clicking the **Create New** or **View/Edit** icon in the Maintenance Group wizard, accessed from the Block Properties window).



It can also be accessed from the Maintenance Groups page of the Resource Manager by choosing **Home > Edit > New**, by selecting a maintenance group and choosing **Home > Edit > View** or by double-clicking a maintenance group.

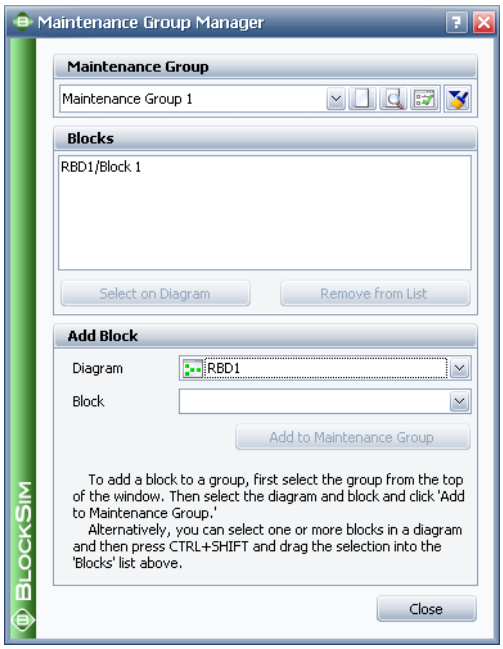
At the top of the window, you can specify the maintenance group name. You can replace the default name with your own name of up to 150 characters, if desired. Users with the "Manage other repository settings" permission can change the default names by choosing **File > Manage Repository > Define Default Names**.

You can also enter comments about the maintenance group in this window.

For an existing resource, a link at the bottom of the properties window indicates how many times it is currently being used. If you need more information, click the link or the **Trace Usage** icon.

This opens the Dependency Viewer, which allows you to explore both where the resource is being used and what other resources are associated with it.

The Maintenance Group Manager allows you to add blocks to or remove blocks from a maintenance group without having to go into each block's properties individually. This window can be accessed by choosing **Project > Data Management > Maintenance Group Manager** or, when you are viewing the Maintenance Groups page of the Resource Manager, by choosing **Home > Actions > Maintenance Group Manager**.



The **Maintenance Group** area allows you to choose an existing maintenance group from the drop-down list. In addition, you can click the **Select Existing** icon to open the Select Resource window. If there is not an existing maintenance group that meets your needs, you can create a new one by clicking the **Create New** icon to open a window that allows you to specify a name for the new maintenance group, as well as any comments. Click the **View** icon to view the characteristics of the selected maintenance group.

You can also click the **Indicator Style** icon to modify the appearance of the corner indicator that will appear on each block that belongs to the maintenance group, including:

- The indicator's background color, fill color and fill style. The background color is the underlying color applied to the indicator; the fill color is applied over the background color in the pattern specified by the fill style.
- The style (e.g., solid, dash, etc.), color and thickness of the indicator border.

The **Blocks** area displays all of the blocks in the maintenance group, in the format [Diagram Name]\[Block Name].

- To add a block to the maintenance group, you can select the diagram and block in the **Add Block** area and click the **Add to Maintenance Group** button. You also can select the block(s) in the diagram, press **CTRL+SHIFT** and drag the block to the list in the **Blocks** area of the Maintenance Group Manager.
- To view a block in the maintenance group, select the block in the list and click the **Select on Diagram** button. The diagram will be displayed with the block selected.
- To remove a block from the maintenance group, select the block in the list and click the **Remove from List** button.

9.3 Universal Reliability Definitions (URDs) in Simulation Diagrams

In simulation diagrams, the universal reliability definitions (URDs) assigned to blocks define reliability and maintenance characteristics. Each URD is a resource that can be shared among analyses and can include two other types of resources:

- **Models** are resources that represent behavior over time. In URDs, they are used to define failure behavior.
- **Tasks** describe the conditions that determine when maintenance actions will be performed on a block. Tasks specify the logistical time and costs associated with those actions as well as their anticipated effectiveness (i.e., how much the component will be restored). Tasks may be configured to use three other types of resources:
  - Tasks use models to describe how long the task takes.
  - **Crews** describe the conditions that determine when personnel will be available to perform specific tasks. These resources also specify the logistical time and costs associated with engaging the crew.
  - **Spare part pools** describe the conditions that determine whether a spare part will be available when needed. These resources also specify the logistical time and costs associated with obtaining the spare part.

  A URD can have one corrective task and/or one or more scheduled tasks. Scheduled tasks are displayed in their priority order, which is the order in which they will be performed; you can use the up and down arrows in the cells in this column to move the task up and down the priority list. *For more information about URDs, see page 170 in Chapter 5.*

9.4 Tasks

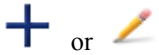
To properly analyze repairable systems, we first need to understand how components in these systems are restored (i.e., the maintenance activities that are performed on the components). In general, maintenance is defined as any action that restores failed units to an operational condition or keeps non-failed units in an operational state. For repairable systems, maintenance plays a vital role in the life of a system. It affects the system's overall reliability, availability, downtime, cost of operation, etc.

In Synthesis applications, maintenance activities are represented using tasks, which are resources that can be shared among analyses and can be managed via the Resource Manager. There are two basic kinds of tasks, which comprise four task classes:

- Corrective tasks are the action(s) taken to restore a failed component to operational status. These cannot be scheduled, as the component's exact failure time is not known before it happens.
- Scheduled tasks can be performed on a known schedule, based on time, component condition or other factors. These include:
  - Preventive tasks
  - Inspection tasks
  - On condition tasks

Tasks are assigned to URDs, which are in turn used to represent a set of properties that can be applied to standard blocks in RBDs and to events in fault trees.

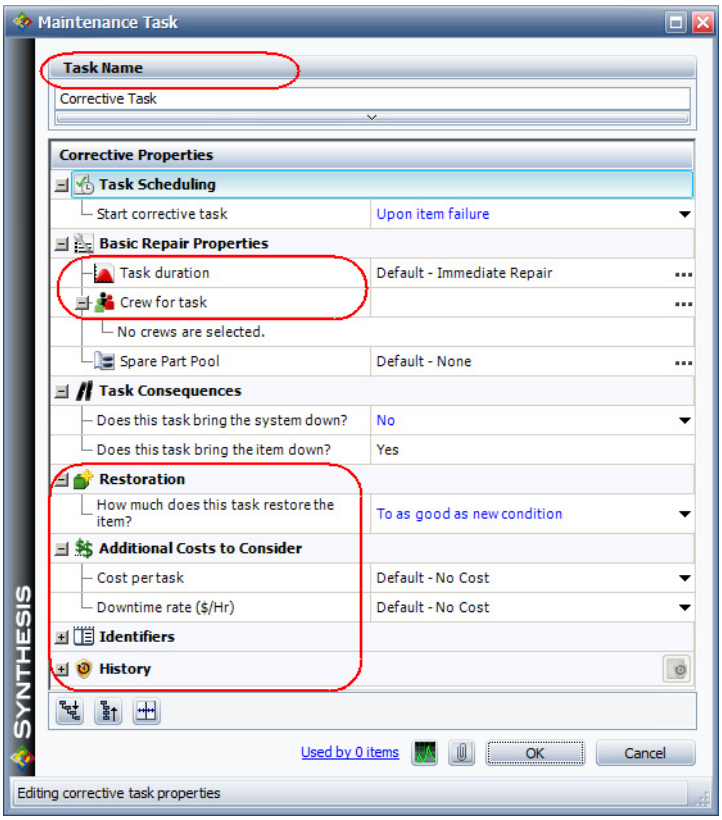
The Maintenance Task window allows you to create, view and edit all classes of maintenance tasks. It can be accessed by clicking the **Create New** or **View/Edit** icon in the Task wizard, which is accessed from **Task** fields in properties windows (e.g., the **Corrective Maintenance Task** field in the Universal Reliability Definition window).

 or

It can also be accessed from the Corrective Tasks and Scheduled Tasks pages of the Resource Manager by choosing **Home > Edit > New**, by selecting a task and choosing **Home > Edit > View** or by double-clicking a task.

For a new resource, a name will be proposed automatically based on the default naming criteria established for the current database (in the Define Default Name Formats window). You can replace this with your own name of up to 150 characters, if desired. Remember that the name and identifiers are the primary way in which your team will be able to find the Synthesis resources you need for your analyses.

Each task class may have properties that apply only to that particular class. This section describes the properties that are common to most task classes. As an example, the following picture shows the properties of a corrective task, with the common properties highlighted.



The common properties must be configured for all classes of tasks. Configuration options that are specific to particular task classes are presented in the corresponding sections.

- **Basic Task Properties**
  - **Task duration** allows you to assign a model, which may represent a fixed time or a distribution, to describe the duration of the task. You can choose an existing model or create a new one. If no model is assigned, it is assumed that the task has a duration of zero (i.e., immediate repair).
  - **Crew for task** allows you to choose or create one or more crews that can perform the task. If multiple crews are assigned to the task and a crew is needed, they will be checked for availability in the order in which they are displayed in this list. Use the **Priority Up** and **Priority Down** arrows that appear when you click a crew name in the list to move it up and down in the list. If a crew is needed and all crews are engaged, the crew with the shortest wait time is chosen to perform the task.

    If no crew is assigned, it is assumed that the work will be done by some undefined crew that is always available.

- **Restoration**. These properties allow you to specify the degree to which the block will be restored after the performance of the task. In the **How much does this task restore the item?** field, you can specify that the item will be returned to as good as new condition (i.e., full repair, equal to a restoration factor of 1), to the same condition it was in when it failed (i.e., "as bad as old" or minimal repair, equal to a restoration factor of 0), or you can choose **Partial restoration**. This option provides you with the ability to model maintenance involving "used parts" or imperfect maintenance. The **Restoration amount** field will appear; you can specify the amount of restoration achieved by the task either by entering the value as a percentage or by clicking the arrow and using the slider bar that appears. If you specify anything other than 0% or 100%, you will need to specify what the restoration effect applies to (i.e., the restoration type). You can select:
  - **Only damage accumulated since last repair**: Each repair will remove only the damage since the last repair (i.e., the nth repair cannot remove the damage incurred before the (n-1)th repair). Note that in this context any task is considered a repair and any damage that has occurred since the last event (corrective task, preventive task, inspection or on condition task) will be reduced.
  - **All accumulated damage**: Each repair can reduce any damage accumulated up to that failure.

  For simulation, the application uses the restoration factor to determine the new age of the block after the maintenance action.

  For example, consider an automotive engine that fails after 6 years. If the engine is rebuilt and the rebuilding task has a 50% restoration factor:

  - If **Only damage accumulated since last repair** is selected, the initial rebuild has the effect of rejuvenating the engine to a condition as if it were 3 years old.

    The engine fails again after 3 years (when it again reaches the effective "age" of 6 years), but the rebuild this time affects only the age accumulated after the first rebuild. Thus the engine has an effective age of 4.5 years after the second rebuild (3 + 3 x (1 - 0.5) = 4.5).

    After the second rebuild, the engine fails again after a period of 1.5 years (when it again reaches the effective age of 6 years) and a third rebuild is required. The effective age of the engine after the third rebuild is 5.25 years (4.5 + 1.5 × (1 - 0.5) = 5.25).
  - If **All accumulated damage** is selected, the initial rebuild has the effect of rejuvenating the engine to a condition as if it were 3 years old.

    The engine fails again after 3 years (when it again reaches an effective age of 6 years) and another rebuild is required. This rebuild also rejuvenates the engine by 50%, thus making it effectively 3 years old again.

    After the second rebuild, the engine fails again after a period of 3 years (when it again reaches the effective age of 6 years) and a third rebuild is required. The effective age of the engine after the third rebuild is 3 years.

  Compare the following tables to see how the two options differ.

Only Damage Accumulated Since Last Repair

Time	Time Since Last Repair	Effective Age Before Repair	Effective Age After Repair
Start = 0	0	0	0
6 years	6	6	3
9 years	3	6	4.5
10.5 years	1.5	6	5.25

All Accumulated Damage

Time	Time Since Last Repair	Effective Age Before Repair	Effective Age After Repair
Start = 0	0	0	0
6 years	6	6	3
9 years	3	6	3
12 years	3	6	3

The ReliaWiki resource portal has more information on restoration factors at: http://www.ReliaWiki.org/index.php/Repairable_Systems_Analysis_Through_Simulation.

- **Additional Costs to Consider** allows you to choose or create models to represent costs that are always associated with the task. **Cost per task** uses a cost model, and **Downtime rate** uses a cost per unit time model. If no models are assigned, it is assumed that there are no additional costs.
- **Identifiers** allows you to enter additional identifying information that can be used to search for this resource.
- **History** provides information about the creation and last modification of the resource. If the history log has been activated at the project level (**Project > Management > History Log**), you can click the **View Item History** icon to open the Record History Log for the resource.

You can collapse or expand all nodes in the hierarchy using the **Collapse All** and **Expand All** icons.

You can resize the columns by pointing to the column splitter. When the pointer becomes a double-headed arrow, drag the splitter to the desired location. You can restore the column widths to their original settings by clicking the **Restore Column Widths** icon.

For an existing resource, a link at the bottom of the properties window indicates how many times it is currently being used. If you need more information, click the link or the **Trace Usage** icon.

This opens the Dependency Viewer, which allows you to explore both where the resource is being used and what other resources are associated with it.

9.4.1 Corrective Tasks

Corrective maintenance consists of the action(s) taken to restore a failed component to operational status. Corrective maintenance is performed at unpredictable intervals because a component's failure time is not known a priori.

Corrective tasks:

- Always bring the block down.
- May bring the system down.
- Require spare parts.

In addition to the common task properties, the following options are used to configure corrective tasks in the Maintenance Task window:

- **Task Scheduling** allows you to specify when the corrective task will be performed.
  - If you choose **Upon item failure**, the task will be initiated upon failure of the component.
  - If you choose **When found failed during an inspection**, the task will be initiated if the component is found to be failed at the next scheduled inspection. This is useful in the case of "hidden failures" (i.e., failures that are not apparent until an inspection is performed) and in cases where the component is not considered mission critical and repair can wait until the next scheduled maintenance.
    - If a preventive task takes place before the next inspection, then preventive maintenance (not corrective maintenance) will be performed to restore the block. The downtime, etc. will depend on the preventive task properties.
    - If a corrective task is performed upon inspection and an inspection finds the block failed, then the total downtime includes the full inspection duration followed by the corrective task duration.

    If you have selected to perform a corrective task upon inspection and an inspection or preventive task does not occur after the failure, then the block will never be restored. This could happen under either of the following circumstances:

    - Neither an inspection nor a preventive task is specified for the block.

    and/or

    - The conditions have not been met to perform an inspection or preventive task.
- **Basic Repair Properties > Spare Part Pool** allows you to choose or create a spare part pool that will be used in performing the task. If a spare part pool is not assigned, it is assumed that unlimited free spares are always immediately available. *For more information about spare part pools, see Section 9.6 on page 294.*

  For corrective and preventive tasks, the simulation requests a crew as soon as a task is initiated; however, the crew does not begin performing the task unless/until the spare part is available.

  The total time for the task consists of:

  - The time to complete corrective and preventive maintenance (based on the corrective task or preventive task properties).

    PLUS

  - The longer of the following times:
    - The time to obtain the spare part (based on the spare part pool).

      OR

    - The time to obtain an available crew (based on crew availability) and any logistical delay time associated with the crew.

  For example, if the block fails at 10 hours (on the system clock), the corrective task will take 5 hours, the time required to obtain the spare will be 48 hours and the logistical delay time associated with the crew is 8 hours, then:

  - The spare will arrive at 58 hours (on the system clock).
  - The crew will be requested at 10 hours and will arrive at 18 hours (on the system clock).

- The maintenance will be completed at 63 hours (on the system clock).
- The total time for the corrective maintenance will be 53 hours (48 for the spare plus 5 for the task).
- The crew will have traveled for 8 hours, waited 40 hours for the spare part and performed the task for 5 hours for a total of 53 hours (including waiting for the spare part).

If the spare part is available without delay and all other conditions are the same, then:

- The crew will be requested at 10 hours and will arrive at 18 hours (on the system clock).
- The maintenance will be completed at 23 hours (on the system clock).
- The total time for the corrective maintenance will be 13 hours (8 for the crew plus 5 for the task).
- The crew will have traveled for 8 hours and performed the task for 5 hours for a total of 13 hours.

- **Task Consequences**
  - **Does this task bring the system down?**: By default, corrective tasks will not bring the system down unless having the block down brings the system down based on the reliability-wise configuration in the diagram. If you answer **Yes**, the task will bring the system down even if the task has a zero duration. This forces the task to be included in the count of system downing events, regardless of the task's duration.
  - **Does this task bring the item down?**: It is assumed that a block will always be down when a corrective task is performed, even a task with a zero duration; thus, this option cannot be changed.

9.4.2 Preventive Tasks

Preventive maintenance is the practice of repairing or replacing components or subsystems before they fail in order to promote continuous system operation or to avoid dangerous or inconvenient failures. The schedule for preventive maintenance is based on observation of past system behavior, component wearout mechanisms and knowledge of which components are vital to continued system operation. In addition, cost is always a factor in the scheduling of preventive maintenance. In many circumstances, it is financially more sensible to replace parts or components at predetermined intervals rather than to wait for a failure that may result in a costly disruption in operations.

Preventive tasks:

- Always bring the block down.
- May bring the system down.
- Require spare parts.

Only one preventive task can be performed on a block at any given time.

In addition to the common task properties, the following options are used to configure preventive tasks in the Maintenance Task window:

- **Task Class** allows you to specify the kind of maintenance that the task performs: preventive, inspection or on condition. The options available in the Maintenance Task window will vary depending on your choice in this field.
- **Task Scheduling** allows you to specify the circumstances under which the task will be performed. *For more details about task scheduling, see Section 9.4.5 on page 290*.
- **Basic Task Properties > Spare Part Pool** allows you to choose or create a spare part pool to assign to the task. The spare part pool describes the conditions that determine whether a spare part will be available when needed

and specifies the time and costs associated with obtaining the spare part. If a spare part pool is not assigned, it is assumed that unlimited free spares are always immediately available).

For corrective and preventive tasks, the simulation requests a crew as soon as a task is initiated; however, the crew does not begin performing the task unless/until the spare part is available. *For a complete explanation of how this affects the total time for the task, see the previous section*.

- **Task Consequences**
  - **Does this task bring the system down?**: By default, preventive tasks will not bring the system down unless having the block down brings the system down based on the reliability-wise configuration in the diagram. If you answer **Yes**, the task will bring the system down, even if the task has a zero duration. This forces the task to be included in the count of system downing events, regardless of the task's duration.
  - **Does this task bring the item down?**: It is assumed that a block will always be down when a preventive task is performed, even a task with a zero duration; thus, this option cannot be changed.
  - **If bringing the item down causes the system to go down, do you still perform the task?**: If you answer **Yes**, the task will be performed even if doing so brings the system down, either because the task itself brings the system down or because the task brings the block down and the block being down causes the system to go down. If you answer No and the task brings the system down, the task will never be performed during simulation unless the system is already down for another reason. For instance, a preventive task that is specified to be performed upon system down will be performed even if it brings the system down regardless of your answer here. This is because the system is already down, which is what triggered the task in the first place.

    A preventive task that does not bring the system down at the preventive maintenance time will still be factored into the simulation even if its duration will bring the system down at a later time.

    For a preventive task that is scheduled to occur based on item age and for which you have answered **No** to this question, if the task is going to bring the system down, then it will not take place. If, however, the block reaches the age again (after restoration by a corrective action, inspection or another type of preventive maintenance) and this time it will not bring the system down, then it will be performed.
  - **Perform this task even if the item failed before this task was scheduled to occur?**: If you answer **Yes**, the task will be performed even if the item has already failed. In other words, the item's "clock" is not stopped upon failure, and the item's "virtual age" is used to trigger the task. This option is available only for preventive and inspection tasks that are scheduled based on item age.
- **RCM** properties are text-based properties that are used to keep track of details that may be helpful in reliability centered maintenance analysis, but are not used in reliability/maintainability simulations. These properties are shown only if they are enabled for the project via the interface style settings in RCM++/RBI.
  - **Status**: The status of the task (choose from a drop-down list). This setting could be used if you want to keep track of all tasks that have been considered, regardless of whether they end up in the actual maintenance plan (e.g. recommended, rejected, assigned).
  - **Proposed Interval**: The interval that was initially proposed for the task. This may be different from the interval that is actually assigned to the task. For example, you may wish to use this property if the team originally suggests a particular interval for the task (perhaps the calculated optimum interval) but then decides to assign a different interval (perhaps an interval that is more convenient for packaging a group of tasks).
  - **Reference Document**: A reference to another document that provides more detailed information about the task (e.g. procedure instructions).
  - **Condition**: A description of the condition that indicates that a failure will occur (e.g. a threshold for a measurement of wear, vibration, etc). Typically, this field is used for on condition maintenance tasks.

- **Zone**: The zone of the system in which the task will be performed. Typically, this field is used for aircraft MSG-3 analyses.
- **Access**: The access that will be required in order to perform the task. Typically, this field is used for aircraft MSG-3 analyses.

Note: A preventive task with a restoration factor of 0 will generate a new failure with the current age.

9.4.3 Inspection Tasks

Inspections are used in order to uncover hidden failures (also called "dormant failures"); they are also used as part of on condition tasks to detect impending failures so that preventive maintenance can be performed.

What's Changed? In previous versions of BlockSim, you could assign failure detection properties to an inspection task and set it to trigger a preventive task, thereby creating an on condition maintenance action. The Synthesis version of BlockSim offers on condition tasks, eliminating the need to create your own. Therefore, failure detection properties are now available only for on condition tasks, not for regular inspection tasks.

In general, no maintenance action is performed on the component during an inspection unless the component is found failed, in which case a corrective maintenance action is initiated. However, there might be cases where a partial restoration of the inspected item would be performed during an inspection. For example, when checking the motor oil in a car between scheduled oil changes, one might occasionally add some oil in order to keep it at a constant level. This sort of restoration is not considered to be preventive maintenance; the deciding factor is that inspection tasks do not use spare parts. Note that an inspection task that triggers a corrective task will not restore the failed block. Only the corrective task will restore the block.

Inspection tasks (including the inspection and associated minor work):

- May bring the block down.
- May bring the system down; if the task brings the system down, it also brings the block down.
- Do not use spare parts.

In addition to the common task properties, the following options are used to configure inspection tasks in the Maintenance Task window:

- **Task Class** allows you to specify the kind of maintenance that the task performs: preventive, inspection or on condition. The options available in the Maintenance Task window will vary depending on your choice in this field.
- **Task Scheduling** allows you to specify the circumstances under which the task will be performed. *For more details about task scheduling, see Section 9.4.5 on page 290*.
- **Task Consequences**: An inspection task will not bring the block down unless one of the following options is selected.
  - **Does this task bring the system down?**: If you answer **Yes**, when the task is performed, the system will be down, even if the task has a zero duration. If not selected, the task will bring the system down only if it brings the block down and having the block down brings the system down based on the reliability-wise configuration in the diagram. If this option is selected, even tasks with a zero duration will bring the system down. This forces the task to be included in the count of system downing events, regardless of the task's duration.
  - **Does this task bring the item down?**: If you answer **Yes**, when the task is performed, the block will be down. Even tasks with a zero duration will bring the block down in this case.

- **Perform this task even if the item failed before this task was scheduled to occur?**: If you answer **Yes**, the task will be performed even if the item has already failed. This option is available only for preventive and inspection tasks that are scheduled based on item age.

- **RCM** properties are text-based properties that are used to keep track of details that may be helpful in reliability centered maintenance analysis, but are not used in reliability/maintainability simulations. These properties are shown only if they are enabled for the project via the interface style settings in RCM++/RBI.
  - **Status**: The status of the task (choose from a drop-down list). This setting could be used if you want to keep track of all tasks that have been considered, regardless of whether they end up in the actual maintenance plan (e.g. recommended, rejected, assigned).
  - **Proposed Interval**: The interval that was initially proposed for the task. This may be different from the interval that is actually assigned to the task. For example, you may wish to use this property if the team originally suggests a particular interval for the task (perhaps the calculated optimum interval) but then decides to assign a different interval (perhaps an interval that is more convenient for packaging a group of tasks).
  - **Reference Document**: A reference to another document that provides more detailed information about the task (e.g. procedure instructions).
  - **Condition**: A description of the condition that indicates that a failure will occur (e.g. a threshold for a measurement of wear, vibration, etc). Typically, this field is used for on condition maintenance tasks.
  - **Zone**: The zone of the system in which the task will be performed. Typically, this field is used for aircraft MSG-3 analyses.
  - **Access**: The access that will be required in order to perform the task. Typically, this field is used for aircraft MSG-3 analyses

To understand how inspections work during simulation, you should be aware of the following:

- Multiple non-downing inspections cannot occur on the same block at the same time.
- A non-downing inspection with a restoration factor greater than 0 restores the block based on the age of the block at the beginning of the inspection (i.e., the task duration is not restored).

9.4.4 On Condition Tasks

On condition maintenance relies on the capability to detect failures before they happen so that preventive maintenance can be initiated. If, during an inspection, maintenance personnel can find evidence that the equipment is approaching the end of its life, then it may be possible to delay the failure, prevent it from happening or replace the equipment at the earliest convenience rather then allowing the failure to occur and possibly cause severe consequences. In BlockSim, on condition tasks consist of an inspection task that triggers a preventive task when an impending failure is detected during inspection.

For on condition tasks, the Inspection properties allow you to configure the inspection portion of the task, and the options are identical to those available for configuring inspection tasks. The On Condition Task (Upon Detection) properties allow you to configure the preventive portion of the task. The options are the same as those available for configuring preventive tasks, except that the Task Scheduling properties are not available. Instead, Failure Detection properties are used to specify the "warning period" that spans from the time when a potential failure can first be detected to the time when the failure occurs.

To define the Failure Detection properties, first specify when the task is likely to detect an imminent failure. If you choose **When a certain percentage of the life of the item has been consumed**, you must then specify the percentage of the item's life that must have elapsed in order for approaching failure to be detected. This is called the *failure detection threshold* (FDT). For example, if the FDT is 0.9 and the item will fail at 1,000 days, the approaching failure can begin to be detected at 900 days. If you choose **Within a fixed time frame prior to failure**, you must then specify the amount of time before a failure when the approaching failure can be detected by inspection. This is called

the P-F interval. For example, if the *P-F interval* is 200 days and the item will fail at 1,000 days, the approaching failure can begin to be detected at 800 days.

Note the following simulation assumptions regarding on condition tasks:

- An inspection that finds a block at or beyond the failure detection threshold or within the range of the P-F interval will trigger the associated preventive task as long as preventive maintenance can be performed on that block.
- If a non-downing inspection triggers a preventive maintenance action because the failure detection threshold or P-F interval range was reached, no other maintenance task will be performed between the inspection and the triggered preventive task; tasks that would otherwise have happened at that time due to system age, system down or group maintenance will be ignored.
- A preventive task that would have been triggered by a non-downing inspection will not happen if the block fails during the inspection, as corrective maintenance will take place instead.
- If a failure will occur within the failure detection threshold or P-F interval set for the inspection, but the preventive task is only supposed to be performed when the system is down, the simulation waits until the requirements of the preventive task are met to perform the preventive maintenance.
- If the on condition inspection triggers the preventive maintenance part of the task, the simulation assumes that the maintenance crew will forego any routine servicing associated with the inspection part of the task. In other words, the restoration will come from the preventive maintenance, so any restoration factor defined for the inspection will be ignored in these circumstances.

9.4.5 Task Scheduling

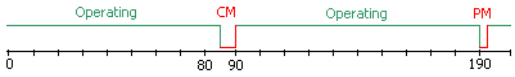
The following information applies to the task scheduling properties for preventive tasks, inspection tasks and the inspection portion of on condition tasks.

Click the arrow in the **When is this task performed?** field to define the criteria for performing the task. The remaining fields in the task scheduling properties will depend on your selection in this field. The task can be performed:

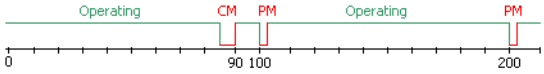
- **At certain intervals**. If you select this option, you must then specify:
  - Whether the interval is fixed or dynamic.
  - Whether the interval is based on the item's age or on calendar time.
- **Upon certain events**. If you select this option, you can choose for the task to be performed:
  - Whenever the system is down for any reason. No further task scheduling properties are required for this option.
  - Based on events in a maintenance group. The task will be performed when user-specified events occur in one or more user-specified maintenance groups. The item that the task is assigned to does not need to be part of the selected maintenance group(s).
  - When a maintenance phase in a phase diagram starts. Please note that if the task is selected to be performed only at the start of a maintenance phase and the item that the task is associated with is not included in the maintenance phase's associated maintenance template, the task will never be performed. No further task scheduling properties are required for this option. *For more information about maintenance phase blocks and templates, see page 342 in Chapter 10*.

The remaining task scheduling properties for each type of schedule are given next. For any interval that you specify, you will be asked to specify the time units used.

- **Fixed interval based on either item age or calendar time**: Enter the interval when the task will be performed (e.g., if you enter 100, the task will be scheduled for 100, 200, 300, etc.).
  - Item age refers to the accumulated age of the block, which gets adjusted each time the block is repaired (i.e., restored). If the block is repaired at least once during the simulation, this will be different from the elapsed simulation time. For example, if the restoration factor is 1 (i.e., "as good as new") and the assigned interval is 100 days based on item age, then the task will be scheduled to be performed for the first time at 100 days of elapsed simulation time. However, if the block fails at 85 days and it takes 5 days to complete the repair, then the block will be fully restored at 90 days and its accumulated age will be reset to 0 at that point. Therefore, if another failure does not occur in the meantime, the task will be performed for the first time 100 days later at 190 days of elapsed simulation time.



  - Calendar time refers to the elapsed simulation time. If the assigned interval is 100 days based on calendar time, then the task will be performed for the first time at 100 days of elapsed simulation time, for the second time at 200 days of elapsed simulation time and so on, regardless of whether the block fails and gets repaired correctively between those times.



  If the task is performed at fixed intervals, you can select the **Override task scheduling properties with a task package** option to assign the task to a task package, which is a group of tasks that are performed together at scheduled intervals. (Note that if a task has been assigned to a task package, the task scheduling properties will be disabled and displayed in italics.)

- **Dynamic interval based on either item age or calendar time**: The task will be performed at variable user-specified intervals. This can be used, for example, to schedule maintenance to be performed with increasing frequency as an item gets older. The **Dynamic intervals** field shows the number of intervals specified. Click the button (**...**) to open the Intervals window, which allows you to enter the item age intervals when the task will be performed. The intervals are the actual time between maintenance (i.e., they are not cumulative).
- **Based on events in a maintenance group**: Specify the events that can trigger the task (i.e., a block fails; a corrective, preventive or inspection task starts; a block is restored to operation), and choose the maintenance group(s) in which the event(s) must occur in order to trigger the task.

Note: For preventive and inspection tasks that are scheduled based on item age, you can use the **Perform this task even if the item failed before this task was scheduled to occur?** property to determine whether actual age or "virtual age" is considered. If you answer **Yes**, the task will be performed even if the item has already failed. In other words, the item's "clock" is not stopped upon failure, and the item's virtual age is used to trigger the task.

9.4.6 Multiple Tasks on the Same Block

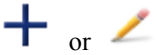
Certain additional simulation assumptions apply when multiple tasks (i.e., corrective, preventive, inspection and/or on condition tasks) apply to the same block:

- Downing events cannot overlap. For example, if a component is down due to preventive maintenance and a trigger suggests another preventive task, the suggestion is ignored.
- Non-downing events can overlap with downing events if they start at different times, provided that the non-downing event starts first. If the downing event starts first, then the non-downing event is ignored. If they start concurrently, the task with the highest priority will be performed.
- Preventive tasks and inspections cannot occur at exactly the time that the block or system comes up from another preventive task or inspection.
- If multiple tasks are scheduled to occur at the same time for a block, the maintenance will be scheduled as follows:
  1. Tasks based on intervals
  2. Tasks based on events in a maintenance group, where the triggering event applies to a block
  3. Tasks based on system down
  4. Tasks based on events in a maintenance group, where the triggering event applies to a subdiagram
- If a failure or preventive task occurs during a non-downing inspection, and the corrective task or the preventive task has a restoration factor, and the inspection task has a restoration factor, then both restoration factors are used (compounded).
- If preventive maintenance occurs after an inspection based on the failure detection threshold or P-F interval and both tasks have restoration factors, the restoration is compounded.
- If a corrective task is scheduled to be performed upon inspection and a preventive task occurs first, the preventive task will restore the block.
- If a corrective task is scheduled to be performed upon inspection and a failure or preventive task occurs during a non-downing inspection, the restoration is compounded.

9.5 Crews

Crews are resources that can be shared among analyses and can be managed via the Resource Manager. You can assign the crews to maintenance tasks.

The Crew window allows you to create, view and edit crews. It can be accessed by clicking the **Create New** or **View/Edit** icon in the Crew wizard, which is accessed from **Crew** fields in properties windows (e.g., the **Crew for task** field in the Maintenance Task window).

 or

It can also be accessed from the Crews page of the Resource Manager by choosing **Home > Edit > New**, by selecting a crew and choosing **Home > Edit > View** or by double-clicking a crew.

For a new resource, a name will be proposed automatically based on the default naming criteria established for the current database (in the Define Default Name Formats window). You can replace this with your own name of up to 150 characters, if desired. Remember that the name and identifiers are the primary way in which your team will be able to find the Synthesis resources you need for your analyses.

The following settings are available to configure the crew:

- **Crew**
  - **Direct cost** allows you to choose or create a model to represent the direct cost per unit time to engage the crew (e.g., the hourly charge for the time that the crew spends performing the task). This field uses a cost per unit time model. If no model is assigned, it is assumed that there is no cost.
  - **Cost per incident** allows you to choose or create a model to represent the cost per incident to engage the crew (e.g., if you have to pay a fixed fee instead of or in addition to the hourly rate). This field uses a cost model. If no model is assigned, it is assumed that there is no cost.
  - **Is there a limit to the number of tasks this crew can perform at the same time?**: If you answer **Yes**, you must specify the limit in the **Number of tasks** field. For example, if the crew can only fix 3 components at any given time and 4 components fail at the same time, then the crew will not be able to fix the fourth component until one of the other components has been restored.
  - **Logistic Delay** allows you to choose or create a model to describe the delay time before the crew can start the task. If no model is assigned, it is assumed that the crew can start the task immediately without any delays (i.e., immediate repair). The crew with the highest priority is always called upon first, regardless of logistic wait times for the crews. If a crew is needed and all crews are engaged, the crew with the shortest wait time is chosen to perform the task.

    For a given simulation, a crew's logistic time is constant across that one simulation for the task. It is taken randomly from its distribution or from its fixed time.

    For on condition tasks, if the same crew performs the inspection and preventive maintenance and there are logistic delays associated with the crew, the logistic delays will be factored into the simulation for both parts of the task.
  - If **Include logistic delay in cost** is selected, then the time spent waiting for the crew (including logistic delay and any delay incurred due to limits on the number of tasks the crew can perform simultaneously) will be included when calculating crew costs, based on the direct cost specified for the crew.
  - If **Include part delay in cost** is selected, then the time spent by the crew waiting for spare parts will be included when calculating crew costs, based on the direct cost specified for the crew.
- **Identifiers** allows you to enter additional identifying information that can be used to search for this resource.
- **History** provides information about the creation and last modification of the resource. If the history log has been activated at the project level (**Project > Management > History Log**), you can click the **View Item History** icon to open the Record History Log for the resource.

You can collapse or expand all nodes in the hierarchy using the **Collapse All** and **Expand All** icons.

You can resize the columns by pointing to the column splitter. When the pointer becomes a double-headed arrow, drag the splitter to the desired location. You can restore the column widths to their original settings by clicking the **Restore Column Widths** icon.

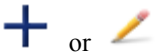
For an existing resource, a link at the bottom of the properties window indicates how many times it is currently being used. If you need more information, click the link or the **Trace Usage** icon.

This opens the Dependency Viewer, which allows you to explore both where the resource is being used and what other resources are associated with it.

9.6 Spare Part Pools

Spare part pools are resources that can be shared among analyses and can be managed via the Resource Manager. A spare part pool describes the conditions that determine whether a spare part will be available when needed and specifies the time and costs associated with obtaining the spare part. You can assign the pools to maintenance tasks.

The Spare Part Pool window allows you to create, view and edit spare part pools. It can be accessed by clicking the **Create New** or **View/Edit** icon in the Spare Part Pool wizard, which is accessed from **Spare Part Pool** fields in properties windows (e.g., in the Maintenance Task window).



It can also be accessed from the Spare Part Pools page of the Resource Manager by choosing **Home > Edit > New**, by selecting a spare part pool and choosing **Home > Edit > View** or by double-clicking a spare part pool.

For a new resource, a name will be proposed automatically based on the default naming criteria established for the current database (in the Define Default Name Formats window). You can replace this with your own name of up to 150 characters, if desired. Remember that the name and identifiers are the primary way in which your team will be able to find the Synthesis resources you need for your analyses.

The following settings are available to configure the spare part pool:

- **Spares**
  - **Direct cost per dispensed item** allows you to specify the direct cost of each spare part in the pool.
  - **Spare acquisition type**: You can select **Unlimited spares** to indicate that an unlimited number of spare parts exist in the pool (i.e., if a spare part is required, it will always be available). If you select **Limited number of spares**, the Pool Restock properties and the Emergency Spare Provisions properties will become available. In addition, you will need to define the following properties:
    - **Initial stock level** is the number of spare parts in the pool at the start of simulation.
    - **Holding Cost ($/Hour in pool)** allows you to choose or create a model to represent the cost of holding a spare part in the pool. The time unit used is the default unit for the database. This field uses a cost per unit time model. If no model is assigned, it is assumed that there is no cost. (To determine which time unit is defined as the default, you can open the Define Units window by choosing **File > Manage Repository > Manage Units**. The default unit is the one that is selected in the Use as Default column.)
    - If you select the **Pool has maximum capacity** check box, you will need to specify the maximum number of items that can exist in the spare part pool at any given time. The restock options will add parts only up to this maximum capacity. For example, suppose that the maximum capacity is 20 units and the restock quantity is 5 units. If the current stock level is 18 when a restock is triggered (either on a scheduled interval or because the stock fell to a specified level), then only 2 units will be added to the spare part pool, even though the restock options call for the addition of 5 units at each restock.

    If you select **Fixed probability of stockout**, the Emergency Spare Provisions properties will become available. In addition, you will need to enter the probability of running out of stock as a decimal value.

- **Logistic Time for Spare Acquisition** allows you to choose or create a model to describe the amount of time required to obtain the part when it is required for maintenance, assuming that the part is in stock. This field uses a duration model. If no model is assigned, it is assumed that the spare part is available for use immediately.

- The **Pool Restock** properties allow you to define how the spare part pool will be restocked. These properties are available only if you have selected the **Limited number of spares** option in the **Spare acquisition type** field. You can select either or both of the following restock schemes:
  - If the **Scheduled restock** option is selected, the pool will be restocked at fixed intervals. The following properties will be available:
    - **Restock every** allows you to specify the time interval at which the pool will be restocked.
    - **Number added per restock** allows you to specify the number of parts that are added to the pool at every scheduled interval.
  - If the **Restock as needed** option is selected, the pool will be restocked when the stock drops to a specified quantity. This option is available only if the initial stock level entered is at least 1. The following properties will be available:
    - **Restock when stock drops to** allows you to specify the number of parts in the pool that will trigger the restock (e.g., when the stock drops to 5, order more parts).
    - **Number added per restock** allows you to specify the number of parts that are added to the pool when the restock condition has been met.
    - **Required time for stock arrival** allows you to choose or create a model to describe the amount of time required for the new parts to arrive after the restock has been initiated. This field uses a duration model. If no model is assigned, it is assumed that the parts arrive immediately.
    - **Off-site spare part pool** allows you to assign an off-site spare part pool to the current pool. If specified, this pool is considered to be the source for restocking the current pool, which allows you to describe properties for the next level of spare parts that can be accessed when needed. If no off-site spares pool is specified, the parts are assumed to be ordered and no properties are described for the source.

Tip: Circular references between spare part pools are not permitted. In other words, if Pool 1 calls Pool 2 as its off-site spare part pool, and Pool 2 calls Pool 1 as its off-site spare part pool, BlockSim will display a message notifying you of the problem upon simulation.

- The **Emergency Spare Provisions** properties allow you to define the circumstances when the spare part pool is empty but a spare part is required for a task. These properties are available only if you have selected either the **Limited number of spares** option or the **Fixed probability of stockout** option in the **Spare acquisition type** field. Selecting the **Can obtain emergency spares if needed** option indicates that a spare part can be obtained if required and the spare part pool is empty; otherwise, the repair will be delayed until more spare parts arrive in the spare part pool. Selecting this option makes the following properties available:
  - **Number added per emergency** allows you to specify the number of parts that are added to the pool when an emergency acquisition occurs.
  - **Additional costs for emergency spares** allows you to choose or create a model to represent a per incident cost for emergency acquisition of spares (e.g., there is a $50 surcharge when emergency spares are required). This field uses a cost model. If no model is assigned, it is assumed that there is no cost.
  - **Required time for emergency spare** allows you to choose or create a model to describe the amount of time required for the emergency spare parts to arrive after the emergency acquisition has been initiated. This field uses a duration model. If no model is assigned, it is assumed that the parts arrive immediately.
  - **Off-site spare part pool** allows you to assign an emergency off-site spare part pool to the current pool. This allows you to create another level of spare parts that can be accessed for the emergency acquisition.

- **Identifiers** allows you to enter additional identifying information that can be used to search for this resource.
- **History** provides information about the creation and last modification of the resource. If the history log has been activated at the project level (**Project > Management > History Log**), you can click the **View Item History** icon to open the Record History Log for the resource.

You can collapse or expand all nodes in the hierarchy using the **Collapse All** and **Expand All** icons.

You can resize the columns by pointing to the column splitter. When the pointer becomes a double-headed arrow, drag the splitter to the desired location. You can restore the column widths to their original settings by clicking the **Restore Column Widths** icon.

For an existing resource, a link at the bottom of the properties window indicates how many times it is currently being used. If you need more information, click the link or the **Trace Usage** icon.

This opens the Dependency Viewer, which allows you to explore both where the resource is being used and what other resources are associated with it.

9.6.1 Spare Part Pools - Simulation Assumptions

The underlying simulation assumptions for spare parts are:

- Spare parts are always required for corrective and preventive tasks.
- If a spare part pool has not been assigned to the task, then it is assumed that unlimited free spares are always available.
- Spare parts are never required for inspection tasks.
- All restocking that occurs during simulation is based on the settings of the spare part pools.
- If the **Restock as needed** check box is selected, a restock will be triggered every time a part is taken from the spare part pool and the number of items in the pool drops below the limit specified in the **Restock when stock drops to** field.
- If the **Scheduled Restock** and **Restock as needed** check boxes are selected, then if a spare part from a scheduled restock can arrive before a spare part from an on condition restock, the spare part from the on condition restock will not be called upon.
- A block that triggers an emergency restock is charged for the additional emergency cost, even if it did not use the restock part(s) (e.g., another part became available from a scheduled restock first).
- The average stock level (ASL) metric in the simulation results is based on how many times the stock level was determined (e.g., every time a request is made, whether it is rejected or accepted, etc.). The average stock level is recorded at the beginning and at the end of each simulation.

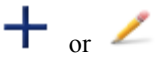
In addition, the following underlying simulation assumptions also apply to off-site spare parts:

- The spare acquisition time for both the on-site pool and off-site pool is taken into consideration when a block orders a spare part. If the off-site spare part will not arrive before the simulation end time, the block will not order it.
- If the off-site emergency spare part will not arrive before the simulation end time, neither the on-site pool nor the off-site pool will order it. However, an on condition restock will be performed as long as the part from the off-site pool will arrive at the on-site pool before the simulation end time.

9.7 Task Packages

Task packages are resources that are available for use throughout the project and can be managed via the Resource Manager. They represent groups of tasks that are performed together at scheduled intervals, for the most efficient allocation of resources and downtime management.

The Task Package window allows you to create, view and edit task packages. It can be accessed by clicking the **Create New** or **View/Edit** icon in the Task Package wizard, which is accessed from the **Task Package** field that appears when you select the **Override task scheduling properties with a task package** option for a preventive, inspection or on-condition task that is scheduled to be performed at fixed intervals.

 or

It can also be accessed from the Task Packages page of the Resource Manager by choosing **Home > Edit > New**, by selecting a task package and choosing **Home > Edit > View** or by double-clicking a task package.

For a new resource, a name will be proposed automatically based on the default naming criteria established for the current database (in the Define Default Name Formats window). You can replace this with your own name of up to 150 characters, if desired. Remember that the name and identifiers are the primary way in which your team will be able to find the Synthesis resources you need for your analyses.

The following settings are available to configure the task package:

- In the **Perform every** field, enter the length of the interval at which the task package will be performed. You must specify time units for this value.
- The **Auto-Packaging Intervals** fields allow you to specify a range of scheduled intervals that will automatically be incorporated into the task package if the user clicks the **Auto-Package Tasks** button at the bottom of the Maintenance Task Packaging window in RCM++/RBI.

  The time units for these fields are the same as those specified for the **Perform every** field. For example, if you enter **150** hours in the **Start time** field and **200** hours in the **End time** field, then tasks that are scheduled to be performed at fixed intervals from 150 to 200 hours can automatically be included in the task package to be performed at the scheduled time of the task package rather than at their individually scheduled times.
- **Identifiers** allows you to enter additional identifying information that can be used to search for this resource.
- **History** provides information about the creation and last modification of the resource. If the history log has been activated at the project level (**Project > Management > History Log**), you can click the **View Item History** icon to open the Record History Log for the resource.

You can collapse or expand all nodes in the hierarchy using the **Collapse All** and **Expand All** icons.

You can resize the columns by pointing to the column splitter. When the pointer becomes a double-headed arrow, drag the splitter to the desired location. You can restore the column widths to their original settings by clicking the **Restore Column Widths** icon.

For an existing resource, a link at the bottom of the properties window indicates how many times it is currently being used. If you need more information, click the link or the **Trace Usage** icon.

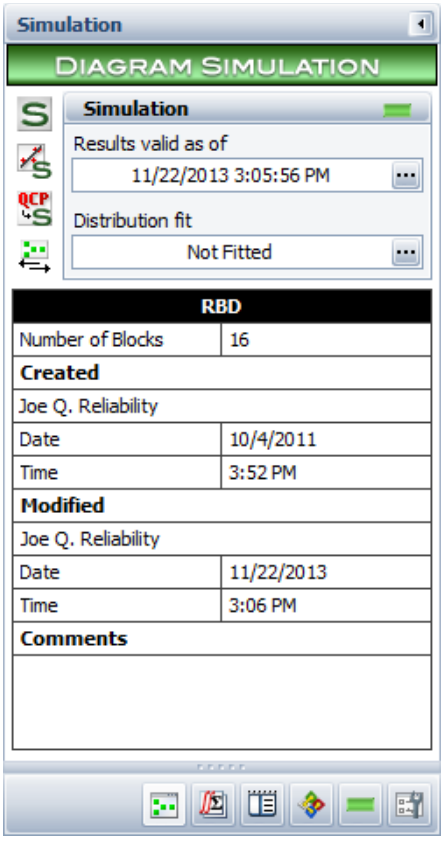
This opens the Dependency Viewer, which allows you to explore both where the resource is being used and what other resources are associated with it.

9.8 Simulation Diagram Control Panel

The control panel associated with BlockSim simulation diagrams consists of multiple pages, each containing options for performing particular tasks. This section focuses on the Simulation page and Analysis Settings page of the simulation diagram control panel, which contains most of the tools you will need to simulate a diagram. (*For more information about the control panel in general, see page 109 in Chapter 3*.)

9.8.1 Control Panel Simulation Page

The Simulation page of the control panel for simulation diagrams is shown next.



The Simulation page of the control panel contains the following options:

- The **Simulation** area displays the status and results of the simulation.
  - **Status** If the light is green, then the simulation has been performed. If the light is red, then the simulation has not been performed. A simulated diagram will remain in the simulated state unless a change is made to the structure of the diagram (e.g., a block is added, a connector is removed, a model is changed, etc.).
  - **Show Simulation Details** (**...**) displays the most recently generated simulation results in the Simulation Results Explorer.
  - **Distribution fit** displays the distribution that has been fitted to the simulation results. Click the button (**...**) to open the Distribution Estimator to fit a distribution and/or for information on the fitted distribution parameters. The fitted distribution can be used to describe the overall behavior of the system and can be published for use as a model elsewhere in the project.
- The **Information** area displays information about the diagram or the currently selected block. If no block is selected in the diagram, the **Information** area displays the number of blocks in the diagram and information about the diagram creation and last modification; in addition, if the diagram results have been published as a model, the name of the model will appear as a link allowing you to view the model in the Synthesis Viewer window.

  If a block is selected in the diagram, it displays a subset of the properties found in the Block Properties window. Items in the table that are resources (e.g., models, switches, etc.) are links; you can click such items to view and/or edit their properties. In both cases, this area also offers a **Comments** field; click the field heading to enter comments. For blocks, these comments are also accessible in the Identifiers properties in the Block Properties window.

- The **Tools** area on the left side of the control panel gives you quick access to the tools you will need to analyze the diagram and generate additional results.

Simulate opens the Maintainability/Availability Simulation window, which allows you to obtain reliability, maintainability, availability and throughput results from simulations.

Plot creates a plot based on the simulation results. Clicking the **Plot** icon before the diagram has been simulated will automatically perform the simulation and then plot the data.

Simulation QCP opens the simulation Quick Calculation Pad (QCP). The QCP allows you to obtain reliability, maintainability and availability metrics based on the simulation results. Clicking the **QCP** icon before the diagram has been simulated will automatically perform the simulation and then open the QCP.

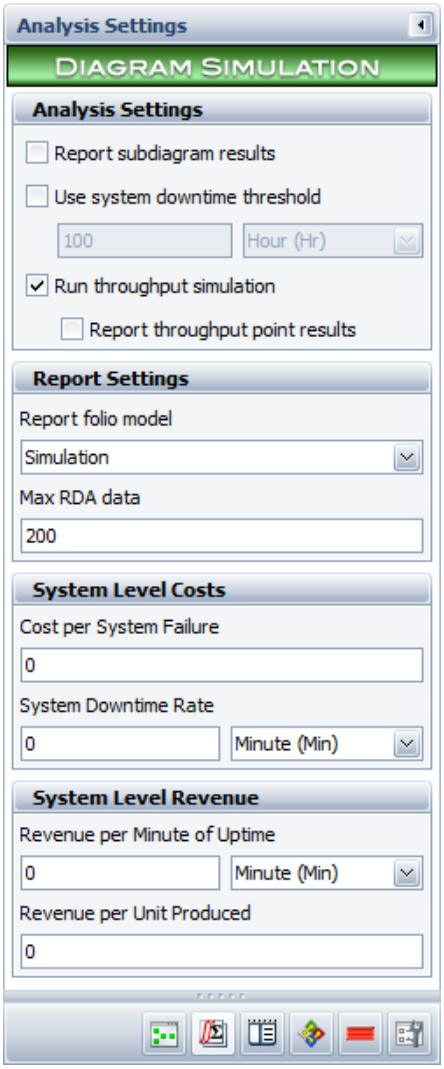
Change Diagram Type changes the current diagram to the opposite diagram type (i.e., changes the current analytical diagram to a simulation diagram, or the current simulation diagram to an analytical diagram). The subdiagram setting used in the change will be the one you specified the last time you used the Change Diagram Type window (**Diagram > Diagram Actions > Change Diagram Type**). In this window, you have the following options:

  - Select **Duplicate using the new type** to create a new diagram using the new type. The new diagram will be named "Copy of [original diagram name]." If you do not select this option, the original diagram itself will be changed to the new type.
  - Select **Perform the same change to any subdiagrams** to change any subdiagrams to the new diagram type. Note that if you select both this and **Duplicate using the new type**, new subdiagrams will be created along with the new main diagram.

  Note that if you create an analytical diagram from a simulation diagram, by either duplication or conversion, the maintenance and throughput characteristics defined in the simulation diagram will not be displayed in the analytical diagram, as they are not relevant to the analysis. However, the properties are not lost, and if you then create a simulation diagram from that analytical diagram, all original properties will be retained.

9.8.2 Control Panel Analysis Settings Page

The Analysis Settings page of the control panel for simulation diagrams is shown next.



The Analysis Settings page of the control panel contains the following options:

- **Analysis Settings**
  - Select **Report subdiagram results** to provide calculated results for subdiagram blocks in the Simulation Results Explorer. If this check box is not selected, subdiagram blocks will not appear in the results.
  - **Use system downtime threshold** allows you to specify a minimum amount of time that the system must be down in order for it to count toward critical downtime metrics.
  - Select **Run throughput simulation** to perform the throughput simulation, which allows you to perform analyses on the number of units that can be processed by the system in the diagram during the simulation time. If you select to run a throughput simulation, you can select **Report throughput point results** to include throughput results in the System Point Results worksheet, which provides results at specified points in time. If this check box is not selected, only the throughput results for the system at the simulation end time will be reported.

- **Report Settings**
  - **Report folio model** allows you to specify the results model that will be used when the diagram is used as the default data source for an analysis workbook or a Word report. You can choose to use either the simulation results or the fitted model.
  - **Max RDA data** allows you to specify the maximum number of events that can be shown in the RDA Data worksheet (which shows the system up/down data for each simulation) in the Simulation Results Explorer. The contents of the RDA Data worksheet can be transferred to a parametric RDA folio in Weibull++ for further analysis.
- **System Level Costs**
  - **Cost per System Failure** allows you to specify a failure cost per incident at the system level (i.e., an additional cost that is not accounted for by any of the block level costs).
  - **System Downtime Rate** allows you to specify a cost per unit time for downtime at the system level (i.e., an additional cost that is not accounted for by any of the block level costs).
- **System Level Revenue**
  - **Revenue per [Unit] of Uptime** allows you to specify an amount of revenue that is brought in by the system for each unit of uptime. This can then be accounted for as a loss when the system is down.
  - **Revenue per Unit Produced** allows you to specify an amount of revenue that is brought in for each unit produced. This can then be accounted for as a loss when the system is down. This option is available only when you have selected the **Run throughput simulation** option in the **Analysis Settings** area of this control panel page.

9.9 Maintainability/Availability Simulation Window

BlockSim's simulations are based on the component characteristics and the reliability-wise system configuration of the diagram. To simulate a diagram, choose **Simulation > Simulation > Simulate** or click the **Simulate** icon.

The Maintainability/Availability Simulation window will open, allowing you to specify the settings to be used in simulating an RBD or fault tree. In addition, it displays the progress of the simulation in the area at the top of the window.

There are two basic types of simulation that you can perform: **standard simulations** and **log of simulations**.

- When you perform standard simulations, the metrics in the Simulation Results Explorer are obtained by calculating the average of the results from all of the simulation runs.
- Saving a log of simulations, on the other hand, provides a set of individual simulation results that can be used to analyze variability within the simulation results and a set of average results in the Simulation Results Explorer.

  For more information about how you might use the log of simulations, please read "An Application of BlockSim's Log of Simulations" at: http://www.weibull.com/hotwire/issue97/relbasics97.htm. (Note that this article uses BlockSim 7, but the theory remains the same.)

In addition to having different purposes, the simulations are based on different randomly generated numbers. To understand this, suppose you are interested in the reliability/maintainability performance for a system and you want to use simulation to estimate useful metrics, such as the mean availability and the total expected downtime after 1,000 hours of operation.

Assume that you run 100 simulations for the 1,000 hour mission time using standard simulations and a seed of 1. In this case, the simulation routine will draw from a single stream of randomly generated numbers, generated using the seed of 1, for the series of 100 simulations. Therefore, the results will be reproducible if you perform the exact same set of simulations again, using the same seed, for the exact same diagram.

Now suppose that you want to perform a more in-depth exploration that will help you to analyze the variability within the simulation results. For that purpose, let's assume that you select to save a log of simulations using 100 seeds with a starting seed of 1. BlockSim will perform one simulation using the seed of 1, then will increment the seed by 1 and perform a second simulation using a seed of 2 (1 + 1 = 2), and so on until the utility has performed 100 separate simulations. In this case, each simulation draws from a separate stream of randomly generated numbers. The first simulation will utilize a stream with seed = 1, the second simulation will utilize a stream with seed = 2, and so on. Because of this, the average results that are shown in the Simulation Results Explorer when you have selected to save a log of simulations will not be the same as those that are shown when you have performed basic simulation.

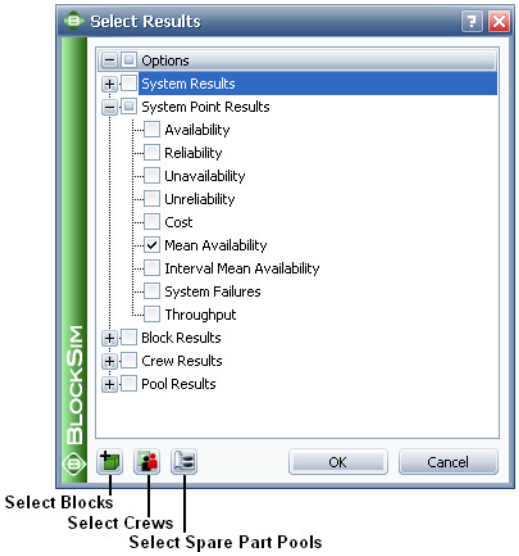
To perform each of these types of simulation, select the simulation type in the **Advanced Options** drop-down and then do the following:

- For all simulation types:
  - Enter the **Simulation End Time**, which is the duration of the simulation. Specify the time and the units of time used. Simulations begin at time = 0.
  - In the **Point Results Every** field, specify the frequency at which the simulation will return results. For example, if you have entered **100 hours** in the **Simulation End Time** field and you enter **10 hours** in the **Point Results Every** field, the simulation will run from 0 to 100 hours and point results will be available at 10 hours, 20 hours, 30 hours and so on.

In addition:

- For standard simulations with a fixed number of simulations:
  - Enter the **Number of Simulations** to be performed.
  - If you need your results to be repeatable, select the **Use a Seed** check box in the **Advanced Options** drop-down and enter a seed, which specifies a starting point from which the random numbers used in simulation will be generated. The same random numbers and, therefore, the same simulation results will be generated when the same seed value is used.
- For standard simulations with a variable number of simulations:
  - Specify the **Maximum Number of Simulations** to be performed.
  - In the **Convergence Criteria** field, specify the standard deviation for a key simulation result. The simulation will continue until this standard deviation target is reached in the simulation results or until the maximum number of simulations have been performed, whichever comes first.
  - If you need your results to be repeatable, select the **Use a Seed** check box in the **Advanced Options** drop-down and enter a seed, which specifies a starting point from which the random numbers used in simulation will be generated. The same random numbers and, therefore, the same simulation results will be generated when the same seed value is used.

- For log of simulations:
  - Specify the **Number of Seeds** to be used (i.e., the number of simulations).
  - Clicking the **Select Results** link opens the Select Results window, which allows you to select the items you want to log individual simulation results for and select which results for those items you want to save in the log.



    In the **Options** area, you can select or clear individual results or select/clear the check box for a result category to select/clear all of the results in that category. You also can select or clear all results by selecting/clearing the Options check box.

    In the lower left corner of the window, click the button corresponding to the desired item type to open a window that allows you to select the items of that type for which results will be saved in the log. For example, click Select Blocks to open the **Select Blocks** window. To select a block so that results for it will be saved in the log, select the box next to the block's name and click **OK**. Selecting or clearing the check box beside the category name will select or clear all of the items.

    If you have selected to save a block result, crew result or pool result, you must specify at least one item of the corresponding type to save results for. For example, if you have selected to save block downtime results but have not selected any blocks to save results for, a warning message will appear telling you that there are no results to be displayed. Click **OK** and select a block.

  - In the **Log of Simulations Results** area, select whether you want the log sent to Excel or saved as a tab-delimited text file. If you save the log as a tab-delimited text file, you can choose to save it as an embedded attachment within the project or you can click **Browse** to specify an external location and filename for the file.
  - If you need your results to be repeatable, select the **Use Starting Seed** check box in the **Advanced Options** drop-down and enter the seed to be used for the first simulation. This specifies a starting point from which the random numbers used in the simulation will be generated. The same random numbers and, therefore, the same simulation results will be generated when the same seed value is used. BlockSim will perform one simulation using the starting seed, then will increment the seed by 1 and perform a second simulation using that seed and so on until the utility has performed the number of simulations specified in the **Number of Seeds** field.

If you have the multi-thread version of BlockSim, then for any simulation type, you can run simulations in multiple threads, which may improve performance and save time when simulating complex diagrams. The **Number of Threads** field in the **Advanced Options** drop-down allows you to specify the number of threads to use for simulation. In general, using twice as many threads as the number of cores in your computer is appropriate; a ratio of threads to cores greater than this 2:1 ratio is likely to degrade performance. To have the application automatically

determine the number of cores in your system and set the number of threads accordingly, click the **Calculate** icon beside the **Number of Threads** field.

The number of threads you use affects repeatability of results. In order for your results to be repeatable, you must use the same number of threads and the same seed each time you simulate the diagram. When you are using multithreading, the seed that you specify in the **Advanced Options** drop-down is the seed for the first thread; seeds for additional threads are sequential (e.g., if you are using 3 threads and your starting seed is 7, the threads will use 7, 8 and 9 as the starting points for random number generation).

The **Analysis Settings** link at the bottom of the Maintainability/Availability Simulation window displays a message that shows the settings you have specified on the Analysis Settings page of the diagram's control panel. These settings determine whether subdiagram results will be included in the results; whether a throughput simulation will be performed and, if so, whether point results will be reported; and whether a system downtime threshold has been specified. You can edit these settings via the diagram's control panel.

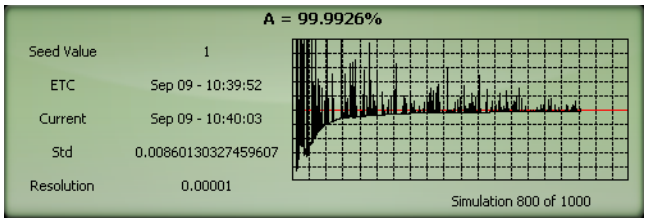
The area at the top of the Maintainability/Availability Simulation window displays the status of the simulation.

During simulation, you can click **Cancel** to stop the simulation at the current point. The results will be provided for the simulations (if any) that were completed before you stopped the process. When simulation is complete, you can click the **Details** button to view the results of the simulation in the Simulation Results Explorer. Click the **Simulate** button to re-run the simulation.

Note: Once you have simulated a diagram, the settings you have specified in the Maintainability/Availability Simulation window are saved along with the diagram so that you or another user can easily simulate the diagram again using the same settings. The first time you simulate a diagram, the settings default to the settings used for the last simulation that was performed on your computer.

9.9.1 Simulation Status Display

The area at the top of the Maintainability/Availability Simulation window displays the status of the simulation.



The following numerical information is always shown:

- **Seed Value** displays the seed value used or, for log of simulations, the starting seed value.
- **ETC** displays the expected date and time of completion for the simulation.
- **Current** displays the current date and time during the simulation. Once all simulations have been performed, this value will display the actual time of completion.

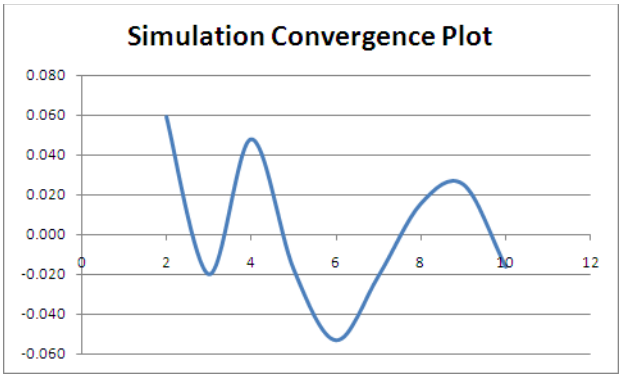
In addition, the current mean availability is shown at the top of the simulation status display. The current number of simulations performed out of the total number of simulations scheduled to be performed (or, in the case of a variable number of simulations, the maximum number of simulations) is shown at the bottom of the simulation status display (e.g., 10 out of 100 simulations).

If you have selected **Full Display** in the **Display Options** drop-down, a plot will also be shown in the simulation status display. This plot shows, for each simulation, the difference between the previous mean availability and the current mean availability, as a percentage of the current mean availability. For example, consider the following table:

Current Simulation	Availability in Current Simulation	Mean Availability Through Current Simulation	Value Shown in Simulation Convergence Plot
1	0.8	0.8	-
2	0.9	0.85	0.059
3	.8	0.833	-0.02
4	1	0.875	0.048
5	0.8	0.86	-0.017
6	0.6	0.817	-0.053
7	0.7	0.8	-0.021
8	0.9	0.813	0.015
9	1	0.833	0.025
10	0.7	0.82	-0.016

To understand how these values are obtained, we can look at simulation 3 as an example. The mean availability at the end of simulation 3 is the average of the availability in simulation 1, simulation 2 and simulation 3, or (0.8 + 0.9 + 0.8) / 3 = 0.833. The value shown in the convergence plot is obtained by subtracting the previous mean availability from the current mean availability and then dividing that number by the current mean availability: (0.833 - 0.850) / 0.833. In this case, this yields a negative value, because the current mean availability is less than the previous mean availability.

The table above would result in the following convergence plot:



As additional simulations are performed, the general pattern is that the mean availability values start to converge, and the line in the convergence plot stabilizes to a straight line. After this point, additional simulations are not especially beneficial.

You can use the **Resolution** field in the **Display Options** drop-down to specify the scaling of the convergence plot. The larger the number, the greater the range of values shown on the y-axis of the plot. The **Update Increments** field

specifies how many times the label below the plot, reflecting the current number of simulations performed out of the total number of simulations, will be updated during the simulation process.

The following additional information is also displayed when **Full Display** is selected:

- **Std** shows the current value in the convergence plot (i.e., the value obtained by subtracting the previous mean availability from the current mean availability and then dividing that number by the current mean availability).
- **Resolution** shows your choice for the scaling of the plot.

9.9.2 Repeatability of Simulation Results

Results obtained through repeated simulations of the same diagram will exhibit some variability because they depend on the randomly generated numbers that are selected for any given simulation run. The use of a seed in simulation forces the software to use the same sequence of random numbers in each simulation, resulting in repeatability. However, you need to be aware that when you use a seed, the same stream of random numbers is utilized, but the order of the utilizations of the stream may differ. This is dependent on the way the diagram was constructed and, for RENO flowcharts, the order in which RENO functions and static functions were created. The random numbers are assigned based on an internal ID number assigned to each block or resource when it was created. For blocks, you can display these internal IDs in the diagram by choosing a display format for the Block ID option on the Background and Grid page of the Diagram Style Window (**Format > Styles > Diagram Style**).

For example, two people may have projects containing identical diagrams using blocks A and B. If in one diagram, A was created before B, A will take the first random value from the stream and B will take the second. If the other person (in the other project) created the same diagram but created B before A, B will take the first random value from the stream and A will take the second, thus yielding potentially different answers, even though both analysts may have used the same seed in simulation. Note that as the number of simulations increases, this difference will be less significant.

9.9.3 Simulation Batch Mode

BlockSim offers a simulation batch mode, which allows you to simulate multiple diagrams consecutively, without user input in between simulations. This enables you to set up a series of simulations and then turn your attention elsewhere, which may be particularly helpful when you need to simulate several very large diagrams.

To use the simulation batch mode, choose **Simulation > Tools > Batch Mode**.

The Batch Mode Simulation window appears; this window displays a list of the available simulation diagrams in the current project (i.e., simulation RBDs, simulation fault trees and phase diagrams). Select the diagrams that you want to simulate; you can select the check box in the Available Diagrams header to select all diagrams.

Each diagram that is selected for simulation in batch mode must have its own simulation settings defined, including:

- The settings specified on the Analysis Settings page of the diagram's control panel. These settings determine whether subdiagram results will be included in the results; whether a throughput simulation will be performed and, if so, whether point results will be reported; whether a system downtime threshold has been specified; and additional system level costs and revenue values.
- The settings specified in the Maintainability/Availability Simulation window. These settings include the simulation end time, point results intervals, the number of simulations to be performed, repeatability options and multithreading options.

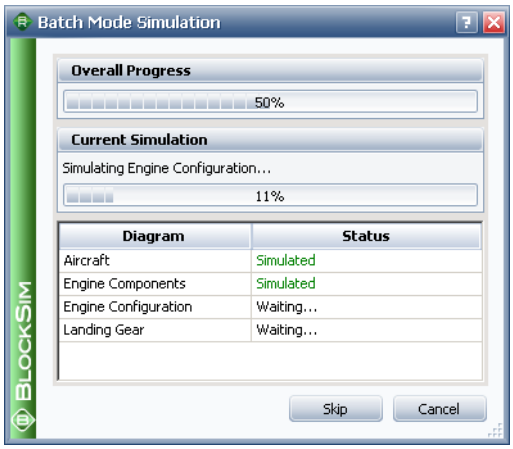
For diagrams that have previously been simulated, the most recent simulation settings are retained. For diagrams that have not yet been simulated, you will need to define these settings. A status light is shown for each available diagram.

If the light is green, simulation settings have been defined for the diagram. If it is red, settings have not yet been defined or are not compatible with batch mode (e.g., the last simulation run was a log of simulations, which is not available in batch mode). To define the simulation settings for one or more diagrams, highlight the diagram row(s) and then click the **Edit Simulation Settings** button at the bottom of the window.

The Batch Mode Simulation Settings window will appear, allowing you to define the settings (this window is a shortened version of the Maintainability/Availability Simulation window, lacking the simulation status display). The settings you define here will be applied to all of the diagrams that were highlighted in the Batch Mode Simulation window. If you want to apply the settings to all of the diagrams that are selected to be run in the batch simulation (i.e., the "checked" diagrams), select the **Apply settings to all highlighted and selected diagrams** check box. Click **OK** to save the settings for the diagram and return to the Batch Mode Simulation window. If you need to adjust the settings specified on the Analysis Settings page of the diagram's control panel, you must exit the Batch Mode Simulation window and adjust the settings for the diagram. Note that if you do not define simulation settings for all diagrams selected to be run in the batch simulation, then when you begin to run the simulations, the Batch Mode Simulation Settings window will be shown for each diagram that needs simulation settings.

Note: The log of simulations simulation type is not available in batch mode.

Click **Run** to run the simulations. The Batch Mode Simulation window will change to show the status of the simulations, as shown next. While simulation is in progress, you can click **Skip** to stop simulating the current diagram and move on to the next one or **Cancel** to stop the simulations entirely.



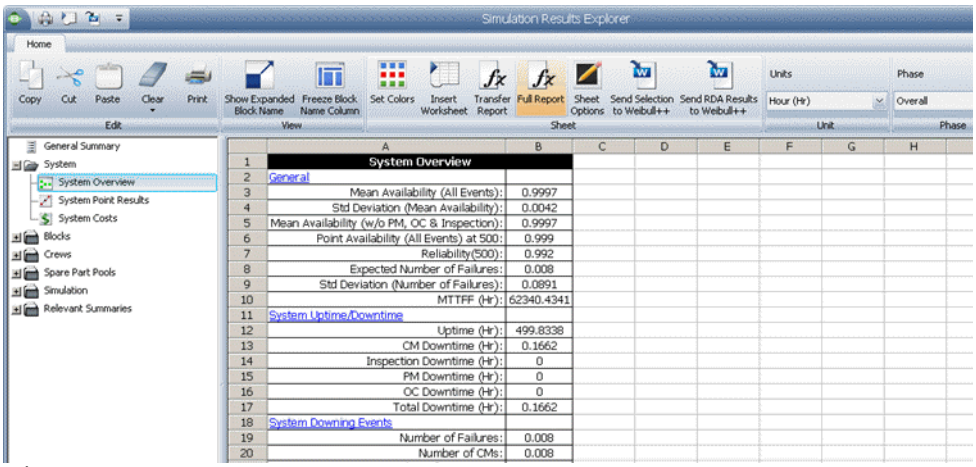
Once the simulations have been completed, you can click **Back** to return to the original view of the Batch Mode Simulation window, where you can select different diagrams or change simulation settings, or **Run** to run the batch simulation process again with the same settings.

You will not be able to perform other operations in the software while the simulations are being performed.

9.10 Simulation Results Explorer

After a simulation diagram or RENO flowchart has been simulated, you can view the results in the Simulation Results Explorer. To access these results from the Maintainability/Availability Simulation window or the RENO Simulation window, click the **Details** button. To access from within any simulation diagram, click the **Show Simulation Details** (**...**) icon on the control panel or choose **Simulation > Tools > Show Simulation Details**.

The navigation panel on the left side of the window shows which results are available from the current simulation. Click any node to display those specific results in the worksheet on the right.



Note the following:

- New simulation results replace all previous results and only the most recent results are saved with the diagram. If you wish to store the current results and then resimulate, you can use the **Transfer Report** command to send the results to an analysis workbook or to Excel before you start the next simulation.
- For BlockSim diagrams, if you selected to perform multiple simulations, some of the results (e.g., expected number of failures, etc.) will be decimals because they are averaged across all simulations. If you selected to perform one simulation, those results will be whole numbers.
- Time Units:
  - For BlockSim diagrams, simulation results that are time-based (e.g., downtime, time to event, etc.) are given in terms of the time unit that is specified in the Unit group of the Ribbon. You can select a different time unit from this drop-down list at any time and the results will automatically be converted to the new time unit.
  - For RENO flowcharts, all results are given in terms of the system base unit (SBU). *For more information, see Chapter 11 on page 354.*

9.10.1 Ribbon Commands

In addition to standard commands for cut, copy, paste, clear and print, the Ribbon in the Simulation Results Explorer also provides the following:

View

Show Expanded Block Name applies only for subdiagram blocks. It displays the name of the diagram represented by a subdiagram block in the Block Name column.

Freeze Block Name Column freezes the Block Name column in the worksheet so that it remains visible as you scroll to the right to view additional columns. If not selected, the Block Name column may not remain visible, depending on your screen resolution and the sizing of the window.

Sheet

Set Colors opens the Set Report Colors window, which allows you to set the fill and colors for worksheet elements such as column titles, sub-headings, etc. These settings are defined separately for BlockSim diagrams and for RENO flowcharts, and apply to all diagrams of that type.

Insert Worksheet inserts a new, blank worksheet where you can perform any calculations desired. Note that this worksheet will not be retained when you close the Simulation Results Explorer.

Transfer Report allows you to send the contents of the Simulation Results Explorer to an analysis workbook or to Excel. In the Transfer Report window that appears, "current item" refers to the simulation results worksheet that is currently selected, while "full report" refers to all worksheets in the Simulation Results Explorer. If you have simulated a phase diagram, the results that are transferred for "current item" or "current phase" are based on your selection in the Phase group of the Ribbon, while "full report" refers to all phases.

Full Report opens all of the currently available worksheets in separate tabs in the Simulation Results Explorer. You can then use either the navigation panel or the tabs to switch between worksheets.

Sheet Options opens the Sheet Options window, which allows you to specify the appearance of the current worksheet and which options are allowed within it. Note that your selections will not be retained when you close the Simulation Results Explorer.

Send Selection to Weibull++ sends the selected data to a standard folio in Weibull++. You must have Weibull++ activated on your computer in order to view the resulting standard folio. To use this functionality, one or two columns of data must be selected in the worksheet.

Send RDA Results to Weibull++ is available only for BlockSim diagrams. It transfers the data in the RDA Data worksheet to a Weibull++ parametric RDA folio within the current project. If Weibull++ is installed on your computer, you can then perform parametric recurrent event data analysis on the data set. The parametric RDA folio uses the General Renewal Process (GRP) model to analyze the failure behavior of a repairable system. The GRP analysis method takes into account the effectiveness of repairs on the condition of the system, and models the cumulative number of failures over time.

Send RGA Results to RGA is available only for BlockSim diagrams. It transfers the data in the RGA Data worksheet to an RGA folio using the Multiple Systems - Concurrent Operating Times data type within the current project. If RGA is installed on your computer, you can then perform reliability growth analysis on the data set.

Units

For BlockSim diagrams only, the units drop-down list allows you to specify the time units used for the simulation results. When you choose a new time unit, the software performs the unit conversion automatically. Authorized users can define the time units that will be available in the database and set up the conversion factors. *See page 64 in Chapter 2*.

Phase

For BlockSim phase diagrams only, the phase drop-down list allows you to select the phase that you want to view simulation results for. You can select an individual phase or select **Overall** to view results for all phases.

9.10.2 Set Report Colors Window

The Set Report Colors window allows you to set the colors that appear in the worksheets in the Simulation Results Explorer. This window can be opened by choosing **Home > Sheet > Set Colors** in the Simulation Results Explorer.

The **Preview** area on the left side of the window displays the colors set for the report. The color boxes on the right side of the window allow you to change the corresponding fill and text colors. Click the corresponding color box to open the color palette, which allows you to set the color.

- **Title** displays the fill and text color for the titles in the Simulation Results Explorer worksheets.
- **Heading** displays the fill and text color for the headings in the Simulation Results Explorer worksheets.

- **Odd Column** displays the fill and text color for the odd columns in the Simulation Results Explorer worksheets.
- **Even Column** displays the fill and text color for the even columns in the Simulation Results Explorer worksheets.
- **Reset** resets the report colors to the default colors that were shipped with the application.

9.10.3 General Summary Results

The General Summary worksheet in the Simulation Results Explorer presents an overview of the simulation settings and total results generated for the system, the blocks, the crews and the spare part pools.

- **Simulation Summary** displays the same information that is displayed in the Simulation Summary worksheet - Section 9.10.8 (p. 324).
- **System Overview** displays the same information that is displayed in the System Overview worksheet - Section 9.10.4.1 (p. 311).
- **Individual Block Summary** displays the same information that is displayed in the Block Summary worksheet - Section 9.10.5.1 (p. 315).
- **Crew Summary** displays the same information that is displayed in the Crew Summary worksheet - Section 9.10.6.1 (p. 321).
- **Spare Part Pool Summary** displays the same information that is displayed in the Spare Part Pool Summary worksheet - Section 9.10.7.1 (p. 323).

9.10.4 System Results

9.10.4.1 System Overview

The System Overview worksheet in the Simulation Results Explorer presents an overview of the calculations performed for the entire system. The values shown are averages of the results of all of the simulations.

- **General**
  - **Mean Availability (All Events)**: Includes all system downing events during the time of the simulation. The standard deviation of this value is provided in the **Std Deviation (Mean Availability)** field.
  - **Mean Availability (w/o PM, OC & Inspection)**: Includes all downing events, but not preventive, on condition or inspection tasks, during the time of the simulation.
  - **Point Availability (All Events) at [Time]**: The point availability of the system, including all downing events, at the simulation end time.
  - **Reliability ([Time])**: The reliability of the system at the simulation end time.
  - **Expected Number of Failures**: The total number of failures expected to occur during the simulation for the system. The standard deviation of this value is provided in the **Std Deviation (Number of Failures)** field.
  - **MTTFF**: The mean time to first failure for the system.
  - **MTBF (Total Time)**: The mean time between failures for the system, based on the total simulation time and including all system downing events.
  - **MTBF (Uptime)**: The mean time between failures for the system, considering only the time that the system was up. This is calculated by dividing system uptime by the number of system failures. You can also think of this as the mean uptime.

- **MTBE (Total Time)**: The mean time between all downing events for the system, based on the total simulation time and including all system downing events. This is calculated by dividing the simulation run time by the number of downing events.
- **MTBE (Uptime)**: The mean time between all downing events for the system, considering only the time that the system was up. This is calculated by dividing system uptime by the number of downing events.

Note: To better understand the difference between total and uptime consider the case of a system that has two time reading displays. One display is based on the actual system run hours (like a Hobbs meter – or time-on meter) and the other is a real-time chronograph recording the true time elapsed. Obviously, as the system goes up and down, the two will no longer be in sync. For example, imagine that the system had two failures and the real time display reads 1000 hours elapsed time while the Hobbs meter reads 500 hours. The MTBF based on total time would be 1000/2=500 hours, while the MTBF based on uptime would be 500/2=250 hours.

- **System Uptime/Downtime**
  - **Uptime**: The total amount of time that the system was up (i.e., operational) during the simulation.
  - **CM Downtime**: The amount of time that the system was down (i.e., not operational) due to corrective tasks during the simulation.
  - **Inspection Downtime**: The amount of time that the system was down (i.e., not operational) due to an inspection during the simulation.
  - **PM Downtime**: The amount of time the system was down (i.e., not operational) due to preventive tasks during the simulation.
  - **OC Downtime**: The amount of time the system was down (i.e., not operational) due to on condition tasks during the simulation.
  - **Total Downtime**: The total amount of time that the system was down (i.e., not operational) during the simulation.
  - **Critical Downtime**: The total amount of time that the system was down where the length of the downtime met or exceeded the system downtime threshold, as specified on the Analysis Settings page of the diagram's control panel.
- **System Downing Events** displays information for all events associated with system downtime that occurred during the simulation. Note that events with zero duration will appear in this section only if the task properties specify that the task brings the system down or if the task properties specify that the task brings the item down and the item’s failure brings the system down.
  - **Number of Failures**: The total number of failures that occurred for the system during the simulation time.
  - **Number of CMs**: The total number of corrective tasks performed on the system during the simulation time.
  - **Number of Inspections**: The total number of inspection tasks performed on the system during the simulation time.
  - **Number of PMs**: The total number of preventive tasks performed on the system during the simulation time.
  - **Number of OCs**: The total number of on condition tasks performed on the system during the simulation time.
  - **Number of OFF Events by Trigger**: The total number of events where the system is turned off by state change triggers. An OFF event is not a system failure but it may be included in system reliability calculations. *For more information on state change triggers, see page 216 in Chapter 7.*
  - **Total Events**: The total number of system downing events that occurred during the simulation time.

- **Costs**
  - **Total Costs**: The total costs that accrued for the system during the simulation time, which is based on the costs for tasks, crews and spare part pools.
  - **Opportunity Costs**: Potential system revenue lost as a result of downtime. This is based on the lost revenue per hour of uptime plus the lost revenue per unit produced (if a throughput simulation has been run). Both are specified on the Analysis Settings page of the diagram control panel
- **Throughput** results will be calculated only if you selected to run a throughput simulation on the Analysis Settings page of the diagram control panel.
  - **Total Throughput**: The total number of units that were processed/produced by the system during the simulation time.
  - **Max. Capacity**: The maximum number of units the system could have processed/produced during the total simulation time if no downtime was considered.
  - **Actual Utilization**: The ratio of actual units produced (Total Throughput) to the maximum that could have been produced (Max. Capacity), expressed as a percent (i.e., Total/Max)
- **Revenue**
  - **Total Revenue**: The total system revenue, including uptime revenue and throughput revenue (If a throughput simulation has been run), based on the System Level Revenue settings on the Analysis Settings page of the diagram control panel.

9.10.4.2 System Point Results

The System Point Results worksheet in the Simulation Results Explorer displays a table of the point reliability and availability results for the system, including all system downing events that occurred up to the specified points in time.

> **Note:** For phase diagrams, this worksheet is available only when you have selected **Overall** from the **Phase** drop-down list in the Ribbon.

- **Time**: The time for each corresponding point availability and reliability value. The points are determined based on the interval specified in the **Point Results Every** field in the Maintainability/Availability Simulation window. The number of points is determined by dividing the simulation time according to that increment, up to the end time you specified in the window.
- **A(t)**: The point availability of the system. This is displayed as a decimal.
- **R(t)**: The point reliability of the system. This is computed based on the first system failure and is displayed as a decimal.
- **(1-A(t))%**: The point unavailability of the system. This is displayed as a percentage.
- **(1-R(t))%**: The point unreliability of the system. This is displayed as a percentage.
- **Labor Cost(t)**: The cost associated with crews at the corresponding system time. This is based on the direct cost per unit time and cost per incident, as specified in the Crew window.
- **Part Cost(t)**: The cost associated with spare parts at the corresponding system time. This is based on the direct cost per item as specified in the Spare Part Pool window, as well as any indirect costs (e.g., holding costs, costs associated with restocking, etc.).
- **Cost(t)**: The cost at the corresponding system time, based on the miscellaneous costs and costs for spare parts and crews.
- **Mean A(t)**: The mean availability of the system up to the corresponding time.

- **Mean A(t[i] - t[i-1])**: The mean point availability of the system during the most recent interval (i.e., the interval between the last specified time and the current specified time).
- **System Failures(t)**: The number of system failures that occurred by the corresponding system time.
- **System Off Events by Trigger(t)**: The number of events where the system is turned off by state change triggers that occurred by the corresponding time. An OFF event is not a system failure but it may be included in system reliability calculations.
- **Throughput(t)**: The number of units that were processed/produced by the system by the corresponding system time. This will be calculated only if you selected to run a throughput simulation and selected to report throughput point results on the Analysis Settings page of the diagram control panel.

9.10.4.3 System Costs

The System Costs worksheet in the Simulation Results Explorer displays the costs that accrued during the simulation time.

For each of the four classes of tasks (i.e., corrective, preventive, on condition and inspection), the following information is presented with regard to costs that accrued during the simulation time:

- **Misc. Costs**: The miscellaneous costs associated with the task class. This is based on the cost per task and downtime rate, specified in the Maintenance Task window.
- **Costs for Parts**: The cost for spare parts due to tasks of this class. This is based on the direct cost per item as specified in the Spare Part Pool window, as well as any indirect costs (e.g., holding costs, costs associated with restocking, etc.). Note that this value is not available for inspection tasks, as inspections do not use spare parts.
- **Costs for Crews**: The cost for crews due to tasks of this class. This is based on the direct cost per unit time and cost per incident, as specified in the Crew window.
- **Total Costs**: The total cost for maintenance of this type, based on the miscellaneous costs and costs for spare parts and crews.

In addition, the following values are calculated:

- **Block Consequential Costs**
  - **Total Block Consequential Costs**: The sum of all individual consequential block costs. Consequential costs are specified in the Block Properties window both for each unit of downtime and for each failure event.
- **Indirect Pool Costs**
  - **Total Pool Holding Costs**: The indirect costs of maintaining the spare part in the pool over time. For example, if it costs $20 per month to keep the part available in the pool and the part remains in the pool for 3 months, then $60 of indirect costs have accumulated for the item.
- **Additional System Level Costs**
  - **System Level Failure Cost (CM)**: The costs associated with system failure events based on the cost per system failure, as specified on the Analysis Settings page of the diagram control panel.
  - **System Level Downtime Costs (CM)**: The total costs associated with system level downtime events due to corrective tasks only. Downtime due to other non-failure events, such as preventive, inspection or on condition tasks, is not included. This is based on the system downtime rate, as specified on the Analysis Settings page of the diagram control panel.
  - **Total System Level Costs (CM)**: The sum of the System Level Failure Costs (CM) and System Level Downtime Costs (CM).
  - **System Level Failure Cost (Total)**: The costs associated with system downing events based on the cost per system failure, as specified on the Analysis Settings page of the diagram control panel.

- **System Level Downtime Costs (Total)**: The total costs associated with all system level downtime events. This is based on the system downtime rate, as specified on the Analysis Settings page of the diagram control panel.
- **System Level Total Costs (Total)**: The sum of the System Level Failure Costs (Total) and System Level Downtime Costs (Total).

- **Total Costs**
  - **Total Costs**: The total cost that accrued during the system simulation time, based on all of the costs shown in the worksheet.
- **Opportunity Costs**
  - **Opportunity Costs**: Potential system revenue lost as a result of downtime. This is based on the lost revenue per hour of uptime plus the lost revenue per unit produced (if a throughput simulation has been run). Both are specified on the Analysis Settings page of the diagram control panel
- **System Revenue Summary**
  - **Uptime Revenue**: The system revenue based on the revenue per unit of uptime, as specified on the Analysis Settings page of the diagram control panel.
  - **Throughput Revenue**: The system revenue based on the revenue per unit of throughput (if a throughput simulation has been run), as specified on the Analysis Settings page of the diagram control panel.
  - **Total Revenue**: The sum of the Uptime Revenue and Throughput Revenue.

9.10.4.4 System Throughput

The System Throughput worksheet will appear in the Simulation Results Explorer only if you selected to run a throughput simulation on the Analysis Settings page of the diagram control panel.

- **Total Throughput**: The total number of units that were processed/produced by the system during the simulation time.
- **Block Name**: The name of each block in the diagram.
- **Uptime Utilization**: The percentage of the block's processing/production capacity while up that it actually processed/produced (i.e. the ratio of the block throughput to the block uptime capacity).

9.10.5 Block Results

9.10.5.1 Block Summary

The Block Summary worksheet in the Simulation Results Explorer displays a summary of the simulation results for each block.

- **Block Name**: The name of each block in the diagram.
- **RS FCI**: The ReliaSoft Failure Criticality Index for each block. This is a relative index showing the percentage of times that a failure of the block caused a system failure (i.e., the number of system failures caused by the block divided by the total number of system failures). Note that this relates only to failures; it does not include every event that could bring the system down (such as routine maintenance).
- **RS DECI**: The ReliaSoft Downing Event Criticality Index for each block. This is a relative index showing the percentage of times that a downing event of the block caused the system to go down (i.e., the number of system downing events caused by the block divided by the total number of system downing events).
- **RS DTCI**: The ReliaSoft Downtime Criticality Index for each block. This is a relative index showing the contribution of each block to the system's downtime (i.e., the system downtime caused by a particular block divided by the total system downtime).

- **Mean Av. (All Events)**: The mean availability of all downing events for each block during the simulation.
- **Mean Av. (w/o PM, OC & Insp.)**: The mean availability of all downing events for each block, not including preventive or inspection tasks, during the simulation.
- **Expected # of Failures**: The total number of failures expected to occur during the simulation for each block.
- **Expected # of OFF Events by Trigger**: The total number of events where the block is turned off by state change triggers. An OFF event is not a failure but it may be included in system reliability calculations.
- **System Downing Events**: The number of events for each block that caused the system to go down during the simulation.
- **Block Downtime**: The total amount of time that each block was down (i.e., not operational) during the simulation.
- **Block Uptime**: The total amount of time that each block was up (i.e., operational) during the simulation.
- **Number of CMs**: The total number of corrective tasks performed on each block during the simulation.
- **CM Downtime**: The total amount of time that each block was down (i.e., not operational) due to corrective maintenance during the simulation.
- **Number of Inspections**: The total number of inspections performed on each block during the simulation.
- **Inspection Downtime**: The total amount of time that each block was down (i.e., not operational) due to an inspection during the simulation.
- **Number of PMs**: The total number of preventive tasks performed on each block during the simulation.
- **PM Downtime**: The total amount of time that each block was down (i.e., not operational) due to a preventive task during the simulation.
- **Number of OCs**: The total number of on condition tasks performed on each block during the simulation.
- **OC Downtime**: The total amount of time that each block was down (i.e., not operational) due to an on condition task during the simulation.
- **Throughput**: The total number of units that were processed/produced by each block during the simulation time. The throughput will be calculated only if you selected to run a throughput simulation on the Analysis Settings page of the diagram control panel.
- **Total Cost**: The total cost for each block, which is based on the task, crew and spare part pool costs.

9.10.5.2 Block Costs

The Block Costs worksheet in the Simulation Results Explorer displays the costs that accrued during the simulation time for each block.

- **Block Name**: The name of each block in the diagram.
- **Total Cost**: The total cost that accrued during the simulation time for each block, which is based on the task, crew and spare part pool costs.
- **CM Misc**: The miscellaneous costs associated with corrective tasks that accrued during the simulation time. This is based on the cost per task and downtime rate, specified in the Maintenance Task window.
- **CM Part**: The cost for spare parts due to corrective tasks that accrued during the simulation time. This is based on the direct cost per item as specified in the Spare Part Pool window, as well as any indirect costs (e.g., holding costs, costs associated with restocking, etc.).
- **CM Crew**: The cost for crews due to corrective tasks that accrued during the simulation time. This is based on the direct cost per unit time and cost per incident, as specified in the Crew window.

- **PM Misc**: The miscellaneous costs associated with preventive tasks that accrued during the simulation time. This is based on the cost per task and downtime rate, specified in the Maintenance Task window.
- **PM Part**: The cost for spare parts due to preventive tasks that accrued during the simulation time. This is based on the direct cost per item as specified in the Spare Part Pool window, as well as any indirect costs (e.g., holding costs, costs associated with restocking, etc.).
- **PM Crew**: The cost for crews due to preventive tasks that accrued during the simulation time. This is based on the direct cost per unit time and cost per incident, as specified in the Crew window.
- **OC Misc**: The miscellaneous costs associated with on condition tasks that accrued during the simulation time. This is based on the cost per task and downtime rate, specified in the Maintenance Task window.
- **OC Part**: The cost for spare parts due to on condition tasks that accrued during the simulation time. This is based on the direct cost per item as specified in the Spare Part Pool window, as well as any indirect costs (e.g., holding costs, costs associated with restocking, etc.).
- **OC Crew**: The cost for crews due to on condition tasks that accrued during the simulation time. This is based on the direct cost per unit time and cost per incident, as specified in the Crew window.
- **IN Misc**: The miscellaneous costs associated with inspection tasks that accrued during the simulation time. This is based on the cost per task and downtime rate, specified in the Maintenance Task window.
- **IN Crew**: The cost for crews that accrued during the simulation time due to inspection tasks. This is based on the direct cost per unit time and cost per incident, as specified in the Crew window.
- **DT Cost**: The costs associated with downtime that accrued during the simulation time. This is based on the block consequential costs, as specified in the Block Properties window (i.e., the block's downtime multiplied by its downtime rate, and its number of failures multiplied by its cost per failure).

9.10.5.3 Block Throughput

The Block Throughput worksheet displays the throughput for each block during the simulation time. This worksheet will appear in the Simulation Results Explorer only if you selected to run a throughput simulation on the Analysis Settings page of the diagram control panel.

- **Block Name (Diagram)**: The name of each block in the diagram.
- **Throughput**: The total number of units that were processed/produced by each block during the simulation.
- **Excess Capacity**: The additional number of units each block could have processed/produced during the simulation.
- **Backlog**: The total number of backlogged units (i.e., units waiting to be processed/produced) for each block at the end of the simulation. This is based on whether the block can accept a backlog and how the backlog units were allocated, as specified in the Block Properties window.
- **Backlog Processed**: The total number of units that were backlogged but eventually processed/produced by each block during the simulation.
- **Excess Backlog**: The number of backlogged units that were rejected (i.e., not processed/produced) by each block during the simulation. Backlogged units will be rejected if the block has a limited backlog (i.e., a limit to the number of units that can be stored in the backlog).
- **Uptime Utilization**: The percent of units the block actually produced/processed of what it could have processed/produced while it was up during the simulation. (i.e., the ratio of the block throughput and the block uptime capacity.)
- **TOCB**: The Throughput Opportunity Cost for Block, which is the amount of total lost throughput revenue that the block is responsible for.

9.10.5.4 Block Details

The Block Details worksheets in the Simulation Results Explorer display the details for each block, or component, in the system. Click the name of the block in the navigation panel to display the block details for that block on a separate worksheet.

- **General Information**
  - **Number of Block Downing Events**: The total number of downing events that occurred during the system simulation time for the block.
  - **Number of System Downing Events**: The number of events for the block that caused the system to go down during the simulation.
  - **Number of Failures**: The total number of failures that occurred during the simulation for the block.
  - **Number of System Downing Failures**: The number of failures for the block that caused the system to go down during the simulation.
  - **Number of OFF Events by Trigger**: The total number of events where the block is turned off by state change triggers. An OFF event is not a failure but it may be included in system reliability calculations.
  - **Mean Availability (All Events)**: The mean availability of the block during the simulation.
  - **Mean Availability (w/o PM, OC & Inspection)**: The mean availability of all downing events for the block, not including preventive, on condition or inspection tasks, during the simulation.
  - **Block Uptime**: The total amount of time that the block was up (i.e., operational) during the simulation.
  - **Block Downtime**: The total amount of time that the block was down (i.e., not operational) during the simulation.
- **Metrics**
  - **RS Criticality Indices**
    - **RS DECI**: The ReliaSoft Downing Event Criticality Index for the block. This is a relative index showing the percentage of times that a downing event of the block caused the system to go down (i.e., the number of system downing events caused by the block divided by the total number of system downing events).
    - **Mean Time Between Downing Events**: The mean time between downing events for the block during the simulation.
    - **RS FCI**: The ReliaSoft Failure Criticality Index for the block. This is a relative index showing the percentage of times that a failure of the block caused a system failure (i.e., the number of system failures caused by the block divided by the total number of system failures). Note that this relates only to failures; it does not include every event that could bring the system down (such as routine maintenance).
    - **MTBF**: The mean time between failures for the block during the simulation.
    - **Mean Downtime per Event**: The mean duration of downtime events for the block during the simulation.
    - **RS DTCI**: The ReliaSoft Downtime Criticality Index for the block. This is a relative index showing the contribution of the block to the system's downtime (i.e., the system downtime caused by the block divided by the total system downtime).
    - **RS BCCI**: The ReliaSoft Block Cost Criticality Index for the block. This is a relative index showing the contribution of the block to the total costs (i.e., the total block costs divided by the total costs).

- **Block Downtime Contribution Index** This is an index showing the factors contributing to the block downtime. Both a crew and a spare part pool must be defined for this to return meaningful results.
  - **Non-Waiting Time CI**: A relative index showing the contribution of repair times to the block's total downtime. (The ratio of the time that the crew is actively working on the item to the total down time).
  - **Total Waiting Time CI**: A relative index showing the contribution of wait factor times to the block's total downtime. Wait factors include crew conflict times, crew wait times and spare part wait times. (The ratio of downtime not including corrective repair time to the total down time)
  - **Waiting for Opportunity/Maximum Wait Time Ratio**: A relative index showing the contribution of crew conflict times. This is the ratio of the time spent waiting for the crew to respond (not including crew logistic delays) to the total wait time (not including the active repair time).
  - **Crew/Part Wait Ratio**: The ratio of the crew and part delays. A value of 100% means that both waits are equal. A value greater than 100% indicates that the crew delay was in excess of the part delay. For example, a value of 200% would indicate that the wait for the crew is two times greater than the wait for the part.
  - **Part/Crew Wait Ratio**: The ratio of the part and crew delays. A value of 100% means that both waits are equal. A value greater than 100% indicates that the part delay was in excess of the crew delay. For example, a value of 200% would indicate that the wait for the part is two times greater than the wait for the crew.

- **Downtime Summary**
  - **Non-Waiting Time**: Time that the block was undergoing active maintenance/inspection.
  - **Waiting for Opportunity**: The total downtime for the block due to crew conflicts (i.e., time spent waiting for a crew while the crew is busy with another task). If no crew is defined, then this will return zero.
  - **Waiting for Crew**: The total downtime for the block due to crew wait times (i.e., time spent waiting for a crew due to logistical delay). If no crew is defined, then this will return zero.
  - **Waiting for Parts**: The total downtime for the block due to spare part wait times. If no spare part pool is defined then this will return zero.
- **Event Count Summary**: For each of the four classes of tasks (i.e., corrective, preventive, on condition and inspection), the following information is presented with regard to tasks performed during the simulation time:
  - **Number of [Task Class]**: The total number of tasks of this class performed on the block during the simulation.
  - **[Task Class] Downtime**: The total amount of time that the block was down (i.e., not operational) due to tasks of this class during the simulation.
- **Cost Summary**: For each of the four classes of tasks (i.e., corrective, preventive, on condition and inspection), the following information is presented with regard to costs that accrued during the simulation time:
  - **Misc. Costs**: The miscellaneous costs associated with the task class. This is based on the cost per task and downtime rate, specified in the Maintenance Task window.
  - **Costs for Parts**: The cost for spare parts due to tasks of this class. This is based on the direct cost per item as specified in the Spare Part Pool window, as well as any indirect costs (e.g., holding costs, costs associated with restocking, etc.). Note that this value is not available for inspection tasks, as inspections do not use spare parts.
  - **Costs for Crews**: The cost for crews due to tasks of this class. This is based on the direct cost per unit time and cost per incident, as specified in the Crew window.

  In addition, the following values are calculated:
  - **Costs (Consequential)**
    - **Block Failure Cost**: The block failure costs that accrued during the simulation time. This is based on the block's number of failures multiplied by its cost per failure, as specified in the Block Properties window.

- **Block Downtime Cost**: The block downtime costs that accrued during the simulation time. This is based on the block's downtime multiplied by its downtime rate, as specified in the Block Properties window.
- **Total Block Downtime Cost**: The block's total costs associated with downtime (i.e., the sum of the Block Failure Cost and the Block Downtime Cost).

❍ **System Level Contribution**

- **System Level Failure Cost (CM)**: The contribution of this block to the overall system level failure costs due to failure events. This is calculated using the system level failure costs (CM) multiplied by the ratio of the number of times this block caused a system failure to the total number of system failures.
- **System Level Downtime Cost (CM)**: The contribution of this block to the overall system level downtime costs. This is calculated using the system level downtime costs (CM) multiplied by the ratio of the system downtime caused by this block to the total system downtime.
- **System Level Total Costs (CM)**: The sum of the System Level Failure Cost (CM) and the System Level Downtime Cost (CM).
- **System Level Failure Cost (Total)**: The contribution of this block to the system level failure costs due to downing events. This is calculated using the overall system level failure costs (total) multiplied by the ratio of the number of times this block caused a system failure to the total number of system failures.
- **System Level Downtime Cost (Total)**: The contribution of this block to the system level downtime costs due to downing events. This is calculated using the overall system level downtime costs (total) multiplied by the ratio of the system downtime caused by this block to the total system downtime.
- **System Level Total Costs (Total)**: The sum of the System Level Failure Cost (Total) and the System Level Downtime Cost (Total).

❍ **Block Opportunity Costs**

- **Block Opportunity Cost (CM)**: The loss of throughput and revenue attributed directly to this block due to failure events. This is calculated by multiplying the system opportunity costs for corrective maintenance (shown in the System Costs worksheet) by the block DTCI.
- **Block Opportunity Cost (Total)**: The loss of throughput and revenue attributed directly to this block due to downing events. This is calculated by multiplying the total system opportunity costs (shown in the System Costs worksheet) by the block DTCI.

❍ **Block Total Costs**

- **Block Total Cost**: The total cost associated with the block, including all costs for all task types, the Total Block Downtime Cost and the System Level Total Costs. Note that the Block Opportunity Cost is not included in this value.

- **Throughput Analysis Summary**

❍ **Throughput** results will be calculated only if you selected to run a throughput simulation on the Analysis Settings page of the diagram control panel.

- **Throughput**: The total number of units that were processed/produced by the block during the simulation time.
- **Excess Capacity**: The additional number of units the block could have processed/produced during the simulation.
- **Backlog**: The total number of units that were backlogged (i.e., waiting to be processed) for the block at the end of the simulation time. This is based on whether the block can accept a backlog and how the units were allocated, as specified in the Block Properties window.
- **Backlog Processed**: The total number of units that were backlogged but eventually processed by the block during the simulation time.

- **Throughput Metrics** will be calculated only if you selected to run a throughput simulation on the Analysis Settings page of the diagram control panel.
  - **Max. Uptime Capacity**: The maximum number of units the block could have processed/produced while it was up during the simulation.
  - **Uptime Utilization**: The percent of units the block actually produced/processed of what it could have processed/produced while it was up during the simulation. (i.e., the ratio of the block throughput and the block uptime capacity.)
  - **Max. Capacity**: The maximum capacity of the block, which is the maximum number of units the block could have processed/produced during the simulation.
  - **Actual Utilization**: The percent of units the block actually produced/processed of what it could have produced/processed (i.e., the ratio of the block throughput and the block maximum capacity).

9.10.5.5 Last Run Details

The Last Run Details worksheets in the Simulation Results Explorer display the details of the last simulation for the selected block. Note that if you are working with a BlockSim 6 file, these details are available only when a single simulation was run. If multiple simulations were run, you must re-simulate in the current version in order for these details to be available.

- **ENUM**: The index number for each event. This is an identifier that indicates the order in which the events occurred.
- **Time to Event**: The time to each event.
- **Waiting for Opportunity**: The time the block was waiting for an opportunity, which is the time associated with the waiting for a crew that is working on another repair.
- **Waiting for Crew**: The time the block was waiting for a crew, which is the time associated with the logistic delay of the crew.
- **Waiting for Spare Part**: The time the block was waiting for a spare part.
- **Under Repair**: The time the block was under repair (i.e., maintenance duration).
- **Event Type**: The type of event. Possible values include:
  - CM: corrective maintenance
  - PM: preventive maintenance
  - I: inspection
  - NDI: non-downing inspection
  - OC: on condition task
  - OFF: block turned off by state change triggers
  - F: block failure, no maintenance

9.10.6 Crew Results

9.10.6.1 Crew Summary

The Crew Summary worksheet in the Simulation Results Explorer displays a summary of the crews.

- **Crew**: The name of each crew assigned to the blocks in the diagram.
- **Calls**: The number of times each crew was called upon to perform maintenance during the simulation time.
- **Accepted**: The number of calls that were accepted during the simulation time by each crew.

- **Percent Accepted**: The percentage of calls that were accepted by each crew during the simulation time.
- **Rejected**: The number of calls that were rejected during the simulation time by each crew. A call may be rejected because the crew is not able to perform simultaneous tasks, it is already performing its maximum number of tasks, etc.
- **Percent Rejected**: The percentage of calls that were rejected by the crew during the simulation time.
- **Time Used**: The total time each crew spent performing maintenance tasks during the simulation time.
- **Crew Delay Time**: The total delay time before each crew started tasks, based on the logistic delay time defined in the crew properties.
- **Spare Delay Time**: The total delay time for spare parts for each crew, based on the logistic delay time defined in the spare part pool properties.
- **Repair Time**: The total active time each crew spent performing maintenance tasks during the simulation time.
- **Direct Costs**: The total direct costs associated with maintenance tasks for each crew, based on the direct costs defined in the crew properties.
- **Incident Costs**: The total incident costs associated with maintenance tasks for each crew, based on the cost per incident defined in the crew properties.
- **Cost**: The total costs associated with each crew during the simulation time, based on the direct and indirect costs.

9.10.6.2 Crew Details

The Crew Details worksheets in the Simulation Results Explorer display the details for each crew assigned to perform tasks on the system. Click the name of the crew in the navigation panel to display the crew details on a separate worksheet.

- **Call Summary**
  - **Calls Received**: The number of times the crew was called upon to perform maintenance during the simulation time.
  - **Accepted Calls**: The number of calls that were accepted during the simulation time by the crew.
  - **Percent Accepted**: The percentage of calls that were accepted by the crew during the simulation time.
  - **Rejected Calls**: The number of calls that were rejected by the crew during the simulation time. A call may be rejected because the crew may not be able to perform simultaneous tasks, it is already performing its maximum number of tasks, etc.
  - **Percent Rejected**: The percentage of calls that were rejected by the crew during the simulation time.
- **Utilization**
  - **Total Utilization**: The total time spent by the crew performing maintenance tasks during the simulation time.
  - **Average Call Duration**: The average duration of the tasks (corrective, preventive or inspection) performed by the crew during the simulation.
  - **Total Wait Time**: The total delay time spent waiting for the crew after the call was made and before the crew was sent to the block. This delay applies only to events where the crew was busy performing maintenance on another item (i.e., it does not include the logistic delay specified in the Crew window).
  - **Total Crew Delay Time**: The total delay time before the crew started tasks, based on the logistic delay time defined in the crew properties.
  - **Total Spare Delay Time**: The total delay time for spare parts for the crew, based on the logistic delay time defined in the spare part pool properties.

- **Total Repair Time**: The total active time the crew spent performing maintenance tasks during the simulation time.

- **Costs**
  - **Direct Costs**: The total direct costs associated with maintenance tasks for the crew, based on the direct costs defined in the crew properties.
  - **Indirect Costs**: The total incident costs associated with maintenance tasks for the crew, based on the cost per incident defined in the crew properties.
  - **Total Crew Costs**: The total crew-related costs for the crew during the simulation time, based on the direct and indirect costs.
  - **Average Cost per Call**: The average cost per call made to the crew during the simulation time.

9.10.7 Spare Part Pools Results

9.10.7.1 Spare Part Pool Summary

The Spare Part Pool Summary worksheet in the Simulation Results Explorer displays a summary of the spare part pools.

- **Pool**: The name of each spare part pool.
- **Avg. Stock Level**: The average number of spare parts in each spare part pool during simulation time.
- **Parts Dispensed**: The number of spare parts in each spare part pool that were dispensed during the simulation time. Spares are required for corrective, preventive and on condition tasks.
- **Avg. Time to Dispense**: The average time it took to dispense the spare parts from each spare part pool during the simulation time. This is based on the logistic time for spare acquisition.
- **Rejected Requests**: The number of spares that were required for corrective or preventive maintenance but were unavailable in the spare part pool. Therefore, no maintenance action was performed.
- **Emergency Spares Dispensed**: The number of emergency spare parts that were used from each spare part pool during the simulation time. Emergency acquisitions are initiated when the spare part pool is empty but a spare part is required for a task.
- **Cost per Part**: The direct cost per dispensed item for each spare part pool, as specified in the Spare Part Pool window.
- **Total Cost**: The total cost for spare parts that were dispensed from each spare part pool during the simulation time.

9.10.7.2 Spare Part Pool Details

The Spare Part Pool Details worksheets in the Simulation Results Explorer display the details for each spare part pool assigned to the blocks in the diagram. Click the name of the spare part pool in the control panel to display the corresponding spare part pool details on a separate worksheet.

- **Dispensed**
  - **Parts Dispensed**: The number of spare parts in the spare part pool that were dispensed during the simulation. Spares are required for corrective, preventive and on condition tasks.
  - **Total Time to Dispense**: The total time it took to dispense the spare parts from the spare part pool during the simulation. This is based on the logistic time for spare acquisition.
  - **Average Time to Dispense**: The average time it took to dispense the spare parts from the spare part pool during the simulation. This is based on the logistic time for spare acquisition.

- **Rejected & Emergency**
  - **Number of Emergency Requests**: The number of emergency spare parts that were used from each spare part pool during the simulation time. Emergency acquisitions are initiated when the spare part pool is empty but a spare part is required for a task.
  - **Rejected Requests**: The number of spares required for corrective or preventive maintenance that were unavailable in the spare part pool. Therefore, no maintenance was performed.
- **Restock Details**
  - **Initial Stock Level**: The number of spare parts that were in the pool at the beginning of the simulation.
  - **Parts Restocked**: The number of spare parts that were restocked in the pool during the simulation.
  - **Average Stock Level**: The average number of spare parts that were in the pool during the simulation.
  - **Current Stock Level**: The number of spare parts that were in the spare part pool at the end of the simulation.
- **Pool Costs**
  - **Total Pool Costs**: The total cost for the spare part pool that was used during the simulation.
  - **Indirect Pool Costs**: The indirect costs of maintaining the spare parts in the pool over time. For example, if it costs $20 per month to keep each part available in the pool and the part remains in the pool for 3 months, then $60 of indirect costs have accumulated for the item.
  - **Direct Pool Costs**: The total direct cost for all spare parts dispensed from the spare part pool during the simulation.
- **Pool Costs per Part**
  - **Total Cost per Part**: The total direct and indirect cost per part for the spare part pool over the duration of the simulation.
  - **Indirect Cost per Part**: The indirect cost per part of maintaining the spare parts in the pool over the duration of the simulation.
  - **Direct Cost per Part**: The direct cost per part for all spare parts dispensed from the pool over the duration of the simulation.

9.10.8 Simulation Summary

The Simulation Summary worksheet in the Simulation Results Explorer displays the details of the simulation.

- **Diagram**: The name of the diagram that was simulated.
- **Simulation Start Time**: The date and time that the simulation began.
- **Simulation Duration**: The total amount of time that simulation took.
- **End Time**: The simulation end time, as defined in the Maintainability/Availability Simulation window.
- **Run Options**: The type of simulation that was run (i.e., standard simulations or log of simulations).
- **Number of Simulations**: The total number of simulations that were performed on the system, as defined in the Maintainability/Availability Simulation window.
- **Seed Value**: The seed value, if any, as defined in the Maintainability/Availability Simulation window.
- **Number of Threads**: The number of threads used during the simulation.

9.10.9 Relevant Summaries

The following worksheets in the Simulation Results Explorer provide additional reports on the simulation results for the system and individual blocks.

- The **Block Failure Criticality Ranking** worksheet ranks each block by its value for ReliaSoft's failure criticality index (RS FCI). This is a relative index showing the percentage of times that a failure of the block caused a system failure (i.e., the number of system failures caused by the block divided by the total number of system failures). Note that this relates only to failures; it does not include every event that could bring the system down (such as routine maintenance).
- The **Block Downtime Criticality Ranking** worksheet ranks each block by its value for ReliaSoft's downtime criticality index for each block in the diagram. This is a relative index showing the contribution of the block to the system's downtime (i.e., the system downtime caused by the block divided by the total system downtime).
- The **Block Availability Ranking** worksheet ranks each block by its availability during the simulation, from highest to lowest.
- The **Block Failures Ranking** worksheet ranks each block by its expected number of failures during the simulation, from highest to lowest.
- The **Block System Downing Failures** worksheet ranks each block by the number of failures of the block that resulted in the downing of the system, from highest to lowest.
- The **Block Downtime Ranking** worksheet ranks each block based on the total amount of time that the block was down (i.e., not operational) during the simulation, from highest to lowest.
- The **Block Uptime Ranking** worksheet ranks each block based on the total amount of time that the block was up (i.e., operational) during the simulation, from highest to lowest.
- The **Block Inspection Count Ranking** worksheet ranks each block based on the total number of inspections performed on the block during the simulation, from highest to lowest.
- The **Block Inspection Downtime Ranking** worksheet ranks each block based on the total amount of time that the block was down (i.e., not operational) during the simulation due to an inspection, from highest to lowest.
- The **Block PM Count Ranking** worksheet ranks each block based on the total number of preventive tasks performed on the block during the simulation, from highest to lowest.
- The **Block PM Downtime Ranking** worksheet ranks each block based on the total amount of time that the block was down (i.e., not operational) during the simulation due to preventive maintenance, from highest to lowest.
- The **Block OC Count Ranking** worksheet ranks each block based on the total number of on condition tasks performed on the block during the simulation, from highest to lowest.
- The **Block OC Downtime Ranking** worksheet ranks each block based on the total amount of time that the block was down (i.e., not operational) during the simulation due to on condition tasks, from highest to lowest.
- The **Block Throughput Ranking** worksheet ranks each block based on the total number of units that were processed/produced by the block during the simulation time, from highest to lowest. This worksheet will appear only if you selected to run a throughput simulation on the Analysis Settings page of the diagram control panel.
- The **Block Cost Ranking** worksheet ranks each block based on the total cost that accrued for the block during the simulation, based on the task, crew and spare part pool costs, from highest to lowest.
- The **System Event Log** worksheet displays information about each system event that occurred during the last simulation. Note that if you are working with a BlockSim 6 file, this worksheet is available only when a single simulation was run. If multiple simulations were run, you must re-simulate in the current version in order for this worksheet to be available.
  - **Event #** displays a sequential number for each system event that occurred during the simulation.
  - **Time-to-Event**: The time-to-event for each system event.

- **Repair Duration**: The repair duration for each system event.

- The **System Failure Event Log** worksheet displays information about each system failure that occurred during the last simulation. Note that if you are working with a BlockSim 6 file, this worksheet is available only when a single simulation was run. If multiple simulations were run, you must re-simulate in the current version in order for this worksheet to be available.
  - **Failure #** displays a sequential number for each system failure that occurred during the simulation.
  - **Time-to-Failure**: The time for each system failure.
  - **Repair Duration**: The repair duration for each system failure.
- **RDA Data** displays system up/down data for each simulation. You can click **Send RDA Results to Weibull++** to transfer the data to a Weibull++ parametric RDA folio within the current project. If Weibull++ is installed on your computer, you can then perform parametric recurrent event data analysis on the data set. The parametric RDA folio uses the General Renewal Process (GRP) model to analyze the failure behavior of a repairable system. The GRP analysis method takes into account the effectiveness of repairs on the condition of the system, and models the cumulative number of failures over time.
  - **System ID** identifies the system that experienced an event. Here, each individual simulation is denoted by a separate system ID.
  - **Event F or E** describes the type of event. An F indicates a failure event and an E indicates that the observation period has ended.
  - **Time to Event** indicates the age of the system when the event occurred.

  You can set the maximum number of events that can be shown in this worksheet using the **Max RDA Data** field on the Analysis Settings page of the simulation diagram control panel.
- **System Downtime Durations** shows the duration of every system downtime during all of the simulations, as well as the part responsible for the downtime.
- **RGA Data** displays system up/down data for each simulation, including start, failure and end events. You can click **Send RGA Results to RGA** to transfer the data to an RGA folio using the Multiple Systems - Concurrent Operating Times data type within the current project. If RGA is installed on your computer, you can then perform reliability growth analysis on the data set.
  - **System ID** identifies the system that experienced an event. Here, each individual simulation is denoted by a separate system ID.
  - **Event F or E** describes the type of event. An S denotes the system start time, an F indicates a failure event and an E indicates that the observation period has ended.
  - **Time to Event** indicates the age of the system when the event occurred.
  - **Responsible Part** identifies the block that was responsible for each failure event.

  The maximum number of events that can be shown in this worksheet is controlled by the **Max RDA Data** field on the Analysis Settings page of the simulation diagram control panel.

9.11 Simulation Quick Calculation Pad

BlockSim includes a Quick Calculation Pad (QCP) for computing useful metrics. You can access the QCP by choosing **Simulation > Simulation > Simulation QCP** or by clicking the icon on the Simulation page of the diagram control panel.

To perform a calculation, select the appropriate option and enter any required inputs in the **Inputs** area, then click **Calculate**. *For more detailed information on how to use the QCP in general, see page 249 in Chapter 7.*

The following calculations are available for simulation diagrams and phase diagrams:

Probability

- **Point Availability**

  Calculates the probability that the system is operational (not failed or undergoing repair) when it is requested for use at a certain point in time. Enter the time at which you wish to calculate the point availability in the **Mission End Time** field. The mission is assumed to start at time = 0.

  For example, a system with a point availability of 90% for a mission end time of 100 hours means that the system is expected to be available for use 90% of the time for a period of 100 hours. In other words, the system is up and running 90 times out of 100.

- **Point Unavailability**

  Calculates the probability that the system will be failed or undergoing repair when it is requested for use at a certain point in time. Enter the time at which you wish to calculate the point unavailability in the **Mission End Time** field. The mission is assumed to start at time = 0.

  The point unavailability is the inverse of the point availability. For example, a 10% point unavailability is equivalent to a 90% point availability.

- **Point Reliability**

  Calculates the probability that the system will operate without failure for a given period of time. Note that this is similar to point availability except that it is assumed that the system has not had a single failure. Other (non-failure) downtime events are also ignored. Enter the time at which you wish to calculate the point reliability in the **Mission End Time** field. The mission is assumed to start at time = 0.

  For example, a reliability of 90% for a mission end time of 3 years means that if 100 identical systems are fielded, then 90 of them will still be operating at the end of 3 years.

- **Point Probability of Failure**

  Calculates the probability that a system will be failed in a given period of time. Enter the time at which you wish to calculate the probability of failure in the **Mission End Time** field. The mission is assumed to start at time = 0.

  Probability of failure is also known as unreliability and it is the inverse of the reliability. For example, a point probability of failure of 10% for a mission end time of 3 years is equivalent to a 90% reliability.

- **Mean Availability**

  Calculates the average availability (i.e., uptime divided by operating time) of the system at the specified time. This metric reflects the average probability, up to that point in time, that the system is operational (not failed or undergoing repair) when it is requested for use during the simulation. Enter the time at which you wish to calculate the mean availability in the **Mission End Time** field. The mission is assumed to start at time = 0.

- **Mean Unavailability**

Calculates the average probability, up to the specified time, that the system will be failed or undergoing repair when it is requested for use during the simulation. It is the inverse of the mean availability. Enter the time at which you wish to calculate the mean unavailability in the **Mission End Time** field. The mission is assumed to start at time = 0.

Time

- **Time for Availability**

  Calculates the time at which a specified availability value will be achieved. Enter the availability goal in the **Required Availability** field.

- **Time for Reliability**

  Calculates the time at which a specified reliability value will be achieved. Enter the reliability goal in the **Required Reliability** field.

Life

- **MTTFF**

  Calculates the mean time to first failure for the repairable system in the diagram. This is the average of the times at which the first system failure occurred for each simulation. Note that when the simulation end time is much less than the time to first failure for the system or when no system failures are observed, the calculation assumes a constant failure rate in order to obtain the MTTFF. If the system does not have a constant failure rate, the simulation end time should be set to a value that is well beyond the MTTF (mean time to failure, as computed analytically) of the system in order to obtain a realistic estimate. As a general rule, the simulation end time should be at least three times larger than the MTTF of the system.

Note: The mean time to first failure is a performance index and does not provide any information about the behavior of the failure distribution of the system.

9.12 Simulation Plots

Plots can be created and viewed by clicking the **Plot** icon located on the Simulation page of the control panel for each diagram.

If you click the **Plot** icon before the diagram has been simulated, you will be prompted to simulate the diagram and then the data will be plotted in the plot sheet. (Note that the **Plot** icon on the Plot sheet control panel refreshes the plot. If data or settings have been changed since the diagram was last simulated, you must resimulate the data before refreshing the plot.)

You can add additional plot sheets to the diagram by right-clicking the sheet index tabs at the bottom of the window and choosing **Insert Simulation Plot** on the shortcut menu. You can delete a plot sheet from the diagram by selecting the plot sheet, right-clicking and choosing **Delete Plot**.

The plots available for simulation diagrams fall into five types:

System Plots

System plots display information about the entire system. These plots are available for all simulation diagrams.

- **Point Availability** shows the system's point availability over time. The point availability is the probability that a system (or component) will be operational (up and running) at any random time, t. Note that unlike reliability plots for analytical systems, the point availability measure incorporates maintainability information.
- **Point Reliability** shows the system's point reliability over time. The point reliability is the probability that the system has not failed by time t. This is similar to point availability with the major exception that it looks only at the probability that the system did not have a single failure. Other (non-failure) downing events are ignored. It is very important to note that this value is not always the same as the reliability computed using the analytical methods, depending on the redundancy present. The reason that it may differ is best explained by the following scenario:

  Assume two units in parallel. The analytical system reliability, which does not account for repairs, is the probability that both units fail. In this case, when one unit goes down, it does not get repaired and the system fails after the second unit fails. In the case of repairs, however, it is possible for one of the two units to fail and get

repaired before the second unit fails. Thus, when the second unit fails, the system will still be up due to the fact that the first unit was repaired.

- **Both Point A & R** shows the system's point availability over time and the system's point reliability over time on the same plot.
- **Cost** shows the system's cost over time. If there are costs associated with crews and with spare part pools, these are also displayed.
- **Mean Availability** shows the system's mean availability over time, which is the proportion of time during a mission or time period that the system is available for use.
- **System Failures** shows the system's failures over time.
- **Throughput** shows the system's throughput over time.
- **System Up/Down** shows the system's uptimes and downtimes over the course of the simulation. Please note that System Up/Down Plots are designed for only one simulation (i.e., **1** must be entered for the **Number of Simulations** in the Maintainability/Availability Simulation window). If you run multiple simulations, the plot shows the results for the last simulation only.
- **Block Up/Down** shows each block's uptimes and downtimes over the course of the simulation. Please note that Block Up/Down Plots are designed for only one simulation (i.e., **1** must be entered for the **Number of Simulations** in the Maintainability/Availability Simulation window). If you run multiple simulations, the plot shows the results for the last simulation only. When you first select the plot, you must specify which blocks to include by using the **Available Blocks** area of the control panel to select which blocks to view.

Metrics Plots

Metrics plots display information about the contributions that individual blocks make to failure and downing events for the entire system. These plots are available for all simulation diagrams.

- **RS FCI** shows the ReliaSoft Failure Criticality Index for each block. This is a relative index showing the percentage of times that a failure of the block caused a system failure (i.e., the number of system failures caused by the block divided by the total number of system failures). Note that this relates only to failures; it does not include every event that could bring the system down (such as routine maintenance). You can specify the number of blocks to display; the blocks with the highest RS FCI value will be shown in the plot. Point to a bar to view the block's name, availability and RS FCI.
- **RS FCI Tableau** shows the ReliaSoft Failure Criticality Index in tableau format. The block name appears on each section of the plot. The green sections represent the blocks with the best RS FCI; the yellow sections the blocks with medium RS FCI and the red sections the blocks with the worst RS FCI. Point to a plot section to view the block's name, availability and RS FCI.
- **RS DECI** shows the ReliaSoft Downing Event Criticality Index for each block. This is a relative index showing the percentage of times that a downing event of the block caused the system to go down (i.e., the number of system downing events caused by the block divided by the total number of system downing events). You can specify the number of blocks to display; the blocks with the highest RS DECI value will be shown in the plot. Point to a bar to view the block's name, availability and RS DECI.
- **RS DECI Tableau** shows the ReliaSoft Downing Event Criticality Index for each block in tableau format. The block name appears on each section of the plot. The green sections represent the blocks with the best RS DECI; the yellow sections the blocks with medium RS DECI and the red sections the blocks with the worst RS DECI. Point to a plot section to view the block's name, availability and RS DECI.
- **RS DTCI** shows the ReliaSoft Downtime Criticality Index for each block. This is a relative index showing the contribution of each block to the system's downtime (i.e., the system downtime caused by a particular block divided by the total system downtime). You can specify the number of blocks to display; the blocks with the highest RS DTCI value will be shown in the plot. Point to a bar to view the block's name, availability and RS DTCI.

- **RS DTCI Tableau** shows the ReliaSoft Downtime Criticality Index for each block in tableau format. The block name appears on each section of the plot. The green sections represent the blocks with the best RS DTCI; the yellow sections the blocks with medium RS DTCI and the red sections the blocks with the worst RS DTCI. Point to a plot section to view the block's name, availability and RS DTCI.

Blocks Plots

Blocks plots display information about the individual blocks that make up the system. These plots are available for all simulation diagrams.

For blocks plots, when you first select the plot, you must specify which blocks to include by using the **Available Blocks** area of the control panel to select which blocks to view. Note that if your simulation does not include throughput then no information will be displayed for those plots.

- **Block Availability** shows the percent of time that the block is available during the simulation. Point to a bar to view the block's name, RS DECI value and availability percentage.
- **Block Costs** shows the cost of the block during the simulation. Point to a bar to view the block's name, RS DECI value and total costs.
- **Block Downtime** shows the amount of time the block is down during the simulation. Point to a bar to view the block's name, RS DECI value and total downtime.
- **Block Expected Failures** shows the expected number of failures for the block during the simulation. Point to a bar to view the block's name, RS FCI value and expected number of failures.
- **Block Expected Downing Events** shows the expected number of downing events for the block during the simulation. Point to a bar to view the block's name, RS DECI value and expected number of downing events.
- **Block Throughput** shows the total number of units processed/produced by the block during the simulation time. Point to a bar to view the block's name, utilization percentage and throughput values.
- **Block Excess Capacity** shows the additional number of units the block could have processed/produced during the simulation. Point to a bar to view the block's name, utilization percentage and excess capacity values.
- **Block Backlog** shows the total number of units that were backlogged (i.e., waiting to be processed) for the block at the end of the simulation time. Point to a bar to view the block's name, utilization percentage and backlog values.
- **Block Backlog Processed** shows the total number of units that were backlogged but eventually processed by the block during the simulation time. Point to a bar to view the block's name, utilization percentage and backlog processed values.
- **Block Bubble Plot** allows you to specify three result types to be plotted against each other for each block. Select one result type for the x-axis and one result type for the y-axis. Select a third result type that will dictate the size of the scatter point for each block; the radius of the point will be proportional to the size of the result value. You can use the slider bar at the bottom of the **Available Results** area on the control panel to set the maximum point size. Point to a data point to view the three selected metrics for the block.

  The result types that are available for use in the plot are:

  - **Uptime**: The total amount of time that the block was up (i.e., operational) during the simulation.
  - **Downtime**: The total amount of time that the block was down (i.e., not operational) during the simulation.
  - **Active Downtime**: For contained standby blocks, the amount of time that the block was down while in an active state during the simulation.
  - **Quiescent Downtime**: For contained standby blocks, the amount of time that the block was down while in a quiescent (i.e., standby) state during the simulation.
  - **Mean Downtime per Event**: The mean duration of downtime events for the block during the simulation.

- **RS DECI**: The ReliaSoft Downing Event Criticality Index for the block. This is a relative index showing the percentage of times that a downing event of the block caused the system to go down (i.e., the number of system downing events caused by the block divided by the total number of system downing events).
- **MTBDE**: The mean time between downing events for the block during the simulation.
- **RS FCI**: The ReliaSoft Failure Criticality Index for the block. This is a relative index showing the percentage of times that a failure of the block caused a system failure (i.e., the number of system failures caused by the block divided by the total number of system failures). Note that this relates only to failures; it does not include every event that could bring the system down (such as routine maintenance).
- **MTBF**: The mean time between failures for the block during the simulation.
- **RS DTCI**: The ReliaSoft Downtime Criticality Index for the block. This is a relative index showing the contribution of the block to the system's downtime (i.e., the system downtime caused by a particular block divided by the total system downtime).
- **Number of CMs**: The total number of corrective tasks performed on the block during the simulation.
- **CM Downtime**: The total amount of time that the block was down (i.e., not operational) due to corrective maintenance during the simulation.
- **Number of Inspections**: The total number of inspections performed on the block during the simulation.
- **Inspection Downtime**: The total amount of time that the block was down (i.e., not operational) due to an inspection during the simulation.
- **Number of PMs**: The total number of preventive tasks performed on the block during the simulation.
- **PM Downtime**: The total amount of time that the block was down (i.e., not operational) due to a preventive task during the simulation.
- **Number of OCs**: The total number of on condition tasks performed on the block during the simulation.
- **OC Downtime**: The total amount of time that the block was down (i.e., not operational) due to an on condition task during the simulation.
- **Number of OFFs**: The total number of times that the block was turned off due to state change triggers during the simulation.
- **OFF Downtime**: The total amount of time that the block was off due to state change triggers during the simulation.
- **Total Cost CM**: The total costs due to corrective tasks that accrued during the simulation time. This includes costs associated with the crews and spare part pools used for the tasks, as well as any downtime costs and additional per-task costs.
- **Total Cost IN**: The total costs due to inspection tasks that accrued during the simulation time. This includes costs associated with the crews used for the tasks, as well as any downtime costs and additional per-task costs.
- **Total Cost PM**: The total costs due to preventive tasks that accrued during the simulation time. This includes costs associated with the crews and spare part pools used for the tasks, as well as any downtime costs and additional per-task costs.
- **Total Cost OC**: The total costs due to on condition tasks that accrued during the simulation time. This includes costs associated with the crews and spare part pools used for the tasks, as well as any downtime costs and additional per-task costs.
- **Total Cost**: The total cost that accrued during the simulation time for the block, which is based on the maintenance properties, policies, crews and spare part pools costs.
- **Throughput Excess Capacity**: The additional number of units the block could have processed/produced during the simulation. The excess capacity will be calculated only if you selected to run a throughput simulation in the Maintainability/Availability Simulation window.

- **Throughput Backlog**: The total number of backlogged units (i.e., units waiting to be processed/produced) for the block at the end of the simulation. This is based on whether the block can accept a backlog and how the backlog units were allocated, as specified in the Block Properties window and Maintainability/Availability Simulation window.
- **Throughput Backlog Processed**: The total number of units that were backlogged but eventually processed/produced by the block during the simulation.
- **Throughput Backlog Excess**: The number of backlogged units that were rejected (i.e., not processed/produced) by the block during the simulation. Backlogged units will be rejected if the block has a limited backlog (i.e., a limit to the number of units that can be stored in the backlog).
- **Throughput Max Uptime Capacity**: The maximum number of units the block could have processed/produced while it was up during the simulation.
- **Throughput Uptime Utilization**: The percent of units the block actually produced/processed of what it could have processed/produced while it was up during the simulation. (i.e., the ratio of the block throughput and the block uptime capacity.)
- **Throughput Max Capacity**: The maximum capacity of the block, which is the maximum number of units the block could have processed/produced during the simulation.
- **Throughput Actual Utilization**: The percent of units the block actually produced/processed of what it could have produced/processed (i.e., the ratio of the block throughput and the block maximum capacity).

Crews Plots

Crews plots display information about the crews that are used in simulation of the system. These plots are available only if your analysis includes maintenance tasks that use defined crews.

For crews plots, when you first select the plot, you must specify which crews to include by using the **Available Crews** area of the control panel to select which crews to view.

- **Crew Accepted Calls** shows the total number of calls that each crew accepted during the simulation time. Point to a bar to view the crew's name, percentage of calls accepted and number of accepted calls.
- **Crew Rejected Calls** shows the total number of calls that each crew rejected during the simulation time. Point to a bar to view the crew's name, percentage of calls accepted and number of rejected calls.
- **Crew Costs** shows the costs incurred by each crew during the simulation time. Point to a bar to view the crew's name, percentage of calls accepted and total costs.
- **Crew Time Used** shows the amount of time used by each crew during the simulation time. Point to a bar to view the crew's name, percentage of calls accepted and time used.

Spare Part Pools Plots

Spare part pools plots display information about the spare part pools that are used in simulation of the system. These plots are available only if your analysis includes maintenance tasks that use defined spare part pools.

For spare part pools plots, when you first select the plot, you must specify which pools to include by using the **Available Pools** area of the control panel to select which pools to view.

- **Pool Average Stock Level** shows the average number of spare parts in the pool during the simulation time. Point to a bar to view the pool name and average stock level.
- **Pool Items Dispensed** shows the average amount of spare parts dispensed during the simulation time. Point to a bar to view the pool name and number of items dispensed.
- **Pool Average Time to Dispense** shows the average amount of time taken to dispense items during the simulation time. Point to a bar to view the pool name and average time to dispense items.

- **Pool Rejected Calls** shows the average number of calls rejected by the pool during the simulation time. Point to a bar to view the pool name and number of rejected calls.
- **Pool Emergency Calls** shows the average number of emergency restock calls by the pool during the simulation time. Point to a bar to view the pool name and number of emergency calls.
- **Pool Costs** shows the average cost of the pools during the simulation time. Point to a bar to view the pool name and pool cost.
- **Pool Cost Per Item** shows the average cost of the items in each pool during the simulation time. Point to a bar to view the pool name and cost per item.

9.13 Dashboard Viewer

The Dashboard Viewer allows you to view Synthesis data in a variety of graphical presentations. In BlockSim, it is accessed by choosing **Simulation > Dashboard > Dashboard Viewer**.

The contents of the Dashboard Viewer will vary depending on the templates that have been created for your application. To choose a template, select it from the **DDF** drop-down list at the top of the window.

Note: Dashboard templates are created via the Dashboard Designer and managed via the Dashboard Manager, both of which are available only to users with the “Create dashboard templates” permission.

BlockSim Phase Diagrams 10

A phase diagram, or more specifically a reliability phase diagram, is an extension of the reliability block diagram (RBD) approach. While an RBD is used for a system with a fixed configuration throughout the entire period of operation, a phase diagram can be used to represent scenarios in which a system may undergo changes over time to one or more of the following:

- Its reliability configuration
- The available repair resources
- The failure, maintenance or throughput properties of its individual components.

For example, a manufacturing company's production may be lower during the night shift than during the day shift. Or a system may use different components during different stages of operation, such as an aircraft during taxiing, takeoff, cruising and landing. In these and many other situations, a phase diagram can be configured so that the simulation will account for the differences in the system during different time periods.

This chapter addresses the following:

- Introduction to phase diagrams - Section 10.1 (p. 336)
- Using the phase diagram control panel - Section 10.2 (p. 337)
- Configuring and using blocks in phase diagrams, including:
  - Operational phase blocks - Section 10.3 (p. 340)
  - Maintenance phase blocks and maintenance templates - Section 10.4 (p. 342)
  - Subdiagram phase blocks - Section 10.5 (p. 345)
  - Node blocks and stop blocks - Section 10.6 (p. 346)
- Assumptions used when simulating phase diagrams - Section 10.7 (p. 347)
- Available results and plots - Section 10.8 (p. 349)
- Performing throughput analysis with phase diagrams - Section 10.9 (p. 351)

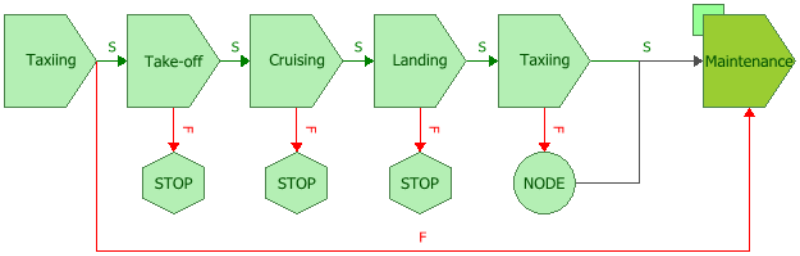
10.1 Introduction to Phase Diagrams

In BlockSim, phase diagrams are built using phase blocks arranged chronologically to represent each sequential stage in the operation of the system. In much the same way that a subdiagram block within an RBD represents another diagram in the current project, operational phase blocks are linked to simulation RBDs that represent the reliability-wise configuration of the system during each operational phase, while maintenance phase blocks are linked to maintenance templates that describe tasks that need to be completed at some point during the system's operation.

As an example, consider an aircraft with three engines and three landing gear assemblies. The aircraft undergoes five operational phases every time it flies: taxiing, take-off, cruising, landing and taxiing. Each phase is modeled using a simulation RBD.

In a typical mission, the airplane taxies for 15 minutes, takes off for 30 seconds, cruises for 4 hours, lands for 30 seconds and taxies for another 10 minutes. Assume that when a critical failure (system failure) occurs during taxiing, the airplane does not take off and is sent for maintenance instead. However, when a critical failure occurs during take-off, cruising or landing, the system is assumed to be lost.

The phase diagram for this aircraft's mission might be configured as shown in the following picture, where each phase block is linked to the RBD that describes the reliability-wise configuration of the aircraft during that activity.



Each of the operational phases in this diagram has two paths leading from it: a success path and a failure path. This graphically illustrates the consequences in each case. For instance, if the first taxiing phase is successful, the airplane will proceed to the take-off phase; if it is unsuccessful, the airplane will be sent for maintenance. The failure paths for the take-off, cruising and landing phases point to stop blocks, which indicate that the simulation of the mission ends. For the final taxiing phase, both the success and failure paths lead to the maintenance path; the node block allows you to model this type of shared outcome.

What's Changed? n BlockSim 7, there was only one path from each phase. The failure outcome for each phase was defined via a drop-down list. The success and failure paths used in the Synthesis version of BlockSim make it easy to see what will happen upon success or failure for each block. They also allow you to create more complex phase diagrams, in which success and failure may lead to entirely different sets of phases. In addition, they offer an additional possible outcome of failure. Previously, the outcome of failure could be maintenance, stopping the simulation or continuing the simulation; now, another possible outcome of failure can be continuing simulation on a different path.

For more information about using success and failure paths, please read "Modeling Nonlinear Systems in Reliability Phase Simulation" at: http://www.weibull.com/hotwire/issue135/hottopics135.htm.

The following table summarizes the operational phases and their properties. By applying a duty cycle of 1.5, the take-off and landing phases show that the system will operate at a 50% greater rate than the taxiing and cruising phases. The On System Failure column shows what will happen if a failure occurs during that phase. For example, if there is a system failure during the take-off, the simulation will end (indicating that the system will be lost) and a new

simulation will begin. Whereas, if the system failure occurs during the taxiing phase, the system will enter a maintenance phase and may be able to resume the mission when the repairs are complete.

Phase Name	Duration	Duty Cycle	On System Failure
Taxiing	15 minutes	1	Go to the maintenance phase
Take-off	30 seconds	1.5	End current simulation, start new simulation if applicable
Cruising	4 hours	1	End current simulation, start new simulation if applicable
Landing	30 seconds	1.5	End current simulation, start new simulation if applicable
Taxiing	10 minutes	1	Go to the maintenance phase

When you simulate the phase diagram, the software will simulate the RBD linked to the Taxiing phase block for the specified period of time and then simulate the RBD linked to the Take-off phase, and so on according to the instructions in the phase diagram.

Most of the techniques for working with objects in a phase diagram, such as selecting blocks within the diagram, copying/pasting blocks and arranging blocks, are the same as the options described for RBDs and fault trees. You can also change the style settings for the blocks and/or diagram by opening the Block Style or Diagram Style windows (**Format > Styles > [Block Style/Diagram Style]**).

The ReliaWiki resource portal has more information on phase diagrams at:
http://www.ReliaWiki.org/index.php/Reliability_Phase_Diagrams_(RPDs).

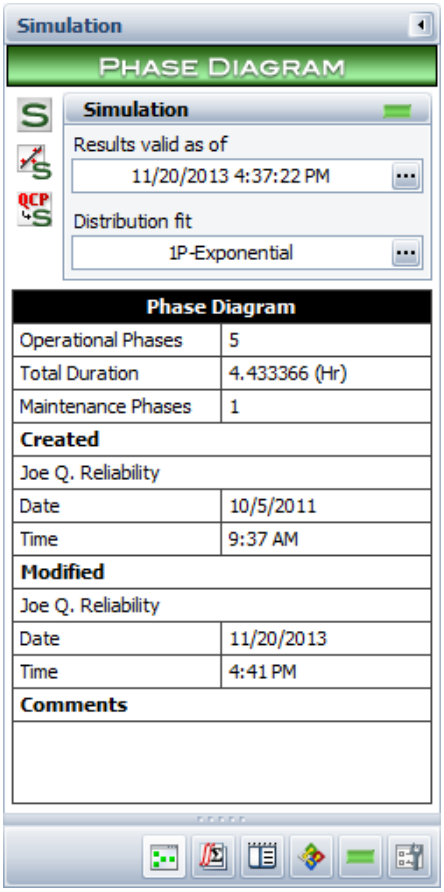
For additional information on how you might use phase diagrams, please read “Reliability Phase Diagram Analysis” at: http://www.ReliaSoft.com/newsletter/v8i1/phases.htm. (Note that this article uses BlockSim 7, but the theory remains the same.)

10.2 Phase Diagram Control Panel

The control panel associated with BlockSim phase diagrams consists of multiple pages, each containing options for performing particular tasks. This section focuses on the Simulation page and Analysis Settings page of the phase diagram control panel, which contains most of the tools you will need to simulate a diagram. (*For more information about the control panel in general, see page 109 in Chapter 3*.)

10.2.1 Control Panel Simulation Page

The Simulation page of the control panel for phase diagrams is shown next.



The Simulation page of the control panel contains the following options:

- The **Simulation** area displays the status and results of the simulation.
  - **Status** If the light is green, then the simulation has been performed. If the light is red, then the simulation has not been performed. A simulated diagram will remain in the simulated state unless a change is made to the structure of the diagram (e.g., a block is added, a connector is removed, etc.).
  - **Show Simulation Details (...)** displays the most recently generated simulation results in the Simulation Results Explorer.
  - **Distribution fit** displays the distribution that has been fitted to the simulation results. Click the button (**...**) to open the Distribution Estimator to fit a distribution and/or for information on the fitted distribution parameters. The fitted distribution can be used to describe the overall behavior of the system and can be published for use as a model elsewhere in the project.
- The **Information** area displays information about the diagram or the currently selected block. If no block is selected in the diagram, the **Information** area displays the number of operational phase and maintenance phase blocks in the diagram, the total duration of all phases in the diagram and information about the diagram creation and last modification. If a block is selected in the diagram, it displays a subset of the properties found in the Block Properties window. In both cases, this area also offers a **Comments** field; click the field heading to enter comments. For blocks, these comments are also accessible in the Identifiers properties in the Block Properties window.

- The **Tools** area on the left side of the control panel gives you quick access to the tools you will need to analyze the diagram and generate additional results.

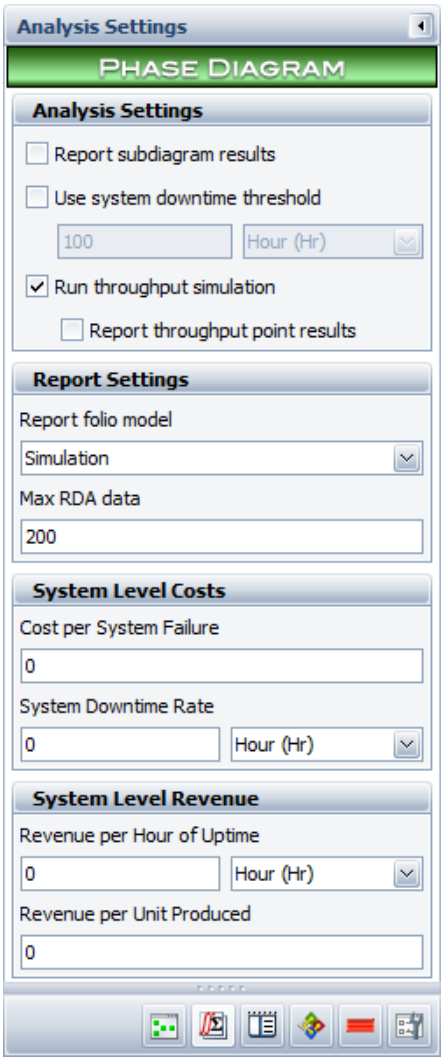
  Simulate opens the Maintainability/Availability Simulation window, which allows you to obtain reliability, maintainability, availability and throughput results from simulations.

  Plot creates a plot based on the simulation results. Clicking the **Plot** icon before the diagram has been simulated will automatically perform the simulation and then plot the data.

  Simulation QCP opens the simulation Quick Calculation Pad (QCP). The QCP allows you to obtain reliability, maintainability and availability metrics based on the simulation results. Clicking the **QCP** icon before the diagram has been simulated will automatically perform the simulation and then open the QCP.

10.2.2 Control Panel Analysis Settings Page

The Analysis Settings page of the control panel for phase diagrams is shown next.



The Analysis Settings page of the control panel contains the following options:

- **Analysis Settings**
  - Select **Report subdiagram results** to provide calculated results for subdiagram blocks in the Simulation Results Explorer. If this check box is not selected, subdiagram blocks will not appear in the results.
  - **Use system downtime threshold** allows you to specify a minimum amount of time that the system must be down in order for it to count toward critical downtime metrics.
  - Select **Run throughput simulation** to perform the throughput simulation, which allows you to perform analyses on the number of units that can be processed by the system in the diagram during the simulation time. If you select to run a throughput simulation, you can select **Report throughput point results** to include throughput results in the System Point Results worksheet, which provides results at specified points in time. If this check box is not selected, only the throughput results for the system at the simulation end time will be reported.
- **Report Settings**
  - **Report folio model** allows you to specify the results model that will be used when the diagram is used as the default data source for an analysis workbook or a Word report. You can choose to use either the simulation results or the fitted model.
  - **Max RDA data** allows you to specify the maximum number of events that can be shown in the RDA Data worksheet (which shows the system up/down data for each simulation) in the Simulation Results Explorer. The contents of the RDA Data worksheet can be transferred to a parametric RDA folio in Weibull++ for further analysis.
- **System Level Costs**
  - **Cost per System Failure** allows you to specify a failure cost per incident at the system level (i.e., an additional cost that is not accounted for by any of the block level costs).
  - **System Downtime Rate** allows you to specify a cost per unit time for downtime at the system level (i.e., an additional cost that is not accounted for by any of the block level costs).
- **System Level Revenue**
  - **Revenue per [Unit] of Uptime** allows you to specify an amount of revenue that is brought in by the system for each unit of uptime. This can then be accounted for as a loss when the system is down.
  - **Revenue per Unit Produced** allows you to specify an amount of revenue that is brought in for each unit produced. This can then be accounted for as a loss when the system is down. This option is available only when you have selected the **Run throughput simulation** option in the **Analysis Settings** area of this control panel page.

10.3 Operational Phase Blocks

In phase diagrams, operational phase blocks are used to represent any stage of the system's mission that is not dedicated exclusively to the execution of maintenance tasks. Operational phases are always defined by (linked to) a simulation RBD in the current project. Each operational phase has a fixed, predefined duration unless a failure occurs during the phase and then it depends on how the properties of the phase block have been configured.

Operational phase blocks can have unlimited incoming connections and up to two outgoing connections, which may include one success path and one failure path. The success path and the failure path must be different; if both success and failure of the block actually lead to the same outcome, there are two ways to model this:

- If the operational phase stops upon failure of the block and the simulation moves to the next phase along the success path, you can use a node block to model this configuration (i.e., the success path links directly to the next phase, and the failure path leads to a node block, which then links to the next phase).
- If the operational phase continues for the specified duration despite failure and the simulation then moves to the next phase along the success path, do not create a failure path.

By default, the first connection that you create from an operational phase block is the success path and the second is the failure path. You can right-click either connector and choose **Set to Success** or **Set to Failure** to swap the two paths.

You can add an operational phase block to a phase diagram by choosing **Phase > Phase > Add Operational Phase**.

The Select Diagram for the Block window will appear. The list of available diagrams will show the simulation RBDs in the project. You can use the **View** area to sort the list hierarchically or alphabetically. The **Filter** field allows you to enter text to search for in the names of the available diagrams. If the diagram that the operational phase block will be linked to does not yet exist, click the **Add RBD** icon to add a new diagram to the project.

To open an operational phase block that already appears in the phase diagram, double-click the block or choose **Phase > Settings > Block Properties** to open the Phase Properties window.

The following properties are available to configure an operational phase block:

- **Phase name** and **Description**: By default, operational phase blocks are named according to the name of the diagram they represent. You can type new text either instead of or in addition to the default name. You can change the default names by choosing **File > Manage Repository > Define Default Names**.

  The **Phase name** field must be populated. Description text is optional. Note that an asterisk is used (*) to represent default block names. The block name that is displayed in the diagram will replace the asterisk with the block's default name; this allows the block name to be updated dynamically.

- **Phase Properties**
  - **Diagram** allows you to select a simulation RBD in the current project that you want the phase block to represent. Note that common components across different RBDs are identified by name. In other words, a component with the exact same name in two RBDs is assumed to be the same component working in two different phases.
  - **Phase duration** allows you to specify how long the system will operate in the configuration defined for this phase. If a failure has not occurred by the end of the specified phase duration, the simulation will proceed along the success path leading from the phase block. If a failure occurs, the simulation will proceed along the failure path leading from the phase block.
  - **Phase duty cycle** allows you to specify a common duty cycle value for the entire RBD that the phase represents, thereby modeling situations where the actual usage of the RBD during system operation is not

identical to the usage for which you have data (either from testing or from the field). This can include situations where the item:

- Does not operate continuously (e.g., a DVD drive that was tested in continuous operation, but in actual use within a computer accumulates only 18 minutes of usage for every hour the computer operates).
- Is subjected to loads that are greater than or less than the rated loads (e.g., a motor that is rated to operate at 1,000 rpm but is being used at 800 rpm).
- Is affected by changes in environmental stress (e.g., a laptop computer that is typically used indoors at room temperature, but is being used outdoors in tropical conditions).

In these cases, continuous operation at the rated load is considered to be a duty cycle of 1. Any other level of usage is expressed as a percentage of the rated load value or operating time. For example, consider the DVD drive mentioned above; its duty cycle value would be 18 min / 60 min = 0.3. A duty cycle value higher than 1 indicates a load in excess of the rated value.

If a duty cycle is specified for the phase and there are also duty cycles specified for blocks within the RBD, their effects are compounded. For instance, consider the aircraft example given in Section 10.1. During the take-off phase, the subsystems experience 1.5 times the normal stress, so you would use a phase duty cycle value of 1.5. We also know that the landing gear is not used continuously during take-off. Assume that the landing gear is actually in use only 30% of the time during take-off. Each landing gear block in the RBD, then, would have a duty cycle value of 0.3. For each block, the effects of the phase duty cycle and the block duty cycle are compounded, yielding an effective duty cycle value of 1.5 x 0.3 = 0.45.

- **Phase Throughput** properties allow you to specify the maximum output that the entire system can process during the particular phase. *For more detail, see Section 10.9 on page 351*.
- **Identifiers**: The **Comments** field allows you to enter additional information about the block.

The **Active Phase** drop-down list at the bottom of the window allows you to choose another phase that you want to edit without the need to close and reopen the Phase Properties window. This option works in exactly the same way as the **Active Block** feature in the Block Properties window.

10.4 Maintenance Phase Blocks

A maintenance phase represents the portion of the overall mission time when the system is down and maintenance actions are being performed on some or all of its components. A maintenance phase block is defined by (linked to) a maintenance template. This template can be thought of as a list of the specific components (blocks) that are designated to undergo inspection, repair or replacement during the maintenance phase, along with their maintenance priority order. For example, if blocks A, B and C are to undergo maintenance during a specific phase, they are placed in a maintenance template in a sequence that defines the priority for completing each task. If three repair crews and three spare parts are available at the same time, the tasks on A, B and C will be carried out simultaneously. However, if only one crew is available, the tasks will be carried out based on the priority order defined in the template.

Note: Maintenance tasks defined in the RBDs represented by operational phase blocks may also be carried out. For more information, see the next section on page343.

Given that many aspects of maintenance can be probabilistically defined (e.g., the time to repair, the time to obtain a spare part, etc.), the duration of a maintenance phase is not fixed; the phase lasts as long as it takes to complete all of the tasks specified in the maintenance template.

Maintenance phase blocks can have unlimited incoming connections and a single outgoing connection.

You can add a maintenance phase block to a phase diagram by choosing **Phase > Phase > Add Maintenance Phase**.

To open a maintenance phase block that already appears in the phase diagram, double-click the block or choose **Phase > Settings > Block Properties** to open the Phase Properties window.

The following properties are available to configure a maintenance phase block:

- **Phase name** and **Description**: Each block is given a default name that is determined based on the block type's default name format. You can type new text either instead of or in addition to the default name. You can change the default names by choosing **File > Manage Repository > Define Default Names**.

  The **Name** field must be populated. Description text is optional. Note that an asterisk is used (*) to represent default block names. The block name that is displayed in the diagram will replace the asterisk with the block's default name; this allows the block name to be updated dynamically.

- **Phase Properties**
  - **Maintenance Template** allows you to select a maintenance template from the current project that you want the phase to represent. Once you have assigned a maintenance template, the number of blocks in the maintenance template is also shown.
  - I**nterval maintenance threshold** allows you to specify the age interval when preventive or inspection tasks based on system or item age intervals (fixed or dynamic) will be performed. This can help you optimize the resources allocated to repair the system during a maintenance phase by performing preventive maintenance or inspections when the system when the system is already down in a maintenance phase. This property provides the ability to add some flexibility to the timing of scheduled preventive maintenance actions and/or inspections.

    For example, imagine that a preventive maintenance action is scheduled for a car (e.g., an oil change, tire rotation, etc.) at 60,000 miles, but a system downing event occurs at 55,000 miles. The Interval Maintenance Threshold allows you to stipulate that the preventive task will be performed upon a system downing event as long as a specified percentage of that 60,000 miles has passed. If, in this case, you specify an Interval Maintenance Threshold of 0.9, the preventive task will be performed if a system downing event occurs after at least 54,000 miles (60,000 miles x 0.9 = 54,000 miles). Thus, the downing event at 55,000 miles will trigger the preventive task. On the other hand, if you were to specify an Interval Maintenance Threshold of 0.95, the preventive task would not be performed upon the system downing event at 55,000, because that event preceded the threshold age of 57,000 miles (60,000 miles x 0.95 = 57,000 miles).
- **Identifiers**: The **Comments** field allows you to enter additional information about the block.

The **Active Phase** drop-down list at the bottom of the window allows you to specify another phase block that you want to edit without the need to close and reopen the Phase Properties window. This option works in exactly the same way as the **Active Block** feature in the Block Properties window.

Note: A maintenance phase will not end until all components are restored. Therefore, if any failed block does not have a task that restores it, the maintenance phase will not end.

10.4.1 Maintenance Phases and Maintenance Tasks in Diagrams

Not all maintenance performed during simulation of a phase diagram is necessarily performed during maintenance phases. There may also be tasks assigned to the blocks within a simulation RBD represented by an operational phase block.

- Scheduled tasks that are assigned to blocks within the simulation RBD are performed as defined, even if they bring the system down.

- For a block that does not bring the system down when it fails, the assigned corrective task (if any) is also performed as defined.
- If the block failure brings the system down, then the connections in the phase diagram determine what happens:
  - If a failure path is defined, the current phase is interrupted and the phase linked to the failure path begins. (This could be a maintenance or another operational phase.) Maintenance properties defined in this new phase now apply. For the block that just failed, the new corrective maintenance properties apply. For other blocks with maintenance in progress, the rules for interrupted maintenances apply (*see Section 10.7 on page 347*).
  - If no failure path is defined, then the phase continues until its specified duration is completed. In this case, the tasks apply as defined in the RBD.

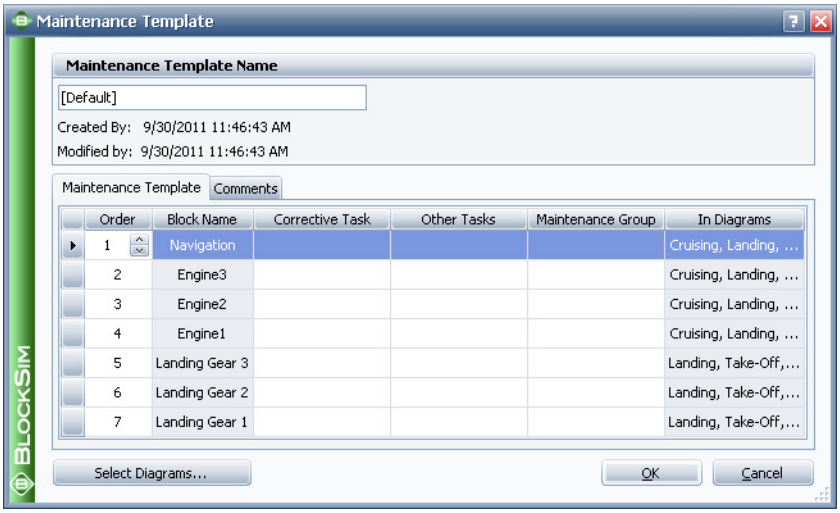
10.4.2 Maintenance Templates

Maintenance templates are resources that serve as the basis for maintenance phases in phase diagrams. The templates contain the repairable blocks of a system that will need maintenance during the phase.

What's Changed? In BlockSim 7, a maintenance template was a separate collection of blocks not associated with any specific diagram. When a template was used in a maintenance phase, the maintenance tasks for each block would apply to any diagram that contained blocks with the same name. In the Synthesis version, the maintenance tasks for each block will apply only to blocks in the diagrams used to create that template. If two blocks in a diagram have the same name, the phase simulation will not run.

To create a new maintenance template, click the **Create New Maintenance Template** icon in the Maintenance Template wizard, which is accessed from the Maintenance Template field in the Phase Properties window,

or, on the Maintenance Templates page of the Resource Manager, choose **Main > Resource > New**. In the Select Diagrams window, select the RBD(s) that the template will pull information from. After you have selected at least one diagram, the Maintenance Template window will open with a row for each block that appears in the selected diagram(s).



For a new resource, a name will be proposed automatically based on the default naming criteria established for the current database (in the Define Default Name Formats window). You can replace this with your own name of up to 150 characters, if desired. Remember that the name and identifiers are the primary way in which your team will be able to find the Synthesis resources you need for your analyses.

The following settings are available to configure the template:

- **Maintenance Template Tab**
  - **Order** allows you to rearrange the priorities for performing maintenance on the blocks listed in the template. Clicking inside a cell in this column will cause an up arrow and a down arrow to appear; you can use these arrows to move the block to the desired priority order, or you can drag the row to the desired location.
  - **Corrective Task** allows you to apply a corrective maintenance task to a block. You can choose an existing task from the drop-down list. In addition, you can click the **Select Existing** icon to open the Select Resource window. If there is not an existing task that meets your needs, you can create a new one by clicking the **Create New** icon. Click the **View/Edit** icon to view the characteristics of the selected task.

  - **Other Tasks** allows you to apply preventive, on condition and/or inspection tasks to each block (multiple tasks can be applied).
  - **Maintenance Group** allows you to identify each block as part of a maintenance group. Events within maintenance groups may trigger maintenance or state changes for other blocks.
  - **In Diagrams** shows where the block appears (i.e., in which of the diagrams that the template pulls information from).

  It is important to note that the settings specified here override those specified via the Block Properties window. Therefore, any tasks or maintenance groups that are assigned directly to a block via the Block Properties window will be ignored when the diagram is simulated as part of the simulation of a phase diagram. Only those tasks and maintenance groups specified in the maintenance template will be used under these circumstances.
- **Comments Tab** allows you to enter comments about the maintenance template.
- **Select Diagrams** allows you to change which RBD(s) the template pulls information from.

For an existing resource, a link at the bottom of the properties window indicates how many times it is currently being used. If you need more information, click the link or the **Trace Usage** icon.

This opens the Dependency Viewer, which allows you to explore both where the resource is being used and what other resources are associated with it.

Once you have created a maintenance template, you will be able to include a maintenance phase in a phase diagram simulation.

Note: If multiple blocks have the same name in a given diagram, a message will be displayed to notify you of this problem. Although you can still create the maintenance template, you will not be able to simulate a phase diagram that uses the maintenance template until you have renamed the blocks in the RBD to avoid duplicate block names.

10.5 Subdiagram Phase Blocks

Subdiagram phase blocks represent other phase diagrams within the project. Using subdiagram phase blocks allows you to incorporate phase diagrams as phases within other phase diagrams. This allows you to break down extremely complex configurations into smaller diagrams, increasing understandability and ease of use and avoiding unnecessary repetition of elements.

Subdiagram phase blocks can have unlimited incoming connections and up to two outgoing connections, which may include one success path and one failure path. The success path and the failure path must be different; if both success and failure of the block actually lead to the same outcome, you can use a node block to model this configuration.

You can add a subdiagram phase block to a phase diagram by choosing **Phase > Phase > Add Subdiagram Phase**.

The Select Diagram for the Block window will appear. The list of available diagrams will show the phase diagrams in the project. You can use the **View** area to sort the list hierarchically or alphabetically. The **Filter** field allows you to enter text to search for in the names of the available diagrams. If the diagram that the subdiagram phase block will be linked to does not yet exist, click the **Add Phase Diagram** icon to add a new diagram to the project.

To open a subdiagram phase block that already appears in the phase diagram, double-click the block or choose **Phase > Settings > Block Properties** to open the Phase Properties window.

The following properties are available to configure a subdiagram phase block:

- **Phase name** and **Description**: By default, subdiagram phase blocks are named according to the name of the phase diagram they represent. You can replace this with your own name, if desired. You can change the default names by choosing **File > Manage Repository > Define Default Names**.

  The **Phase name** field must be populated. Description text is optional. Note that an asterisk is used (*) to represent default block names. The block name that is displayed in the diagram will replace the asterisk with the block's default name; this allows the block name to be updated dynamically.

- **Phase Properties**
  - **Phase Diagram** allows you to select a phase diagram in the current project that you want the block to represent.
- **Identifiers**: The **Comments** field allows you to enter additional information about the block.

The **Active Phase** drop-down list at the bottom of the window allows you to choose another phase that you want to edit without the need to close and reopen the Phase Properties window. This option works in exactly the same way as the **Active Block** feature in the Block Properties window.

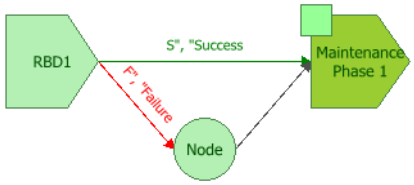
10.6 Node Blocks and Stop Blocks

In the Synthesis version of BlockSim, the two possible outcomes of an operational phase block are modeled using success and failure paths. Where previously a failure outcome was defined as part of the operational phase block's properties, it is now graphically represented within the diagram. Node blocks and stop blocks are provided to allow you to build configurations that are both accurate and readable.

Node Blocks

The purpose of a node block is simply to enable configurations that would otherwise not be possible due to limitations on connecting blocks. For example, consider an instance where maintenance is scheduled to be performed after the operational phase has completed successfully, and if a failure occurs during simulation, that maintenance will take place upon failure. In this case, the operational phase block's success and failure outcomes are identical. Success paths and failure paths cannot be identical in phase diagrams, however, so you would model this configuration in one of two ways:

- If the operational phase stops upon failure of the block and the simulation moves to the next phase along the success path, you would use a node block to model this configuration, as shown next.

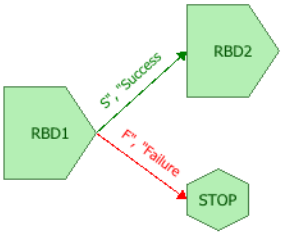


- If the operational phase continues for the specified duration despite failure and the simulation then moves to the next phase along the success path, you would simply not create a failure path. If there is only a success path, then upon system failure during this phase, the corrective maintenance specified begins and the time to restore the system is part of the operational phase's time. In other words, the repairs continue in the operational phase until the system is up again. If the repairs are not completed before the phase ends, the repairs continue into the next phase. Thus, in this case, the duration of an operational phase is not affected by a system failure. For example, consider a production line operating in two phases: a day shift and a night shift. A failure occurs in the day shift that renders the production line non-operational. Repair of the production line begins immediately and continues beyond the day shift. The production line is back up after midnight. In this case, the repair of the production line exhausts all of the duration of the day shift phase from the time of the failure to the end of the phase. Some part of the night shift phase is also exhausted.

Node blocks can have unlimited incoming connections and a single outgoing connection. They have no properties other than their block name. By default, node blocks are named “Node.” You can replace this with your own name, if desired. You can change the default names by choosing **File > Manage Repository > Define Default Names**.

Stop Blocks

Stop blocks indicate that the simulation of the mission ends. A new simulation may then begin, if applicable. This is useful in situations where maintenance is not possible upon failure.



Stop blocks can have unlimited incoming connections. No outgoing connections can be defined for stop blocks.

10.7 Phase Diagram Simulation Assumptions

The underlying assumptions when performing simulations are:

- Phase duration is based on calendar time.
- Age is carried over from phase to phase.

Maintenance tasks in progress during one operational phase can be interrupted if that phase ends before the repair is completed. For example, a crew delay or spare parts order may extend the duration of a repair beyond the duration of

the phase. As described next, the software handles these interruptions differently, based on the stage in which the repair was interrupted and whether or not the failed block is present in the next contiguous phase.

- If a phase ends during the repair of a failed block and the block is present in the next contiguous phase:
  - If the same task is present in both phases, then the task will continue as-is in the next phase. This is considered an uninterrupted event, and counts as a single unique event at both the block and the system level.
  - If the interrupted task is not used in the next phase, then the task is cancelled and new tasks are applied as needed. In this case, all crew calls are cancelled and spare parts are restocked.
    - If the repair has started or the crew is delayed (crew logistic delay), the call will be assumed accepted and the component will be charged for it. If the crew was occupied with another component's repair, the call will be assumed rejected and hence not charged to the component.
    - If the call for spare parts incurred emergency charges, those are charged to the block; otherwise, there are no other charges to the block.
- If a phase ends during the repair of a failed block and the block is not present in the next contiguous phase, then the task is cancelled and new tasks are applied as needed. All crew calls are cancelled and spare parts are restocked.
  - If the repair has started or the crew is delayed (crew logistic delay), the call will be assumed accepted and the component will be charged for it. If the crew was occupied with another component's repair, the call will be assumed rejected and hence not charged to the component.
  - If the call for spare parts incurred emergency charges, those are charged to the block; otherwise, there are no other charges to the block.
  - Discontinuous events are counted as two distinct events at both the block and the system level.
  - When the system fails in a phase that has a failure path leading to a stop block, the system will remain down for the remainder of the simulation. From that point on, the blocks that are down are assumed unavailable and the blocks that are up are assumed operational for availability calculations.

A system is considered down and unavailable during the execution of a maintenance phase and remains down until all components have been repaired or maintained according to the properties specified for the maintenance phase. A maintenance phase is executed when the simulation reaches the phase while progressing through the phase diagram, either following a success path or a failure path. The following assumptions apply to both cases.

- When a component enters a maintenance phase in a down state, the following rules apply:
  - If a task is in progress for this component, the event will transfer to the maintenance phase provided that the same task is present in the maintenance phase. The rules for interrupted tasks apply as noted above.
  - If the component is failed but no corrective maintenance is in progress (either because the component was non-repairable in the phase where it failed or because it had a task scheduled to be executed upon inspection and was waiting for an inspection), a repair is initiated according to the corrective maintenance properties specified for the component in the maintenance phase.
  - Failed components are fixed in the order in which they failed.
- When a component enters a maintenance phase in an operating state, the following rules apply:
  - Maintenance will be scheduled as follows:

    1. Tasks based on intervals or upon start of a maintenance phase
    2. Tasks based on events in a maintenance group, where the triggering event applies to a block
    3. Tasks based on system down
    4. Tasks based on events in a maintenance group, where the triggering event applies to a subdiagram

Within these categories, order is determined according to the priorities specified in the maintenance template (i.e., the higher the task is on the list, the higher the priority).

- An inspection or preventive task may be initiated, if applicable, with inspections taking precedence over preventive tasks. Inspections and/or preventive tasks are initiated if one of the following applies:
  - Upon certain events:
    - The task is set to be performed when a maintenance phase starts.
    - The policy is set to be performed based on events in a maintenance group and one of those events occurs within the one of the specified maintenance groups. Note that such a triggered maintenance does not follow the priorities specified in the maintenance template, but is sent to the end of the queue for repair.
    - The task is set to be performed whenever the system is down.
  - At certain intervals:
    - The task is set to be performed at a fixed time interval, based on either item age or calendar time, and the maintenance falls within the maintenance threshold specified in the maintenance phase.

  If the inspection task is not set to bring either the item or the system down, the inspection will still be considered a downing inspection.

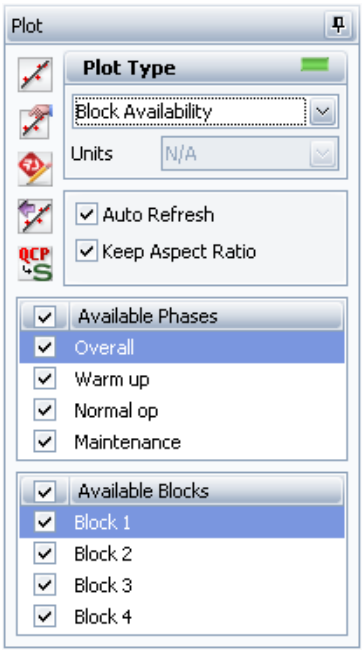
Finally, if a block enters a maintenance phase in a failed state:

- If the block does not have a corrective task in the maintenance phase but does have an on condition task, the preventive portion of the on condition task is triggered immediately in order to restore the block.
- A maintenance phase will not end until all components are restored. Therefore, if any failed block does not have a task that restores it, the maintenance phase will not end.

10.8 Results and Plots

The results available for phase diagrams, including those available in the Simulation Results Explorer and those available in the simulation QCP, are identical to the results available for simulation diagrams.

The plots available for phase diagrams are the same as the simulation diagram plots (*page 328 in Chapter 9*), with the exception that there are also system - phases plots available. In addition, for certain plots, you must specify which phase(s) you want to show in the plot.



- **Available Phases** allows you to specify which phases to display on the plot. To select all phases, click the check box in the table header. To select individual phases, click the check box in each row.

  The **Available Phases** option is available for all system - phases plots, metrics plots and blocks plots (other than the Block Backlog plot, which provides information about the backlog at the end of the simulation time, and the Block Bubble plot). It is also available for all crews plots and spare part pools plots if your analysis includes those items.

The system - phases plots display information about the individual phases that make up the phase diagram.

- **Availability - All Events** shows the percent of time that the system is available during the simulation, taking into account all system downing events. Point to a bar to view the system availability during that phase.
- **Availability w/o PM, OC & Insp** shows the percent of time that the system is available during the simulation, taking into account all downing events, but not preventive, on condition or inspection tasks. Point to a bar to view the system availability during that phase.
- **Costs** shows the system costs during the simulation. Point to a bar to view the system availability and costs accrued during that phase.
- **Downtime** shows the amount of time the system is down during the simulation. Point to a bar to view the system availability and downtime during that phase.
- **Expected Failures** shows the expected number of failures for the system during the simulation. Point to a bar to view the system availability and expected number of failures during that phase.
- **Expected Downing Events** shows the expected number of downing events for the block during the simulation. Point to a bar to view the system availability and expected number of downing events during that phase.
- **Throughput** shows the total number of units processed/produced by the block during the simulation time. Point to a bar to view the system availability and throughput during that phase. If your simulation does not include throughput then no information will be displayed for this plot.

10.9 Throughput Analysis - Phase Diagrams

If you are performing throughput analysis for a phase diagram, you will need to specify, for each operational phase, how its possible throughput is calculated. The phase throughput properties in the Phase Properties window allow you to specify the maximum throughput that the entire system can process during the particular phase. *For a detailed discussion of throughput at the block level, see page 277 in Chapter 9.*

A phase's throughput can be thought of as the initial throughput that enters the system modeled in the linked RBD. For example, imagine a textile factory that receives different quantities of raw materials during different seasons. These seasons could be treated as different phases. In this case a phase may be seen as sending a certain quantity of units to the first component of the system (the textile factory in this case). Depending on the capacity and availability of the factory, these units may be all processed or a backlog may accumulate.

Phase throughput can be one of the following:

- **Unrestricted throughput** specifies that the amount of output the system can process during the phase will be equal to the amount that the linked diagram can process during the specified phase duration.
- **Constant throughput** allows you to specify an amount of throughput per unit time that is the same throughout the entire simulation.
- **Variable throughput** allows you to specify an amount of throughput per unit time that varies over the course of the simulation. For example, the flow of oil from a well may drop over time as the oil reserves are depleted so the amount of throughput per unit time will decrease as the simulation time increases. If you select this option you must also specify the following:
  - **Throughput model** allows you to choose the model that describes the change in throughput, y, as a linear function of time, x. You can choose from three general models for this purpose:
    - Linear
    - Exponential
    - Power

    The parameters, a and b, for each of these models are chosen by the user based on engineering judgment and on any previous data.
  - **Maximum throughput capacity** allows you to specify the maximum throughput capacity of the phase. Once this maximum throughput capacity value is reached, the throughput per unit time becomes constant and equal in value to the maximum throughput capacity specified by the user. In this situation, the variable throughput model would then act as a constant throughput model.

    For example, suppose a machine has a constant throughput when functioning normally. When an outage happens, it takes about 12 days to restore the machine. Every time the machine fails, it is repaired and then the throughput goes through a warm up period of 5 days until it reaches normal production capacity again. During this warm up period, the machine is not operating at its maximum level, but after the warm up period and the machine returns to its normal function and achieves its maximum and constant throughput.

  Even though only three models are available, they provide considerable modeling flexibility. This flexibility is achieved by using the functions as building blocks for more complex functions. As an example, you can create a step model by using multiple phases that each have a constant throughput. A ramp model would use phases with linearly increasing functions in conjunction with constant phases, and so forth.

The ReliaWiki resource portal has more information on throughput analysis in phase diagrams at: http://www.ReliaWiki.org/index.php/Reliability_Phase_Diagrams_(RPDs).

RENO Flowcharts 11

RENO flowcharts are used to build models of probabilistic or deterministic problems, processes or scenarios in order to help you understand and/or quantify them. The flowchart is a graphical representation of the mathematical process of predicting an outcome. Possible applications include risk and safety analysis, complex reliability modeling, decision making, maintenance planning, optimization, operational research, financial analysis and much more.

A RENO flowchart consists of various types of blocks that are connected in order to model the scenario. In addition, RENO resources serve to store or generate values used in the flowchart. Resources cannot be placed directly into the flowchart; instead, they are referenced or called by the blocks.

This chapter addresses the following:

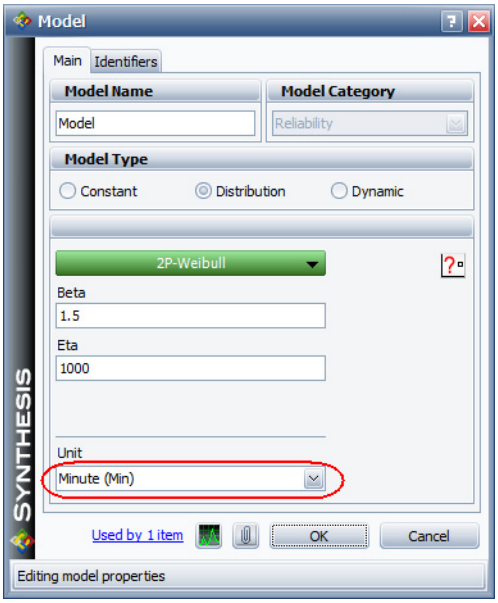
- Using time units in RENO - Section 11.1 (p. 354)
- Configuring and using blocks in RENO flowcharts - Section 11.2 (p. 355)
- Configuring and using RENO resources - Section 11.3 (p. 380)
- Using the Function Selector and Equation Editor to simplify building equations - Section 11.4 (p. 390)
- Standard and special operators available for use in equations in RENO blocks and resources - Section 11.5 (p. 391)
- Flowchart configurations and execution - Section 11.6 (p. 393)
- Debugging a flowchart - Section 11.7 (p. 396)
- Using the control panel - Section 11.8 (p. 397)
- Flowchart simulation - Section 11.9 (p. 399)
- Simulation results and plots - Section 11.10 (p. 402)

11.1 Using Time Units in RENO

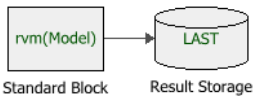
For each database, authorized users can configure which time units will be available for time-related inputs and results, and they can define conversion factors that allow the software to automatically convert data from one time unit to another (see Manage Units window).

In RENO, this means three things:

- Results are always given in terms of the system base unit (SBU).
- Resources that do not require you to define a time unit (e.g., RENO tables, variables, cost models, etc.) are assumed to be in terms of the base unit. This also applies to all constants and parameters used in equations, RENO predefined functions and RENO internal functions.
- Resources that require you to define a time unit (e.g., reliability models, duration models, etc.) will have their values automatically converted to the base unit during simulation. For example, let's assume that in your database, an hour is equal to 1 SBU. Now suppose your model is a Weibull distribution that is defined in terms of minutes, as shown next.



The following flowchart shows a block that obtains a random value from this model.



If rvm(Model) = 60 minutes, then the output of the model in a RENO flowchart would be 1 hour (automatically converted from 60 minutes).

Tip: Whenever you don't want the software to convert the value obtained by RENO from a model resource, make sure the model uses a unit that is defined as equal to 1 SBU. To view the conversion factors for the units, open the Manage Units window by choosing **File > Manage Repository > Manage Units**.

For an example, you can open the "Snow Blower" project in the RENO9_Examples.rsgz9 database that is installed with the software. In this analysis, the "Depth" model describes snow depth (in inches) and the "Area" model describes the area that needs to be cleared (in square feet). These models use the "Hour" unit, with SBU = 1, to ensure that values obtained from the models do not get converted.

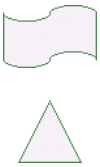
11.2 Flowchart Block Types

In order to provide the flexibility required to create flowcharts that model scenarios as realistically as possible, there are several different types of blocks. For the different types of blocks, the appearance and options available in the Block Properties window will vary. This section is intended to help you identify the basic types of blocks found in flowcharts.

What's Changed? In previous versions of the software, the blocks used in RENO flowcharts were known as constructs.

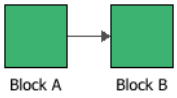
Note that the shapes, colors and locations shown here represent the default settings. These settings may be changed via the Block Style window (**Format > Styles > Block Style**) and/or the Diagram Style window (**Format > Styles > Diagram Style**).

Visual Indicator	Block Type	Description
	Flowchart Standard Block	A flowchart standard block evaluates a mathematical expression, and then passes the result (output value) of the expression to the next block(s) in the flowchart.
	Result Storage Block	A result storage block stores numerical values passed to it during simulations, and then computes or holds a result
	Conditional Block	A conditional block functions as an "if" statement. It checks the incoming value against a conditional expression, with possible outcomes of true and false.
	Binary Node	A binary node takes an incoming value and multiplies it by a specified value. The resulting value is passed down the "true" path. The value passed down the "false" path is the incoming value multiplied by (1 minus the specified value). Binary nodes are primarily intended to facilitate building decision trees.
	Logic Gate	A logic gate checks multiple incoming values against a conditional expression, with possible outcomes of true and false. It determines the overall block outcome based on the gate type, which specifies how many of the incoming values must meet the condition in order for the outcome to be true.

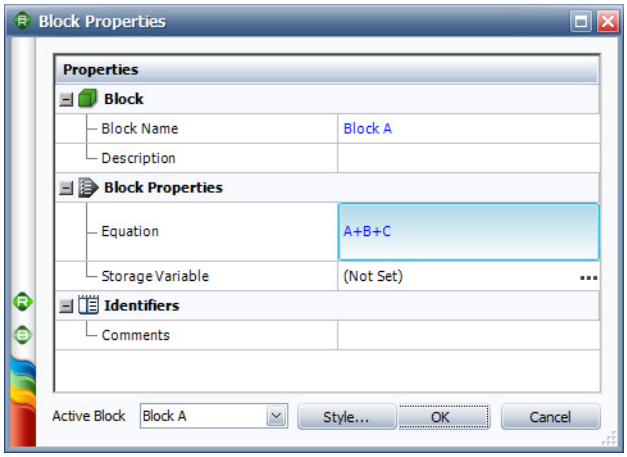
Visual Indicator	Block Type	Description
	Branch Gate	A branch gate functions as a “switch” statement. It checks the incoming value against a number of cases, and passes different outputs depending on which case or branch evaluates to true.
	Summing Gate	A summing gate performs a simple mathematical operation on all incoming values (addition, subtraction, multiplication, division, averaging incoming values or finding the maximum or minimum of the incoming values), and returns a single value to all outgoing paths.
	Flag Marker and Go-to-Flag Blocks	Flag markers and go-to-flags are used to build subroutines and loops that repeat steps in a flowchart until a certain condition is reached. The flag marker marks a spot in the path of the flowchart, and the go-to-flag transfers the flow of execution to that spot.
	Stop Flag	A stop flag halts the simulation process when it is encountered. No further simulations, runs or analyses will be performed until you next simulate the flowchart.
	Counter Block	A counter block records the number of times the simulation has passed through it, and then passes the recorded value to the next block in the flowchart.
	Reset Block	A reset block forces RENO to generate new values for all static functions in the flowchart while simulation is in progress.
	UI Block	A UI block pauses flowchart simulation until the user provides some form of input.
	Subchart Block	Subchart blocks represent other flowcharts within the project. Subchart blocks are generally used to break down a larger flowchart into simpler steps to reduce complexity and improve traceability.

11.2.1 Flowchart Standard Blocks

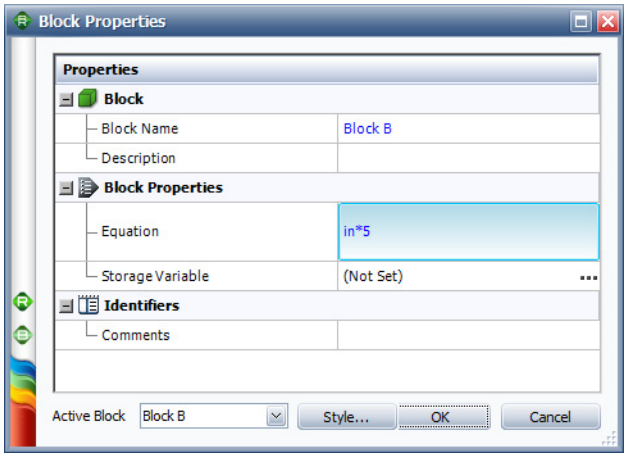
A standard block evaluates a mathematical expression, and then passes the result (output value) of the expression to the next block(s) in the flowchart. For example, the following configuration shows two standard blocks in a flowchart. The direction of the arrow indicates that the output of Block A is passed to Block B.



The following picture shows the Block Properties window of Block A. The **Equation** field contains the expression that the block evaluates. In this case, the expression is A+B+C.



The following picture shows the Block Properties window of Block B. In this block, the output of the expression in Block A (represented by the RENO internal function in) is multiplied by 5.



Standard blocks may accept up to one incoming path (input value), and may have multiple outgoing paths (i.e., the same output value is passed to more than one block). In addition to passing the output to the next block(s) in the flowchart, the standard block may be configured to store its output in a RENO variable. The **Storage Variable** field in the Block Properties window of the block displays the name of the variable, if any.

The expression in the standard block can include:

- Numerical values
- Standard and special operators
- Predefined mathematical functions (exp, log, sin, etc.)
- RENO internal functions
- RENO resources
- References to any analysis workbooks

You can enter the equation manually, or you can use the function selector or the equation editor to assist you.

RENO resources, analysis workbooks and internal functions are automatically color-coded to assist in reading the equation. When the cursor is positioned on one of these items, a preview area appears and displays a summary of the item. If the item is a resource, you can click the **View** icon in the Preview area to open that resource's properties window.

IMPORTANT: All values and results are assumed to be in terms of the default unit specified for the database. See the "Using Time Units in RENO" section for details on how this may affect your analysis.

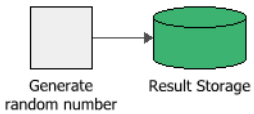
11.2.2 Result Storage Blocks

A result storage block stores numerical values passed to it during simulations, and then computes or holds a result based on one of the following options:

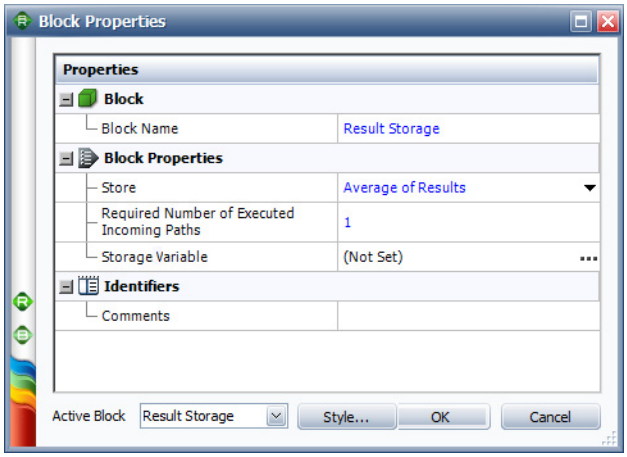
- **Average of Results** computes the average value of the numbers that were passed to the result storage block. Each time a new value is passed to the block, the average is recomputed. If the block is called during simulation, it will return the latest average of the values.
- **Sum of Results** computes the sum of the numbers that were passed to the result storage block. The total sum is recalculated with each new value that is passed to the block. If the block is called during simulation, it will return the latest sum of the values.
- **Array of Results** allows the result storage block to hold an array of values. These values may be used to perform various statistical calculations, such as calculating the variance, standard deviation, etc. The values stored in the block are displayed in the Simulation Results Explorer. If the block is called during simulation, it will return only the most recent value that was added to the array.
- **Minimum of Results** holds the smallest numerical value that was passed to the result storage block. Each new value that is passed to the block is compared to the value that is currently stored. If the new value is less than the stored value, the block discards the stored value and replaces it with the new value; otherwise, the new value is discarded.
- **Maximum of Results** holds the largest numerical value that was passed to the result storage block. Each new value that is passed to the block is compared to the value that is currently stored. If the new value is greater than the stored value, the block discards the stored value and replaces it with the new value; otherwise, the new value is discarded.

- **Last Value** holds only the latest value that was passed to the result storage block. Each new value that is passed to the block replaces the old value.
- **Normalized Sum of Results** adds the values that were passed to the result storage block and divides by the number of simulations. If the block is called during simulation, it will return the summed value currently stored divided by the total number of simulations to be performed.

For example, if you select the **Average of Results** option, and then run 100 simulations on the following configuration (a single simulation is a single pass through the flowchart), then the block that generates a random number will pass a total of 100 random values to the result storage block, which will compute the average value of the numbers as they come in.



The following picture shows the Block Properties window of the result storage block in the example. The **Store** field allows you to select the output of the block. In this case, it stores the average of results.



A result storage block can have any number of incoming paths and any number of outgoing paths (i.e., the same output value is passed to more than one block). In addition to storing the results, the block may be configured to store the output in a variable. The **Storage Variable** field in the Block Properties window of the block displays the name of the variable, if any. If the result storage block is holding an array of results, only the most recent value that was added to the array will be stored in the variable.

Advanced Configurations

- If your block has multiple incoming paths, you can configure the block to not pass an output or compute a result until a certain number of those incoming paths have been executed. This allows you to control the number of inputs that the block processes at any given time. To do this, enter the required number of inputs in the **Required Number of Executed Incoming Paths** field in the Block Properties window. Starting in version 9.0.10, an asterisk (*) in this field means that all incoming paths are required. See the "Flowchart Configurations and Execution" section for an example.

11.2.3 Conditional Formatting

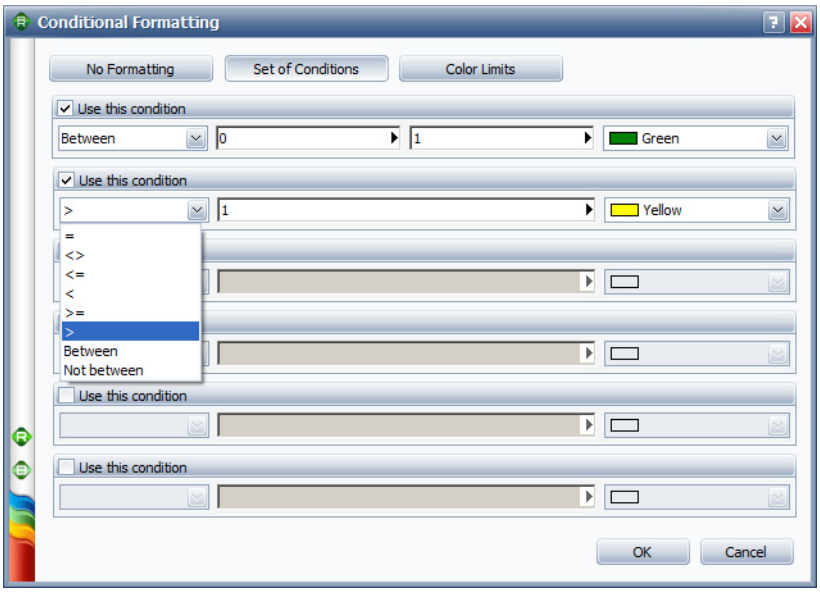
Conditional formatting allows you to mark the result storage block with a corner indicator in a particular color when it meets a certain condition. For instance, you might wish to mark the block with red if the computed result is less than a particular value. The conditional formatting corner indicator appears at the upper left corner of the block. To apply conditional formatting, select the result storage block(s) and choose **Format > Styles > Conditional Formatting** or right-click the result storage block(s) and select **Conditional Formatting** on the shortcut menu.

There are two types of conditional formatting that you can apply in the Conditional Formatting window:

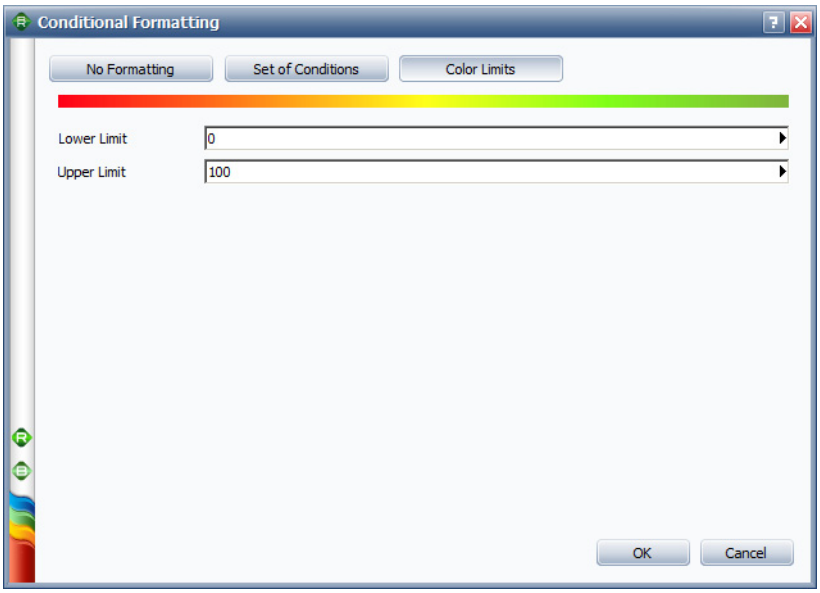
- You can click the **Set of Conditions** button to define conditions and specify the color of the block under each condition. To enable each condition, select the **Use this condition** check box. The fields directly below will be enabled, allowing you to select the conditional operator, enter the value(s) against which the value of the result storage block will be checked and specify a color to be used when the value meets the condition. Available conditional operators are:

=	equal to
<>	not equal to
<=	less than or equal to
<	less than
>=	greater than or equal to
>	greater than
Between	
Not between	

  You can define up to six conditions, and can enable and disable them with the **Use this condition** check boxes. The conditions you have defined will be saved even when disabled. Enabled conditions do not need to be immediately consecutive. Keep in mind, however, that the conditions are evaluated in the order in which they are displayed in this window and that if a result storage block meets more than one condition, the first true condition will be applied.



- You can click the **Color Limits** button to define a range of values across which the range of colors from red to green will be applied. Result storage blocks with values close to the specified lower limit will be marked with red corner indicators; those with values close to the specified upper limit will be marked with green corner indicators.



In both types, for the value(s) against which the result storage block is checked, the expression can include:

- Numerical values
- Standard and special operators
- Predefined mathematical functions (exp, log, sin, etc.)
- RENO internal functions
- RENO resources
- References to any analysis workbooks

You can enter the equation manually, or you can use the function selector or the equation editor to assist you.

RENO resources, analysis workbooks and internal functions are automatically color-coded to assist in reading the equation. When the cursor is positioned on one of these items, a preview area appears and displays a summary of the item. If the item is a resource, you can click the **View** icon in the Preview area to open that resource's properties window.

IMPORTANT: All values and results are assumed to be in terms of the default unit specified for the database. See the "Using Time Units in RENO" section for details on how this may affect your analysis.

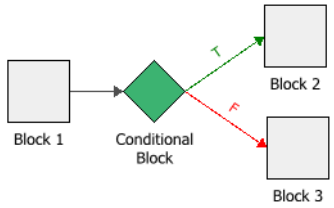
You can click the **No Formatting** button to turn off conditional formatting for the selected block(s).

Note that for result storage blocks of type array, the conditional formatting will be based on the most recent value. The size of the conditional formatting corner indicators can be changed via the Block Corner Indicators pages of the Diagram Style window (**Format > Styles > Diagram Style**).

11.2.4 Conditional Blocks

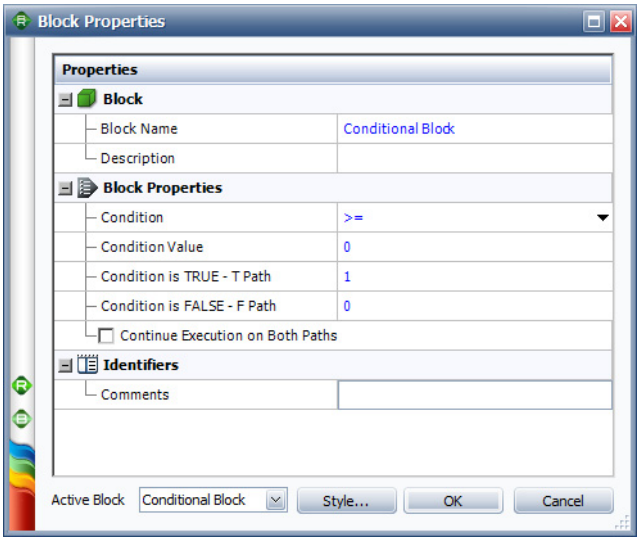
A conditional block functions as an "if" statement. It checks the incoming value against a conditional expression, with possible outcomes of true and false. A conditional block accepts one incoming path and may have one or two outgoing paths.

For example, in the configuration shown next, the conditional block evaluates the output from Block 1 against a conditional expression. If the outcome is true, then an output is passed down the "true" (T) path to Block 2, but if the outcome is false, an output is passed down the "false" (F) path to Block 3.



Tip: RENO designates the first connector drawn from a conditional block, binary node or logic gate as the "true" path and the second connector as the "false" path. You can switch these designations by right-clicking the connector and choosing **Set to Failure** or **Set to Success** on the shortcut menu.

The following picture shows the Block Properties window of the conditional block in the example. In this case, if the output of Block 1 is greater than or equal to 0, then the value "1" is passed down the "true" path. However, if the output of Block 1 is less than 0, then the value 0 is passed down the "false" path.



The **Condition** and **Condition Value** fields allow you to configure the conditional expression of the block. The drop-down list in the Condition field includes the following relational signs:

=	equal to
<=	less than or equal to
<	less than
>=	greater than or equal to
>	greater than
FP<=%	a special case where the conditional block itself draws a random number uniformly distributed from 0 to 100, and then evaluates whether the number is less than or equal to the condition value.
<>	not equal to

For the condition value, as well as the outgoing "true" and "false" values, the expression can include:

- Numerical values
- Standard and special operators
- Predefined mathematical functions (exp, log, sin, etc.)
- RENO internal functions
- RENO resources
- References to any analysis workbooks

You can enter the equation manually, or you can use the function selector or the equation editor to assist you.

RENO resources, analysis workbooks and internal functions are automatically color-coded to assist in reading the equation. When the cursor is positioned on one of these items, a preview area appears and displays a summary of the item. If the item is a resource, you can click the **View** icon in the Preview area to open that resource's properties window.

IMPORTANT: All values and results are assumed to be in terms of the default unit specified for the database. See the "Using Time Units in RENO" section for details on how this may affect your analysis.

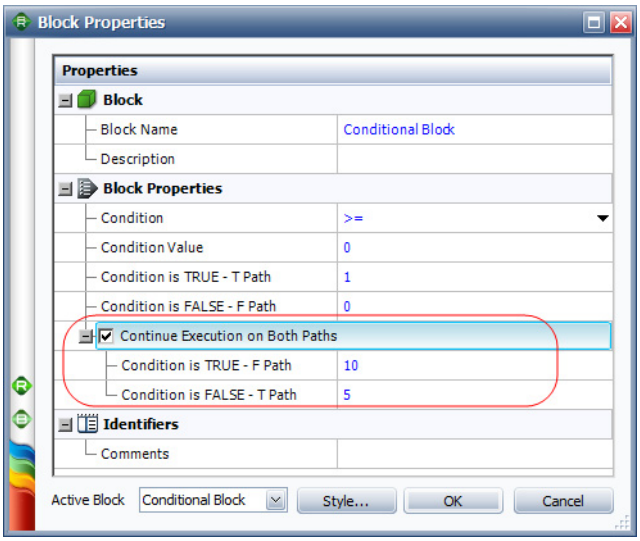
Advanced Configurations

- You can configure the conditional block to have only one outgoing path. This is useful for situations where you want the simulation to end based on a certain condition so that the next simulation, if any, can begin. For example, in the following configuration, if the condition evaluates to true, then the simulation will continue down the "true" path, but if the condition evaluates to false, then the simulation will end because no "false" path exists.

- By default, if the condition evaluates to true, then the specified value will be passed down the "true" path and nothing will be passed down the "false" path. Similarly, if the condition evaluates to false, a value will be passed down the "false" path only. If you wish to pass a value down both paths regardless of the outcome of the

evaluation, select the **Continue Execution on Both Paths** check box. This allows you to specify the values to pass to the opposite paths in each case.

In the example shown next, if the incoming value is greater than or equal to zero, the value 1 will be passed down the "true" path and the value 10 will be passed down the "false" path. Similarly, if the condition evaluates to false, the value 0 is will be passed down the "false" path and the value 5 will be passed down the "true" path. (*See also Section 11.6 on page 393 for a discussion on how flowcharts are executed.*)



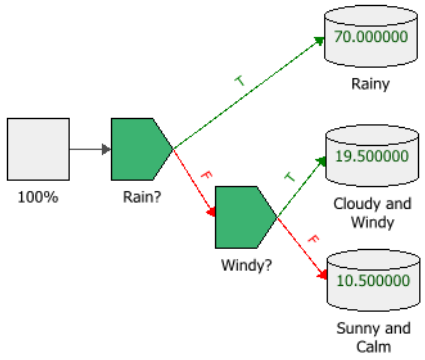
11.2.5 Binary Nodes

A binary node takes an incoming value and multiplies it by the number specified in the node. The resulting value is passed down the "true" path. The value passed down the "false" path is the incoming value multiplied by (1 minus the specified value). In this case, "true" and "false" are simply the names of the paths, and do not indicate anything other than how the values passed down them are computed. Note that both the incoming value and the number specified in the node are intended to represent probabilities $0 < P < 1$, although RENO places no constraints on the values entered.

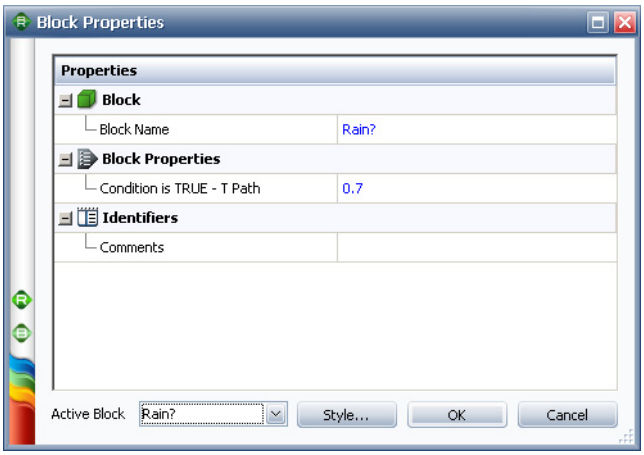
Each binary node accepts one incoming path and may have one or two outgoing paths.

Tip: RENO designates the first connector drawn from a conditional block, binary node or logic gate as the "true" path and the second connector as the "false" path. You can switch these designations by right-clicking the connector and choosing **Set to Failure** or **Set to Success** on the shortcut menu.

In the example shown next, a meteorologist predicts that there is a 70% chance of rain tomorrow. If it doesn't rain, there is a 65% chance that the day will be cloudy and windy, and a 35% chance that the day will be sunny and calm. The flowchart below shows the probability of each outcome.



Here, the first block in the example has a value of 100, representing the 100% probability that the day will have weather of some sort. This value is passed to the Rain? binary node, shown next.



The incoming value of 100 is multiplied by the value in the **Condition is TRUE - T Path** field (i.e., 0.7) and the resulting value of 70 is passed to the "true" path. This results in a 70% chance that tomorrow will be rainy. The Rain? binary node then multiplies the incoming value of 100 by (1 minus the value in the Condition is TRUE - T Path field) (i.e., 1 - 0.7 = 0.3) and passes the resulting value of 30 to the "false" path. The Windy? binary node performs a similar operation, but this time, it multiplies the incoming value (30) by the probability of a cloudy and windy day (0.65). This results in the conclusion that there is a 19.5% chance that it will be cloudy and windy, and a 10.5% chance of a sunny, calm day.

For the **Condition is TRUE - T Path** value, the expression can include:

- Numerical values
- Standard and special operators
- Predefined mathematical functions (exp, log, sin, etc.)
- RENO internal functions
- RENO resources
- References to any analysis workbooks

You can enter the equation manually, or you can use the function selector or the equation editor to assist you.

RENO resources, analysis workbooks and internal functions are automatically color-coded to assist in reading the equation. When the cursor is positioned on one of these items, a preview area appears and displays a summary of the item. If the item is a resource, you can click the **View** icon in the Preview area to open that resource's properties window.

IMPORTANT: All values and results are assumed to be in terms of the default unit specified for the database. See the "Using Time Units in RENO" section for details on how this may affect your analysis.

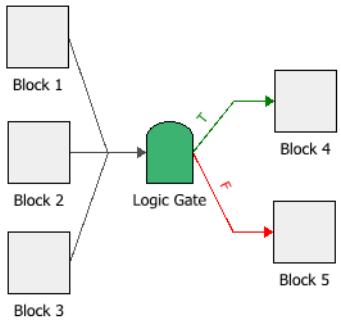
Advanced Configurations

- You can configure the binary node to have only one outgoing path (i.e., only a "true" path). This is a shortcut for having a "false" path terminated by a stop flag.

11.2.6 Logic Gates

A logic gate is similar to a conditional block in that it checks incoming values against a conditional expression, with possible outcomes of true and false, and can have one or two outgoing paths. It differs, however, in that it accepts multiple incoming paths and determines the overall block outcome based on the gate type, which specifies how many of the incoming values must meet the condition in order for the outcome to be true.

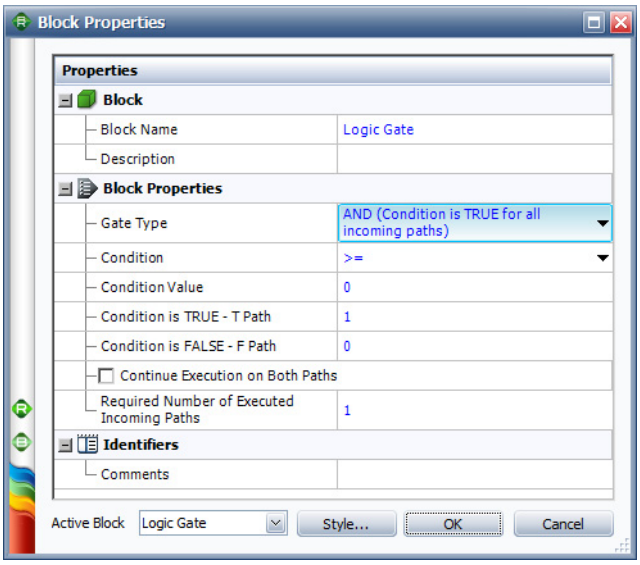
For example, in the configuration shown next, the logic gate evaluates the outputs of Blocks 1, 2 and 3 against a conditional expression, and then evaluates the overall outcome based on the gate type. If the outcome is true, then an output is passed down the "true" (T) path to Block 4, but if the outcome is false, an output is passed down the "false" (F) path to Block 5.



Tip: RENO designates the first connector drawn from a conditional block, binary node or logic gate as the "true" path and the second connector as the "false" path. You can switch these designations by right-clicking the connector and choosing **Set to Failure** or **Set to Success** on the shortcut menu.

The following picture shows the Block Properties window of the logic gate in the example. In this case, the logic gate is an AND gate, meaning that all incoming paths must meet the condition in order for the block outcome to be true. If the outputs of Blocks 1, 2 and 3 are all greater than or equal to 0, then the block outcome is true and the value "1" is

passed down the “true” path. However, if one of the outputs is less than 0, then the block outcome is false and the value 0 is passed down the “false” path.



The logic gate can be configured for any of the following gate types:

- **AND**: If all incoming paths meet the condition, then the block outcome is true; otherwise, it is false.
- **OR**: If at least one incoming path meets the condition, then the block outcome is true; otherwise, it is false.
- **XOR**: If one, and only one, of the incoming paths meets the condition, then the block outcome is true. If more than one incoming path meets the condition, then the block outcome is false. This gate is also known as an *exclusive OR*.
- **K-out-of-N**: If *k out of n* incoming paths (e.g., 2 out of 3 incoming paths) meet the condition, then the block outcome is true; otherwise, it is false. When you select this gate type, the **Required Number of Paths** field will appear, allowing you to specify the value of *k*.

The **Condition** and **Condition Value** fields allow you to configure the conditional expression of the block. The drop-down list in the Condition field includes the following relational signs:

=	equal to
<=	less than or equal to
<	less than
>=	greater than or equal to
>	greater than
FP<=%	a special case where the conditional block itself draws a random number uniformly distributed from 0 to 100, and then evaluates whether the number is less than or equal to the condition value.
<>	not equal to

For the condition value, as well as the outgoing “true” and “false” values, the expression can include:

- Numerical values
- Standard and special operators

- Predefined mathematical functions (exp, log, sin, etc.)
- RENO internal functions
- RENO resources
- References to any analysis workbooks

You can enter the equation manually, or you can use the function selector or the equation editor to assist you.

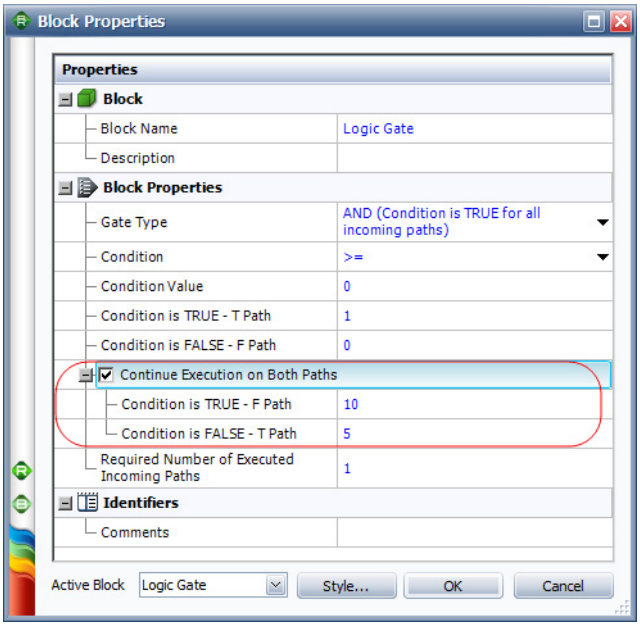
RENO resources, analysis workbooks and internal functions are automatically color-coded to assist in reading the equation. When the cursor is positioned on one of these items, a preview area appears and displays a summary of the item. If the item is a resource, you can click the **View** icon in the Preview area to open that resource's properties window.

IMPORTANT: All values and results are assumed to be in terms of the default unit specified for the database. See the "Using Time Units in RENO" section for details on how this may affect your analysis.

Advanced Configurations

- By default, if the condition evaluates to true, then the specified value will be passed down the "true" path and nothing will be passed down the "false" path. Similarly, if the condition evaluates to false, a value will be passed down the "false" path only.

  If you wish to pass a value down both paths regardless of the outcome of the evaluation, select the **Continue Execution on Both Paths** check box. This allows you to specify the values to pass to the opposite paths in each case. For example, in the picture shown next, if the outcome is true, then the value "1" will be passed down the "true" path and the value 10 will be passed down the "false" path. Similarly, if the outcome is false, then the value 0 is will be passed down the "false" path and the value 5 will be passed down the "true" path.



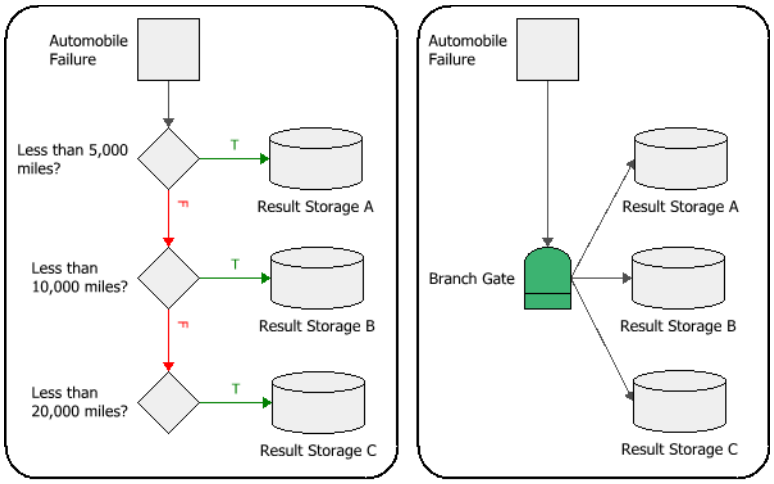
- If your block has multiple incoming paths, you can configure the block to not pass an output or compute a result until a certain number of those incoming paths have been executed. This allows you to control the number of inputs that the block processes at any given time. To do this, enter the required number of inputs in the **Required**

Number of Executed Incoming Paths field in the Block Properties window. Starting in version 9.0.10, an asterisk (*) in this field means that all incoming paths are required. See the "Flowchart Configurations and Execution" section for an example.

11.2.7 Branch Gates

A branch gate functions as a "switch" statement. It checks the incoming value against a number of cases and passes different outputs depending on which case or branch evaluates to true. It has one incoming path and multiple outgoing paths.

A branch gate may be used in place of a series of conditional blocks. For example, the flowchart on the left, as shown next, has three conditional blocks that evaluate an automobile's mileage. If the first condition is met, the value is passed to the "true" (T) path to "Result Storage A," but if the condition is not met, the value is passed down the "false" (F) path to the next conditional block for evaluation. The evaluation continues until all paths end. The flowchart on the right shows an example of how a branch gate may be used to replace the conditional blocks.

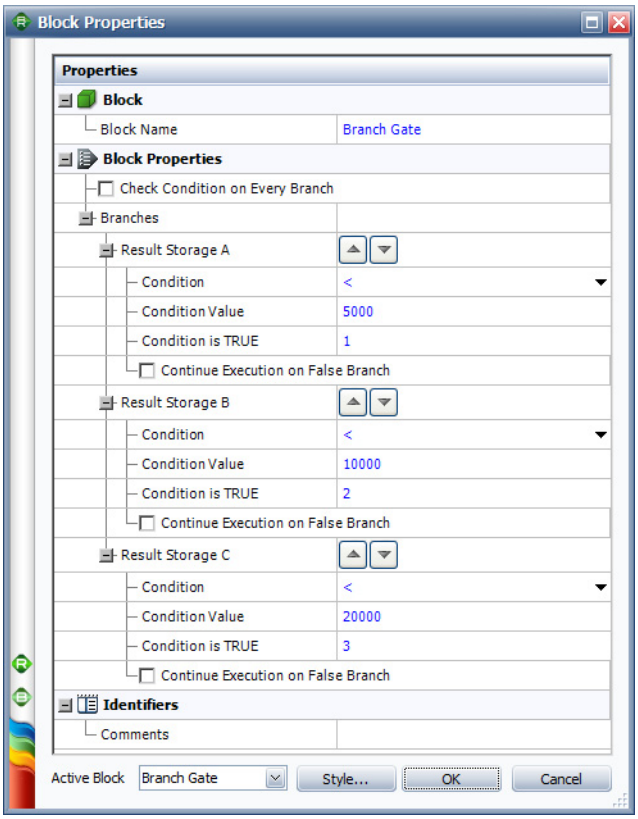


A chain of conditional blocks

A branch gate replaces the conditional blocks

The following picture shows the Block Properties window of the branch gate in the example. The branch gate checks the incoming value against each branch sequentially, based on the order shown in the Block Properties window (you

can move a block up or down in priority by clicking the arrows that appear next to the block's name). The branch gate then passes only the output of the first branch that evaluates to true.



The **Condition** and **Condition Value** fields allow you to configure the conditional expression of the block. The drop-down list in the Condition field includes the following relational signs:

=	equal to
<=	less than or equal to
<	less than
>=	greater than or equal to
>	greater than
FP<=%	a special case where the conditional block itself draws a random number uniformly distributed from 0 to 100, and then evaluates whether the number is less than or equal to the condition value.
<>	not equal to

For the condition value, as well as the outgoing values, the expression can include:

- Numerical values
- Standard and special operators
- Predefined mathematical functions (exp, log, sin, etc.)

- RENO internal functions
- RENO resources
- References to any analysis workbooks

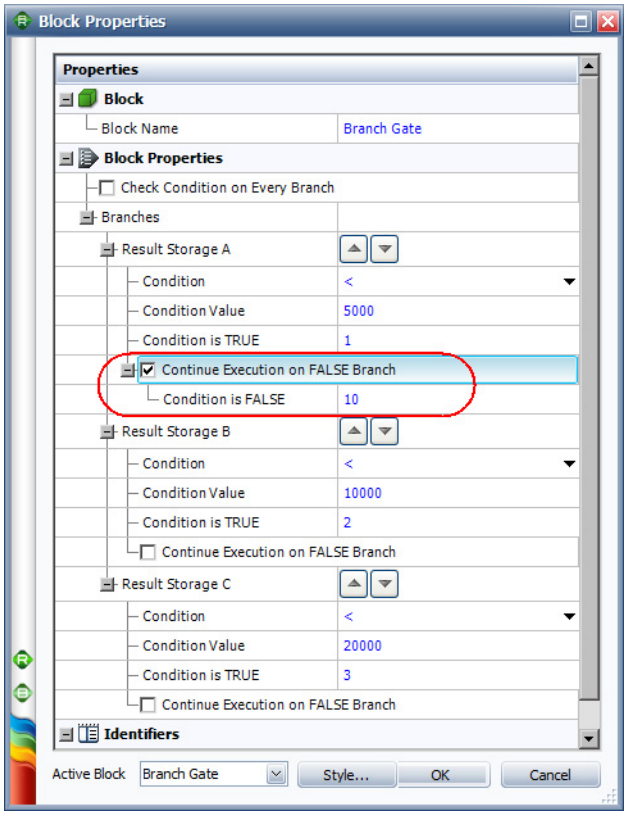
You can enter the equation manually, or you can use the function selector or the equation editor to assist you.

RENO resources, analysis workbooks and internal functions are automatically color-coded to assist in reading the equation. When the cursor is positioned on one of these items, a preview area appears and displays a summary of the item. If the item is a resource, you can click the **View** icon in the Preview area to open that resource's properties window.

IMPORTANT: All values and results are assumed to be in terms of the default unit specified for the database. See the "Using Time Units in RENO" section for details on how this may affect your analysis.

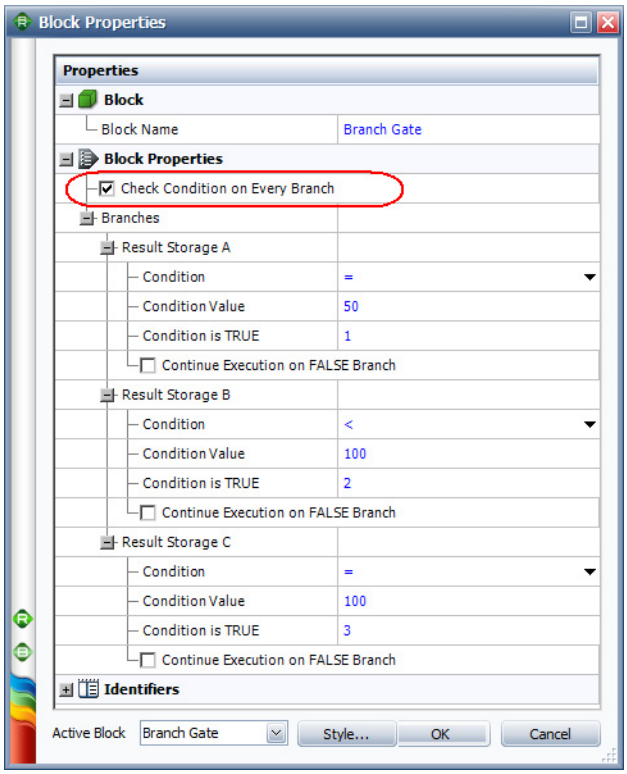
Advanced Configurations

- If you wish to have the branch gate pass an output when a branch evaluates to false, select the **Continue Execution on FALSE Branch** check box and enter a value in the **Condition is FALSE** field that appears, as shown next. In this example, if the incoming value is 6,000, then the branch gate will pass two outputs: the value 10 to the "Result Storage A" branch, and the value 2 to the "Result Storage B" branch.



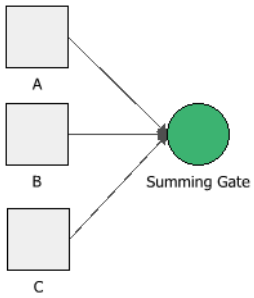
- By default, the branch gate passes only the output of the first branch that evaluates to true. You can configure the branch gate to pass the output of every branch that evaluates to true by selecting the **Check Condition on Every Branch** check box. With this setting, the order of the branches does not affect the output of the branch gate. In the

following example, if the incoming value is 50, then the branch gate will pass two outputs: the value 1 to the "Result Storage A" branch and the value 2 to the "Result Storage B" branch.



11.2.8 Summing Gates

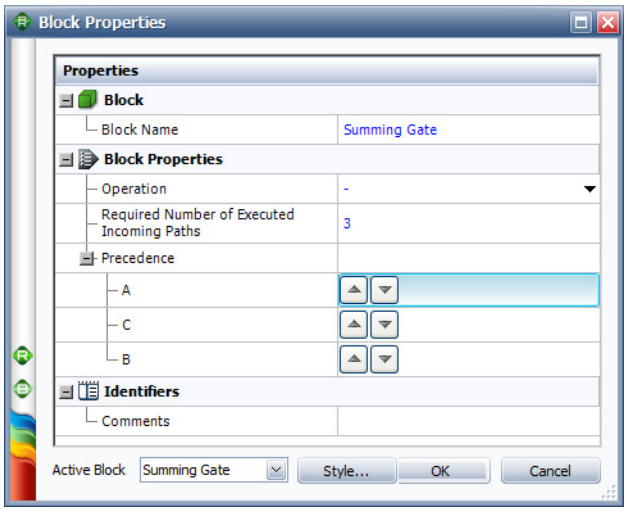
A summing gate is a block that performs a simple mathematical operation on the incoming values and returns a single value to all outgoing paths. These operations include addition, subtraction, multiplication or division. Starting in version 9.0.11, they also include averaging incoming values and finding the maximum or minimum of the incoming values. For example, the following configuration shows three blocks that pass their output values to a summing gate.



If you select to perform addition or multiplication, the values can be passed to the summing gate in any order without affecting the result (i.e., the gate follows the associative and commutative properties of addition and multiplication). The order of incoming values is also not relevant when averaging the values or finding the maximum or minimum value.

If you select to perform subtraction or division, you can define the appropriate order of operation by using the **Precedence** area of the gate's Block Properties window. For example, the following picture shows that the order of subtraction will be: (A - C) - B. To move a block up or down in priority, click the arrows that appear next to the block's name.

The **Required Number of Executed Incoming Paths** field allows you to control the number of inputs that the summing gate processes at any given time. See the "Flowchart Configurations and Execution" section for an example on how this block setting affects the results of a flowchart.



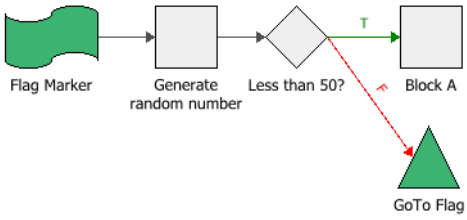
11.2.9 Flag Marker and Go-to-Flag Blocks

Flag markers and go-to-flags are blocks that are used to build subroutines and loops that repeat steps in a flowchart until a certain condition is reached. The flag marker marks a spot in the path of the flowchart, and the go-to-flag transfers the flow of execution to that spot. A flag maker can have any number of incoming and outgoing paths. A go-to-flag can have any number of incoming paths, and no outgoing paths. The flow of execution will always go from the go-to-flag to the designated flag marker.

In the case of multiple incoming paths, the diagram and/or the conditions set in prior blocks will dictate when the transfer happens (i.e., the number of paths that must reach the go-to-flag in order to initiate the transfer of flow). In other words, if multiple incoming paths are present, the behavior will depend on the connections and settings prior to the flag. You should use the Simulation Debugger to review the execution sequence for your diagram in order to assure that the transfer happens as intended. *See Section 11.7 on page 396.*

Loops

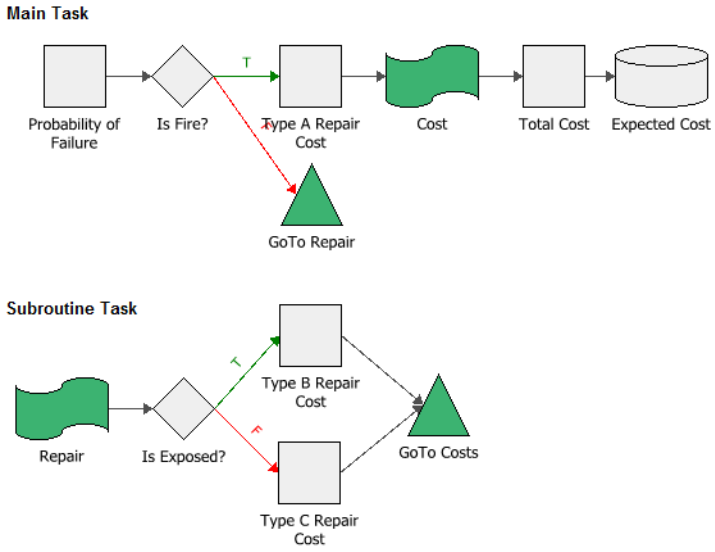
The following flowchart shows a simple example of how a flag marker and go-to-flag pair can form a loop. In this example, if the number generated by the standard block is higher than 50, the go-to-flag block transfers the flow of execution to the flag marker. The loop will continue to repeat until the task produces a value less than 50.



IMPORTANT: Avoid configurations that would create infinite loops. It may cause RENO to be unresponsive or create unresolved issues. You can test your flowchart for any errors before running a simulation by using the Debug tool.

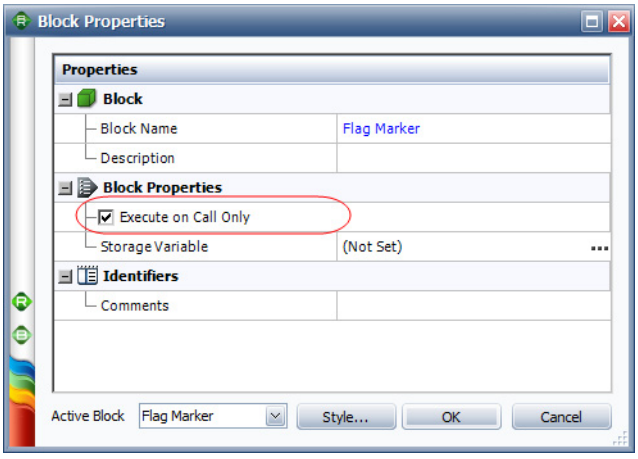
Subroutines

Subroutines are specific tasks that are separate from the main task of the flowchart. The following configuration, for example, shows a flowchart with a main task and a subroutine task. In this case, the main task is to compute for the total cost of repairs. The subroutine task is executed only when the product failure does not cause a fire, in which case, the subroutine computes for either type B or C costs, and then passes that value back to the main task so that it can be counted as part of the total cost. Notice that due to the location of the flag marker called "Costs," the main task skips the computation of the type A repair whenever the subroutine is executed.



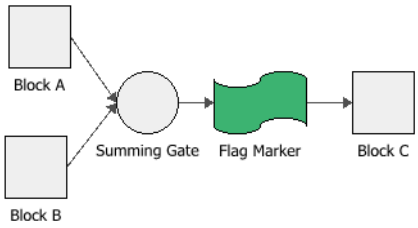
To build a subroutine in RENO, follow these two steps:

1. Create the subroutine task in the same diagram as the main flowchart. Use a flag marker as the starting block for the subroutine, as shown in the example above.
2. Double-click the flag marker of the subroutine task to open its Block Properties window, and then select the **Execute on Call Only** check box, as shown next. This setting allows the subroutine to be executed only when it is called by a go-to-flag block. Note that this check box will appear only when the flag marker is used as a starting block. If you do not select this setting, the subroutine task may be executed before it is called by the main task (see also "Flowchart Configurations and Execution" for a discussion on how flowcharts are executed).

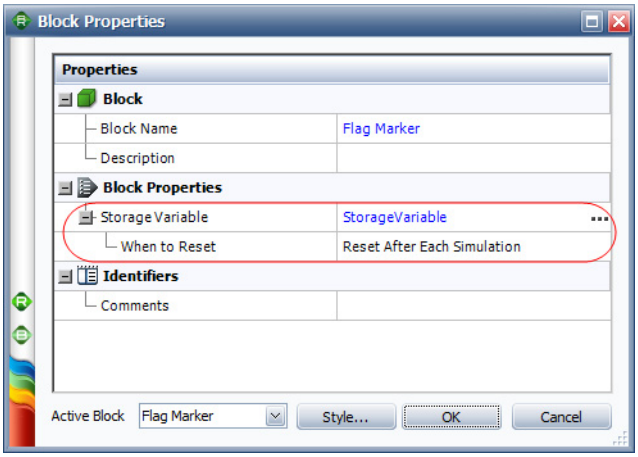


Advanced Configurations

- Aside from marking a spot in the path of a flowchart, a flag marker may be used to store the value passed to it during simulation into a variable. For example, in the configuration shown next, the summing gate adds the outputs of Blocks A and B, but it cannot store the resulting value; it only passes the result to the next block in the flowchart. In this case, the flag marker can store the value from the summing gate into a storage variable, as well as pass that same value to Block C.

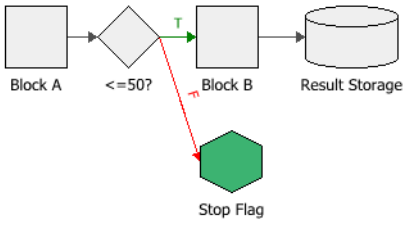


The following picture shows the Block Properties window of the flag marker in the example.



11.2.10 Stop Flags

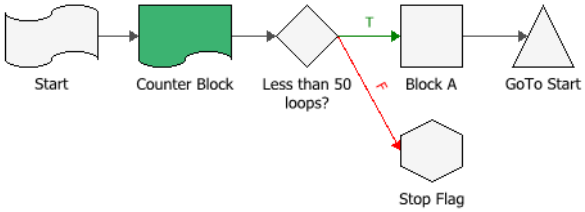
A stop flag halts the simulation process when it is encountered. No further simulations, runs or analyses will be performed until you next simulate the flowchart. A stop flag can have any number of incoming paths, and no outgoing paths. The following example shows how a stop flag may be used to stop the simulation from progressing when a certain condition is met.



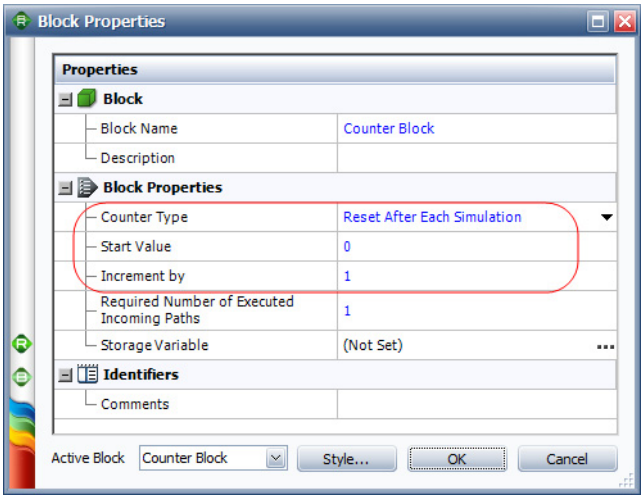
11.2.11 Counter Blocks

A counter block increments its value by a specified amount each time the simulation passes through it, and then passes the recorded value to the next block in the flowchart. It may have multiple incoming and outgoing paths. The

following example shows a counter block that records the number of loops that it goes through. The counter block passes its tally to the conditional block that stops the loop when the condition is met.



The following picture shows the Block Properties window of the counter block in the example. In this case, the **Start Value** field indicates that the count starts at 0, and the **Increment By** field indicates that the start value increments by 1 every time the simulation passes through the block.



The **Counter Type** field allows you to configure how the block will keep count. There are three options:

- **Reset After Each Simulation**. A simulation is a single pass through the flowchart. However, if the flowchart contains one or more loops, and the counter block is within a loop, the simulation may pass through the block multiple times. In this case, the block will count the number of loops repeated, and then reset once the simulation has reached the end of the flowchart.
- **Reset After Each Run**. A run is a set of simulations. The counter block will reset after the specified number of simulations have been completed.
- **Reset After Analysis**. An analysis is a set of runs. The counter block will reset after each set of runs has been completed.

In addition to passing the recorded value to the next block in the flowchart, the counter block may also be configured to store its output in a variable. The **Storage Variable** field displays the name of the storage variable, if any.

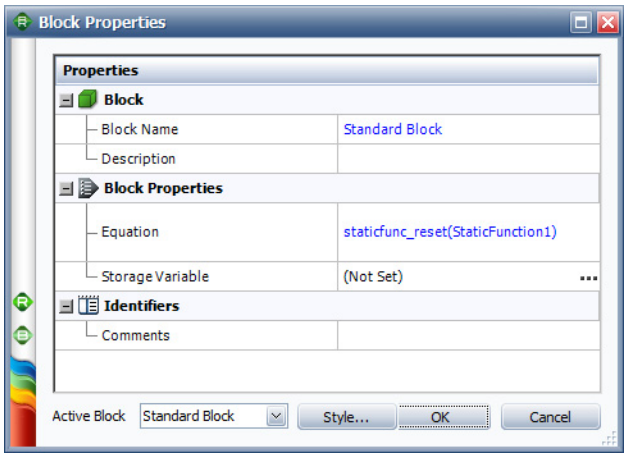
Advanced Configurations

- If your block has multiple incoming paths, you can configure the block to not pass an output or compute a result until a certain number of those incoming paths have been executed. This allows you to control the number of inputs that the block processes at any given time. To do this, enter the required number of inputs in the **Required Number of Executed Incoming Paths** field in the Block Properties window. Starting in version 9.0.10, an asterisk (*) in this field means that all incoming paths are required. See the "Flowchart Configurations and Execution" section for an example.

11.2.12 Reset Blocks

A reset block forces all RENO static functions in the flowchart to generate new values during simulation. Any other resources that are dependent on these values are also recalculated.

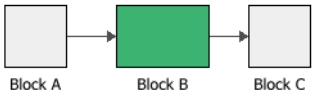
You can also recompute only the static function(s) that you specify by using a standard block and entering the following RENO internal function: “staticfunc_reset(NAME),” where NAME is the name of the static function, as shown in the example below.



A reset block can have one incoming path and multiple outgoing paths.

11.2.13 UI Blocks

A user interface (UI) block pauses flowchart simulation until the user provides some form of input, then passes its value to the next block(s) in the flowchart. For example, consider the following configuration.

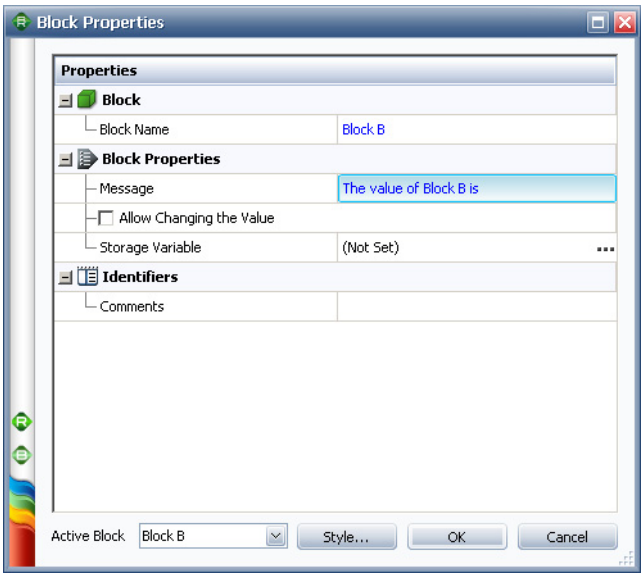


Let's assume that Block A passes a value of 3 to Block B, the UI block. At this point, the simulation will be paused and a message will be displayed to the user. Depending on how the UI block is configured, the message can do one of two things:

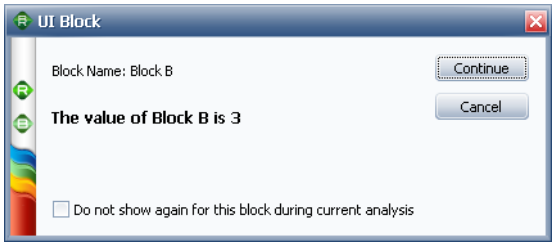
- Display the UI block's value (3).
- Give the user the opportunity to enter a value for the UI block.

In both cases, the user can then either continue simulation or cancel simulation. Additionally, the user can specify that no further messages will be shown for the UI block during the current analysis. In the following picture, the UI block

is configured to display a message showing its value. The **Message** field contains the wording that will be displayed in the message.

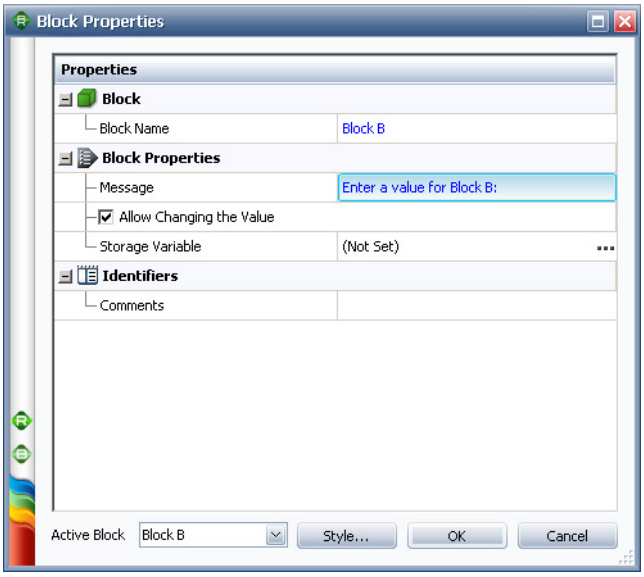


The message is shown next.

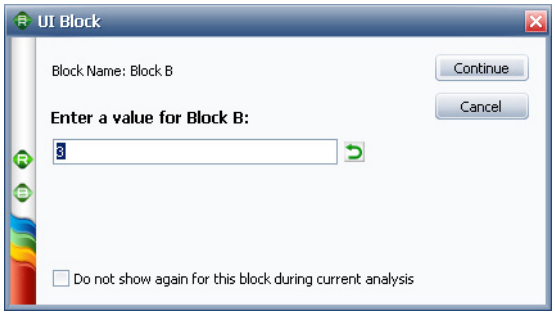


In this case, if the user continues the simulation, the value of 3 will be passed to Block C.

In the following picture, the UI block is configured to allow the user to enter a value for the UI block. The **Message** field contains the wording that will be displayed in the message. The **Allow Changing the Value** check box is selected, which adds an input field to the message.



The message is shown next.



In this case, the user can keep the original value (3) or enter a new value for the UI block. The Reset icon resets the input field to its original value.

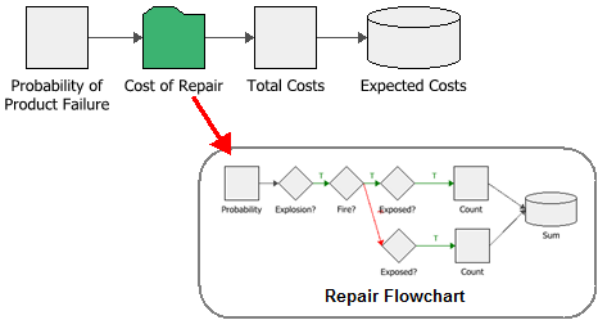
If the user continues the simulation, the value in the input field will be passed to Block C.

UI blocks may accept up to one incoming path (input value), and may have multiple outgoing paths (i.e., the same output value is passed to more than one block). In addition to passing the output to the next block(s) in the flowchart, the UI block may be configured to store its output in a RENO variable. The **Storage Variable** field in the Block Properties window of the block displays the name of the variable, if any.

11.2.14 Subchart Blocks

Subchart blocks represent other flowcharts within the project. Subchart blocks are generally used to break down a larger flowchart into simpler steps to reduce complexity and improve traceability.

For example, in the flowchart shown next, the subchart block called “Cost of Repair” represents another flowchart in the project called “Repair Flowchart” (as shown in the inset). The subchart block processes the incoming value by using a copy of the flowchart that it represents, and then passes the output to the next block in the path.



When you add a subchart block, you will be required to select the flowchart that the block represents or calls. Circular references are not permitted (i.e., a subchart block cannot call the flowchart in which it is placed), and any flowchart called by a subchart block must have a single starting block and a single end block.

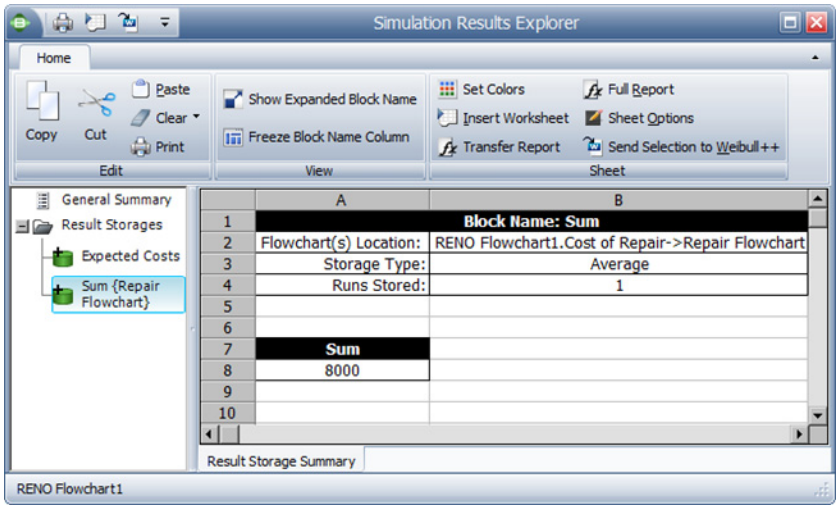
During simulation, subchart blocks use a copy of the flowchart they represent, and not the actual flowchart that was created. This means that:

- You do not need to run a simulation of the actual flowchart before using it in a subchart block.
- You can create configurations where several subchart blocks call the same flowchart. Each copy of the flowchart will return different outputs based on the variables passed to it during simulation.

- If a subchart block calls a flowchart that has a result storage block, then the output value of that result storage block is also reported in the Simulation Results Explorer. For example, the flowchart shown above has a result storage block called “Expected Costs.” In addition, the “Repair Flowchart” also has a result storage block called “Sum.” The values stored in both results storage blocks are reported in the Simulation Results Explorer, as shown in the navigation panel in the next picture.

  As you can see, for any result storage block not located in the main flowchart:

  - The navigation panel displays the name of the block and the name of the flowchart (in curly brackets) in which the block is used.
  - The block's worksheet includes a **Flowchart(s) Location** field, which displays the name of the main flowchart, the name of the subchart block and the name of the flowchart in which the block is used.



Note: Double-clicking a subchart block will open the flowchart that the block is based on. Open the Block Properties window for a subchart block by selecting the block and choosing the command on the Ribbon or pressing **CTRL+E**.

11.3 RENO Resources

RENO resources are objects that serve to store or generate values used in flowcharts. They cannot, however, be placed directly into flowcharts; instead, they are referenced or *called* by the blocks within the flowcharts. Any RENO resource is available to be called by any flowchart within the project.

Like all other types of Synthesis resources, RENO resources can be created and edited via the Resource Manager. In addition, when the resource is updated with new information, the change is reflected automatically in all flowcharts in which it is used. There are several types of RENO resources:

- Variables and Models
- RENO functions and static functions
- RENO simulation definitions
- RENO tables

Note that variables and models may be used by other Synthesis applications. The remaining types are exclusive to RENO.

Each type of RENO resource has an identifier, or name. All identifiers must meet the following requirements.

Note: The following requirements apply to models only if you have selected the Enforce RENO Compliance check box in the Synthesis Setup window.

- Must be unique within the project.
- Must be text; cannot be a number or anything that evaluates to a number, such as Pi or RENO internal functions.
- Cannot start with a number.
- Cannot contain spaces or any of the following characters: \ / : * ? " < > | ‘ ~ ! @ # $ % ^ & ( ) ;

While the values of some resources (variables and tables) may change during the simulation process, once simulation has been completed, each resource contains only the original value(s) that you entered in its properties window (unless it is a variable set to never reset). If you wish to keep the value(s) of the resource used during simulation, you must use a result storage block within the flowchart to store the value(s). This means that the underlying value of the resource is unchanged by simulation, and therefore simulation of one flowchart will not affect other flowcharts that use the resource.

What's Changed? In previous versions of the software, RENO resources were known as definitions.

11.3.1 Variables

A variable is a resource that stores a numerical value and allows you to assign a name to that value. You can then use the variable name in place of the actual value in the equations that you create. You can also use variables in functions and/or as conditional output values in conditional blocks, logic gates and branch gates.

A variable starts with an assigned initial value, which may be fixed or varied during simulation. You can use variables:

- In RENO, to act as a constant (i.e., maintain a fixed numerical value during simulation).
- In RENO, to temporarily store the output value of a block during simulation. *See the next section for more information.*
- In RENO, to perform a sensitivity analysis, which involves varying the value of one or two variables between *runs* (i.e., sets of simulations). For example, if you are analyzing potential investment strategies, you may wish to vary the number of years that you will be investing and/or the percentage of your income that will be invested to see how different inputs will affect the final results.
- As the basis for a dynamic model (*see page 172 in Chapter 5*), which represents a fixed probability, duration or cost. The variable can be programmatically varied during simulation, thereby changing the value of the model for successive simulation runs.
- As inputs for certain fields in BlockSim simulation RBDs via simulation worksheets, allowing programmatic variation of the field value during simulation.

What's Changed? In previous versions of RENO, variables that stored fixed values were known as constants, and variables that stored outputs were known as storage variables.

11.3.1.1 Using Variables to Store Output Values

RENO Only

You can use variables to hold numerical values passed to them during simulation. In RENO, the following blocks are able to store their output values in variables:

- Standard blocks
- Result storage blocks
- Flag markers
- Counter blocks

Depending on your choice in the **When to Reset** field of the variable's properties window, you can configure the variable to retain the value passed to it during simulation or reset the variable to its initial value at specific times. The options are:

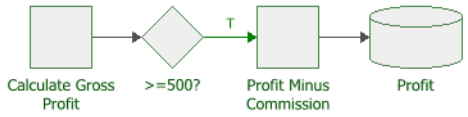
- **Reset After Each Simulation** (i.e., reset after a single pass through the flowchart)
- **Reset After Each Run** (i.e., reset after each set of simulations)
- **Reset After Analysis** (i.e., reset after each set of runs)
- **Never Reset**

When the simulations end, the variable always returns to its initial starting value unless you have selected **Never Reset**. If you wish to keep the value used during simulation, you must use a result storage block to store it.

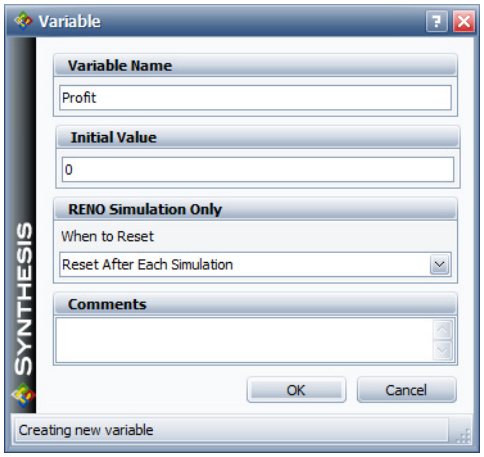
Example

The following example demonstrates two things: a) how a variable may be used to store an output value and b) how a variable may be used in an equation.

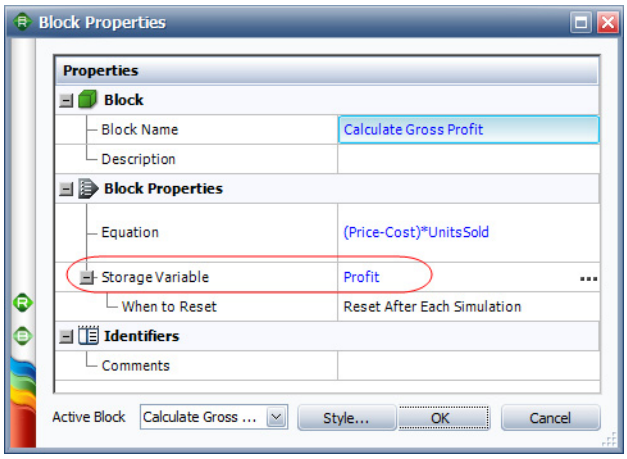
In the following configuration, the first block calculates the gross profit from a sale, and then stores its output into a variable called “Profit.” The conditional block determines whether the gross profit exceeds 500. If the output is true, the third block deducts the sale's commission from the profit, and overwrites the stored value with the new output.



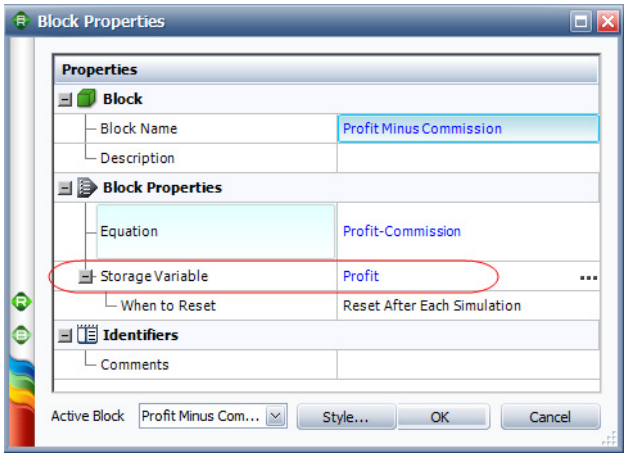
The following picture shows the properties window of the variable called “Profit.” The initial value of this variable is set to 0.



The following picture shows the Block Properties window of the first block in the example. The output of the equation is stored in the variable.



The following picture shows the Block Properties window of the third block in the flowchart. The equation shows that the commission is deducted from the existing value in the variable, and then the variable is overwritten with the new result. At the end of the simulation, the result storage block in the flowchart stores the final value of the variable, and then the variable resets to 0.



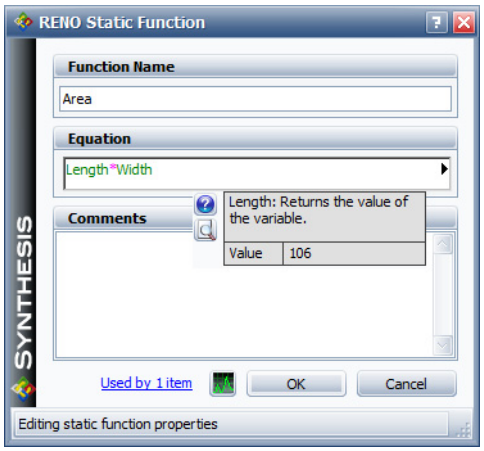
11.3.2 RENO Functions and Static Functions

RENO functions and static functions store equations. The key difference is that a RENO function is always recomputed whenever it is encountered during simulation, and it allows you to pass input variables (also known as *parameters* or *arguments*) to the equation while simulation is in progress. A static function, on the other hand, is computed only once, before the simulation begins, and its value will remain constant while the simulation is in progress. It will be recomputed only after the current simulation ends and before the next simulation begins (unless manually forced to recompute).

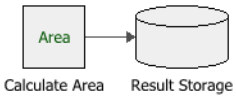
As a rule of thumb, use a RENO static function if you need the *final* value of the equation before the simulation even begins, and use a RENO function if you expect the equation to *find* the values of its parameters during the simulation. The following example illustrates these concepts.

What's Changed? In previous versions of the software, RENO static functions were known as equation variables.

Suppose you wish to calculate the area of an object: Area = Length * Width, where Length = 106 and Width = 112. To use a static function to calculate the area, you can define the numerical values directly within the equation or use other RENO resources (e.g., variables, models, etc.) to represent those values, as shown in the following example.



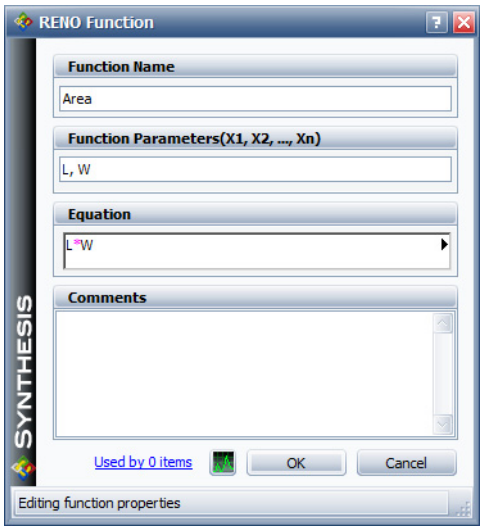
Once you have defined the static function, you can use it in a flowchart by referencing its name, as shown in the following example.



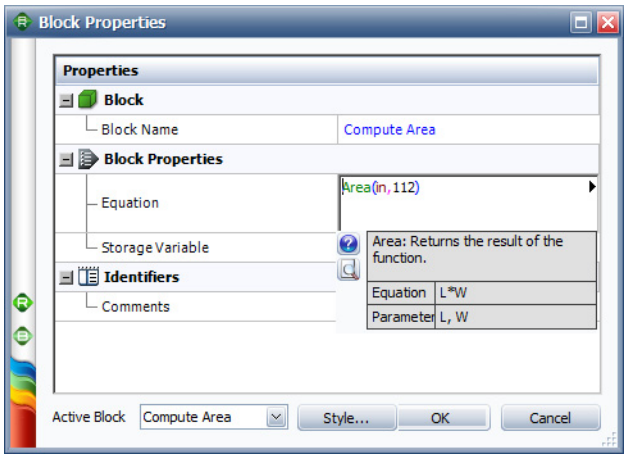
Tip: You can force all static functions in a flowchart to recompute during simulation by using a reset block. You can also recompute only the static function(s) that you specify by using the RENO internal function called "staticfunc_reset."

Now suppose that Length is a random number from 100 to 150, and Width = 112. In this case, you first need to obtain the length of the object before you can compute its area, but the value for length can be obtained only during simulation. A RENO function would therefore be more appropriate for handling this problem.

There are two ways to use RENO functions: you can use other RENO resources to supply the values for the equation (similar to the example shown above for the RENO static function), or you can choose to explicitly define the parameters of the equation by using the optional Function Parameters field, as shown in the example below.



When you explicitly define the parameters, you will be required to define their inputs whenever you use the equation in a flowchart. You can do this by referencing the function name as shown in the example below (where "in" represents the output of the block that obtains the length, and width is set to a value of 112). The Preview area shows the order in which the inputs must be entered.



The following picture shows the modified flowchart:

The expressions in RENO functions and static functions can include:

- Numerical values
- Standard and special operators
- Predefined mathematical functions (exp, log, sin, etc.)
- RENO internal functions
- RENO resources
- References to any analysis workbooks

You can enter the equation manually, or you can use the function selector or the equation editor to assist you.

RENO resources, analysis workbooks and internal functions are automatically color-coded to assist in reading the equation. When the cursor is positioned on one of these items, a preview area appears and displays a summary of the item. If the item is a resource, you can click the **View** icon in the Preview area to open that resource's properties window.

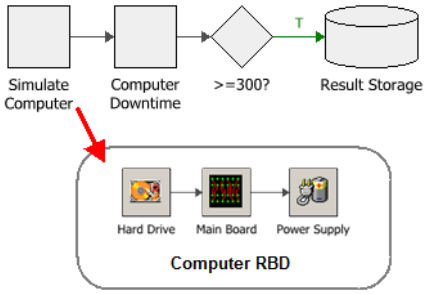
IMPORTANT: All values and results are assumed to be in terms of the default unit specified for the database. See the "Using Time Units in RENO" section for details on how this may affect your analysis.

11.3.3 RENO Simulation Definitions

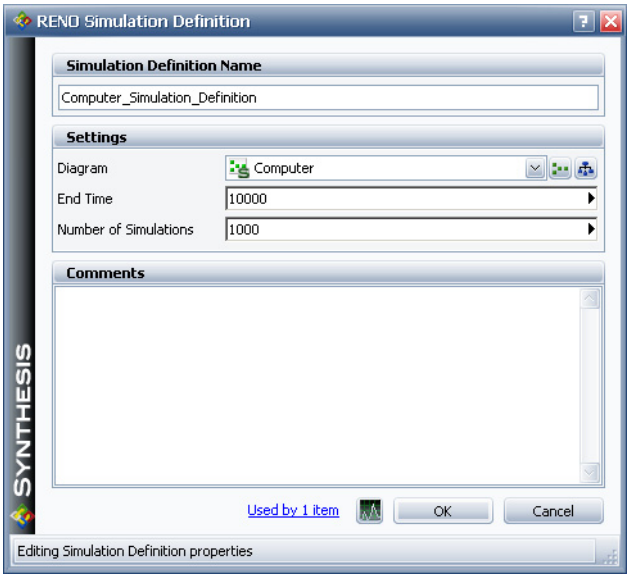
A RENO simulation definition enables you to trigger simulation of a BlockSim diagram from within a RENO flowchart by using certain RENO internal functions, so that you can use one or more results from the simulation in the flowchart.

Note: You can access BlockSim analytical diagrams for use in RENO flowcharts by using the analytical diagram's published model. *See page 174 in Chapter 5.*

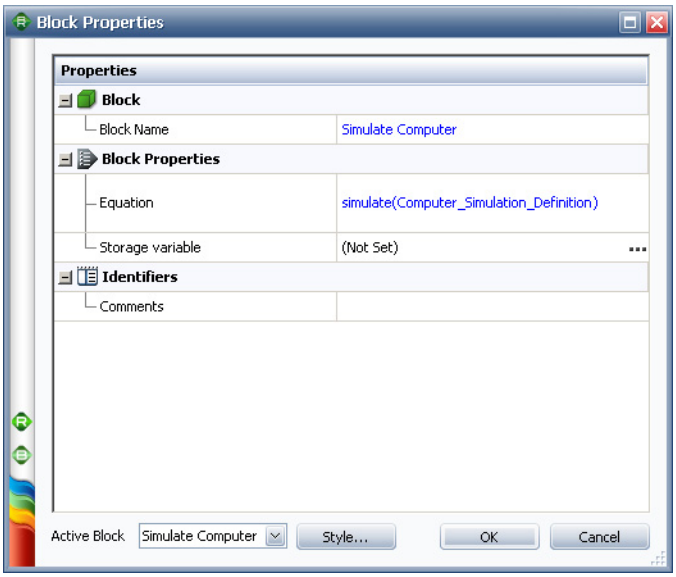
The following example shows a scenario where an RBD of a computer is simulated by the block called "Simulate Computer." The resulting system downtime is retrieved by the block called "Computer Downtime" and evaluated by the conditional block. If the downtime is greater than or equal to 300 hours over the 10,000 hour simulation period, then the computer is deemed to have insufficient availability.



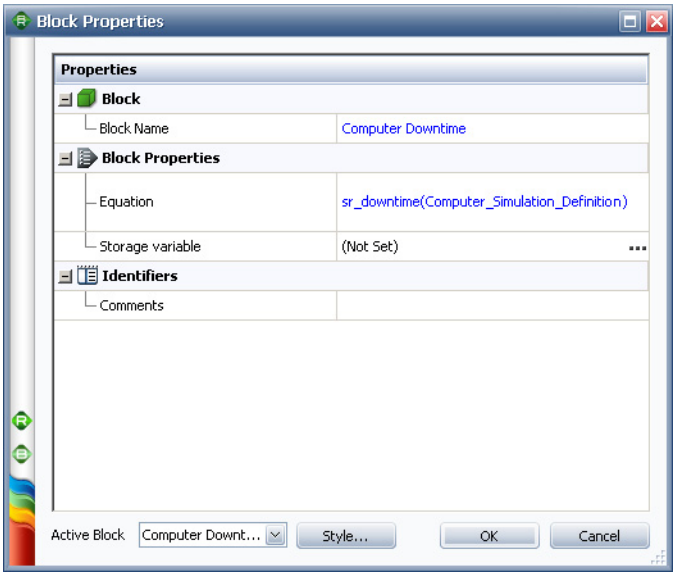
The following picture shows the simulation definition that is used to simulate the "Computer" RBD. This performs a run of 1,000 simulations of the RBD called "Computer," each simulation for a period of 10,000 hours.



The “Simulate Computer” block uses this simulation definition as the parameter for the RENO internal function called “simulate,” as shown next.

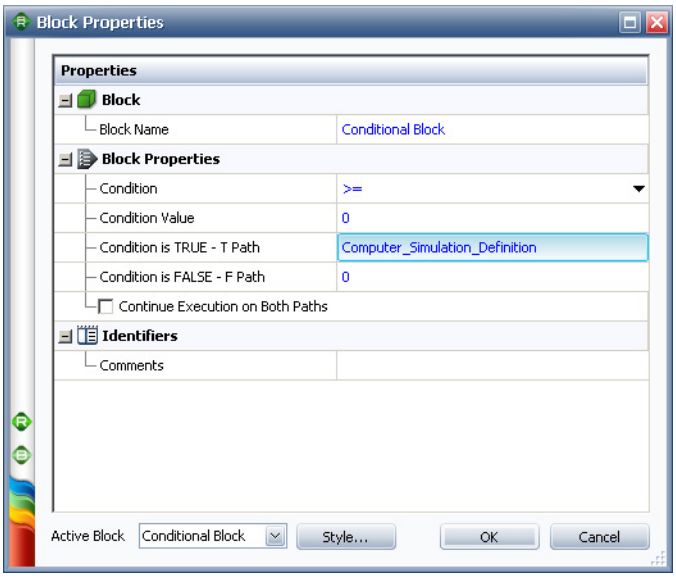


The “Computer Downtime” block can then retrieve the downtime for the “Computer” RBD by using the same simulation definition as the parameter for the RENO internal function called “sr_downtime,” as shown next.



The value retrieved is then passed to the conditional block for evaluation.

Another way to use a simulation definition is demonstrated next, using the same simulation definition as in the example above.



This conditional block checks the incoming value to determine if it is greater than 0. If it is, then the "Computer" RBD is simulated 1,000 times, each for a period of 10,000 hours, as specified by the simulation definition. The resulting mean availability value for the system is then passed along the outgoing "true" path.

The BlockSim diagram called by the simulation definition can be either a simulation RBD or a simulation fault tree. If the desired diagram does not exist, you can use the icons beside the Diagram field to create it.

For the **End Time** and **Number of Simulations** fields, the expression can include:

- Numerical values
- Standard and special operators
- Predefined mathematical functions (exp, log, sin, etc.)
- RENO internal functions
- RENO resources
- References to any analysis workbooks

You can enter the equation manually, or you can use the function selector or the equation editor to assist you.

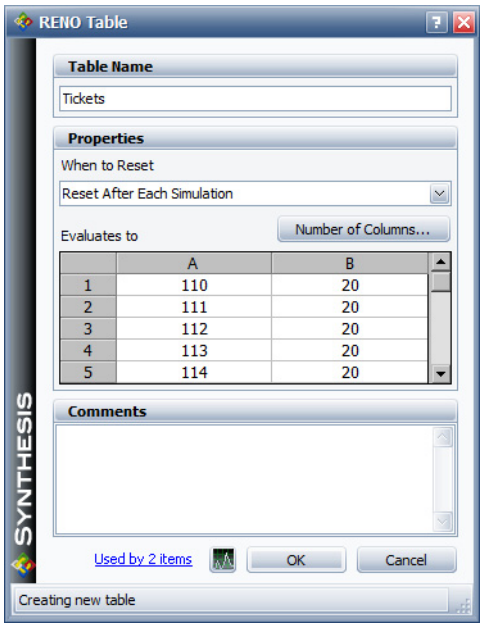
RENO resources, analysis workbooks and internal functions are automatically color-coded to assist in reading the equation. When the cursor is positioned on one of these items, a preview area appears and displays a summary of the item. If the item is a resource, you can click the **View** icon in the Preview area to open that resource's properties window.

IMPORTANT: All values and results are assumed to be in terms of the default unit specified for the database. See the "Using Time Units in RENO" section for details on how this may affect your analysis.

Note that the seed used in the simulation is taken from the current RENO simulation and a single thread is always used.

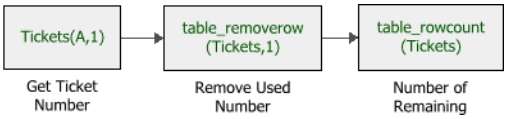
11.3.4 RENO Tables

A RENO table stores an array of values in rows and columns. This table can have up to 65,536 rows and 256 columns. The following example shows the properties of a RENO table called "Tickets." A reference to a RENO table requires inputs for the column number or name and the row number. For example, both Tickets(1,1) and Tickets(A,1) are references to the cell located in the first column and first row of the table.



IMPORTANT: You can change the name of a column by double-clicking its heading; however, making this change can break existing references to table cells. For this reason, it is preferred to use column numbers in references. Additionally, it is recommended that you edit the column names only upon creating the table.

RENO tables store only numerical values, and the values can be used or manipulated during simulation by using certain RENO internal functions. The following configuration, for example, shows how to use two particular internal functions. In this example, the first block evaluates to a table reference, which draws the value stored in the specified cell. The second block evaluates the "table_removerow" function, which deletes the specified row from the specified table. The third block evaluates the "table_rowcount" function, which returns the number of rows that contain data in the specified table.



The values in the RENO table can be retained during simulation or reset to their original values at specific times, depending on your choice in the **When to Reset** field of the table's Block Properties window. The options are:

- **Reset After Each Simulation** (default setting)
- **Reset After Each Run** (i.e., reset after each set of simulations)
- **Reset After Analysis** (i.e., reset after each set of runs)

When the simulations end, the table always returns to the original values. If you wish to keep the values stored in the table during simulation, you must use a result storage block to store them.

11.4 Function Selector and Equation Editor

The function selector and the equation editor can be used to create string expressions (i.e., equations) to be evaluated in flowchart standard blocks, RENO functions and static functions, or to specify conditional inputs or outputs for conditional blocks, binary nodes, logic gates and branch gates.

To access the function selector or the equation editor, you can:

- Right-click inside the equation, input or output field and choose **Show [Function Selector/Equation Editor]**.
- Press **CTRL+SPACE** (function selector) or **SHIFT+SPACE** (equation editor).
- Click the arrow that appears when you click inside the equation, input or output field and choose **Show [Function Selector/Equation Editor]** from the shortcut menu that appears.

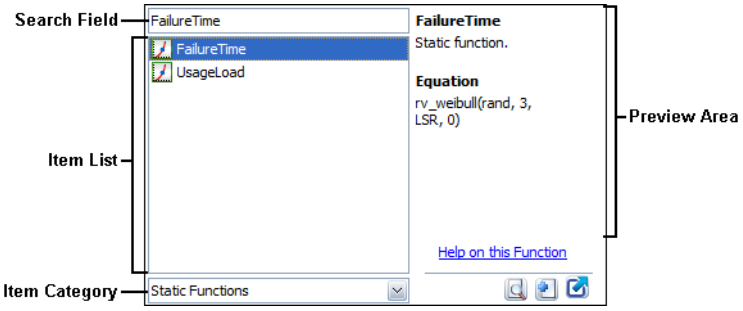
11.4.1 Function Selector

You can use the function selector to quickly add any of the following to the equation:

- Variables
- RENO functions and static functions
- RENO simulation definitions
- RENO tables
- Analysis workbooks
- Models
- RENO internal functions
- RENO predefined functions

IMPORTANT: All values and results are assumed to be in terms of the default unit specified for the database. See Section 11.1 on page 354 for details on how this may affect your analysis.

In the picture shown next, the function selector is being used to display a list of RENO static functions that have been created for use in the project.



Choose the item type from the **Item Category** drop-down list at the bottom of the function selector (or choose **All** if you want to see all items available for inclusion). The **Item List** will display the items of that type that are currently available in the project. You can search within the item list by typing the search term in the field above the item list.

When you select an item, basic information about it will be displayed in the **Preview** area on the right side of the function selector. You can double-click the item to insert it into the equation and close the function selector. If the selected item requires parameter inputs, you may be prompted to provide those inputs depending on whether the **Add parameters when inserting functions using function selector** check box is selected on the Display page of the Application Setup (**File > Application Setup**). If you are not prompted, you will need to enter the parameters manually.

While using the function selector, you can also:

- View or edit the properties of a resource by selecting the resource and clicking the **View/Edit Resource** icon .
- Add a new resource by clicking the Add New Resource icon . If a resource type is currently selected in the item category list, the new resource will be of that type. Otherwise, you can choose the resource type from the shortcut menu that appears.
- Open the equation editor by clicking the Open Equation Editor icon .

11.4.2 Equation Editor

The equation editor functions similarly to the function selector, but it has an equation area on the left side of the window that displays the equation that you are building. You can add to the equation by typing directly in this area, or by double-clicking items in the list on the right.

To view or edit the properties of a resource, select the resource and choose **Edit Resource**.

To add a new resource, choose the relevant **Add New** command. If a resource type is currently selected in the item category list, you can create a resource of that type only. Otherwise, all **Add New** commands will be available. New resources will appear in the item list.

When you are finished building the equation in the equation area, click **OK** to insert the equation into the block or resource properties.

11.5 RENO Standard and Special Operators

Starting in version 9.0.8, the following operators can be used in expressions in RENO blocks and RENO resources. The standard operators are available for use in other locations, such as reports; special operators are reserved for use only in RENO blocks and RENO resources.

Standard Arithmetic Operator	Meaning (Example)
+ (plus sign)	Addition (3+3=6)
- (minus sign)	Subtraction (3-1=2)
	Negation (-1)
* (asterisk)	Multiplication (3*3=9)
/ (forward slash)	Division (3/3=1)
% (percent sign)	Percent (20%=0.2)
^ (caret)	Exponentiation (3^2=9)

RENO Special Arithmetic Operator	Meaning (Example)
\ (backward slash)	Returns the integer result of a division (11\2=5)
] (right bracket)	Modulus, remainder of the division 12]5=2
[(left bracket)	n-th root (2[16=4, 3[64=4)
! (exclamation mark)	Factorial (3!=6)
# (number sign)	Logarithm (b#n where b is the base and n is the number, or 2#8=3)

RENO Comparison Logic Operator	Meaning (Example)
All comparison operators return False=0, True=1	
= (equal)	Equal (3=2 returns 0, 2=2 returns 1)
> (greater than sign)	Greater than (3>2 returns 1, 3<2 returns 0)
< (less than sign)	Less than (3<2 returns 0, 3>2 returns 1)
>= (greater than or equal to sign)	Greater than or equal to (3>=2 returns 1)
<= (less than or equal to sign)	Less than or equal to (3<=2 returns 0)
<> (not equal to sign)	Not equal to (2<>2 returns 0)
{ (left curly bracket)	Smallest of the two (3{2 returns 2)
} (right curly bracket)	Largest of the two (3}2 returns 3)
| (pipe)	OR (0|1 returns 1, 1|0 returns 1, 1|1 returns 1, 0|0 returns 0)
& (ampersand)	AND (0|1 returns 0, 1|0 returns 0, 1|1 returns 1, 0|0 returns 0)

Reserved Constants	Meaning (Example)
Pi	Return Pi to the precision used (3.141592654)
e	Return e to the precision used (2.718281828)

Reserved for Future Use	Meaning (Example)
? (question mark)	Reserved (10? returns 10)
' (apostrophe)	Reserved (10' returns 10)
" (quotation mark)	Reserved (10" returns 10)
: (colon)	Reserved (10:10 returns 10)

11.6 Flowchart Configurations and Execution

RENO's simulations are based on the properties of the flowchart blocks, RENO resources and the configuration of the flowchart. There are almost no inherent limits on the configuration of a RENO flowchart; this provides you with the flexibility to model any scenario you wish to examine.

In general, each path in the flowchart is placed in a queue, and the simulation executes each path based on its order of priority in the queue. However, there may be situations where the order in which the paths are executed is important to the outcome of the simulation. The following examples demonstrate how to use RENO's advanced configuration settings to control the order of execution.

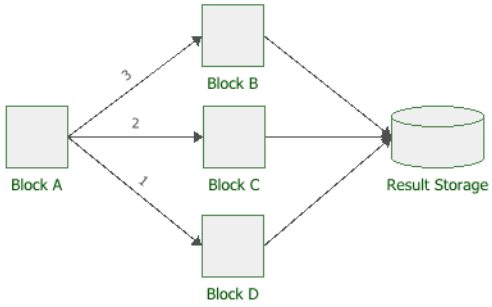
- Example 1: Blocks with multiple outgoing paths - Section 11.6.1 (p. 393)
- Example 2: Blocks that require a certain number of executed incoming paths - Section 11.6.2 (p. 395)
- Example 3: Flowcharts with multiple starting paths - Section 11.6.3 (p. 396)

Tip: Use the Debug tool to run a step-by-step simulation on your flowchart to trace the output and verify whether the flowchart behaves as intended.

11.6.1 Example 1: Blocks with Multiple Outgoing Paths

Consider a block (Block A) that has three outgoing paths. The order of execution of the outgoing paths is based on the connector priority, as shown in the following example. In this case, the path represented by connector 1 will be simulated first, followed by the path for connector 2 and then for connector 3. The simulation continues down each

path until it reaches an end, and only then is the next path in the queue executed. A path reaches an end when it is stopped by a block that requires a certain number of executed incoming paths (see Example 2) or it reaches the last block in the path.



To display or hide the connector priorities, select or clear the **Show Connector Priorities** check box on the diagram's control panel.

What's Changed? In the previous version, the Ordered Relations command was used at the block level to show connector priorities.

By default, the connector priority is determined by the order in which the connectors were connected. You can change a connector's priority number by right-clicking the connector and choosing any of the following commands on the shortcut menu: **Increase Priority**, **Decrease Priority**, **Set to First Priority** or **Set to Last Priority**. You can also specify the priority by choosing **Flowchart > Selection > Connector Priority**.

Note the following special cases:

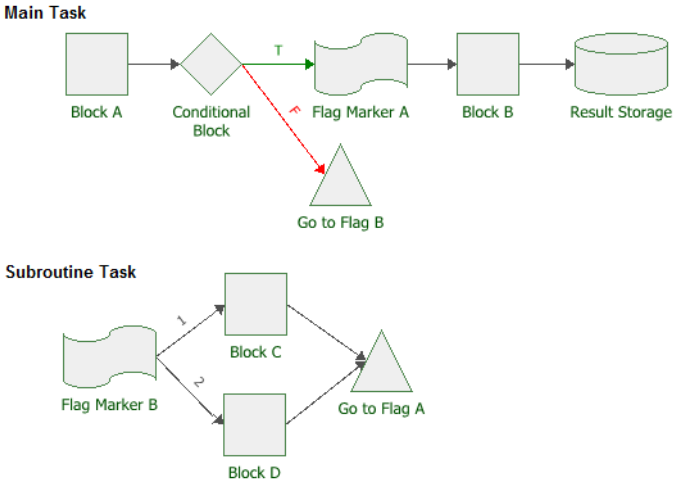
- Conditional blocks and logic gates have advanced configurations that allow you to continue execution on both true and false paths. In this case, the simulation will execute the "true" path first. Once it reaches the end of that path, the simulation will then execute the "false" path and continue until that path also ends.
- Connector priority also applies to subroutines. The simulation will execute each path in the subroutine queue until it ends (i.e., makes a full pass through the main flowchart). For example, in the following flowchart, the simulation will execute the subroutine as follows:
  - **Main Path**:

    Block A→ Conditional Block→ Go to Flag B→ Flag Marker B→ Subroutine Path 1→ Subroutine Path 2
  - **Subroutine Path 1**:

    Block C→ Go-to-Flag A→ Flag Marker A→ Block B→ Result Storage.

- **Subroutine Path 2**:

  Block D→ Go-to-Flag A→ Flag Marker A→ Block B→ Result Storage.

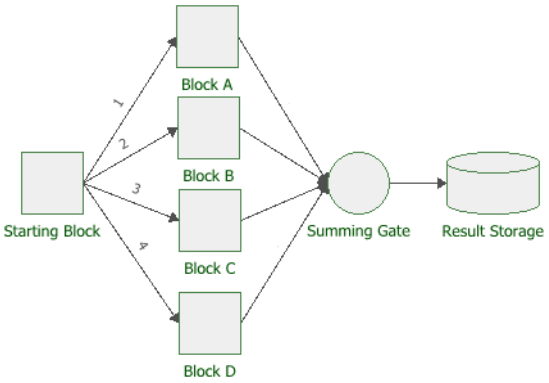


11.6.2 Example 2: Blocks that Require a Certain Number of Executed Incoming Paths

If a block has multiple incoming paths, you can configure the block to not pass an output or compute a result until a certain number of those incoming paths have been executed. This configuration is available for result storage blocks, logic gates, summing gates and counter blocks.

Enter the required number of paths in the **Required Number of Executed Incoming Paths** field in the Block Properties window of the applicable block.

The following example shows a summing gate that is configured to require 2 out of the 4 incoming paths to be executed before it passes any output to the result storage block. (The connector priorities of the starting block's outgoing paths are also shown to provide context.) In this case, the summing gate will pass two results to the result storage block: once with the value of blocks A+B, and once with the value of blocks C+D.



If the summing gate were configured to require 3 executed incoming paths, then the result of blocks A+B+C would be passed to the result storage block, and the value from block D would be ignored.

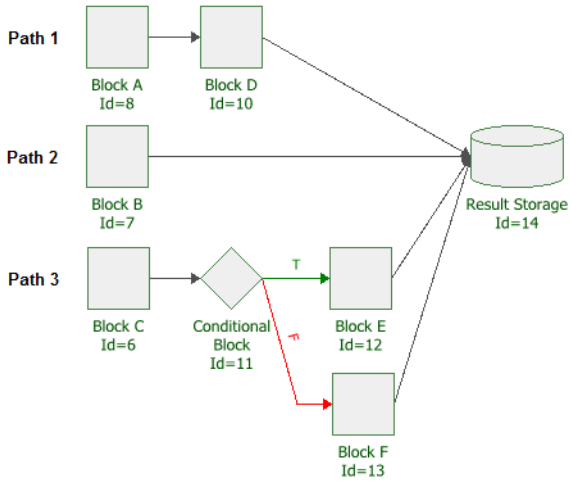
If the summing gate were configured to require only 1 executed incoming path, then the value of blocks A, B, C and D would be passed to the result storage block, without the summing gate performing any mathematical operation on the values.

11.6.3 Example 3: Flowchart with Multiple Starting Paths

In a flowchart with a single starting path, the blocks are simulated in the order they were added to the path. However, if the flowchart has multiple starting paths, then each path is placed in a queue and the simulation executes each path based on the block IDs of the starting blocks. A block ID is a number assigned to the block when it is created. You can display or hide the IDs of all the blocks in a flowchart by selecting the option in the Block ID field on the Background and Grid page of the Diagram Style window (**Format > Styles > Diagram Style**).

Note: Block IDs cannot be edited; therefore, to have full control over the execution of a flowchart, we recommend that you add a single starting block to the flowchart and define the connector priorities, as demonstrated in Example 1.

The following example shows a flowchart that has three starting blocks (with the block IDs shown). Each starting block represents an individual path. The starting block with the lowest block ID number has the highest priority in the queue; therefore, in this case, the first path to be simulated is Path 3, followed by Path 2 and then Path 1.



Note that you can have more than one flowchart within the same diagram, allowing you to test more than one case in the same diagram or create subroutines. The separate flowcharts will be put into the queue based on the block IDs of their starting blocks.

If your flowchart contains subroutines, you must configure the starting blocks of the subroutines to be executed only when they are called by go-to-flag blocks; otherwise, the simulation will consider the subroutines to be separate flowcharts (*for specific instructions, see Section 11.2.9 on page 373*).

11.7 Debugging a Flowchart

RENO's Simulation Debugger is a simple tool that allows you to trace and examine the output of a flowchart by running a step-by-step simulation through each block. You can access the tool by clicking the **Debug** icon on the control panel of the diagram or by choosing **Simulation > Debug**.

This opens a window that starts the debugging session. The flowchart being evaluated is displayed in the upper window, while details about the progress of the session are displayed in the bottom window.

Tip: RENO's simulations are based on the properties of the flowchart blocks, RENO resources and the configuration of the flowchart. If your flowchart does not behave as intended, see the previous section for help.

The Debug window includes the following commands:

Run begins a simulation of the flowchart and displays the result of each step in the simulation. You can control the speed of the simulation and other factors via the Debug Options window.

Step Into allows you to view one step of the simulation at a time. Each time you click the **Step Into** command, RENO executes the next step in the flowchart. The current block is highlighted in the flowchart, and the details of the step are displayed. If the next block in the flowchart is a subchart block, the simulation steps inside the subchart to show the details.

Step Over allows you to execute the subchart in the simulation without stepping inside the subchart.

Step Out is available only when you are viewing the progress of the simulation of a subchart. It allows you to step out of the subchart and continue the flow of the simulation in the parent flowchart. The subchart will be executed, but that part of the simulation will not be shown.

Stop halts the simulation.

Options opens the Debug Options window, which allows you to control how the debug session simulates the flowchart.

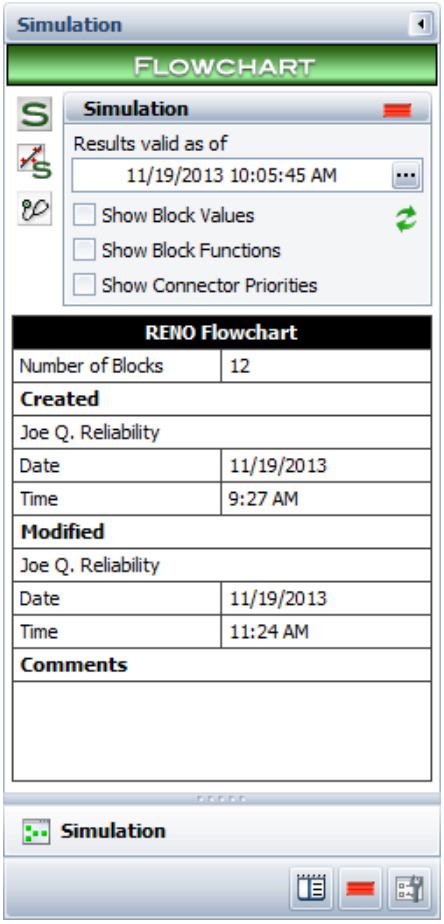
- The **Use a Seed** check box allows you to specify a starting point from which the random numbers used in the debug session will be generated. The same random numbers and, therefore, the same simulation results will be generated when the same seed value is used.
- The **Number of Simulations** field allows you to specify how many individual simulations in the flowchart should be performed during the debug session.
- The **Show Inside Subdiagrams** check box allows you to display the simulation of the subchart when using the Run command. If cleared, the subchart will be executed, but that part of the simulation will not be shown.
- The **Time Delay** field allows you to specify the amount of time, in seconds, between steps when using the Run command.

End Session closes the Debug window.

11.8 Flowchart Control Panel

The control panel associated with RENO flowcharts consists of multiple pages, each containing options for performing particular tasks. This section focuses on the Simulation page of the RENO flowchart control panel, which contains most of the tools you will need to simulate a diagram. (*For more information about the control panel in general, see page 109 in Chapter 3*.)

The Simulation page of the RENO flowchart control panel is shown next.



The Simulation page includes the following settings:

- The **Simulation** area displays the status and results of the simulation.
  - **Status** If the light is green, then the simulation has been performed. If the light is red, then the simulation has not been performed. A simulated diagram will remain in the simulated state unless a change is made to the structure of the diagram (e.g., a block is added, a connector is removed, etc.).
  - **Show Simulation Details (...)** displays the most recently generated simulation results in the Simulation Results Explorer.
  - Select the **Show Block Values** check box to display the most recent value for each block (except for reset blocks and subchart blocks) at the end of the last simulation in the block's description. The location and appearance of the block description can be changed via the Diagram Style window or the Block Style window (**Format > Styles > [Diagram Style/Block Style]**). If you have made any changes to the flowchart since the last simulation, no values will be displayed. To perform a single simulation of the flowchart using the seed that is currently specified in the RENO Simulation window, click the **Update Block Values** icon.

- Select the **Show Block Functions** check box to display information about the function the block performs in the block's description. Specifically, for each block type, the following will be shown. (For block types not shown in this list, nothing is displayed.)
  - Flowchart standard blocks: equation
  - Result storage blocks: result type (e.g., average, sum, etc.)
  - Conditional blocks: condition (e.g., =, >, etc.)
  - Logic gates: gate type (e.g., AND, OR, etc.)
  - Summing gates: operation
  - Go-to-flag blocks: the target flag marker
  - Counter blocks: counter type (i.e., when the counter resets)
  - Subchart blocks: the flowchart that is being used as a subchart
- Select the **Show Connection Priorities** check box to display the order of execution of the outgoing paths from each block. This order is based on the connector priority and can be changed by selecting a connector and then choosing **Flowchart > Selection > Connector Priority**.

- The **Information** area displays information about the diagram or the currently selected block. If no block is selected in the diagram, the **Information** area displays the number of blocks in the diagram and information about the diagram creation and last modification. If a block is selected in the diagram, it displays a subset of the properties found in the Block Properties window. In both cases, this area also offers a **Comments** field; click the field heading to enter comments. For blocks, these comments are also accessible in the Identifiers properties in the Block Properties window.
- The **Tools** area on the left side of the control panel gives you quick access to the tools you will need to analyze the diagram and generate additional results.

Simulate opens the RENO Simulation window, which allows you to specify simulation settings for the current diagram and run the simulation.

Plot creates a plot based on the simulation results. Clicking the **Plot** icon before the diagram has been simulated will automatically perform the simulation and then plot the data.

Debug opens the RENO Simulation Debugger, which allows you to trace and examine the output of a flowchart by running a step-by-step simulation through each block.

11.9 Flowchart Simulation Window

To simulate a RENO flowchart, choose **Simulation > Simulation > Simulate** or click the **Simulate** icon.

Tip: The following terms apply to simulating RENO flowcharts: a *simulation* is a single pass through the flowchart (note that this may pass through blocks multiple times if your flowchart includes one or more loops), a *run* is a set of simulations, and an *analysis* is a set of runs.

When simulating, you have three basic choices of what to do:

- **Perform a single run**, consisting of anywhere from 1 to 100,000,000 simulations.
- **Perform a sensitivity analysis**, which involves varying either one or two variables and performing a run at each setting. This allows you to evaluate the effect of each of the variables on the results.
- **Perform multiple analyses**, which involves performing multiple sensitivity analyses in order to minimize or maximize the value of a specified result storage block.

The area at the top of the RENO Simulation window displays the status of the simulation. For information on configuring this display, see the next section on page 402.

During simulation, you can click **Cancel** to stop the simulation at the current point. The results will be provided for the simulations (if any) that were completed before you stopped the process. When simulation is complete, you can click the **Details** button to view the results of the simulation in the Simulation Results Explorer. Click the **Simulate** button to re-run the simulation.

Note: Once you have simulated a flowchart, the settings you have specified in the RENO Simulation window are saved along with the diagram so that you or another user can easily simulate the flowchart again using the same settings. The first time you simulate a flowchart, the settings default to the settings used for the last simulation that was performed on your computer.

Performing a Single Run

To perform a single run, you will need only the settings on the General Settings tab of the window. First, specify the **Number of Simulations** that you want the run to consist of.

If you need your results to be repeatable, select the **Use a Seed** check box and enter a seed, which specifies a starting point from which the random numbers used in simulation will be generated. The same random numbers and, therefore, the same simulation results will be generated when the same seed value is used.

In the Advanced Simulation Settings area, make sure that both sensitivity analysis and multiple analyses are disabled, then click **Simulate**.

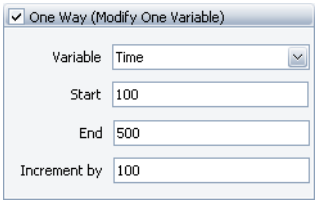
Performing a Sensitivity Analysis

Sensitivity analysis allows you to evaluate the effect of one or two variables on the results by performing a run at each of a range of settings.

To perform a sensitivity analysis, first specify the settings for each run on the General Settings tab of the window.

On the Sensitivity Analysis tab, specify whether you want to modify one or two variables by selecting either the **One Way** check box or both the **One Way** and **Two Way** check boxes. In each case, choose the variable from the **Variable** drop-down list, then specify the range of settings for the variable. The **Start** value will be used in the first run. The **End** value is the maximum value that can be used in the simulations; the largest value used will be less than or equal to this. **Increment by** is the amount by which the start value will be increased for each subsequent run.

These settings determine how many runs will be performed. For example, let's say that the number of simulations specified on the General tab is 1,000. You select the **One Way** check box and, for a selected variable, you enter a **Start** value of 100, an **End** value of 500 and an **Increment by** value of 100, as shown next.



With these settings, the 1,000 simulations will be run with the variable valued at 100, again with it valued at 200, again at 300 and so on until they have been run with the variable at its maximum value of 500. In this case, that would mean that the analysis will consist of 5 runs, with 1,000 simulations performed for each run.

Note: In the **Variable** drop-down list, you can also choose to vary the number of simulations performed or the seed used.

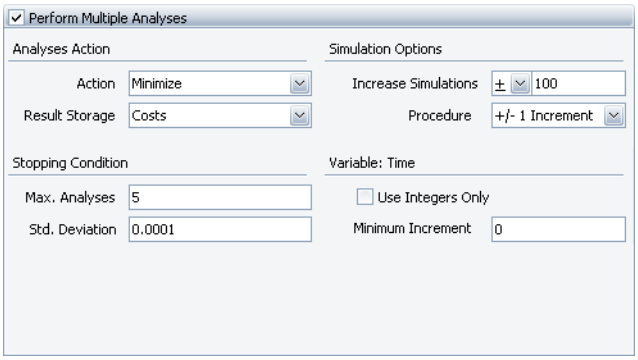
Performing Multiple Analyses

You can perform multiple sensitivity analyses in order to minimize or maximize the value of a result storage block. When performing multiple analyses, RENO performs the first sensitivity analysis and finds the value of the variable at which the result storage block of interest is either lowest or highest (depending on whether you are minimizing or maximizing the value). It then takes an interval surrounding that value, divides it up according to the increment specified for sensitivity analysis and performs the specified number of runs on that interval. This process is repeated until specified stopping conditions are reached, refining the interval and the minimum/maximum result with each analysis.

To perform multiple analyses, first specify the settings for each run on the General Settings tab of the window, then specify the sensitivity analysis settings.

On the Multiple Analyses tab, select the **Perform Multiple Analyses** check box, then specify the following settings:

- In the **Analyses Action** area, choose the **Result Storage** block that you want to work with, and the **Action** you want to perform on it (i.e., whether you want to minimize or maximize the value of the selected result storage block). Note that you cannot choose a result storage block that is set to store an array of values.
- In the **Simulation Options** area:
  - Use the **Increase Simulations** field to specify a number by which the number of simulations is either incremented or multiplied for each consecutive sensitivity analysis. Click the operator (+ or x) to toggle between incrementing and multiplying. This option allows you to, for example, perform more simulations at each consecutive analysis so that early analyses over a broad range take less time, while final analyses are performed in detail.
  - In the **Procedure** field, specify how to calculate the range for each consecutive analysis. For instance, if you choose **+/- 2 Increments**, then RENO will take the "best" value of the variable from the most recent analysis (i.e., the one that yields the most desirable result for the result storage block) and, for the next analysis, will use a range that spans from two increments below that value to two increments above it. The increments are those used in the previous analysis. For example, let's assume that you use the sensitivity analysis settings specified above (i.e., variable = Time, start = 100, end = 500, increment = 100) and the multiple analyses settings shown next.



The results of the simulation are shown next.

Analysis 1 - Time	Cost	Analysis 2 - Time	Cost	Analysis 3 - Time	Cost	Analysis 4 - Time	Cost	Analysis 5 - Time	Cost
100	998.442183	300	539.357805	400	459.768421	400	459.830628	412.5	456.846666
200	718.817572	350	487.060469	425	455.620649	412.5	456.885721	418.75	456.077303
300	539.326056	400	459.884154	450	457.581814	425	455.659714	425	455.583603
400	459.821511	450	457.570687	475	465.866379	437.5	455.788598	431.25	455.50615
500	480.336322	500	480.419894	500	480.464505	450	457.58326	437.5	455.839078

With these settings, Analysis 1 consists of runs performed with the Time variable at 100, 200, 300, 400 and 500. The minimum value for the Costs variable in the first analysis is found at 400. The range for Analysis 2 is +/- 1 increment around 400 (i.e., 300 - 500), and that range is divided into the same number of steps used in

the previous analysis. The minimum value for the Costs variable in the second analysis is found at 450. The range for Analysis 3 is +/- 1 increment around 450, where the increment is the one used in Analysis 2 (i.e., +/- 50, yielding a range of 400 - 500). This continues until the stopping condition is reached, as explained next.

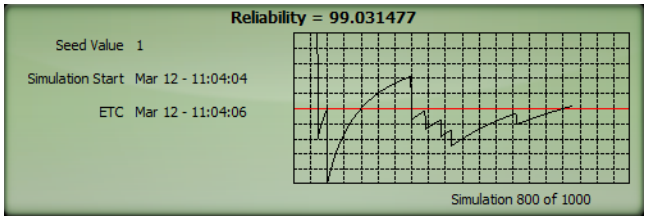
- In the **Stopping Condition** area, enter the maximum number of sensitivity analyses that will be run in the **Max. Analyses** field. In the **Std. Deviation** field, enter the convergence threshold at which the result is considered accurate enough. Simulation will stop when the standard deviation is reached or when the maximum number of analyses have been performed, whichever comes first.

For each variable being modified in the sensitivity analyses, you can select the **Use Integers Only** check box to ensure that the analyses will use only integer values of the variable. You can also specify a **Minimum Increment** that can be used to determine the values of the variable for the analyses. Consider the multiple analyses example above. In Analysis 5, the range of values for the variable is 412.5 - 437.5. Suppose, however, that you can adjust Time only to a 10-hour window. If you specify a minimum increment of 10, you obtain the results shown next.

Analysis 1 - Time	Cost	Analysis 2 - Time	Cost	Analysis 3 - Time	Cost	Analysis 4 - Time	Cost	Analysis 5 - Time	Cost
100	998.442183	300	539.357805	400	459.768421	400	459.830628	412.5	456.846666
200	718.817572	350	487.060469	425	455.620649	412.5	456.885721	422.5	455.7451
300	539.326056	400	459.884154	450	457.581814	425	455.659714	432.5	455.527354
400	459.821511	450	457.570687	475	465.866379	437.5	455.788598		
500	480.336322	500	480.419894	500	480.464505	450	457.58326		

11.9.1 RENO Simulation Status Display

The area at the top of the RENO Simulation window displays the status of the simulation.



The following numerical information is always shown:

- **Seed Value** displays the seed value used, if any (i.e., the starting point from which the random numbers used in simulation are generated).
- **Simulation Start** displays the date and time that the simulation began.
- **ETC** displays the expected date and time of completion for the simulation.

In addition, the current number of simulations performed out of the total number of simulations scheduled to be performed is shown at the bottom of the simulation status display (e.g., 10 out of 100 simulations). You can control how often this number is updated by specifying the number of increments to divide the total number of simulations into in the **Update Increments** field in the **Display Options** area.

If you have selected **Full Display** in the **Display Options** area, a plot will also be shown in the simulation status display. This plot shows the value of the result storage block that you have chosen from the **Display Result Storage Block** drop-down list. Result storage blocks of all types except array will be available in this list. The range of values represented on the y-axis is controlled by your entries in the **Y-Axis** fields. During simulation, the current value of the selected result storage block is displayed above the plot.

11.10 Simulation Results and Plots

The Simulation Results Explorer is generally the same for BlockSim diagrams and for RENO flowcharts, with a few minor differences. *For details on using this window, see page 308 in Chapter 9*.

The available worksheets are presented next.

General Summary Results

The General Summary worksheet presents an overview of the simulation settings.

- **Number of Simulations**: The number of simulations per run in the last analysis that was performed.
- **Seed Value**: The seed value, if any, used when generating random numbers during simulation.
- **Simulation Variable 1(2)**: The name of the variable(s) modified during simulation, if any.
- **Variable Start**: The starting value of the modified variable.
- **Variable End**: The ending value of the modified variable.
- **Number of Runs**: The number of runs that were performed for the modified variable in the last analysis. Unless you performed one or more sensitivity analyses, this will be one.

Result Storage Blocks

The Result Storage Blocks worksheets display the simulation results for each result storage block in the flowchart or in any subcharts called during simulation. Note that for any block not located in the main flowchart, the name of the flowchart in which it is used is displayed in the navigation panel. Click the name of the block in the navigation panel to display the block details on a separate worksheet for that corresponding block.

The first table displays the following information:

- **Flowchart(s) Location**: The path by which the result storage block was accessed during simulation. For example, consider a result storage block located in Flowchart 1. If you simulated Flowchart 1, then the location shown would simply be “Flowchart 1.” However, if Flowchart 1 is called by “Subchart Block” in Flowchart 2, and you simulated Flowchart 2, then the location shown would be “Flowchart 2.Subchart Block->Flowchart 1.”
- **Storage Type**: The type of data that the result storage block contains (i.e., average, sum, array, minimum, maximum, last value or normalized sum of results).
- **Runs Stored [per Analysis]**: The number of runs of simulations performed. If you performed multiple analyses, this information is for each analysis.

The second table displays the value (or values, if the result storage block is an array) of the result storage block at the end of each simulation run. If you have performed only one run, only the result storage block name and value will be displayed. If you have performed multiple runs (i.e., a single sensitivity analysis), the table will contain values for the varied variable(s) and for the result storage block. (Where two variables are varied, the values of the variable named first are in the first column (column A) and the values of the variable named second are in the heading row of the table.) If you have performed multiple analyses, the table(s) will contain values for the varied variable(s) and for the result storage block for each analysis.

11.10.1 Flowchart Plots

Plots can be created and viewed by clicking the Plot icon located on the Simulation page of the control panel for each flowchart.

If you click the **Plot** icon before the flowchart has been simulated, you will be prompted to simulate the flowchart and then the data will be plotted in the plot sheet. (Note that the **Plot** icon on the Plot sheet control panel refreshes the plot. If data or settings have been changed since the flowchart was last simulated, you must resimulate before refreshing the plot.)

You can add additional plot sheets to the flowchart by right-clicking the sheet index tabs at the bottom of the window and choosing **Insert Simulation Plot** on the shortcut menu. You can delete a plot sheet from the flowchart by selecting the plot sheet, right-clicking and choosing **Delete Plot**.

There are five plot types available in RENO. Not all plot types are available for all simulation settings. In addition, there are restrictions on the types of data that can be plotted on each plot type at each simulation setting.

- **Result Storages Plot** shows the value(s) of one or more result storage blocks plotted against either the result index (in the case of a single simulation run) or the varied variable (in the case of sensitivity analysis or multiple analyses). You can show results for single-value and array result storage blocks within the same plot.
- **Result Comparison Plot** shows the value(s) of two result storage blocks plotted against each other. Both result storage blocks must be either single-result or array; you cannot compare a single-value result storage block against an array result storage block. For single-run simulations, only array result storage blocks can be compared.
- **Pie Chart** displays the value(s) of one or more result storage blocks in pie chart format. This plot is available only for single-run simulations and for analyses where one variable is modified. For single-run simulations, you can plot the results of multiple single-value result storage blocks or of one array result storage block. For sensitivity/multiple analyses, you can plot the results of one single-value result storage block.
- **Bar Chart** displays the value(s) of one or more result storage blocks in bar chart format. This plot is available only for single-run simulations and for analyses where one variable is modified. For single-run simulations, you can plot the results of multiple single-value result storage blocks or of one array result storage block. For sensitivity/multiple analyses, you can plot the results of one single-value result storage block.
- **Probability Plot** plots the values of one or more result storage blocks against the cumulative probability of the values, based on the ranking method (Median Ranks or Kaplan-Meier) selected on the Plot page of the Application Setup (**File > Application Setup**). For single-run simulations, you can plot the results of one or more array result storage blocks. For sensitivity/multiple analyses where one variable is modified, you can plot the results of one or more single-value and/or array result storage blocks. For sensitivity/multiple analyses where two variables are modified, you can plot the results of one single-value or array result storage block.

The Plot Sheet control panel changes depending on the selected plot type and the simulation settings that have been used, so that it is easy for you to meet the requirements for the plot you are generating. When only one item can be plotted on an axis or on a plot, the selection area will be a drop-down list. When multiple items can be plotted, the selection area will be a list with check boxes. In both cases, the type of each result storage block will be marked in parentheses next to the block name. For those plots where you can plot only single results or only arrays, RENO will display only the result storage blocks of the correct type. For the Result Comparison plot, once you have selected the result storage block for the X-axis, only result storage blocks of the same type (i.e. single-value or array) will be displayed in the selection area for the Y-axis, as you cannot plot single values against arrays.

The Probability plot offers one additional option. You can use the **Scale** drop-down list to select the scale you would like to use for the plot.

Plots 12

Plots are available for analytical data and for simulation data. These plots can be created and viewed by clicking the **Plot** icon located on the Main page of the control panel for each diagram.

If you click the **Plot** icon before the diagram has been analyzed or simulated, it will automatically be analyzed or you will be prompted to simulate the diagram and then the data will be plotted in the plot sheet. (Note that the **Plot** icon on the plot sheet control panel refreshes the plot. If data or analysis settings have been changed since the diagram was last analyzed or simulated, you must re-analyze the data before refreshing the plot.) You can control which items (lines, points, etc.) are displayed on the plot via the Show/Hide Plot Items window, which is accessed by choosing **Plot > Actions > Show/Hide Plot Items**.

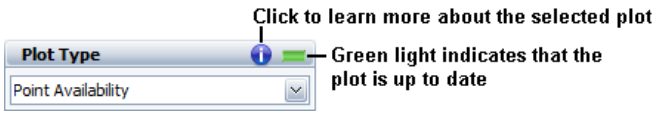
You can delete a plot sheet from the diagram by right-clicking the plot sheet and choosing **Delete Plot**.

Note: You can copy the plot graphic to the Clipboard for use in other locations. Starting in version 8.0.5, choose **Plot > Copy Plot Graphic** and select an image type. You can copy the plot as a bitmap image or as a metafile image optimized for pasting into either a Synthesis spreadsheet (e.g., analysis workbook) or an external application. Choosing **Home > Copy** or pressing **CTRL+C** will also copy the plot graphic, using the default copy type selected in the **Plot Copy Type** field in the Synthesis Setup window (**File > Synthesis Setup**).

12.1 Plot Sheet Control Panel

The control panel tools include the following options:

- The **Plot Type** drop-down list provides a choice of applicable plots. To learn more about the selected plot, click the blue (i) icon. This area also contains an indicator light that displays the status of the plot sheet in relation to the analysis it is associated with. If the light is green, then the plot reflects the current analysis. If the light is red, then you will have to update the plot by refreshing it and/or reanalyzing the associated diagram. *For more information about the available plot types, refer to the specific section covering the diagram type.*



- **Units** (available for all analytical plots and for simulation System plots) allows you to choose the units to display the plot in. Choose the unit from the drop-down list.

- **Auto Refresh** if selected, automatically refreshes the plot if any changes are made to the plot.
- **Keep Aspect Ratio** if selected, the proportion of horizontal size to vertical size will remain constant when you resize the plot. If not selected, the plot graphic will stretch to fill the window if the window is resized.
- **Scaling** (available for all analytical plots other than the Static RI and Static RI Tableau plots, for all BlockSim simulation System plots, for the BlockSim simulation Block Results plot and for all RENO flowchart plots except the Pie chart)
  - **X and Y Scaling Boxes** shows the minimum and maximum values for the x- and y-axes. You can change these values if the check box beside the value range is not selected. If it is selected, the software will automatically choose appropriate values for the range.
- **Other**
  - **Show Intervals** (available only for Mean Availability and Throughput plots) shows the time intervals used on the plot. The number of intervals is set by the simulation settings.
  - **Resolution** (available for all analytical plots other than the Static RI and Static RI Tableau plots) allows you to specify how many points are used to construct the plot. You can specify any number between 2 and 300.
  - **Show System** (available only for simulation Block Up/Down plots and all analytical Block plots) if selected, displays the system values on the plot.
- **Static RI set for**
  - **At Time** (available only for analytical Static RI and Static RI Tableau plots) allows you to specify the time shown.
  - **Blocks** (available only for analytical Static RI and simulation RS FCI and RS DECI plots) allows you to specify how many blocks to include in the plot.
- **Available Blocks** (available only for Block Up/Down plots and all Blocks plots other than the Block Results plot) allows you to specify which blocks to display on the plot. To select all blocks, click the check box in the table header. To select individual blocks, click the check box in each row.
- **Available Crews** (available only for Crew simulation plots and only if at least one crew has been assigned to a block in the current diagram) allows you to specify which crews to display on the plot. To select all crews, click the check box in the table header. To select individual crews, click the check box in each row.
- **Available Pools** (available only for Spare Part Pools simulation plots and only if at least one spare part pool has been assigned to a block in the current diagram) allows you to specify which pools to display on the plot. To select all pools, click the check box in the table header. To select individual pools, click the check box in each row.
- **Available Results** (available only for Block Results plots) allows you to specify which values the x-axis, y-axis and bubble size will be based on.
- **Scale** (available only for flowchart Probability plots) allows you to specify the type of scale used for the plot axes.
- **Available Result Storages** (available only for flowchart plots) allows you to specify the result storage block(s) that will be included in the plot. To select all result storage blocks, click the check box in the table header. To select individual result storage blocks, click the check box in each row.

The folio tools are arranged on the left side of the control panel:

Redraw Plot updates the plot to reflect any changes that have been made.

Plot Setup opens the Plot Setup window, which allows you to customize most aspects of the plot including the titles, colors, sizes, etc.

RS Draw launches ReliaSoft Draw, which allows you to view the plot in greater detail, add annotations and modify selected plot elements.

Export Plot Graphic allows you to save the current plot graphic in one of the following formats: *.jpg, *.gif, *.png or *.wmf.

QCP opens the Quick Calculation Pad (QCP), which allows you to obtain calculated results, such as the block unreliability or the expected number of system failures, based on the associated diagram.

12.2 Show/Hide Plot Items Window

The Show/Hide Plot Items window allows you to select the data you want displayed on the plot. It is accessed by choosing **Plot > Actions > Show/Hide Plot Items**.

The appearance of this window will vary depending on the plot type and on the data type used, but it will function in the same way regardless.

- **Subset** allows you to select the system, block or container that the settings in this window will apply to. In an overlay plot, Subset allows you to select the diagram that the settings in this window will apply to.
- **Name** displays each item (e.g., Failure Rate Line, Block 1, etc.) available for display on the plot. Clicking the header of this column will sort the items by name. Clicking again will reverse the sort.
- **Points/Line/Bar/Area/Slice** allows you to select how the values for the item are displayed. Clicking the header of this column will toggle between selecting all options in the column and clearing all options in the column.
- **Legends, upper and lower titles**
  - **Legends** displays all legends on the plot.
  - **Titles** displays the Header and Footer titles on the plot.

12.3 Overlay Plots

BLOCKSIM Only

BlockSim gives you the ability to place results from more than one RBD, fault tree and/or phase diagram on a single plot using an overlay plot. To add an overlay plot to a project, choose **Insert > Reports and Plots > Overlay Plot**.

The Select Diagrams window will appear, allowing you to select the RBDs, fault trees and/or phase diagrams to be associated with the overlay plot from any diagram in the project. You must have two or more analyzed diagrams, fault trees and/or phase diagrams within the current project in order to plot multiple diagrams in an overlay plot diagram. You can include a maximum of 20 diagrams in the plot. The caption bar on the Select Diagrams window indicates how many remaining diagrams can be added to the plot.

Note that if you select a simulation plot before simulating all selected diagrams, fault trees and/or phase diagrams, a window will appear asking if you want to simulate the diagram. If you click **Yes**, the Maintainability/Availability Simulation window will appear, which allows you to perform the simulation. After running the simulation, the overlay plot displays the plot.

What's Changed? In previous versions of the software, overlay plots were known as MultiPlots.

Plot Types

You can select which plot type you want to view by choosing one from the **Plot Types** drop-down list on the overlay plot control panel. The available plots depend on the diagrams used in the plot.

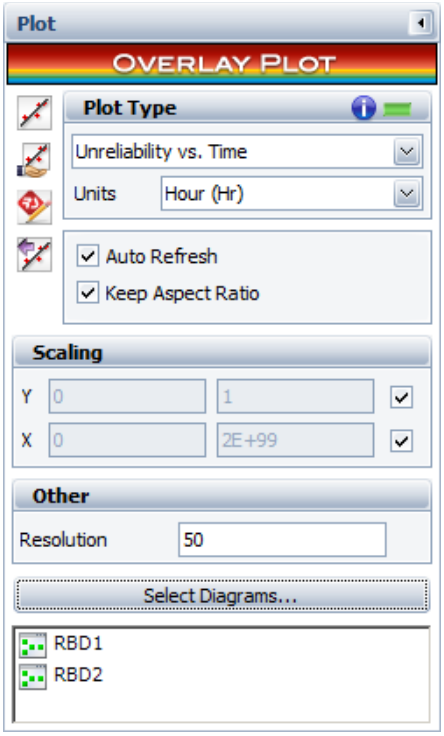
- Analytical Plots
  - Unreliability vs. Time
  - Reliability vs. Time
  - PDF Plot
  - Failure Rate vs. Time
- Simulation Plots
  - Point Availability
  - Point Reliability
  - Cost
  - Mean Availability
  - System Failures
  - Throughput
  - System Bubble Plot (This report is similar to the Block Results plot, but it shows results for systems instead of blocks.)

When the overlay plot shows results from both analytical and simulation diagrams, the Reliability plot shows system reliability over time for analytical diagrams and point reliability over time for simulation diagrams. These differ in that the analytical system reliability does not account for repairs. Point reliability reflects the fact that simulation diagrams may include repairs.

Consider the following scenario: assume two units in parallel. The analytical system reliability, which does not account for repairs, is the probability that both units fail. In this case, when one unit goes down, it does not get repaired and the system fails after the second unit fails. In the case of repairs, however, it is possible for one of the two units to fail and get repaired before the second unit fails. Thus, when the second unit fails, the system will still be up due to the fact that the first unit was repaired.

Overlay Plot Control Panel

The control panel in the overlay plot shares several options with the control panel in standard plots (*see Section 12.1 on page 405*). In addition, the control panel in the overlay plot includes a **Select Diagrams** button that allows you to select multiple diagrams to be displayed in a single plot, as shown in the following example.

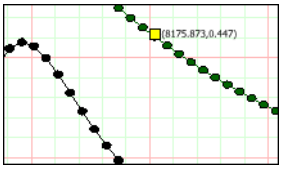


You can select calculated diagrams from any RBD, fault tree or phase diagram in the project for inclusion in the overlay plot. To remove a data set from the plot, click **Select Diagrams** and clear the diagram's check box.

12.4 Tips on Working with Plots

There are several additional commands that you can use when you are working with plots.

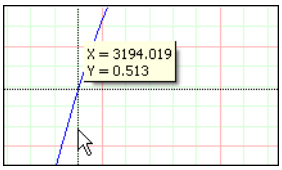
- To show the coordinates of any location on the plot, press **SHIFT** and click in the plot. A box displaying the coordinates of the selected location will appear, as shown next.



  You can release the **SHIFT** key once the coordinates are displayed. The coordinates will continue to be displayed as long as you hold down the mouse button. You can move the pointer around in the plot and the coordinates will change to reflect its current location.

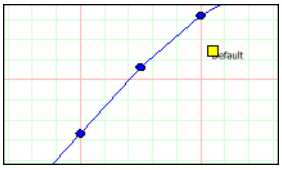
- If the pointer is on a plot line (e.g., a failure intensity line, a point availability line, etc.), pressing **SHIFT** and clicking the line will track the line when you move the pointer. A crosshair will be displayed showing the current

location on the line. As you move the pointer left or right, the crosshair will mirror the movement but will stay on the line and the current coordinates on the line will be displayed (i.e., Y is calculated based on X), as shown next.



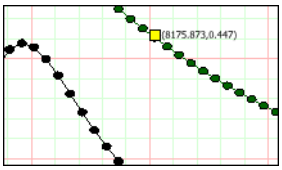
Pressing **CTRL+SHIFT** and clicking the line does the same thing, but X is calculated based on Y.

- To add a custom label to the plot, press **CTRL** and click in the plot. A new label will appear in the plot, as shown next.



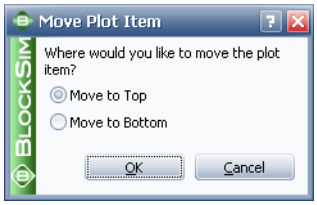
The yellow box at the upper left corner of the label indicates that the label is selected. When the label is selected, you can edit the label text directly in the plot and can also click the box and drag it to a new location, if desired. Clicking elsewhere in the plot will de-select the label.

- To add a label that displays coordinates to the plot, position the pointer at the location you want labeled then press **CTRL+ALT** and click the location. A coordinate label will be added to the plot, as shown next.



Note that a coordinates label includes a < sign pointing to the location that the coordinates refer to.

- Pressing **ALT** and clicking a plotted line or point will open a window like the one shown next.



You can choose to bring the line to the front or send it to the back. If you have clicked a point, this window will enable you to bring the group of points to the front or send them to the back. This feature is useful in cases where lines or points are obscuring one another.

- You can click any plot element (i.e., point, line, etc.) to open the Plot Setup window. The pointer will change from an arrow to a pointing hand when you move the pointer over an element for which this is possible. Note that the element you clicked is automatically selected on the last page of the Plot Setup.

Plot Setup 13

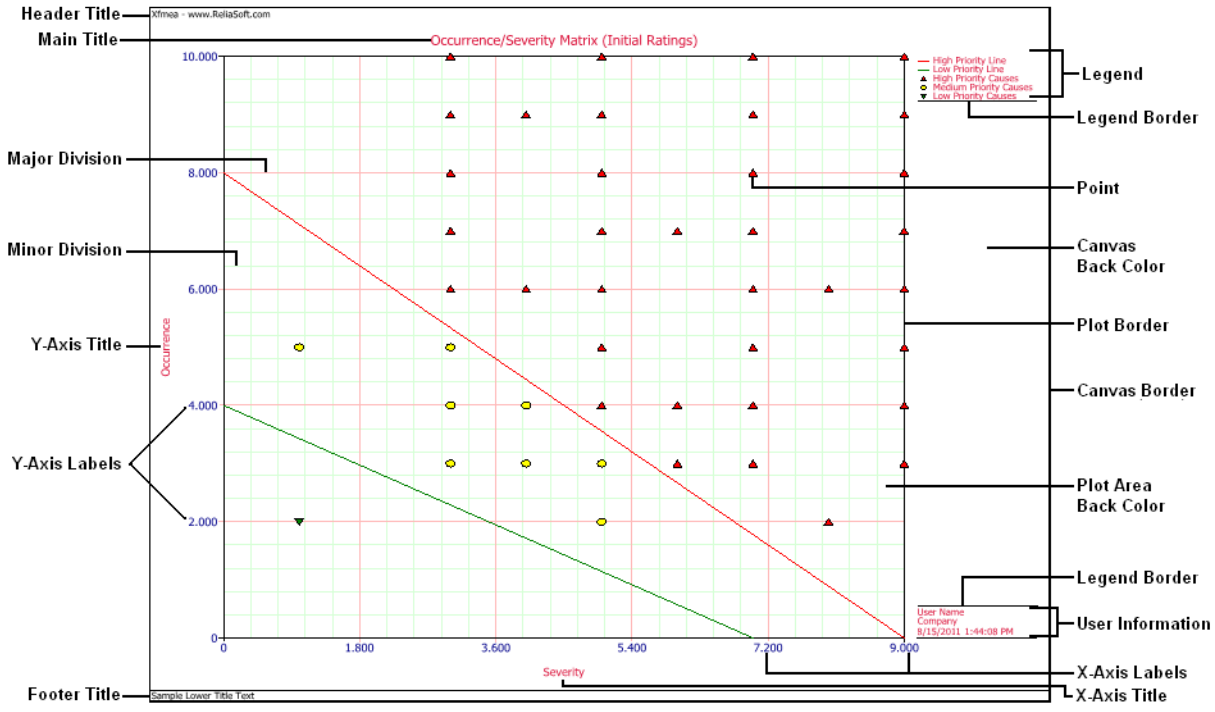
The software allows you to customize plots to meet your needs. Depending on the size and resolution of your monitor, you may find that different setups work better for you or you may simply prefer settings other than those shipped with the application. The Plot Setup window gives you full control over the settings used both for individual plots and the default settings for all new plots.

Open the Plot Setup window by clicking the **Plot Setup** icon on the Plot control panel.

In addition, clicking an element (e.g., point, line, etc.) in the plot will open the Plot Setup window.

Each time you open the Plot Setup window, the changes that you make will apply only to the current plot unless you specify otherwise. To specify settings to be used as defaults for all subsequent plots or to re-apply default settings to the current plot, click the **Defaults** button to open the Plot Defaults window. *See Section 13.11 on page 422*.

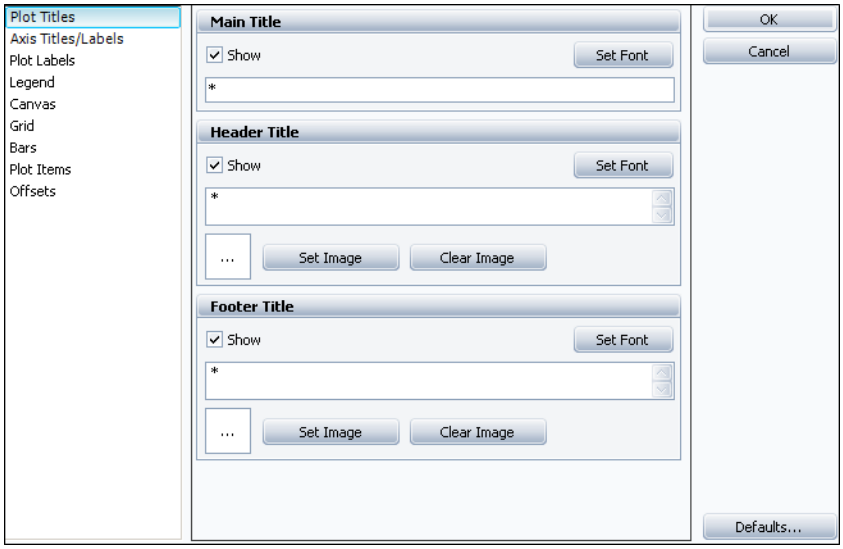
The Synthesis applications that include plots use the applicable plot elements (e.g., there are no confidence bounds in Lambda Predict plots, so those lines will not be shown on plots.) For your reference, the following picture shows a line plot with its components labeled. (Note that the plot is generated from Weibull++, but the components settings are the same for all Synthesis desktop applications.)



You can change these settings on the pages of the Plot Setup window.

13.1 Plot Titles Page

The Plot Titles page allows you to define the main, header and footer titles used in the plot and to change the text font and color.



Select or clear the **Show** check box to determine whether the title will be shown on the plot. Click the **Set Font** button next to the corresponding input box to open the Font window, which allows you to set the font type, style, size, color and text orientation.

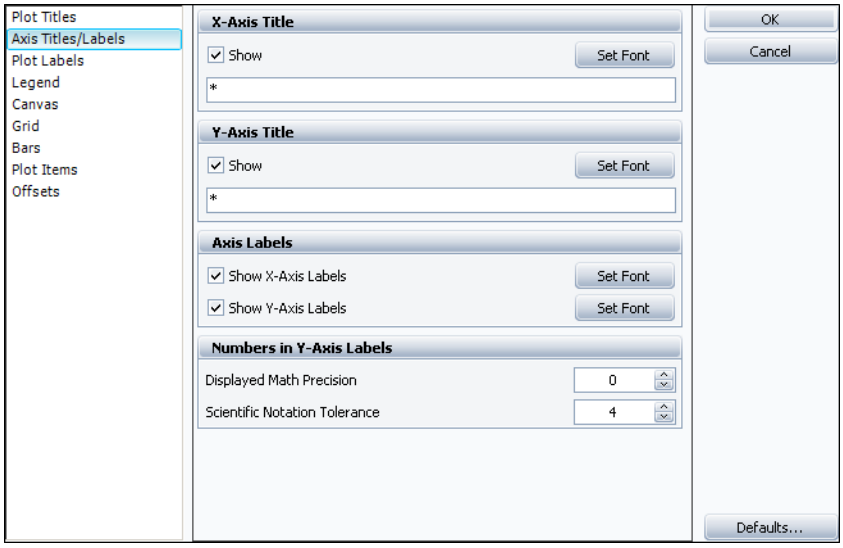
Each asterisk (*) represents the default title text. For the main title, this is defined on the Titles Text page of the Plot Defaults window. (*See Section 13.11.1 on page 423*.) If you want to append additional text to the default title, keep the asterisk and add the additional text before or after it. For example, "* (Product A)" will result in a plot title like "Default Main Title (Product A)." If you want to completely replace the default title, delete the asterisk before typing the new text.

You can use the **ENTER** key to add additional lines to the header title and the footer title.

To add an image file to the header title or footer title, click **Select Image** then browse for the file you want to include. Click **Open**. (You can select an image that uses one of the following formats: *.bmp, *.gif, *.jpg, *.jpeg, or *.ico.) The selected image appears next to the **Select Image** button. The image will also appear at the left side of the header or footer text. To remove the image, click **Clear Image**.

13.2 Axis Titles/Labels Page

The Axis Titles/Labels page allows you to define the information used with the x-axis and y-axis, including the titles, axis labels and the numbers displayed in the axis labels. This page is not available when you are working with a pie chart.



The options on this page will vary depending on the plot style you are working with. The image above shows the options for a line chart.

- **Axis Titles**
  - Select or clear the **Show** check box to determine whether the title will be shown on the plot. Click the **Set Font** button next to the corresponding input box to open the Font window, which allows you to set the font type, style, size, color and orientation of the text.

  Each asterisk (*) represents the default title text. If you want to append additional text to the default title, keep the asterisk and add the additional text before or after it. For example, "* (Product A)" will result in an axis title like "Axis Title (Product A)." If you want to completely replace the default title, delete the asterisk before typing the new text.

- **Axis Labels**
  - **Axis Labels** options, when selected, displays the numerical labels for the x-axis and/or the y-axis in the plot.
  - If available for the plot type, click the **Custom Labels** button next to the corresponding option to open the Custom Axis Labels window, which allows you to add user-defined numerical labels to the x-axis or y-axis of the plot. Note that values outside the plot's display range are accepted but will not be shown. This option is available starting in 9.0.10.

  Click the **Set Font** button next to the corresponding option to open the Font window, which allows you to set the font type, style, size, color and orientation of the text.

- **Numbers in Axis Labels**

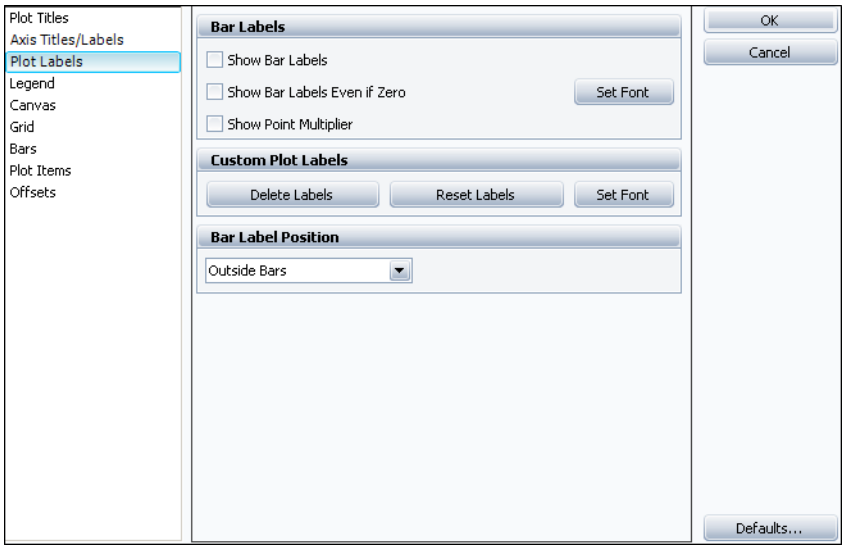
  Use the options in this area to configure the mathematical precision (number of decimal places) and scientific tolerance of the values shown on the axes.

The scientific tolerance sets the point at which the numbers will be converted to normalized scientific notation. For example, setting the scientific tolerance to 3 means that all numbers with a value of 1,000 or more will be converted to normalized scientific notation (e.g., 1.0E +3).

Note: When you are working with a bar chart, the **Numbers in Axis Labels** area will contain settings for only the axis representing the dependent variable. This is affected by the orientation of your bar chart; if you are using vertical bars, this area will contain settings for the y-axis, and if you are using horizontal bars, it will contain settings for the x-axis. It is important to be aware that the settings for one axis will not transfer to the other axis when you change the bar orientation. You will need to return to this page and re-enter the settings.

13.3 Plot Labels Page

The Plot Labels page allows you to customize the labels for items (e.g., bars. slices, points) shown in the plot as well as for custom labels used in the plot.



The options on this page will vary depending on the plot style you are working with. The image above shows the options for a bar chart. All available options are presented below.

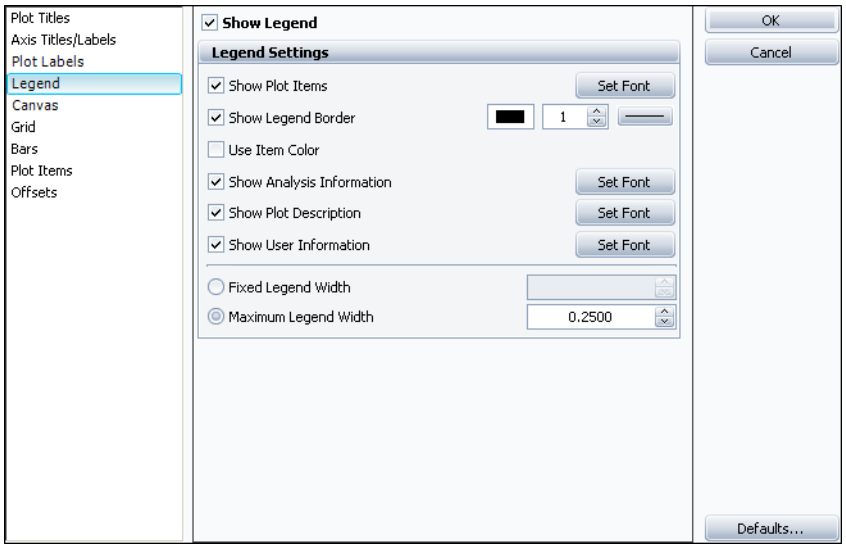
- **Bar Labels** (available only for bar charts)
  - To display the y-axis value of each bar with the bar, select the **Show Bar Labels** check box. (You can set the bar style using the **Bar Orientation** field on the Bars page of the Plot Setup. *See Section 13.7 on page 419*.)
  - **Show Bar Labels Even if Zero** if selected, bars with a y-axis value of 0 will be shown with a label of "0." The number of decimal places in the label will conform to the y-axis math precision value specified on the Axis Titles/Labels page.
- **Point Labels** (available only for line plots)
  - To display the coordinates for each point at the lower right of the point, select the **Show Point Coordinates** check box. If this option is not selected, you can still display the coordinates for each point in a pop-up box by pointing to the point.
  - To display the point label in the same color as the border of the point itself, select the **Use Point Border Color** check box.
  - If there are overlapping points, select the **Show Point Multiplier** check box to display the number of points to the right of the point.

- **Slice Labels** (available only for pie charts)
  - To label the slices of the pie chart according to the components they represent, select the **Show Slice Labels** check box.
  - To label the slices of the pie chart with the percentage of the whole that they represent, select the **Show Slice Size** check box.
- **Custom Plot Labels** allows you to control the appearance of custom plot labels. You can add custom labels to your plot by pressing **CTRL** and clicking the plot. *For more tips on working with plots, see page 409 in Chapter 12.*
  - **Delete Labels** deletes all custom labels on the plot.
  - **Reset Labels** immediately resets all custom labels on the plot to use the settings specified via the **Set Font** button. This allows you to apply the settings to existing custom labels rather than just new labels created after changing the settings.
- **Bar Label Position** (available only for bar charts) allows you to specify whether you want the bar labels located within the bars or outside of the bars.

For all labels, you can click the **Set Font** button in the section to open the Font window, which allows you to set the font type, style, size and color for labels of that type.

13.4 Legend Page

The Legend page allows you to customize the display of the legend on the current plot.



The options on this page will vary depending on the plot style you are working with. All available options are presented below.

- To display the legend on the plot, select the **Show Legend** check box.
  - To include on the legend a color definition for each point/line/bar/slice shown in the plot, select the **Show Plot Items** check box.
  - To include lines above and below the legend and the user information, select the **Show Legend Border** check box. Use the options to the right to modify the appearance of the border. To change the color of the border, click the **Color** box to the left of the **Thickness** box. Adjust the thickness of the border by entering a positive integer in the input box. Change the style of the line (e.g., solid, dashed, etc.) by clicking the **Line Style** box to the right of the **Color** box.

- To display the text of the legend in the same color as the lines, points, bars and/or slices on the plot, select the **Use Item Color** check box. If not selected, the text will be red.
- **Show Analysis Information** if selected, the legend will display information about the analysis shown on the plot, including the data source and/or the settings used in the calculation, if applicable.
- To display the plot description (including the plot type) on the plot, select the **Show Plot Description** check box.
- To show the user display name, company and the date and time the plot was generated on the plot, select the **Show User Information** check box. (You set the display name and company using the User Login and Contact Information Window by choosing **My Portal > Users > My Profile**.)

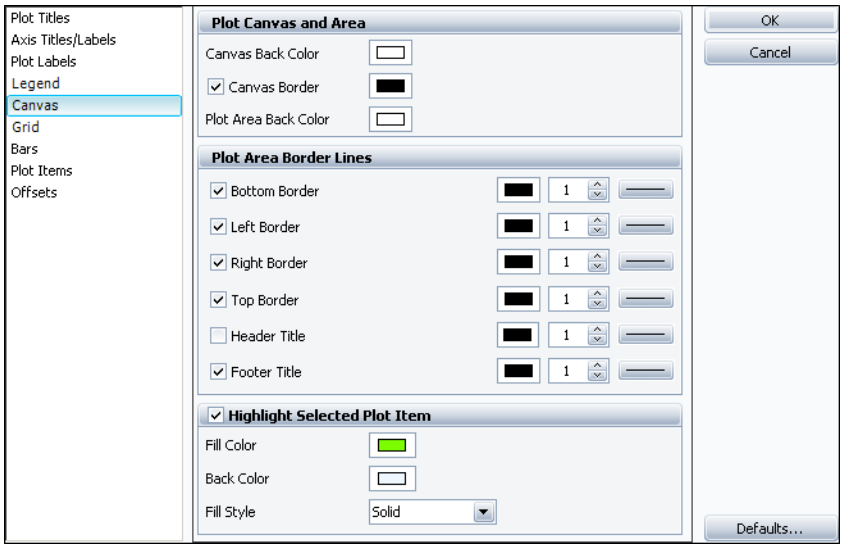
- The legend area allows you to specify the amount of space used for the legend.
  - **Fixed Legend Width** if selected, the legend will remain the same size regardless of the information it contains. You can set the percentage of the canvas width that the legend will occupy.
  - **Maximum Legend Width** if selected, the legend will automatically recalculate its width based on the information it contains. You can set the maximum percentage of the canvas area that the legend will occupy.

Click the **Set Font** button next to the corresponding option to open the Font window, which allows you to set the font type, style, size and color of the text.

- **Color Spectrum** allows you to set the color limits for the plot. This option is available only for bar and pie charts.
  - You can set both lower and upper color limits by dragging the markers on the scale between 0% and 100%.
  - **Full Spectrum Scale** if selected, the legend will show only the values between the upper and lower limits that you have specified; if not selected, the legend will show the default range of values.

13.5 Canvas Page

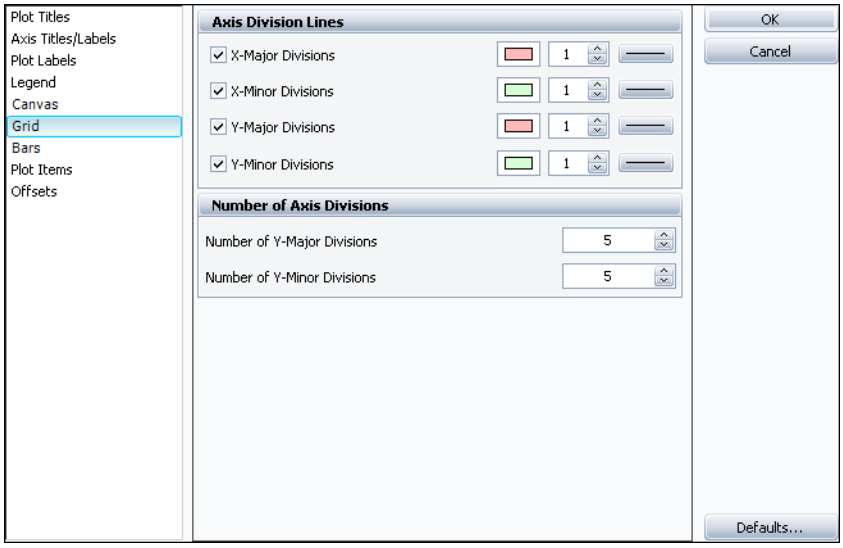
The Canvas page allows you to customize the color, style and thickness of the plot borders and title lines, along with the appearance of the rest of the plot sheet.



- **Plot Canvas and Area**
  - **Canvas Back Color** allows you to choose the color for the area outside the plot (i.e., the area containing the titles, labels, legend, etc.).
  - **Canvas Border** allows you to determine if the border around the plot field will be displayed and, if so, to set the color.
  - **Plot Area Back Color** allows you to choose the color for the plot background.
- **Plot Area Border Lines** allows you to hide or show each of the specified lines, as well as to choose a color, a thickness and a style for each. The borders referred to in this area are the borders of the plot area itself (i.e., the bottom border is the x-axis, etc.).
  - To show or hide a line, select or clear the associated check box.
  - To change the color of any of the lines, click the **Color** field. The drop-down list that appears allows you to choose from custom colors, web-safe colors or the colors used in the current Windows system settings. You can add colors to the Custom page by right-clicking one of the color boxes in the bottom two rows.
  - You can adjust the thickness of a line by entering a positive integer in the input box. A value of 1 will draw the thinnest possible line and other values such as 2, 3, etc. will draw thicker lines. Note that the size of the thinnest possible line is dependent upon your screen resolution.
  - In addition, you can change the style of each line by clicking the **Line Style** box to the right of the **Thickness** box. A list providing line style options (e.g., solid, dashed, etc.) will appear. Choose the line style you want and it will appear in the **Line Style** box.
- To highlight each object (i.e., point, line, bar or slice) on the plot as you point to it, select the **Highlight Selected Plot Item** check box. When an object is active, you can click it to open the Plot Setup window and edit the properties specific to that object. The remaining options in this area are available only if this option is selected.
  - **Fill Color** allows you to specify a foreground color to be used on the highlighted active object.
  - **Back Color** allows you to specify a background color to be used on the highlighted active object. This color is not used if the selected fill style does not use a background color (i.e., solid or transparent fill style).
  - **Fill Style** allows you to select a style of hatching to be used on the highlighted active object.

13.6 Grid Page

The Grid page allows you to customize the color, style and thickness of the grid lines and to set the number of axis divisions used. This page is not available when you are working with a pie chart.



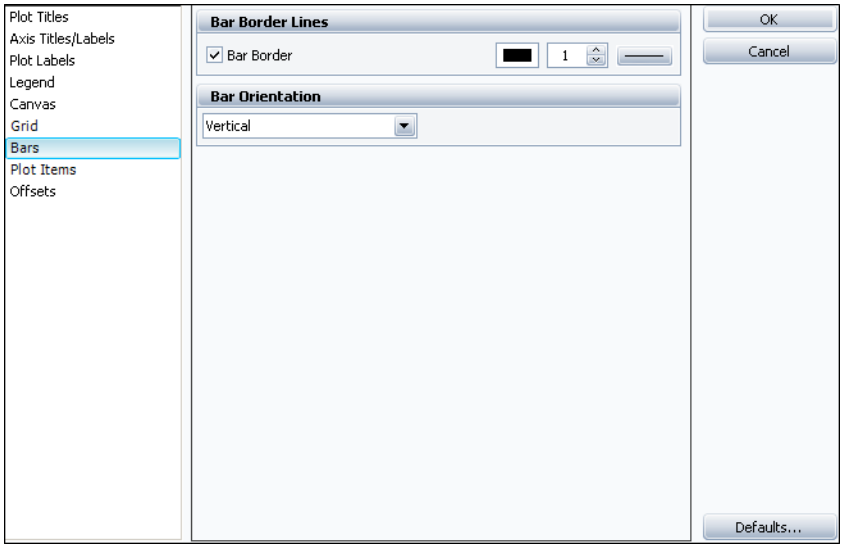
The options on this page will vary depending on the plot style you are working with. All available options are presented below.

- **Axis Division Lines** allows you to hide or show each of the axis division lines, as well as to choose a color, a thickness and a style for each.
  - To show or hide a line, select or clear the associated check box.
  - To change the color of any of the grid lines, click the **Color** box. The drop-down that appears allows you to choose from custom colors, web-safe colors or the colors used in the current Windows system settings. You can add colors to the Custom page by right-clicking one of the color boxes in the bottom two rows.
  - You can adjust the thickness of a grid line by entering a positive integer in the input box. A value of 1 will draw the thinnest possible line and other values such as 2, 3, etc. will draw thicker lines. Note that the size of the thinnest possible line is dependent upon your screen resolution.
  - In addition, you can change the style of each line by clicking the **Line Style** box to the right of the thickness box. A list providing line style options (e.g., solid, dashed, etc.) will appear. Choose the line style you want and it will appear in the **Line Style** box.
- **Number of Axis Divisions** allows you to set the number of major and minor divisions for each axis.

> **Note:** When you are working with a bar chart, the Grid page will contain settings for only the axis representing the dependent variable. This is affected by the orientation of your bar chart; if you are using vertical bars, this area will contain settings for the y-axis, and if you are using horizontal bars, it will contain settings for the x-axis. It is important to be aware that the settings for one axis will not transfer to the other axis when you change the bar orientation. You will need to return to this page and re-enter the settings.

13.7 Bars Page

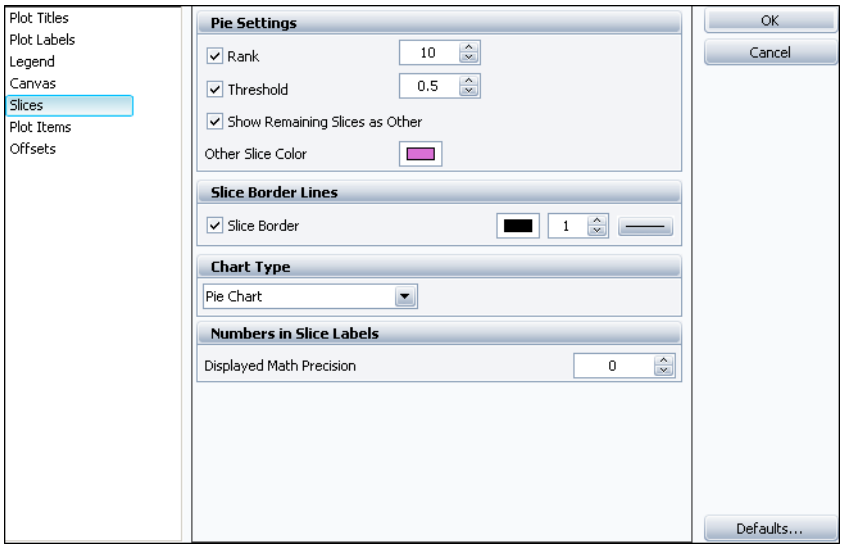
The Bars page allows you to customize the appearance of the bars in the plot. This page is available only when you are working with a bar chart.



- **Bar Border Lines** allows you to hide or show the border line for the bars, as well as to select a color, a thickness and a style for the border.
- **Bar Orientation** allows you to select how you want the plot laid out. You can choose to use vertical bars or horizontal bars.

13.8 Slices Page

The Slices page allows you to customize the appearance of the slices in the plot. This page is available only when you are working with a pie chart.



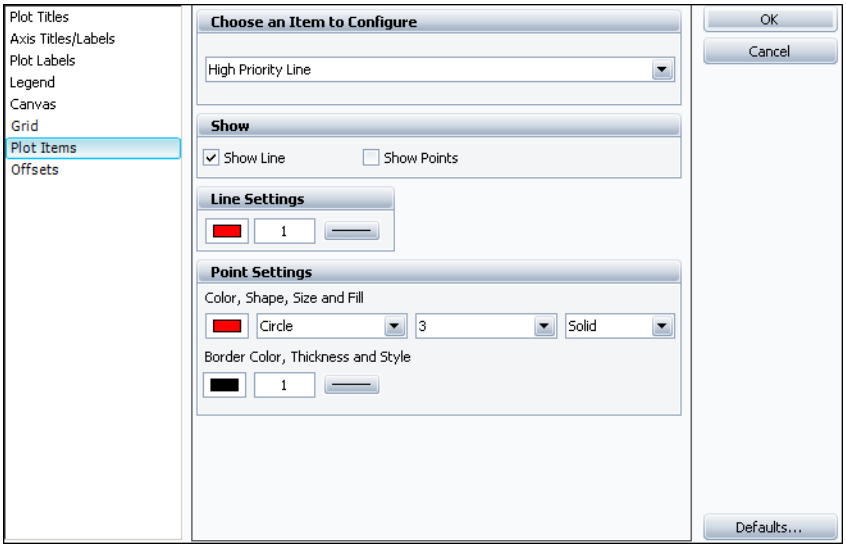
- **Pie Settings**
  - **Rank** if selected, you can specify the ranking of the slices that you want to view. For example, entering **5** will cause only the five largest slices to be shown in the plot.

- **Threshold** if selected, you can specify a minimum size (in percentage of the whole) for the slices that you want to view. For example, entering **0.1** will cause only slices accounting for ten percent or more of the whole to be shown in the plot.
- **Show Remaining Slices as Other** if selected, all remaining slices that do not meet the rank or threshold criterion specified will be shown in the plot as "other." This option is available only if you have specified a rank, a threshold or both.
- **Other Slice Color** allows you to choose the color used to represent "other" slices. This option is available only if you have selected the **Show Remaining Slices as Other** option.

- **Slice Border Lines** allows you to hide or show the border line for the slices, as well as to select a color, a thickness and a style for the border.
- **Chart Type** allows you to select how you want the plot laid out. You can choose to use an area chart (blocks), cake chart (layers) or pie chart (wedges).
- **Numbers in Slice Labels** allows you to set the number of decimal places to be displayed in slice labels.

13.9 Plot Items Page

The Plot Items page allows you to specify details of the appearance of the points, lines, bars and/or slices on the plot.



The options available on this page will vary depending on the plot style that you are currently working with. The image above shows the options for a line plot.

The following options are available for all plot styles:

- **Choose an Item to Configure** allows you to choose which plot element the settings on this page apply to.
- **Show** allows you to select whether or not the plot element you currently chose to configure will be shown on the plot. Depending on the plot style that you are currently working with, you may select to show the bars, slice, line and/or points. This option is not available for all plots.

For bar charts and pie charts, the following option applies.

- **Area Fill Color and Style** allows you to specify the appearance of the bar or slice representing the selected item. To change the color of the bar or slice, click the Color box. You can select a fill style from the drop-down list.

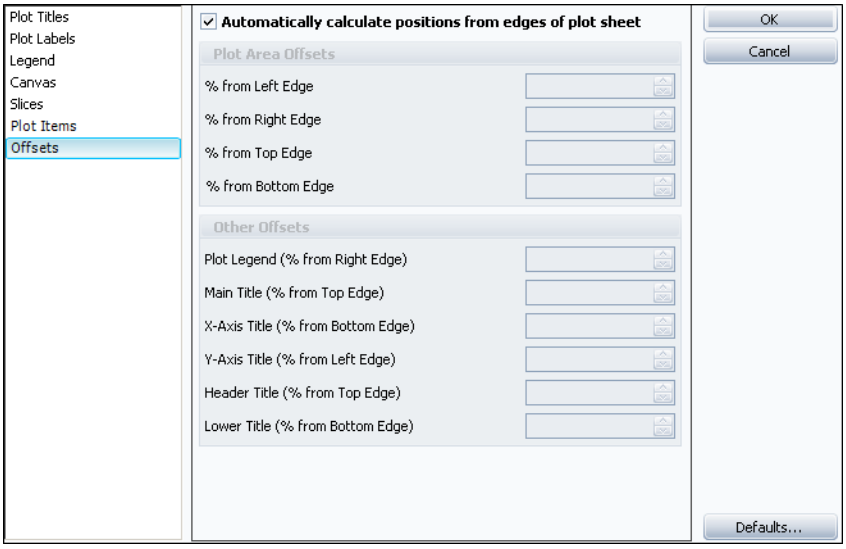
For line plots, the following options apply:

- **Line Settings** allows you to specify the appearance of the line representing the selected item. To change the color of the line, click the **Color** box. Adjust the thickness of the line by entering a positive integer in the input box. Change the style of each line such as solid, dashed, etc., by clicking the **Line Style** box.
- **Point Settings**
  - **Color, Shape, Size and Fill** allows you to specify the appearance of the point body. To change the color of the point, click the Color box. You can select a point shape from the drop-down list, select the size of the point (with 1 being the smallest point) and select a fill style from the drop-down list.
  - **Border Color, Thickness and Style** allows you to modify the appearance of the border of the points. To change the color of the border, click the Color box. Adjust the thickness of the border by entering a positive integer in the input box. Change the style of each line such as solid, dashed, etc., by clicking the **Line Style** box.

  Certain point shapes are drawn using only the border color. These include minus, pike, plus and x-cross.

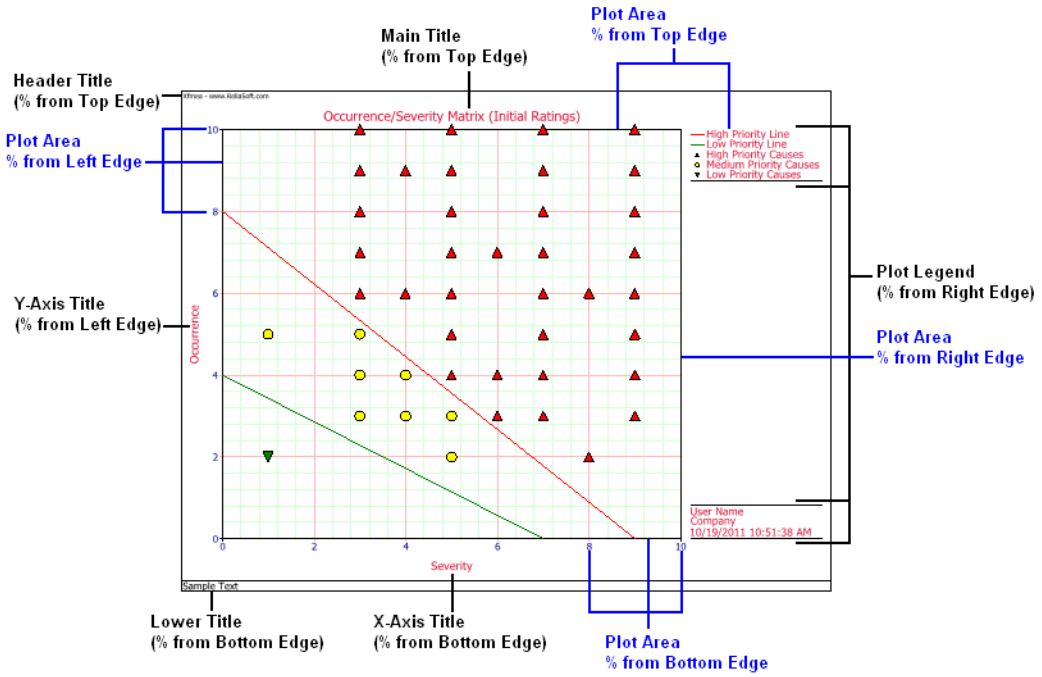
13.10 Offsets Page

The Offsets page allows you to specify the distance of various components from the edge of the plot area.



Automatically calculate positions from edges of plot sheet if selected, automatically calculates the offsets for maximum plot display and readability. If not selected, you can manually specify the offsets of the plot's left, right, top and bottom edges; the offset of the legend from the right edge of the plot area; and the offset of the main, x-axis, y-axis, header and footer titles. The offset values are set as a percentage of the canvas area.

For your reference, the following picture shows which plot elements are affected by each of the offset settings. (Note that the plot is generated from Weibull++, but the settings are the same for all Synthesis desktop applications.)



13.11 Plot Defaults Window

Each time you open the Plot Setup window, the changes that you make will apply only to the current plot unless you specify otherwise. To specify settings to be used as defaults for all subsequent plots or to re-apply default settings to the current plot, click the **Defaults** button to open the Plot Defaults window.

The Plot Defaults window gives you the flexibility to set the default settings for the three main areas of a plot: the plot titles, the plot item settings and the general display areas. You can choose to separately save, load or restore the default settings of each main area or you can set the default settings for all areas at once.

The Plot Defaults window consists of nine pages:

- All Defaults page. This page controls the settings for all the other plot setup pages. When you click a button on this page, the effects apply to all pages. This allows you to quickly make changes without having to individually change each of the other pages.
- Titles Text page. This page allows you to customize default plot titles for each plot type without having to view a plot of that type. For example, you can display an RD Questions plot while changing the titles for a Control Type plot. *See Section 13.11.1 on page 423*.
- Plot Items Display page. This page allows you to customize the details of the default appearance of the lines, points, bars and/or slices of every plot type without having to individually change each plot. *See Section 13.11.2 on page 424*.
- General page. This page controls the settings for the five section pages below it. When you click a button on this page, the effects apply to the Titles Display, Labels, Legend, Canvas and Grid and Offsets pages.
- The Titles Display, Labels, Legend, Canvas and Grid and Offsets pages control the settings of their individual sections. When you click a button on one of these pages, the effects apply only to that specific section.

Three Defaults buttons appear on every page of the Plot Defaults window. The scope of their effects differs slightly depending on the page you are working with.

- **Save Defaults** saves settings for use as the default settings for all subsequent plots. This will overwrite the previous default settings.
- **Load Defaults** enters the saved default values for the settings. You can then click **OK** in the Plot Setup window to apply the default settings to the current plot. This is an easy way to undo changes you have made in the Plot Setup window and re-apply default settings.
- **Restore Defaults** clears the saved default settings and restores the default values that are shipped with the application.

The scope of the Defaults buttons' effects changes as follows:

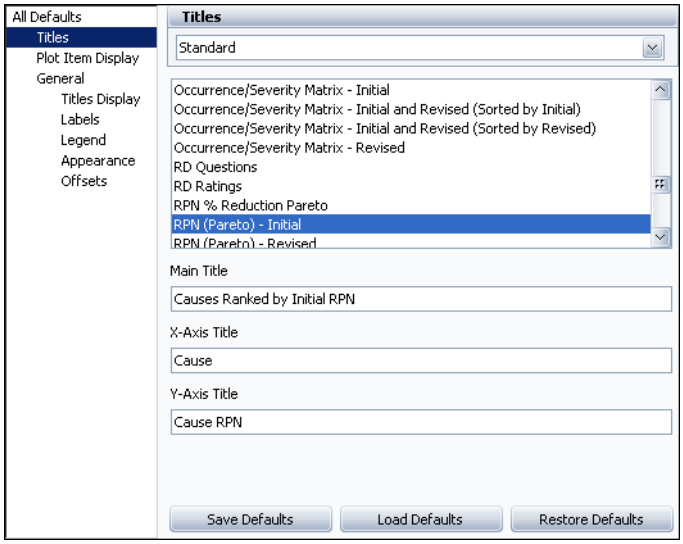
- For the Titles and Plot Item Display pages, these buttons save/load/restore the settings specified on the current page.
- For the Titles Display, Labels, Legend, Canvas and Grid and Offsets pages, these buttons save/load/restore all current settings on the corresponding page(s) of the Plot Setup window (e.g., clicking **Save Defaults** on the Canvas and Grid page of the Plot Defaults window will save the settings from the Canvas, Grid, Bars and Slices pages).

> **Note:** The settings affected by the Defaults buttons on the Titles Display, Labels, Legend, Canvas and Grid and Offsets pages include all settings relevant to the section, regardless of whether they are currently available or not. For example, if you have previously specified settings for a pie chart and are now specifying settings for a bar chart, the settings that you created for the pie chart will be saved as defaults when you click **Save Defaults** and will be applied to subsequent pie charts.

- For the All Defaults and General pages, clicking these buttons is equivalent to clicking them on all of the subordinate pages.

13.11.1 Plot Defaults: Titles Page

The Titles Text page of the Plot Defaults window allows you to customize default plot titles for each plot type.

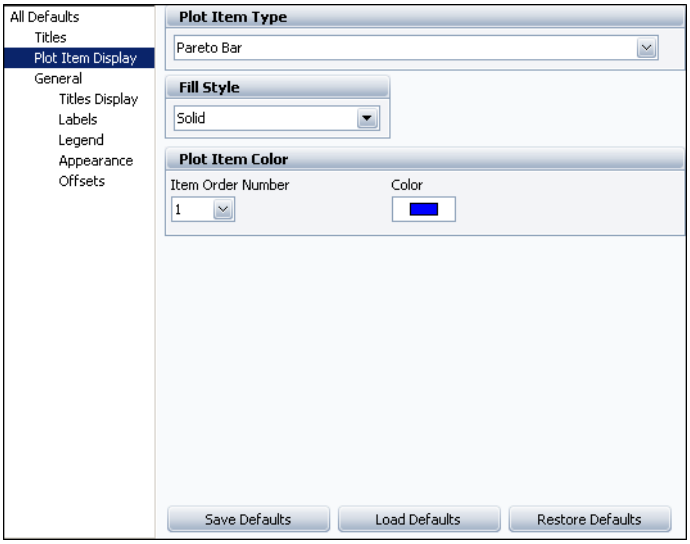


- **Titles** allows you to select the plot type that you are changing the default titles for.
- **Main** allows you to type the default main title for the plot.

- **X-Axis** allows you to type the default title for the x-axis for the plot.
- **Y-Axis** allows you to type the default title for the y-axis for the plot.

13.11.2 Plot Defaults: Plot Item Display Page

The Plot Items Display page of the Plot Defaults window allows you to customize details of the default appearance of the lines, points, bars and/or slices on the plot.

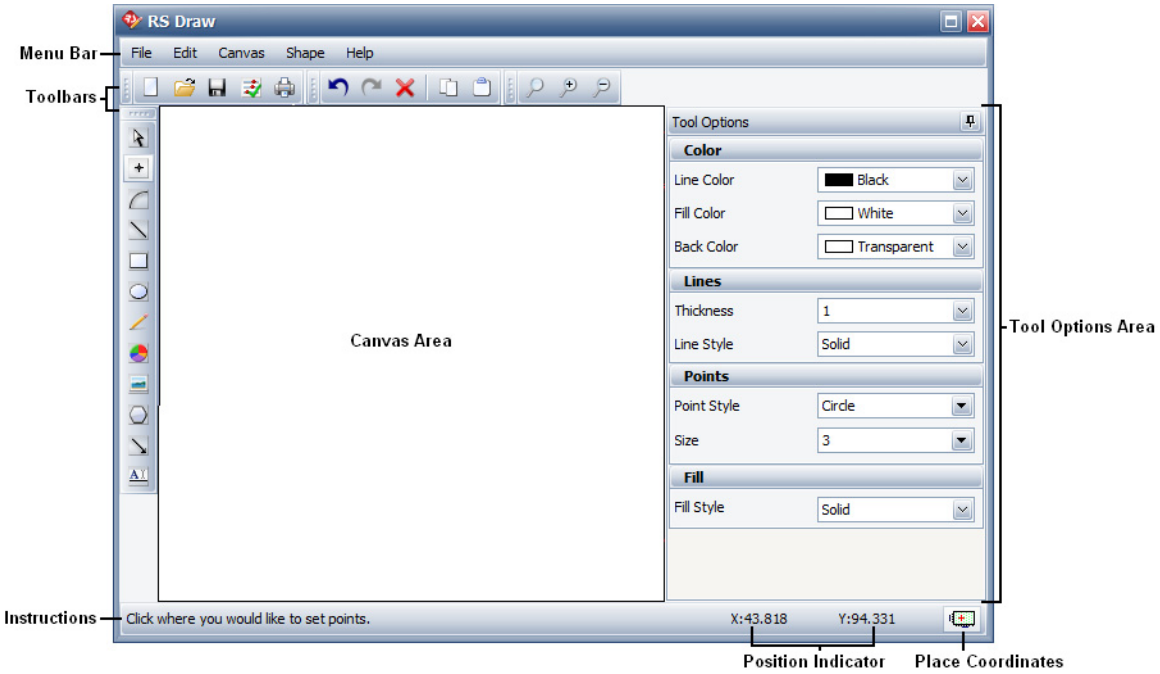


Plot Item Type allows you to select the style of plot that you want to specify default settings for. The settings you specify are retained in the background when you choose a new item in this list, so you can specify settings for each type and save all of them, if desired.

The options on this page are identical to those on the Plot Items page of the Plot Setup window for the corresponding plot style.

ReliaSoft Draw 14

ReliaSoft Draw (RS Draw) is a full graphics metafile editor that allows you to annotate and customize your plots. With RS Draw, you can insert text, draw an object, mark a particular point or paste another picture into your plot. You can also re-arrange your objects by selecting and moving them to the position you desire. The figure below shows the RS Draw interface and its components.



14.1 Linking RS Draw

RS Draw is available by clicking the **RS Draw** icon on the Plot control panel.

The plot will be linked to RS Draw and will appear in the canvas area for editing. You will then be able to save any changes made to the plot in RS Draw to the current plot sheet by selecting **File > Apply Changes and Close** or by clicking the **Apply Changes and Close** icon on the toolbar.

14.2 RS Draw Menu Bar

The menus in the RS Draw menu bar belong exclusively to the RS Draw utility. Each menu item and its subitems are discussed next.

14.2.1 File Menu

The RS Draw File menu contains the following commands.

- **New** creates a new, blank RS Draw plot area. Only one canvas can be open in RS Draw at a time.
- **Open** opens a previously saved RSDrawCanvas (*.rdc) graphic.
- **Save** saves the current image in one of five formats: JPEG graphic (*.jpg), CompuServe Graphics Interface (*.gif), Portable Network Graphics (*.png), Windows Meta File (*.wmf) or RSDrawCanvas (*.rdc) graphic. If the graphic has never been saved, you will be prompted for a filename.
- **Apply Changes and Close** exports the current annotated graph to the current Plot Sheet.
- **Print** opens the Print window, which allows you to send the image to the printer.
- **Exit** closes RS Draw.

14.2.2 Edit Menu

The RS Draw Edit menu contains the following commands.

- **Undo** cancels the previous action. You can cancel multiple actions by choosing Undo as many times as necessary.
- **Redo** reapplies the previously canceled action. You can redo multiple actions by choosing Redo again.
- **Cut** cuts the selected object(s) and puts them on the Clipboard.
- **Copy** copies the selected object(s) to the Clipboard. Objects stored in the Clipboard can be pasted into this and other applications.
- **Copy Image** allows you to copy the plot to the Clipboard as a graphic. You must select an image type using one of the following options.
  - Select **As Bitmap** when you need a bitmap version to be used in an external application.
  - Select **As Metafile Optimized for Synthesis Spreadsheet** when you will be copying and pasting the image into Synthesis spreadsheets (e.g., analysis workbooks).
  - Select **As Metafile Optimized for External Use** when you need a metafile that will be used by an external application.
- **Paste** pastes the contents of the Clipboard onto the canvas area.
- **Delete Selected** deletes the selected object(s).
- **Select All** selects all objects in the plot area.
- **Preferences** opens the Preferences window where you can set various options about the RS Draw interface.

14.2.2.1 Preferences Window

The Preferences window allows you to set various options about the RS Draw interface. You can access it by choosing **Edit > Preferences**.

The window contains the following settings:

- **Hot Spots**
  - **Point** allows you to choose the color of the point handles that appears when a line, free-form line or polygon is selected. These handles represent the end points of the line segments that are used to draw the object, and can be moved independently of each other to reconfigure the object.
  - **Object** allows you to choose the color of the object handles that appear when an object is selected. They are used to move or resize the entire object.
- **View**
  - **Instructions** if selected, turns on the help text that displays in the Status Bar at the bottom of the RS Draw interface when you click an option on the Draw Toolbar.
- **Grid Spacing** sets the grid's width. The grid spacing is a percentage of the grid's width. For example, if you want the canvas to show 20 points, the grid spacing must be set to 0.05.

14.2.3 Canvas Menu

The RS Draw Canvas menu contains the following commands.

- **Background Color** opens the Color Palette, which allows you to choose the canvas color.
- **Show Grid** toggles the display of the dotted-line grid on and off.
- **Snap to Grid** sets the movement and placement of the selected object to be in accordance with the spacing of the grid points. In other words, each object moved or placed in the graphic will be automatically moved to align with the nearest grid lines. Note that this command is available only when the grid is displayed.
- **Zoom** opens a submenu that allows you to set the degree of magnification on the RS Draw canvas.

14.2.4 Shape Menu

The RS Draw Shape menu contains the following commands.

- **Move to Front** moves the selected object(s) to the front (i.e., displayed over the other layers).
- **Move to Back** moves the selected object(s) to the back (i.e., displayed behind the other layers).
- **Bind Group** groups the selected objects together as one. To ungroup the select objects, use the **Unbind Group** command.
- **Clone** creates a duplicate of each currently selected object and places it next to the original.
- **Align Vertically** opens a submenu that allows you to vertically align the selected object(s): Left, Center or Right. When aligning objects within the graphic, the first selected object selected is used as the reference point.
- **Align Horizontally** opens a submenu that allows you to horizontally align the selected object(s): Top, Center or Bottom. When aligning objects within the graphic, the first selected object selected is used as the reference point.
- **Rotate Right** and **Rotate Left** rotate the selected object(s) 90º in the indicated direction. To flip the selected object(s) by 180º, select **Rotate 180º**.

14.2.5 Help Menu

The RS Draw Help menu contains the following commands.

- **Contents** displays RS Draw's online help contents.
- **Visit ReliaSoft on the Web** allows you to have direct access to ReliaSoft's website at http://www.ReliaSoft.com.

14.3 RS Draw Toolbars

Four toolbars are available on the RS Draw window. Each toolbar contains quick access icons that activate different parts of the application. Depending on the current active window or type of data, certain icons may be unavailable and will appear dimmed. You can choose which toolbars you want to appear in the interface by customizing the toolbars. *See Section 14.3.5 on page 430*. The toolbars and their icons are shown and explained next.

14.3.1 Standard Toolbar

New creates a new, blank canvas in RS Draw. Only one canvas can be open in RS Draw at a time.

Open opens a previously saved RSDrawCanvas (*.rdc) graphic.

Save saves the current graphic. If no name is currently selected, you will be prompted for a filename.

Apply Changes and Close applies the changes in the annotated graph to the current Plot Sheet and closes the RS Draw window.

Print opens the Print window, which gives you printing options and allows you to send the current graphic to the printer.

14.3.2 Edit Toolbar

Undo cancels the previous actions. You can cancel multiple actions by choosing **Undo** again.

Redo reapplies the previously canceled action. You can redo multiple actions by choosing **Redo** again.

Delete deletes the selected object(s).

Copy copies the selected object(s) to the Clipboard. Objects stored in the Clipboard can be pasted into this and other applications.

Paste pastes the contents of the Clipboard onto the canvas area.

14.3.3 Zoom Toolbar

Reset Zoom sets the degree of magnification, the zoom, to 100%.

Zoom In increases the degree of magnification.

Zoom Out decreases the degree of magnification.

14.3.4 Draw Toolbar

Select Object allows you to select an object. When an object is selected, you can move it to a new position, change its properties or delete the object. (To select multiple objects, hold down the **CTRL** key while you click the other objects you want to select.)

Point allows you to place a point on the canvas. With the tool selected, click the desired location on the canvas. The point displays at that location.

Arc allows you to draw an arc. With the tool selected, click the desired location on the canvas and drag the arc to the appropriate size.

Line allows you to draw a line. With the tool selected, click the desired start point for the line and drag to the desired length.

Rectangle allows you to draw a rectangle. With the tool selected, click the desired location on the canvas and drag the rectangle to the appropriate size.

Ellipse allows you to draw an ellipse. With the tool selected, click the desired location on the canvas and drag the ellipse to the appropriate size.

Free allows you to draw a free-form line. With the tool selected, click the desired start point for the line and draw the line. You can continue to draw until you release the mouse button.

Pie allows you to draw a pie segment. With the tool selected, click the desired location on the canvas and drag the pie segment to the appropriate size.

Image allows you to insert an image onto the canvas. You can import files of the following types: jpg, gif, png, bmp, wmf or emf. With the tool selected, click the desired location on the canvas. The Open window displays. Navigate to the appropriate folder, then choose the image and click **Open**. The image appears at the desired location.

Polygon allows you to draw a polygon. With the tool selected, click the start location on the canvas, and then click each additional end point of a line segment. As soon as you have three points an additional line is automatically drawn, closing the shape, but you can continue to add points and the polygon will be redrawn accordingly. To stop adding points, right-click the canvas area.

Arrow allows you to insert an arrow into the plot. With the tool selected, click the desired start point for the arrow line and drag to the desired length. The arrow head is automatically added at the end point of the arrow line.

Text allows you to insert a single line of text into the plot. With the tool selected, click the start point and type the text.

For all objects, you can change and set the object's properties using the Tool Options area. *See Section 14.5 on page 431*. Note that you can use the Tool Options area in two ways:

- If an existing object on the canvas is selected, changing the settings in the Tool Options area will alter that object's properties only.
- If no object on the canvas is selected, the settings in the Tool Options area will be applied to all subsequent objects created using the selected tool.

14.3.5 Customizing Bars in the RS Draw Interface

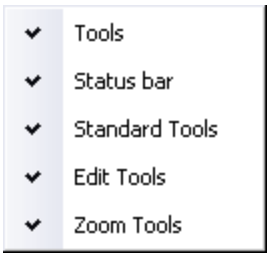
RS Draw's interface provides you with the flexibility to customize the workspace to meet your needs. You can hide or display any of the toolbars. You can also change the location and shape of any of the toolbars.

14.3.5.1 Moving Toolbars

You can move any of the toolbars by clicking the handle at the top or left side of each toolbar and dragging the bar into the position you prefer. When you move a bar, the shape may change to fit the new location.

14.3.5.2 Hiding and Showing Toolbars

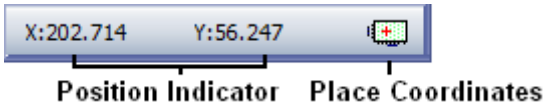
You can choose to display or hide any of RS Draw's toolbars by right-clicking inside the grey toolbar area. A Toolbar shortcut menu like the one shown next will appear.



Click a toolbar name to select or deselect it. If selected, the toolbar will appear in the RS Draw Interface. If not selected, the toolbar will not appear.

14.4 Additional Tools

Two additional tools are located on the bottom of the RS Draw window.



- **Position Indicator** indicates the exact position of the pointer within the plot area. The Position Indicator is available only if RS Draw is linked to a data set.
- **Place Coordinates** if selected, allows you to mark a point on the plot area by right-clicking at the spot you would like to mark. This option is available only if RS Draw is linked to a data set.

14.5 Object Properties — The Tool Options Area

You can set the properties of objects in RS Draw using the Tool Options area.

The Tool Options area includes the Lines Properties, the Fill Properties, the Points Properties, the Angles Properties, the Text Properties and the Color Properties. The properties that will be available will vary depending on the selected object (text, line or a shape).

The functionality of this area changes depending on what is selected on the Draw toolbar. If the Select tool is active, any changes you make are applied to the currently selected object. If any other tool is active, then any changes you make will be used as the properties of that tool the next time you use that tool to draw something.

For example, to set the default options for all rectangles, you would first click the **Rectangle** icon and, in the Tool Options area, set the property values that will be applied to all further instances of the items of that type.

To change the properties of an existing object, first click the **Select** icon, then select the object and change the desired properties in the Tool Options area. The changes are applied to the selected object immediately after you make them. To undo the change, choose **Edit > Undo**.

14.5.1 Lines Properties

The Lines properties area of the Tool Options area is available when a point, arc, line, rectangle, ellipse, free-form line, pie, polygon or arrow is the selected object. Note that when working with an object other than a line or a free-form line, these properties apply to the object's outline.

The following options are available:

- **Thickness** allows you to set the line thickness, in pixels, for the selected line. The line thickness can range from 1 to 9, with 1 being the thinnest and 9 being the thickest.
- **Line Style** allows you to choose a style for the selected line from the drop-down menu.

14.5.2 Fill Properties

The Fill properties area of the Tool Options area is available when a point, rectangle, ellipse, pie, polygon or arrow is the selected object.

The following option is available:

- **Fill Style** allows you to choose a fill style from the drop-down menu. The setting determines how the Fill and Back colors interact on the display. Note that if you choose **Transparent** then neither color displays.

14.5.3 Points Properties

The Points properties area of the Tool Options area is available when a point is the selected object.

The following options are available:

- **Point Style** allows you to set the style for the point from the drop-down menu.
- **Size** allows you to set the point size from the drop-down menu.

14.5.4 Angles Properties

The Angles properties area of the Tool Options area is available when an arc or pie is the selected object.

The following options are available:

- **Start** allows you to set the starting point for the object, where 0 is a horizontal line. The Start value determines the orientation of the object. For example, a Start value of 0 indicates that the object will start at the 3 o'clock position, while a Start value of 270 indicates that the object will start at the 12 o'clock position.
- **Sweep** allows you to set the angle of the object. The Sweep value determines how big the pie slice or arc angle is. For example, a Sweep value of 45 indicates that the object is one-eighth of a circle, while a Sweep value of 270 indicates that the object is three-quarters of a circle.

14.5.5 Text Properties

The Text properties area of the Tool Options area is available when text is the selected object.

The following options are available:

- **Font** allows you to choose the specific font.
- **Font Size** allows you to set the text point size.
- **Orientation** allows you to set the angle at which the text is displayed. For example, a value of 0 indicates that the text will start at the 3 o'clock position, while a value of 90 indicates that the text will start at the 12 o'clock position.
- **Bold** if selected, bolds the text.
- **Italics** if selected, italicizes the text.
- **Underline** if selected, underlines the text.
- **Vertical Alignment** allows you to set the text's vertical alignment relative to the insertion point. The options are: Left, Center or Right.
- **Horizontal Alignment** allows you to set the text's horizontal alignment relative to the insertion point. The options are: Top, Center or Bottom.

14.5.6 Color Properties

The Color properties area of the Tool Options area is available when a point, arc, line, rectangle, ellipse, free-form line, pie, polygon, arrow or text is the selected object.

The following options are available:

- **Line Color** opens the Color Palette, which allows you to choose a color for the selected line.
- **Fill Color** opens the Color Palette, which allows you to choose the fill color of the selected object from the drop-down menu. **Transparent** indicates that there is no fill color.

Note: If the selected object is text, the Fill Color sets the font's color. The Line Color and Back Color settings are not used with text.

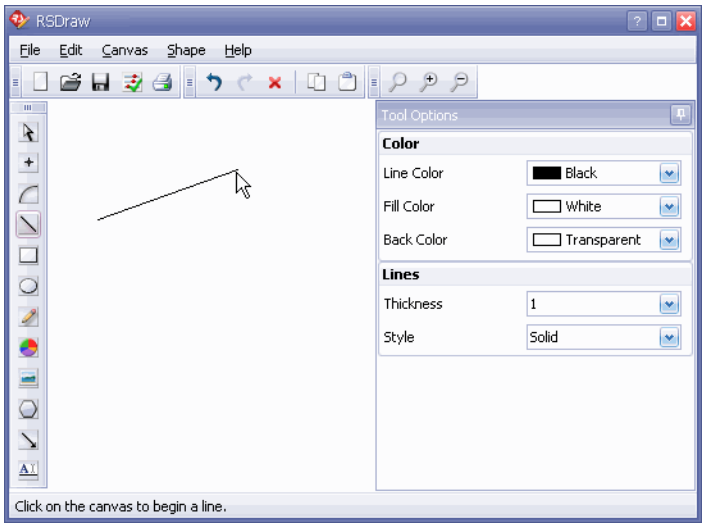
- **Back Color** opens the Color Palette, which allows you to choose the background color for the selected object from the drop-down menu. **Transparent** indicates that there is no background color.

14.6 Working with Objects: Examples

14.6.1 Drawing a Line

To draw a line on a graphic that is being edited in RS Draw, click the **Line** icon on the Draw toolbar. (The other RS Draw tools are used in a similar fashion.)

On the canvas, click the desired start point for the line and drag to the desired orientation and length. A line will appear, as shown next.



14.6.2 Editing an Object

To edit an object in RS Draw, click the **Select** icon on the Draw toolbar and then click the object. Handles appear on the object. These handles allow you to move and/or resize the object. The number and type of handles depends on the selected object. For more information about the handle types and how to set the handle colors, refer to the Preferences Window. *See Section 14.2.2.1 on page 427.*

14.6.2.1 Example: Editing a Line

When you click a line, three handles will appear on the line: one point handle at each end of the line and one object handle in the middle of the line.

Drag one of the point handles to edit the line's direction and length with respect to the other end handle. You can move the pointer around to get a feel for what you can do. When you are satisfied with the new line, release the button.

To move the line, drag the object handle to the appropriate place.

14.6.2.2 Example: Resizing a Rectangle

When you click a rectangle, five object handles will appear on the rectangle: one at each corner and one at the rectangle's center. To resize the rectangle, click one of the corner handles and move the pointer. When you are satisfied with the new size, release the button.

To move the rectangle, drag the central handle to the appropriate place.

14.7 Saving an Image

You can save an image for future use either in RS Draw or in another application.

Note: If you want to edit the image in RS Draw again, you must save the image as an RSDrawCanvas (*.rdc) graphic image. Otherwise, you will not be able to open the graphic file in RS Draw.

To save an image, do the following:

- Choose **File > Save**. The Save As window displays.
- In the **File name** field, type the image's name.
- In the **Save as type** field, choose the image type.
- Click **Save**.

System Improvements 15

BlockSim offers two tools to help you plan improvements to your system:

- **Allocation analysis** helps to decide how the system's reliability can best be improved, based on the reliability importance of the blocks selected for consideration
- The **Optimum Replacement window** allows you to determine the most cost-effective time to replace one or more blocks in one or more systems, based on their planned and unplanned replacement costs.

15.1 Allocation Analysis

The allocation analysis tool can help you determine the reliability requirements for individual components/assemblies that would be needed to achieve an overall reliability goal. To use the tool, you specify a target reliability for the system at a particular time and select which of the individual blocks in the system you would like to consider for improvement. You then choose the allocation type or method that you want to use and perform the optimization.

What's Changed? In BlockSim 7, this kind of system optimization was performed by selecting certain blocks to be considered in the calculation and specifying their maximum achievable reliability and the feasibility of achieving that reliability via their properties windows; the QCP was then used to perform the calculations. The allocation analysis tool provided in the Synthesis version allows you to select the blocks for consideration, specify the required values and perform the calculations in a single window, thereby making it simpler to set up and modify your calculations.

To add an allocation analysis to the project, choose **Insert > Tools > Allocation Analysis**.

In the Select Diagram window that appears, select the diagram that you want to perform an allocation analysis for. Allocation analysis can be performed only for analytical diagrams, so all of the diagrams that are available in this window will be analytical diagrams. You can select any diagram that is currently in an analyzed state; diagrams that have not been analyzed will be displayed in the list but cannot be selected. (You can hide these diagrams by selecting the **Hide unavailable diagrams** check box at the bottom of the window.) When the allocation analysis folio is created, the name of the diagram that it is based on will be displayed at the bottom of the control panel; you can click the diagram name to open it.

In the allocation analysis folio, you will see a table that displays the current reliability and the reliability importance of each of the blocks in the selected diagram. Reliability importance is a measure of the relative importance of each component in a system with respect to the overall reliability of the system. These values are calculated at the mission time specified in the control panel; you can change that value in the control panel and click the Refresh button to update the values in the table accordingly. The current reliability for the entire system is displayed in the **Outputs** area of the control panel, and is also dependent upon the specified mission time.

The color coding used in the Reliability Importance column provides a quick visual indicator of how important the blocks are to the system reliability. You can adjust the threshold for the colors by clicking the **Color Limits** field in

the control panel. Move the sliders to indicate the values below which you want everything to be displayed in green (i.e., least important) and above which you want everything to be displayed in red (i.e., most important). To automatically determine a min/max for the Color Limits based on the values in the Reliability Importance column of the current tab, click the **Calculate Color Limits** button.

Select the check box in the first column for each block that you want to consider for improvement in the allocation analysis. Any block that has a model assigned to it can be considered in the optimization process; standby and load sharing configurations are considered at the container/gate level, not at the contained block/dependent event level. You can select or clear the check box in the column header to select or clear all of the individual check boxes. The goal of an allocation analysis is to improve the reliability of the system; this purpose is best served if you do not select blocks with very low reliability importance or with very high current reliability values.

In the **Inputs** area of the control panel, enter the following information:

- The **Target Reliability** is the system reliability that you want to reach.
- **Time** is the mission time, or the time at which the optimization will be conducted. You must specify the units for this measurement. For example, if you want to determine the optimum scenario for increasing component reliability in order to achieve a system reliability of 0.98 at 100 days, you would enter **100** in the **Time** field and choose **Day** in the **Units** field.
- **Iterations** allows you to specify the maximum number of iterations of the optimization algorithm to be conducted in order to obtain a solution.

The software provides three allocation types that can be used to logically apportion the system reliability across the selected blocks. Select the desired allocation type in the **Allocation Type** field in the control panel. Available allocation types include:

- The **equal** allocation type achieves the specified target reliability for the system by determining a single target reliability value that all selected blocks must reach, regardless of their current reliability or reliability importance. If you select this allocation type, you do not need to enter any additional information for each block.
- The **weighted** allocation type allows you to specify a weight for each selected block, which is used in determining the block's target reliability. If you select this allocation type, you will need to enter a weight for each selected block. If all selected blocks are given the same weight, they will all have the same target reliability.
- The **cost optimized** allocation type allows you to meet the system's target reliability at the lowest possible cost. This allocation type determines the target reliability for each selected block based on the maximum reliability that can be achieved for the block and the relative difficulty or cost of increasing the block's reliability. If you select this allocation type, you will need to enter two additional values for each selected block:
  - In the Maximum Achievable Reliability column, enter the greatest reliability that the component can possibly reach at the mission time specified in the control panel. This value is a limiting reliability that the component will approach, but not reach. It should be based on engineering judgment and experience, taking into account the current state of the art of the component and other similar factors.
  - In the Feasibility column, specify the difficulty of increasing the component's reliability, relative to that of increasing the reliability of the rest of the components in the system. Depending on the design complexity, technological limitations, etc., certain components can be very hard to improve. Clearly, the more difficult it is to improve the reliability of the component, the greater the cost. In the drop-down for each cell in this column, there are two ways to specify this value. You can select to use a predefined feasibility value, in which case you simply move the marker on the scale to the desired location between Easy (1) and Hard (9). You can also

select to enter your own feasibility equation. If you choose to do this, the equation, or function, must meet the following requirements:

- The function must include a reference to the reliability of the component, which is denoted by an R.
- The function must return a positive value.
- The function must be described in notation similar to that used in Excel.

Since the evaluation is relative, any feasibility functions that you enter should be correlated with one another. In addition, you should not use your own functions for some components and the predefined feasibility values for others unless you are sure that your functions are correlated to the function used to calculate the predefined values.[1] For this reason, it is preferable to use the predefined values unless you have a compelling reason to do otherwise.

You can then click the **Optimize** icon.

Once optimization has been performed, the following information will be displayed for each selected block:

- The target reliability, which is the reliability of the component at the specified mission end time that would be required in the optimum scenario to increase component reliability to reach the specified reliability goal.
- Equivalent parallel units, which indicates the number of identical blocks that would be required in a parallel configuration for the particular block to meet the specified reliability goal, in lieu of increasing the component's reliability.

In addition, the **System Reliability** field in the **Outputs** area of the control panel will display the reliability of the system if the optimum scenario is implemented (i.e., if all selected blocks are improved to their target reliability values).

Note: Subdiagram blocks are displayed as hyperlinks. You can click the name of the subdiagram block to add another tab to the allocation analysis folio, where you can perform allocation analysis for the subdiagram.

You can click the **Send to Workbook** icon to save the results in an analysis workbook.

The Select Workbook window that appears allows you to select an existing analysis workbook or add a new one to the project. If you select an existing workbook, you can select the **Overwrite Existing Sheets** check box to overwrite the current contents of the workbook. If you do not select this check box, a new sheet will be added to the workbook. If you select to add a new workbook, you will be prompted for the workbook name.

You can click the **Send to Excel** icon to save the results as an Excel file.

In either an analysis workbook or an Excel file, the results will look like the ones shown next.

	A	B	C	D	E
1	Fault Tree (Equal)				
2	Block Name	RI (100)	Reliability (100)	Target Reliability (100)	N.E.P.U. *
3	Cause 1	0.9387	0.9689	0.9473	0.8481
4	Cause 2	0.9387	0.9689	0.9473	0.8481
5	Cause 3	0.9387	0.9689	0.9473	0.8481
6					
7					
8	System Results				
9	Reliability (100)	Target Reliability (100)	Units		
10	0.9095	0.8500	Hour		
11					
12					
13	* N.E.P.U.				
14	Number of Equivalent				
15	Parallel Units				
16	Indicates the number of blocks				
17	that would be required in a				
18	parallel configuration for this				
19	particular block, in lieu of				
20	increasing the block's reliability				

The first row displays the name of the optimized diagram and the allocation type that was used in the optimization. For each selected block, the following information is shown:

- The block name.
- Any inputs that you entered:
  - For the Equal allocation type, no block-level inputs are shown.
  - For the Weighted allocation type, the **Weight** for each block is displayed.
  - For the Cost Optimized allocation type, the **Maximum Achievable Reliability** and the **Feasibility** for each block are displayed.
- **RI (Time)** is the block's reliability importance at the specified mission time.
- **Reliability (Time)** is the block's reliability at the specified mission time if no improvements are made.
- **Target Reliability (Time)** is the block's goal reliability at the specified mission time in order to meet the target system reliability.
- **Equivalent parallel units** is the number of identical blocks that would be required in a parallel configuration for the particular block to meet the specified reliability goal, in lieu of increasing the component's reliability.

The System Results include the following:

- **Reliability (Time)** is the system's reliability at the specified mission time if no improvements are made.
- **Target Reliability (Time)** is the system's reliability at the specified mission time if the optimum scenario is implemented.
- **Units** displays the units that were specified for the mission time.

The ReliaWiki resource portal has more information on reliability allocation and optimum replacement at: http://www.ReliaWiki.org/index.php/Reliability_Importance_and_Optimized_Reliability_Allocation_(Analytical).

Watch a video example of allocation analysis at: http://www.ReliaSoft.tv/blocksim/appexamples/blocksim_app_ex_2.html.

15.1.1 About Reliability Importance

Reliability importance is a measure of how much effect each component has on the overall reliability of the system. One simple way to demonstrate this concept is to look at a series system. In general, the least reliable component in a series system has the greatest effect on the reliability of the system.

Consider three components arranged reliability-wise in series, where R1 = 70%, R2 = 80% and R3 = 85% (for a given time). In the table shown next, we can examine the effect of each component's reliability on the overall system reliability.

Component 1	Component 2	Component 3	System
0.7	0.8	0.85	0.476
0.8	0.8	0.85	0.544
0.7	0.9	0.85	0.536
0.7	0.8	0.95	0.532

The first row of the table shows the given reliabilities for each component and the corresponding system reliability. In each of the following rows, the reliability of one component has been increased by 10% while keeping the reliabilities of the other components at the given values. This allows us to observe the effect of each component's reliability on the overall system reliability. It is clear that the highest value for the system's reliability was achieved when the reliability of Component 1, which is the least reliable component, was increased by a value of 10%. In other words, Component 1 has a higher reliability importance.

In simple systems such as this series system configuration, it is easy to identify the weak components. However, in more complex systems this becomes quite a difficult task. The value of the reliability importance depends both on the reliability of a component and its position in the system. BlockSim calculates this value automatically.

Mathematically, reliability importance is the partial derivative of the system reliability equation with respect to a component's reliability. The reliability importance, IR, of component i in a system of n components is given by:

$$I_{R_i} = \frac{\partial R_s}{\partial R_i}$$

where:

- R_s is the system reliability.
- R_i is the component reliability.[1]

The ReliaWiki resource portal has more information on reliability importance at: http://www.ReliaWiki.org/index.php/Reliability_Importance.

For more information about reliability importance and the various ways it is used within BlockSim, please read "Reliability Importance Measures of Components in a Complex System" at: http://www.weibull.com/hotwire/issue66/relbasics66.htm.

1. Leemis, L.M., Reliability - Probabilistic Models and Statistical Methods, Prentice Hall, Inc., Englewood Cliffs, New Jersey, 1995.

15.2 Optimum Replacement Window

The Optimum Replacement window allows you to specify costs for planned (i.e., preventive) and unplanned (i.e., corrective) replacement of the component(s) of interest in the current diagram. It then uses this information to determine the most cost-effective time to replace the components, either individually or as a group.

Preventive maintenance is a logical choice if, and only if, the following two conditions are met:

- Condition #1: The failure rate of the component in question increases with time, implying wearout.
- Condition #2: The overall cost of the preventive maintenance action must be less than the overall cost of a corrective action. (The overall cost for a corrective action should include ancillary tangible and/or intangible costs, such as downtime costs, loss of production costs, lawsuits over the failure of a safety-critical item, loss of goodwill, etc.)

If both of these conditions are met, then preventive maintenance makes sense. The Optimum Replacement window then helps you to determine the appropriate time frame for the preventive maintenance.

What's Changed? In BlockSim 7, optimum replacement calculations were performed for individual blocks. The utility in the Synthesis version allows you to perform calculations either for an individual block or for multiple blocks in a diagram simultaneously. When working with multiple blocks, the calculations can be for individual blocks or a group of blocks.

To access the Optimum Replacement window for an entire diagram, choose **[Diagram/Fault Tree] > Calculation > Optimum Replacement**.

To access the Optimum Replacement for a single block, click the **Optimum Replacement** icon in the Block Properties window. This functionality is available for standard blocks (including contained blocks) in RBDs and for events (including dependent events) in fault trees.

If you are working with a diagram instead of with a single block, the **Available Blocks** area on the left side of the window lists all of the standard blocks (in RBDs) or the events (in fault trees), as well as the switches for any standby containers. The following requirements must be met for an item to be available for inclusion in the optimization calculations:

- Blocks must have a failure distribution.
- Switches must have a failure model assigned. Note that in the standby configuration, the switch is the item that is optimized, not the container or the contained blocks/dependent events.
- The failure model must meet the following requirements.
  - Fixed reliability and exponential distributions are not allowed.
  - Weibull distributions must have beta > 1.
  - Gamma distributions must have k > 1.
  - Loglogistic distributions must have std < 1.
  - For generalized gamma distributions and mixed Weibull distributions, there must be an increasing failure rate within at least one time period. A local optimum can then be found within intervals with an increasing failure rate.

You can select an individual item's check box or drag the block into the table on the right side of the window to include the item in the optimization calculations. You can also select the check box in the Available Blocks header to select all check boxes. To remove items from the optimization calculations, you must clear the check box(es); you cannot drag items out of the table.

For each item that is included in the optimization calculations, you will need to specify the cost for a planned replacement and the cost for an unplanned replacement. For analytical diagrams, this is done manually. For simulation diagrams, it can be done manually or by clicking **AutoFill Simulation Costs**. This command is available only if the diagram has been simulated. It estimates the replacement costs based on the average preventive maintenance costs and average corrective maintenance costs for the block, and the system level costs, multiplied by the block's ReliaSoft Failure Criticality Index. The results will be used to populate the Calculated Costs portion of both the Planned Replacement Cost and the Unplanned Replacement Cost columns (if an Additional Cost is already entered, the Calculated Cost will be added to it). If you subsequently decide that you do not want to use the calculated replacement costs, you can clear these columns by clicking **Clear Values**.

If you are working with a diagram instead of with a single block, when you click **Calculate**, you can choose from the following options:

- Calculate the individual optimum replacement time for each item.
- Calculate a common optimum replacement time, which is the most cost-effective time to replace all of the selected items concurrently.
- Calculate clustered optimum replacement times, which allows you to specify a number of times to replace groups of the selected items. For example, if you enter **3** in the **Number of clusters** field, the software will determine the three best times to replace groups of items such that all of the selected items are replaced at one of the three times.

If you are working with a single block, no choice is necessary.

The software will calculate the optimum replacement time for the items based on their failure characteristics and on the defined costs. This value will be displayed in the Optimum Replacement Time column of the table. In addition, the Cost per Unit Time column will display the estimated cost per unit time for a replacement performed at the calculated optimum replacement time. Multiply the replacement time by the cost per unit time to obtain the cost of replacing the item at the optimum replacement time.

If you have accessed the Optimum Replacement window from a simulation diagram or a block within a simulation diagram, you can click **Create Maintenance Task(s) at Replacement Time** to create a preventive task for one or more of the items. The window that appears allows you to select which items to create tasks for. The preventive tasks created in this way restore the item to as good as new condition and are scheduled at fixed intervals based on item age, equal to the optimum replacement time.

You can click **Cost vs. Time Plot** to show a plot of the results. If you have calculated individual optimum replacement times, the plot will show the cost for each item as a function of time. If you have calculated a common optimum replacement time, the plot will show the common cost as a function of time. If you have calculated clustered optimum replacement times, the plot will show the cost for each cluster as a function of time.

The ReliaWiki resource portal provides more information on preventive maintenance and the Optimum Replacement window at: http://www.ReliaWiki.org/index.php/Introduction_to_Repairable_Systems.

Failure Modes and Reliability Analysis (FMRA) 16

The Xfmea/RCM++/RBI and BlockSim applications provide a failure modes and reliability analysis (FMRA) feature that is used to:

- Display, for the current project, the system hierarchy and Failure Modes and Effects Analysis (FMEA) records that could have an impact on system reliability/maintainability. This includes anything that could cause an item to fail to perform one of its intended functions or cause the failure of one of its sub-items. FMEAs can be created in Xfmea/RCM++/RBI.
- View and set the reliability/maintainability properties of these records.
- Perform different calculations based on these records, including:
  - Generating a preliminary baseline reliability estimate for the system based on occurrence ratings that have been assigned to the cause records.
  - Performing a quantitative criticality analysis to obtain a criticality number for each failure mode in the analysis.
  - Other reliability, availability and cost calculations. The specific functionality that is available will vary depending on the application(s) you have installed. For example, if you installed both Xfmea and BlockSim, you can synchronize the FMRA you built in Xfmea with diagrams in BlockSim to perform a more advanced analysis.

The following sections will discuss:

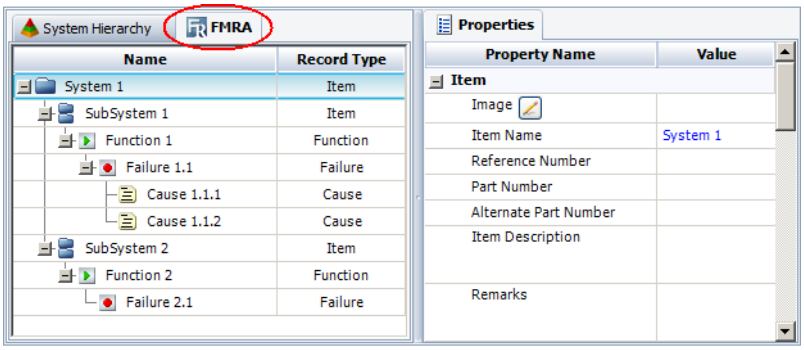
- Viewing the FMRA - Section 16.1 (p. 443).
- Working with records in the FMRA (e.g., adding/deleting records) - Section 16.2 (p. 445).
- Setting FMRA record properties (e.g., reliability policies) - Section 16.3 (p. 447).
- FMRA diagrams and the reliability-configuration of FMRA records - Section 16.4 (p. 453).
- Performing FMRA calculations - Section 16.5 (p. 455).

16.1 Viewing the FMRA

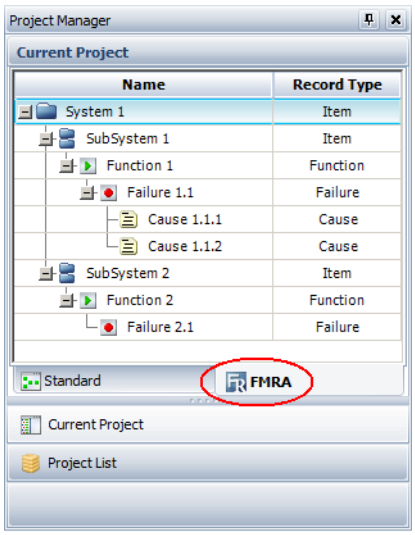
The FMRA can be enabled or disabled by choosing **View > FMRA > Show FMRA**. This command toggles the FMRA on and off.

Once enabled, the way you access the FMRA depends on the application you are using.

- To access the FMRA in Xfmea/RCM++/RBI, go to the FMRA tab of the System panel, as shown next.



- To access it in BlockSim, go to the FMRA tab of the current project explorer, as shown next.



Interpreting the FMRA Hierarchy

The FMRA hierarchy displays all of the items in the system hierarchy, in addition to any functions, failures and causes that have been defined for those items. These are all the records that can impact the reliability/maintainability of the system. They are arranged in a tree-like configuration like the one shown above.

Each branch of the tree displays items and FMEA records down to the level at which the reliability is defined. In the example shown above, an FMEA has been created for SubSystem 1, and both cause records have URDs assigned to them, defining their probability of occurrence. SubSystem 2 also has an FMEA, but in that case the URD is assigned at the failure level. (Although cause records may be associated with that failure record, they are not shown because the reliability is calculated based on the URD assigned to the failure record.)

By default, the software assumes a reliability-wise series configuration of these records. Thus, System 1 fails if either of its two subsystems fails. SubSystem 1 fails if Cause 1.1.1 or Cause 1.1.2 occurs, and SubSystem 2 fails if Failure 2.1 occurs. If you do not want to assume a series configuration and you have BlockSim 9 installed on your computer, you can synchronize the hierarchy with RBDs, which allows you to arrange the records in a more complex configuration.

Available FMRA Columns

The FMRA hierarchy also contains columns that display calculation results and additional information about each record in the hierarchy. To hide or display columns, right-click the column headers, then click **Customize Columns** to select which columns you want to display. In Xfmea, RCM++ and RBI, these settings are stored per computer/username on the FMRA page of the Application Setup, so any project that you open on this computer will have the same columns displayed.

The following columns can be displayed in any application that supports FMRAs:

- **Name** displays the record description/name. This column cannot be hidden.
- **User Access** displays one of the following status indicators when the record is inaccessible.
  - If the record is being edited by another user, an in-use icon will be displayed. Point to the in-use icon to display the name of the user who is accessing the item.
  - If you do not have the item permissions needed to edit the record, a restricted access icon will be displayed.
  - If the record is locked by an FMEA change log, a locked FMEA icon will be displayed. This means you will not be able to make any changes to the FMRA that would affect the locked FMEA (e.g., adding/removing records, changing the name of the locked record, etc.).
- **Record Type** displays whether the record is an item from the system hierarchy, or a function, failure or cause from an FMEA.
- **Synchronized with BlockSim** displays if the record is being synchronized with an FMRA in BlockSim.
- **Reliability Policy** displays “Inherit” if the record's reliability will be determined by the reliability of its lower-level records. If a universal reliability definition (URD) is assigned directly to the record, this column will display the URD's name instead. If a record is set to “Define at this level” but has no assigned URD, then “Default (Not Set)” will be displayed and the software will assume that the reliability is 100% (e.g., for a cause record, the software would assume the cause never occurs). *See Section 16.3.1 on page 447*.
- **Mirror Group** displays the name of the mirror group that a cause belongs to, if any. All the causes in a mirror group are treated as one and the same event. *See Section 16.3.4 on page 452*.

The remaining columns pertain to different calculation results and are discussed in Section 16.5 on page 455.

16.2 Working with Records in the FMRA

A record will be automatically added to the FMRA hierarchy whenever you add an item to the system hierarchy or add a function, failure or cause to an FMEA. If you prefer, you can also add items, functions, failures and causes to your project (as well as delete them from the project) directly from the FMRA.

To move records within a project or across projects in the same database, you can cut/copy them and then paste them in the new location. (Please note that if you want to share records between different databases, you must use the import/export functionality.)

Note: If you set a record's reliability policy to “Define at this level,” you will not be able to add dependents to that record via the FMRA. For example, if an item's policy is set to “Define at this level,” then you will not be able to add any sub-items or FMEA records to that item. This is because once you define the reliability policy at a given level, any records at the lower level would be irrelevant to the software's calculations.

The following sections describe how to add, delete, cut, copy and paste records in the FMRA using the Xfmea/RCM++/RBI application. However, you can also perform these functions within BlockSim's FMRA using similar commands.

Adding and Deleting Records

Use any of the following commands to add an item to the FMRA. Any items you add will also appear in the system hierarchy.

- To add a new top-level (system) item, choose **FMRA > Add System Items > Add System**.

- To add a new item to the next level below a selected item, choose **FMRA > Add System Items > Add Next Level Item**.

- To add a new item to the same level as a selected item, choose **FMRA > Add System Items > Add Same Level Item**.

Use any of the following commands to add an FMEA record to an item. If you add a record to an item that does not already have an associated FMEA, the software will create a new FMEA for that item. (Note that you cannot add effects because they do not affect reliability or maintainability.) When applicable, the software will add an effect record to the FMEA as a placeholder.

- To add a function to a selected item, choose **FMRA > Add FMEA Items > Add Function**.

- To add a failure to a selected function, choose **FMRA > Add FMEA Items > Add Failure**.

- To add a cause to a selected failure, choose **FMRA > Add FMEA Items > Add Cause**.

To delete a record from the FMRA, select it and press **DELETE** (or choose **Home > Edit > Delete**). The record will be deleted, along with any dependents. ***There is no undo for deleting a record.***

Cutting/Copying Records

First select the record(s), then use the standard keyboard shortcuts for cut/copy (**CTRL+X** and **CTRL+C**). You can also choose **Home > Clipboard > Cut** to cut them or **Home > Clipboard > Copy** to copy them. When a record is cut or copied, all dependents also will be cut or copied, along with any associated analyses and attachments.

Tip: In Xfmea/RCM++/RBI, you can also choose **Home > Clipboard > Copy Active View** to copy the currently visible contents of the System or Analysis panel to the Clipboard as a graphic that can be pasted into other applications. The entire visible area within the active panel will be copied, but any information that is not currently visible will be excluded.

Pasting Records

You can paste a record as a dependent to another record in the same project or to another project within the same database by pressing **CTRL+V** or choosing **Home > Clipboard > Paste**. The record will be pasted as a dependent to the selected record in the hierarchy.

In Xfmea/RCM++/RBI, you can paste an item as a top-level system by choosing **Home > Clipboard > Paste > Paste As Top Level**.

Once a record has been cut or copied, any edits that are made to the original will also be applied to its copy in the Clipboard. If a record is cut or copied and then deleted from the project, the copied data will also be deleted from the Clipboard. Also note that you will not be able to cut or copy a record in a project that is locked by another user.

16.3 Setting FMRA Record Properties

How you view and modify the properties of a selected record in the FMRA hierarchy will depend on the software you are using.

- In Xfmea/RCM++/RBI, a record's properties are accessed via the Analysis panel. (Note that the fields that are enabled in the interface and the options available in configurable drop-down lists will depend on the interface style settings that have been defined for the current project. You can choose **Project > Management > Configurable Settings > Interface Style** to open the Edit Interface Style (This Project) window and change the settings for the current project.)
  - For all FMEA records and system items, the settings under the **Reliability Policy** heading on the Properties tab are used to set the reliability policy of that record.
  - All item properties can be edited via the Properties tab of either the FMRA or the system hierarchy.
  - For FMEA records that can be viewed in the FMRA (i.e., functions, failures and causes), the remaining properties are the same as those that can be edited from an FMEA, with the addition of the **Mode Ratio** and **Prob of Loss** fields for failure records that are used for quantitative criticality analysis (*see Section 16.5 on page 455*).
- BlockSim's abilities to view and modify a record's properties are limited. You can change the level at which the reliability is defined and edit the URDs that are used to define the reliability. You can also double-click a block in an FMRA diagram to modify the name of the corresponding record and other properties.

The following sub-sections discuss:

- Setting reliability policy types - Section 16.3.1 (p. 447).
- Using rating scales in the FMRA - Section 16.3.2 (p. 450).
- Using the Batch Update Reliability Policy window to simultaneously update multiple policies - Section 16.3.3 (p. 451).
- Using the Add to Mirror Group window to place the same cause in multiple locations - Section 16.3.4 (p. 452).

16.3.1 Setting Reliability Policy Types

Before you can perform any calculations with the FMRA, you must specify how the reliability/maintainability is calculated for each record in the hierarchy. An "Inherited" policy means the record's reliability/ maintainability is rolled up from its dependents. In all other cases, the calculated values depend on the URD that is assigned to the record. *See Section 16.5 on page 455*.

Note: The information in this section assumes that you understand how to configure URDs and use them to define reliability/maintainability characteristics within Synthesis applications.

Available Policy Types

The reliability policy type determines how a record's reliability/maintainability characteristics will be defined. The following policy types are available in any application that supports FMRAs:

- **Define at this level**: This option is available for all records. When selected, you will be required to set the reliability/maintainability characteristics of the record by assigning a universal reliability definition (URD) to it. If the record does not yet have an assigned URD, "Default (Not Set)" will show in the Reliability Policy column of the FMRA. The software assumes that items with the default policy never fail (i.e., the reliability is always 100%) and failures with the default policy never occur (i.e., the probability of failure is always 0%).

  Note: If you use this option with a record that has dependents, the software will hide those dependents in the FMRA. This is because they will be irrelevant to the FMRA's calculations.

- **Inherit**: This option is available for all records except causes. When selected, the record will inherit its reliability characteristics from the URDs assigned to its dependents. For example, in the hierarchy shown below, System 1 inherits its reliability from its two components. SubSystem 1 inherits its reliability from its two cause records, and SubSystem 2 inherits its reliability from its failure record. Thus, the reliability of System 1 is determined by URD_1, URD_2 and URD_3.

Name	Reliability Policy
System 1	Inherit
SubSystem 1	Inherit
Function 1	Inherit
Failure 1.1	Inherit
Cause 1.1.1	URD_1
Cause 1.1.2	URD_2
SubSystem 2	Inherit
Function 2	Inherit
Failure 2.1	URD_3

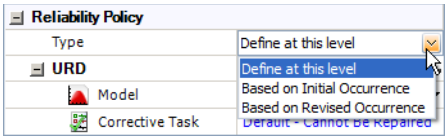
  If you calculate the reliability characteristics using Xfmea/RCM++/RBI, the software will assume a reliability-wise series configuration (i.e., it will assume that the occurrence of any cause, or the failure of any sub-item, is sufficient for the entire system to fail). If you have BlockSim 9 installed on your computer, you can also synchronize the FMRA hierarchy with RBDs in BlockSim to arrange the records in a more complex reliability-wise configuration.

The policy types discussed next pertain to qualitative rating scales and can be selected only in Xfmea/RCM++/RBI. *See Section 16.3.2 on page 450.*

- **Based on Failure Probability**: This option is only available for failure records, and only when the Failure Probability criticality analysis field is enabled for the project. When selected, a read-only probability of failure model (i.e., the probability that the failure mode will occur) will be automatically assigned to the record based on the rating selected in the Failure Probability drop-down list.
- **Based on Initial/Revised Occurrence**: These options are only available for causes, and only when the Initial Occurrence and/or Revised Occurrence properties are enabled on the FMEA > RPNs page of the interface style. When selected, a read-only probability of failure model will be automatically assigned to the record based on the rating selected in the Initial/Revised Occurrence drop-down list.
- **Based on RBI analysis**: This option is only available for RBI components, and only when the option to do so is selected when generating the RBI results. When selected, the model generated by the RBI analysis will be assigned to the component.

Selecting a Policy Type

In Xfmea/RCM++/RBI, you can change the reliability policy type of a selected record by choosing an option in the Type drop-down list on the Properties tab. The policies available for a cause record are shown next.



Tip: To change the policy types of multiple records at once in Xfmea/RCM++/RBI, open the Batch Update Reliability Policy window by choosing **FMRA > RAM Analysis > Policy Update**.

In BlockSim, you can change the policy type in the following ways:

- For any item or record that currently inherits its reliability information from a lower level item or record (i.e., any subdiagram block in a diagram), you can select it in the FMRA hierarchy or select the block in the diagram and choose **FMRA > Inheritance > Define at This Level**. The block in the diagram will change from a subdiagram block to a standard block; you can then open the Block Properties window and assign a URD to the block.
- For any item or record above the cause level that currently has reliability defined, you can select it in the FMRA hierarchy or diagram and choose **FMRA > Inheritance > Inherit**. You can then add the appropriate item(s) or record(s), either in the FMRA hierarchy or in the new subdiagram, and define their properties.

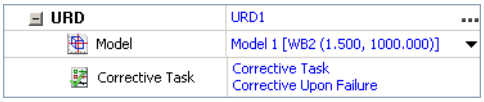
Assigning a Universal Reliability Definition

When **Define at this level** is selected as the policy type, you will have the option to assign a universal reliability definition (URD) to the selected record. The URD can include a probability model. RCM++/RBI and BlockSim can also include corrective and scheduled tasks.

When creating a new model for a record, please note the following:

- An item's model represents the item's reliability (i.e., the probability that the item will not fail).
- A function's model represents the probability that the item will fail to perform its intended function.
- A failure's model represents the probability that the failure will occur.
- A cause's model represents the probability that the cause will occur.

To edit a record's URD, select the record and then, on the Properties tab, double-click the URD, Model or Corrective Task field to open the associated resource wizard.



Tip: If you want to add a scheduled task to the URD in RCM++/RBI, you can do so via the URD resource wizard. The task will then appear on the Properties tab.

To edit a record's URD in BlockSim, double-click the record in the FMRA hierarchy or double-click the block in the diagram. Then double-click the desired field in the **Universal Reliability Definition** area. Note that, to edit a record's URD in BlockSim, the FMRA must be synchronized with diagrams.

16.3.2 Using Xfmea/RCM++/RBI Rating Scales in the FMRA

Xfmea, RCM++ and RBI allow you to use qualitative ratings (e.g., "Level B - Reasonably Probable," "Level E - Extremely Unlikely," etc.) to rate causes and failure modes based on their relative probabilities of occurrence. In addition to being used in RPN calculations and qualitative criticality analysis, these ratings can also be used to assign quantitative models to cause and failure records in the FMRA (in the absence of more specific data).

By using rating scales to assign quantitative models to FMRA records, you can:

- Generate a preliminary baseline reliability estimate for a system based on probabilities of failure that are assigned via qualitative occurrence ratings.
- Perform a quantitative criticality analysis based on probabilities of failure that are assigned via qualitative failure probability ratings.

To use a rating scale in the FMRA, an authorized user must first configure the scale by associating a quantitative model (e.g., a constant failure probability of 5%) with each qualitative rating (e.g., "Level C - Occasional"). Then any user can choose to base an FMRA record's reliability policy on the record's assigned rating. The following sections explain how to edit the rating scales and how to base a policy on a rating scale.

Editing Rating Scales

Authorized users can edit the occurrence scale in the Rating Scale window for the current project or the library.

- To edit the scale for the project, choose **Project > Management > Configurable Settings > Occurrence**.
- To edit the scale for the library, open the Profiles/Library Manager and then click the **Edit** button on the Occurrence page. The window also will be displayed when you double-click one of the scales displayed on these pages.

The failure probability scale can only be edited for the current project. To do this, choose **Project > Management > Configurable Settings > Failure Probability**.

In the Rating Scale window (regardless of whether it is used to edit the occurrence scale or the failure probability scale), the **Quantitative Value** will be treated as either a fixed probability (that is the same for all times) or a probability at a specified time (assuming an exponential distribution). You select how the value is treated in the **Treat Quantitative Values As** area.

> **Note:** When you convert an existing library or project from Version 5 of Xfmea/RCM++, the software will set the quantitative values of these scales to match those in the "RS DFR Focus" profile that ships with Version 9. Thus, after converting data, you may have to re-assign or manually adjust the associated profiles.

- To be able to assign fixed (i.e., time-independent) failure probabilities, select the **Fixed Probability (Q)** option.

  For example, suppose you assigned a quantitative value of **0.01** to an occurrence rating of "7 - High." In this case, if a cause is assigned an occurrence rating of "7 - High" and has a reliability policy based on the occurrence rating, then it will be automatically assigned a model that represents a fixed unreliability of 1%.

- To be able to assign time-dependent distributions with constant failure rates, select the **Exponential Distribution at R(T) = 1-Q, and T is** option. The quantitative value will then be treated as an unreliability point at the time you specify in the **Time** field. This unreliability point will be used to solve for the exponential distribution's parameter.

For example, suppose you entered **1,000** for the time and **Hour** for the time units. Moreover, suppose you assigned a quantitative value of 0.01 to a rating of "Level C - Occasional." In this case, if a failure has a rating of "Level C - Occasional" and a reliability policy based on the failure probability rating, then the software will solve for the exponential distribution's parameter given an unreliability of 1% at 1,000 hours.[1] A model using this distribution will then be automatically assigned to the failure.

Note: If you are using the Rating Scale window to edit the occurrence scale in an enterprise library, the time specified for this option will always be entered in base units. This is because the same standard library might be used by different databases.

Basing a Reliability Policy on a Rating Scale

When a user chooses to set the reliability policy of a failure or cause record based on a qualitative rating, the software will use the rating scales defined for the project to create a probability model for that record.

- To base a cause's reliability policy on an occurrence rating, first make sure that the Initial Occurrence and/or Revised Occurrence properties are enabled on the FMEA > RPNs page of the interface style and that the cause has an occurrence rating assigned. Then select **Based on Initial Occurrence** or **Based on Revised Occurrence** as the policy type.
- To base a failure's reliability policy on a failure probability rating, first make sure that the Failure Probability criticality analysis field is enabled for the project and that the failure has a failure probability rating assigned. Then select **Based on Failure Probability** as the policy type.

Tip: You can also view and select a rating scale by clicking the **Select Existing** icon that appears when you click inside the **Initial/Revised Occurrence** or **Failure Probability** field on the Properties tab for the selected failure/cause record.

If you later change the rating that the reliability policy is based on, or if you change the qualitative value associated with the rating in the rating scale, the software will automatically update the associated model.

16.3.3 Batch Update Reliability Policy Window

The Batch Update Reliability Policy window in Xfmea/RCM++/RBI is used to change the policy type for all the records in the FMRA of the selected types.

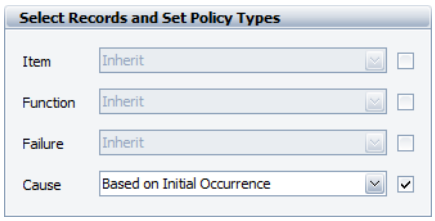
For example, if all of the system's causes have reliability policies that are currently based on their initial occurrence ratings, and you wish to update them based on their revised ratings instead, you could use this window to change all of their policies at once.

To open the window, choose **FMRA > RAM Analysis > Policy Update**.

The check boxes in the window determine which kinds of records will be updated, and the drop-down lists determine how the selected records will be updated (i.e., what policy type they will have). For example, to update all cause

1. To solve for the parameter of the exponential distribution given the unreliability point, the software uses the same calculation method used in the Quick Parameter Estimator. For example, if the given unreliability point is 1% at 1,000 hours, then the model is an exponential distribution with lambda = 0.00001. (You can choose to use mean time for exponential distributions on the Settings page of the Application Setup.)

records so their reliability is based on their initial occurrence ratings, you would select the **Cause** check box and then choose **Based on Initial Occurrence** from the drop-down list.



Please note the following regarding the available policy types.

- If you select **Inherit**, all records of the specified type will have the "Inherit" policy. Note that all records with this policy are required to have dependents with assigned URDs.
- If you select **Define at this level**, then all records of the specified type will have the "Define at this level" policy. Records that previously had a different policy will have "Default (Not Set)" status, and the software will assume that their reliability is 100%.
- If you select **Based on Initial Occurrence**, you will only update causes that have an initial occurrence rating.
- If you select **Based on Revised Occurrence**, you will only update causes that have a revised occurrence rating.

16.3.4 Add to Mirror Group Window (FMRA)

By adding multiple cause records to a mirror group, you can place the same cause in more than one location in the FMRA hierarchy. This is useful when you need to consider common cause failures (i.e., single faults that result in the failure of multiple components) in your analysis. Typical examples of common cause failures include impact, vibration, temperature, contaminants, miscalibration, improper maintenance, and so forth.

Adding Causes to Mirror Groups

In the FMRA, mirroring is accomplished by adding causes to a mirror group via the Mirror Group window. To add a cause to a mirror group in Xfmea/RCM++/RBI, select it in the FMRA and choose **FMRA > RAM Analysis > Mirrors > Add Cause to Mirror Group**. For BlockSim, choose **FMRA > Mirror Groups > Mirrors > Add Cause to Mirror Group**.

- To create a new mirror group for the cause, select **Add to new** in the **Options** area and enter a name for the group.
- To add a cause to an existing group, select **Add to existing** in the **Options** area and then click the group in the **Select Mirror Group** area.

All cause records in a mirror group (i.e., "mirrored causes") are treated as representing the very same cause. To accommodate this, the use of mirror groups will change the properties of cause records in the following ways. ***Note that there is no undo for changed cause properties.***

- When a cause is added to a new mirror group or an existing empty group (i.e., one that includes no other causes), that cause's properties will determine the group's properties.
- When a cause is added to a mirror group that already includes other causes, that cause's properties (e.g., the short description, reliability policy, etc.) will be changed to match the group's properties.
- If you change the properties of a mirrored cause, that change will also be applied to all other causes in the same mirror group.

Removing Causes from Mirror Groups

To remove a cause from a mirror group in Xfmea/RCM++/RBI, select it and choose **FMRA > RAM Analysis > Mirrors > Remove Cause from Mirror Group**. For BlockSim, choose **FMRA > Mirror Groups > Mirrors > Remove Cause from Mirror Group**.

To remove all causes from all mirror groups, choose **FMRA > RAM Analysis > Mirrors > Clear All Mirror Groups**. For BlockSim, choose **FMRA > Mirror Groups > Mirrors > Clear All Mirror Groups**. (Note that this does not delete mirror groups; it only removes causes from them. To delete mirror groups, use the Resource Manager.)

16.4 FMRA Diagrams and Reliability-Wise Configurations

BLOCKSIM Only

By default, the records in an FMRA are assumed to have a reliability-wise series configuration. In other words, if any failure mode occurs or any component fails, then the entire system fails. However, using BlockSim's FMRA diagrams, you can represent more complex configurations.

To do this, first make sure that the FMRA shown in BlockSim and the FMRA shown in Xfmea/RCM++/RBI are identical. In BlockSim, choose **FMRA > Refresh > Synchronize FMRA** to refresh the FMRA hierarchy and create the associated diagrams and blocks.

Note the following about FMRA synchronization:

- Changes that are made in BlockSim are synchronized and propagated to Xfmea/RCM++/RBI automatically, although you may need to refresh the view in Xfmea/RCM++/RBI to see them (**View > Refresh**).
- When items or records are added to the current FMRA in Xfmea/RCM++/RBI, they will not be displayed in BlockSim's FMRA hierarchy until you do one of two things:
  - Refresh the view (**View > Refresh**). When you do this, no associated diagrams or blocks will be created. You will not be able to open the new items/records as diagrams until you synchronize the FMRA.
  - Synchronize the FMRA (**FMRA > Refresh > Synchronize FMRA**). This will refresh the FMRA hierarchy and create the associated diagrams and blocks.
- If you no longer wish to synchronize the FMRA with BlockSim diagrams, you can choose **View > FMRA > Disassociate FMRA** to delete all of the diagrams and blocks associated with the current FMRA hierarchy and close the FMRA view. Once you have disassociated the FMRA, you can subsequently re-open the FMRA view, but none of the records will be synchronized with Xfmea/RCM++/RBI and there will not be any associated diagrams or blocks. You can resynchronize at any time.

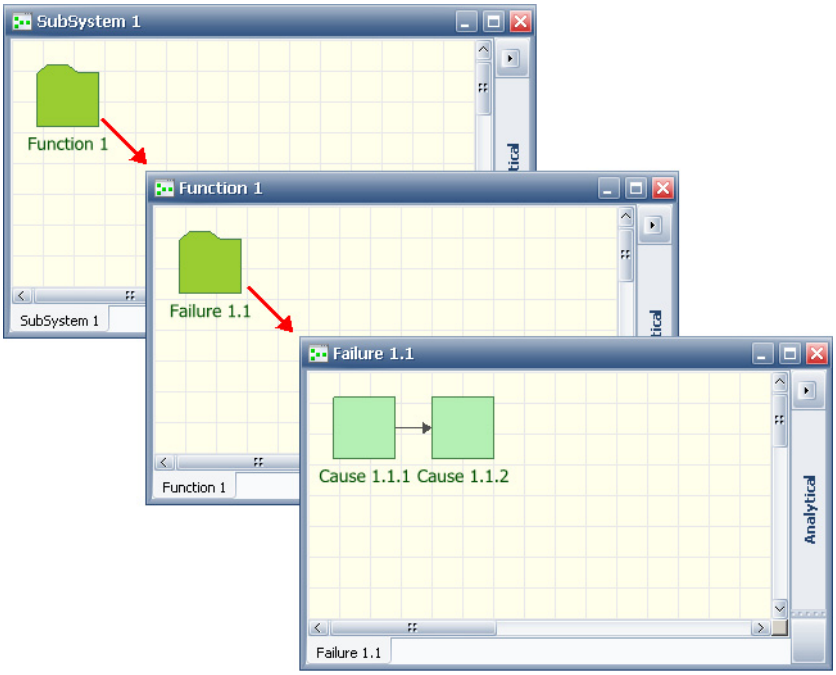
Working with FMRA Diagrams

IMPORTANT: The diagrams that you work with from within the FMRA view are distinct from the BlockSim diagrams in the project. FMRA diagrams relate to the analysis being performed in the project using Xfmea/RCM++/RBI. They are accessible only via the FMRA view, while all other diagrams are accessible only from the Standard tab of the current project explorer.

Each item and record in the FMRA hierarchy represents a reliability block diagram, with the exception of the lowest-level item in each branch, which represents a block in the parent diagram. The levels of the hierarchy are expressed using subdiagrams. For example, consider the FMRA hierarchy shown next.

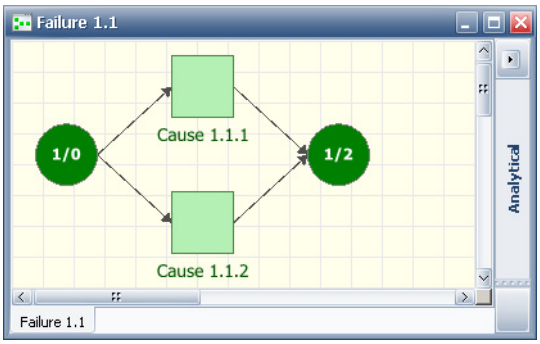
Name	Reliability Policy
System 1	Inherit
SubSystem 1	Inherit
Function 1	Inherit
Failure 1.1	Inherit
Cause 1.1.1	URD_1
Cause 1.1.2	URD_2
SubSystem 2	Inherit
Function 2	Inherit
Failure 2.1	URD_3

Now, let's look at the branch of the FMRA that begins with SubSystem 1. The associated FMRA diagrams are:



As you can see, the causes are assumed to be in a series configuration, as is the case with all reliability-wise configurations built in Xfmea/RCM++/RBI. Working with the FMRA in BlockSim allows you to change the reliability-wise configuration. For instance, let's imagine that the causes are actually in a parallel configuration—in

other words, the item will not fail if only Cause 1.1.1 happens OR only Cause 1.1.2 occurs, but the item will fail if both occur. To represent this, you would change the Failure 1.1 diagram as follows:



Note: In this example, the first node will function as a starter block because there are no paths leading into it.

By default, the FMRA diagrams are created as analytical diagrams. You can choose FMRA > Inheritance > Toggle Diagram Type to change them to simulation diagrams (or to change simulation diagrams to analytical diagrams). This command changes all of the FMRA diagrams in the project; you cannot have both analytical and simulation FMRA diagrams in the same project. You can analyze or simulate an FMRA diagram just as you would any other RBD, and can generate plots and QCP results for each diagram. You also can perform analytical calculations or simulation at the FMRA level.

16.5 Performing FMRA Calculations

The FMRA can perform a variety of analytical and simulation-based calculations. These calculations are based on the configuration of the FMRA hierarchy and the properties that you define for the records in the hierarchy. How you set up the FMRA and perform calculations will depend on which application you are using.

This section describes how to perform FMRA calculations, including:

- Setting up the FMRA
- Reliability calculations
- System availability and cost calculations
- Criticality calculations

Most of these calculations can also be performed within BlockSim, but it may be more convenient to perform most or all of these steps in Xfmea/RCM++/RBI unless you need to use BlockSim to customize the configuration of the FMRA records. This will minimize how often you have to switch back and forth between applications.

Setting up the FMRA

To perform any of the FMRA's calculations, you must first set up the FMRA hierarchy and define the relevant properties for each record. Typically, the initial setup is done in Xfmea/RCM++/RBI; it is possible, however, to perform basic functions in BlockSim.

- Add the system and all the records that could impact its reliability to the FMRA hierarchy. *See Section 16.2 on page 445*.

- In the FMRA, set the reliability policy for all records.
  - Items will typically inherit their reliability from their sub-items and cause records. Thus, URDs will typically be assigned at the item level and cause level. In some cases, you may have to assign URDs at the failure or function level. For example, you would assign a URD at the failure level for quantitative criticality analysis, or if you knew the probability of a particular failure mode occurring, but you didn't have that information for its causes.

    By default, when you select the "Inherit" policy, the software assumes a reliability-wise series configuration (i.e., the software assumes an item will fail if any of its sub-items fail or if any of its causes occur). To perform more complex calculations that do not assume a series configuration, you can use BlockSim's FMRA diagrams to customize the configuration of the FMRA records.
  - In Xfmea/RCM++/RBI, you can also select policies that assign models to records based on occurrence ratings or failure probability ratings. See Using Rating Scales in the FMRA.
- In Xfmea/RCM++/RBI: On the Properties tab of the Analysis panel, define the following properties for each item.

  Note: The fields that are enabled in the interface will depend on the interface style settings that have been defined for the current project. You can choose **Project > Management > Configurable Settings > Interface Style** to open the Edit Interface Style (This Project) window and change the settings for the current project.

  - **Operating Time**: The amount of time for which the system is expected to operate. The reliability calculated for the system is the reliability at the specified operating time. This can be modified at the system level only. When you perform a simulation-based calculation, you can also edit this value in the simulation settings.
  - **Current Age**: Defines the age of the item at the start of the analysis. It allows you to define items that have already accumulated age prior to the start of the current mission (i.e., used components).
  - **Duty Cycle**: Allows you to model situations where the actual usage of an item during system operation is not identical to the usage for which you have data. This can include situations where the item:
    - Does not operate continuously (e.g., a DVD drive that was tested in continuous operation, but in actual use within a computer accumulates only 18 minutes of usage for every hour the computer operates).
    - Is subjected to loads that are greater than or less than the rated loads (e.g., a motor that is rated to operate at 1,000 rpm but is being used at 800 rpm).
    - Is affected by changes in environmental stress (e.g., a laptop computer that is typically used indoors at room temperature, but is being used outdoors in tropical conditions).

    Continuous operation at the rated load is considered to be a duty cycle of 1. Any other level of usage is expressed as a percentage of the rated load value or operating time. For example, consider the DVD drive mentioned above; its duty cycle value would be 18 min / 60 min = 0.3. A duty cycle value higher than 1 indicates a load in excess of the rated value.
  - **Maintenance Group**: This field is available only in RCM++/RBI, and it allows you to choose or create a maintenance group to assign to the item. A maintenance group is a set of items where some event within the group can trigger maintenance for one or more items, either within the group or outside of it. (This field is not applicable for reliability and criticality calculations.)
  - **Target Reliability/Availability**: These optional fields are not required to perform calculations, but they allow you to record the reliability and, for RCM++/RBI, the availability that you hope to achieve and compare it to the calculated value.
- In BlockSim: Make any necessary changes to the diagrams to reflect the actual reliability-wise configuration. Additionally, if you have not already defined the operating time at which calculations will be performed in Xfmea/RCM++/RBI, you can do so in BlockSim by selecting the system in the FMRA hierarchy or opening the system diagram and choosing **FMRA > Inheritance > System Settings**. In the System Settings window, enter a value in the **Operating Time** field.

Reliability Calculations (Analytical)

After you set up the FMRA hierarchy, you can calculate the reliabilities of the records.

- In Xfmea/RCM++/RBI, these calculations are always available. Simply choose **FMRA > RAM Analysis > Calculate (Reliability)**. If there are multiple systems in the FMRA, or if the FMRA is synchronized with BlockSim, the Calculation Options window will appear so you can specify how the calculation will be performed.
- In BlockSim, these calculations are available when you are using analytical FMRA diagrams. To calculate results for the entire FMRA, choose **FMRA > Calculations > Calculate Tree**. You can also choose **FMRA > Calculations > Calculate Branch** to calculate results only for items within the selected system.

The following values are obtained from the calculation and may be shown in the FMRA. To change which results columns are visible, right-click the column headers of the FMRA, then click **Customize Columns**.

- **Analytical Results Status** displays a green indicator light if the analytic results are based on the current inputs. It displays a red light if they are not.
- **Reliability (Analytical)** displays the record's calculated reliability at the system's specified operating time. In Xfmea/RCM++/RBI, the reliability for each item will also be displayed in the **Current Reliability** field of the Properties tab when the item is selected. This value will be highlighted in red if it is below the specified target reliability.
- **Probability of Failure (Analytical)** displays the record's probability of failure at the system's specified operating time. This is the inverse of reliability (1-R).

Tip: If you are using the FMRA to estimate the system's baseline reliability, you may wish to thoroughly review and vet the first draft of the FMRA in order to improve the results. The ReliaWiki resource portal has more information on estimating reliability with the FMRA at: http://www.ReliaWiki.org/index.php/Using_FMRA_to_Estimate_Baseline_Reliability.

System Availability and Cost Calculations

Once it has been set up, the FMRA in RCM++/RBI and BlockSim can also be used to perform simulation-based calculations to predict the system's availability, the cost of operation and other values based on the reliability and the maintenance characteristics of the FMRA records. These calculations are not available in Xfmea.

IMPORTANT: When you perform simulation-based calculations in RCM++/RBI, the software always assumes a reliability-wise series configuration. If you modified the configuration using BlockSim, then you should also perform the simulation within BlockSim in order to take the new configuration into account.

- In simulation, random failure times are generated from each component's failure distribution. These failure times are then considered in accordance with the way the components are reliability-wise arranged within the system. The overall results are analyzed in order to determine the behavior of the entire system. Simulation can be used for complex scenarios involving a multitude of probabilistic events, such as corrective maintenance, preventive maintenance, inspections, imperfect repairs, crew response times, spare part availability, etc.
- In RCM++/RBI, simulation-based calculations are always available. Simply choose **FMRA > RAM Analysis > Simulate (Availability)**. A window will open where you can change the simulation settings. Click **OK** to perform the simulation.
- In BlockSim, simulation-based calculations are available when you are using simulation FMRA diagrams. Choose **FMRA > Calculations > Simulate Tree Elements** to open the Simulation Batch Mode window. You can select to include any or all of the systems in the FMRA in the simulation. Each diagram that is selected for simulation in batch mode must have its own simulation settings defined. Once you have specified all necessary settings, click **Run** to perform the simulation.

The following values are obtained from the simulation and may be shown in the FMRA. To change which results columns are visible, right-click the column headers of the FMRA, then click **Customize Columns**.

- **Simulation Results Status** displays a green indicator light if the simulation results are based on the current inputs. It displays a red light if they are not.
- **Operating Cost** displays the total costs that accrued during the simulation.
- **Cost per Operating Time** displays the operating cost divided by the total operating time.

> **Note:** It is not possible to simulate failure times for components whose reliability is defined by a constant value. Failure times can be simulated only in contexts where reliability is time-dependent. Thus, the calculated operating costs for any component with a constant reliability will be zero. In these situations, it may be advisable to change the model from a constant value to a time-dependent distribution. *See Section 16.5.2 on page 459*.

- **Availability** displays the average availability (i.e., uptime divided by total operating time). This is a metric that reflects the probability that the item is operational (not failed or undergoing repair) when it is requested for use during the simulation.
- **RS FCI** shows the ReliaSoft Failure Criticality Index for each item. This is a relative index showing the percentage of times a failure of the component caused a system failure (i.e., the number of system failures caused by the component divided by the total number of system failures). Note that this relates only to failures; it does not include every event that could bring the system down (such as routine maintenance).
- **RS DECI** shows the ReliaSoft Downing Event Criticality Index for each item. This is a relative index showing the percentage of times that a downing event of the component caused the system to go down (i.e., the number of system downing events caused by the component divided by the total number of system downing events). Note that this only relates to downing events with a non-zero duration.
- **RS DTCI** shows the ReliaSoft Downtime Criticality Index for each item. This is a relative index showing the contribution of each component's downtime to the system's downtime (i.e., the system downtime caused by the component divided by the total system downtime).

Criticality Calculations

In Xfmea/RCM++/RBI, if you wish to perform a quantitative criticality analysis, the software can calculate the probabilities of failure and use that information to calculate the criticality.

The following values are used only in criticality analysis and can be shown in the FMRA only if the applicable criticality analysis fields have been enabled for the project. (The Probability of Failure column, described in the Reliability Calculations section, is also relevant because it is used to solve for these values.) To change which results columns are visible, right-click the column headers of the FMRA, then click **Customize Columns**.

- **Mode Ratio displays** the probability that the item failure will be due to the failure mode under consideration. This value is used to perform criticality analysis.
- **Prob of Loss** displays a factor from 0 to 1 that describes the probability that this failure mode would result in a system failure. By default, this is set to 1 (Actual Loss).
- **Criticality** displays the calculated criticality value. This column must be displayed when the mode ratio or probability of loss is displayed.

16.5.1 Calculation Options Window (Xfmea/RCM++/RBI)

In Xfmea/RCM++/RBI, the Calculation Options window lets you choose which system(s) in the failure modes and reliability analysis (FMRA) will be calculated and which application will be used to perform the calculations. If the FMRA has multiple systems, or it has records synchronized with BlockSim, this window will appear when you choose **FMRA > RAM Analysis > Calculate (Reliability)**.

In the **What items do you want to calculate?** area, you can choose to perform calculations only for the system that is currently selected, or you can choose to calculate all the systems in the hierarchy.

In the **Calculate Method** area, two options will be available if there are records synchronized with BlockSim:

- The **Calculate using BlockSim** option is available only when there are records in the FMRA hierarchy that are currently synchronized with BlockSim. If you calculate using this option, the software synchronizes all records and then returns results based on BlockSim's representation of the FMRA, which uses synchronized RBDs.
- The **Calculate using current application** option will use the software you are currently using (i.e., Xfmea, RCM++ or RBI) to perform the calculations. This means that the software will assume a reliability-wise series configuration among all the records in the FMRA. For example, the software will assume that the occurrence of any one failure, or the failure of any one subsystem, is sufficient to make the entire system fail.

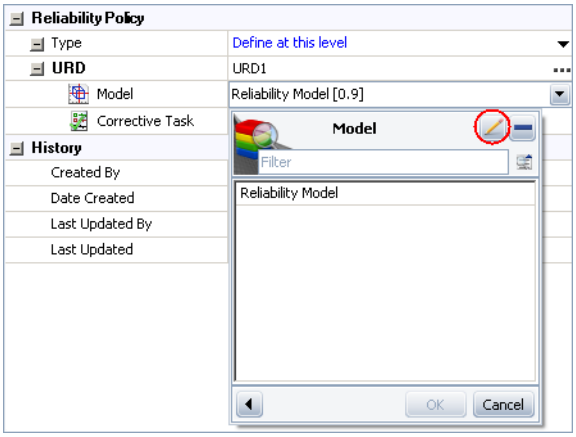
16.5.2 Changing Constant Probability Models to Distributions

At times you may need to change a constant probability model to a distribution. For example, to perform simulation-based cost calculations, the software must be able to simulate failure times for the product based on a time-dependent life distribution. Thus, if you try to perform cost calculations using constant (i.e., time-independent) reliability models, the results of the cost calculation will always be zero. To perform the calculations, you must change any constant model that is considered in the calculation to a time-dependent distribution, as described next.

This section explains how to change the constant probability model to a distribution in Xfmea/RCM++/RBI, but you can follow similar steps in BlockSim by opening the diagram for the record and editing its model in the Block Properties window.

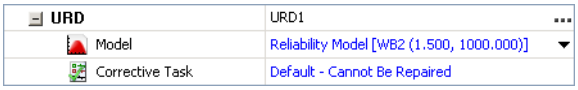
- In the FMRA, select the record that has the model you wish to change. Then double-click the **Model** field under the **Reliability Policy** heading of the Properties tab.

- In the Model wizard that appears, click the **Remove** icon, as shown next.



> **Tip:** As shown in the above example, you can tell whether a model is time-independent by looking at the text displayed in the **Model** field. If the name of the model is followed by a single decimal number in brackets (e.g., "[0.1]"), then the model is time-independent (in this case, the probability of failure is constant at 10%).

- After the model is removed, double-click inside the **Model** field again, then use the Model wizard to create a new distribution and link it to the record.
- After you save the new model, the text in the **Model** field of the Properties tab will indicate the new distribution and parameters. For example, if the model now uses a 2-parameter Weibull distribution (WB2), with beta = 1.5 and eta = 1,000, then the following text would be shown in the field.



Simulation Worksheets 17

Starting in Version 9, simulation worksheets allow you to vary values that are used when simulating a BlockSim RBD or a RENO flowchart. This allows you to investigate the effect of one or more settings on the simulation results.

Simulation worksheets consist of columns that can be defined as inputs or outputs. The input values are used during simulation, and they can be entered manually or transferred from a DOE++ experiment design. Values in output columns are determined through simulation. For example, you might design an experiment in DOE++ and transfer all the factor level combinations to the simulation worksheet. Then you could simulate the experiment in BlockSim and transfer the output values back to DOE++ to analyze the results. If, on the other hand, you do not want to test all combinations of all factor settings, you can enter the desired combinations of settings by hand.

In general, using a simulation worksheet will include the following steps:

- Create the Variables and the Diagram in BlockSim/RENO - Section 17.1 (p. 462)
- Define the Factor Settings - Section 17.2 (p. 463)
- Define the Simulation Worksheet Columns in BlockSim/RENO - Section 17.3 (p. 465)
- Simulate the Worksheet in BlockSim/RENO - Section 17.4 (p. 466)
- Analyze the Results in DOE++ - Section 17.5 (p. 467)
- Verify the Results in BlockSim/RENO - Section 17.6 (p. 469)

Depending on what you are trying to achieve, some of the steps may not be necessary.

In this chapter, we will consider an example in which we explore options for maintaining a spare part pool, with a goal of maximizing system availability while minimizing the cost of the pool. The factors that we will vary are the number of spare parts initially in the pool, the frequency with which the pool is restocked and the number of parts added to the pool at each restock.

17.1 Create the Variables and the Diagram in BlockSim/RENO

Build the diagram for the system in BlockSim/RENO. Use variables to represent the factors whose settings you want to investigate. Depending on the factors you are studying, you might use the variables directly (as field inputs for BlockSim blocks and resources, or in equations and functions in RENO) or via a dynamic model. In BlockSim diagrams, the following fields can be defined using a variable:

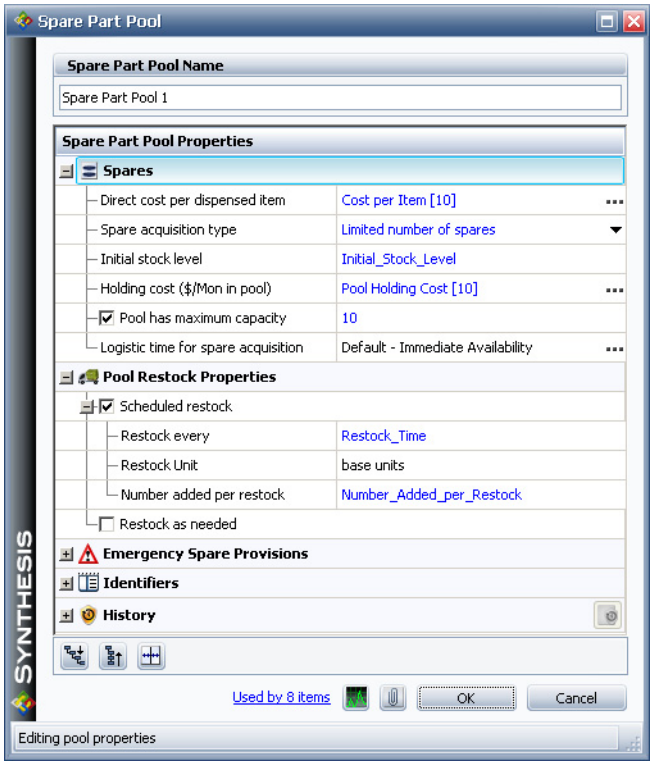
- In standard blocks:
  - **Throughput**
- In crews:
  - **Number of tasks** (visible when there is a limit to the number of tasks the crew can perform at the same time)
- In spare part pools:
  - **Initial stock level**
  - Capacity if **Pool has maximum capacity** is selected
  - For scheduled restocks:
    - **Restock every**
    - **Number added per restock**
  - For restocking as needed:
    - **Restock when stock drops to**
    - **Number added per restock**
  - For emergency spare provisions:
    - **Number added per emergency**
- In scheduled tasks performed at fixed or dynamic intervals:
  - Interval value

Any field that takes a model can use a dynamic model; it is up to you to decide if exploring various values for that field will yield useful results.

IMPORTANT: In RENO flowcharts, you will need to store the results of interest in variables. The **When to Reset** field for output variables must be set to **Never Reset** in order for the results to be available to the simulation worksheet. This ensures that the results are available to the simulation worksheet, rather than the variables being reset to their initial values.

Example

The spare part pool that is used by the tasks in our example is configured as follows:



The cost per item is specified as a constant model equal to $10 and the holding cost is specified as a constant model equal to $10 per month. Since the factors of interest (i.e., the initial stock level, the frequency of restocking and the number added per restock) will not be held constant, they are assigned to variables. These variables represent factor values that will be specified in the simulation worksheet.

17.2 Define the Factor Settings

For the factors of interest, you will decide on the settings to be used in simulation and the combinations of factor settings that should be tried (i.e., that will be used as inputs to the simulation).

- One way to do this is to build an experiment design in DOE++ and then transfer the factor settings to a new simulation worksheet by choosing **Data > Simulation Worksheets > Transfer to Simulation Worksheet**.

  This will ensure that the factor settings are tested in combinations appropriate to determine the effects of each factor and of factorial interactions.

- In some situations, not all factor setting combinations may be possible. For instance, let's say that you are considering two versions of a part, one that costs $100 and has a reliability of 0.97 at 1,000 hours and another that costs $85 and has a reliability of 0.92 at 1,000 hours. You can vary the cost of replacement and the block reliability to account for these scenarios. You would not, however, want to run a simulation using a cost of $85 and a reliability of 0.97, since that combination does not reflect an available part. In cases like this, you can add a new simulation worksheet to the project by choosing **Insert > Simulation Worksheets > Add Simulation Worksheet**.

After adding the new worksheet, you can enter the desired factor values manually, as shown next. (Note that the column names were changed by double-clicking each column header.)

	Replacement Cost	Reliability at 1,000 Hours
1	100	0.97
2	85	0.92

While you have the simulation worksheet open, it will be read-only in other applications. For instance, if you create the simulation worksheet in DOE++, it will be read-only in BlockSim and RENO until you do one of the following:

- Close the simulation worksheet, or
- Click **Commit and Lock** on the control panel. This will commit the data in the simulation worksheet to the database and lock the simulation worksheet within the current application so that it will be available within other applications.

In either case, if you already have the simulation worksheet open in a read-only state, you will need to click **Unlock and Edit** on the control panel once you have released the lock in the original application.

Example

The DOE++ experiment used in our example is a general full factorial design using the following settings:

General Full Factorial

Response Properties	
Name	**Units**
System Availability	
Pool Cost	

Factor Properties									
Abbreviation	**Name**	**Units**	**Type**	**Number of Levels**	**Level 1 (Low)**	**Level 2**	**Level 3**	**Level 4**	**Level 5 (High)**
A	Initial Stock Level		Quantitative	5	1	2	3	4	5
B	Restock Time		Quantitative	3	500	1000	2000	-	-
C	Number Added per Restock		Quantitative	5	1	2	3	4	5

Additional Settings	
# of Unique Runs	75
# of Blocks	1
Center Points / Block	0
# of Replicates	1
Block on Replicates	No
Total # of Runs	75

The first fifteen rows of the simulation worksheet that is created in the transfer process are shown next.

	Initial Stock Level	Restock Time	Number Added per Restock	System Availability	Pool Cost
1	1	500	4		
2	2	500	3		
3	3	2000	1		
4	1	2000	4		
5	5	1000	2		
6	1	2000	1		
7	1	1000	5		
8	5	2000	3		
9	4	500	3		
10	3	500	1		
11	1	500	1		
12	5	2000	4		
13	4	500	5		
14	4	2000	2		
15	5	500	3		

Note: Complete data set is not shown.

17.3 Define the Simulation Worksheet Columns in BlockSim/RENO

Once the factor settings have been entered in the simulation worksheet, you will simulate the worksheet in BlockSim/RENO. The software will perform a set of simulations for each combination of factor settings in order to obtain response values. In BlockSim/RENO, open the simulation worksheet and click the **Simulate** icon on the control panel.

The Select Diagram window will appear, allowing you to select the diagram that will be used to simulate the response values (i.e., the data source) for the simulation worksheet.

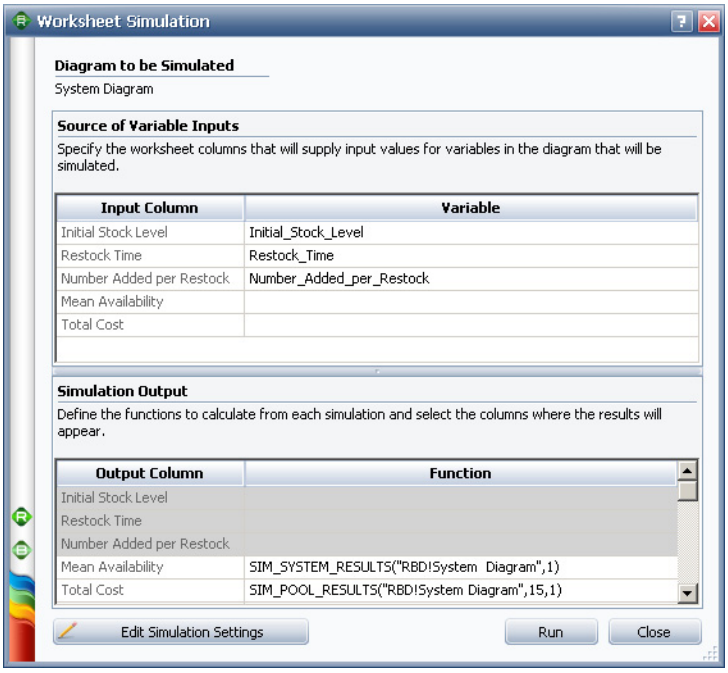
Note: You can also click the **Associated Data Sources** icon to specify or change the source diagram.

The Worksheet Simulation window then appears, allowing you to specify the information contained in each column of the simulation worksheet.

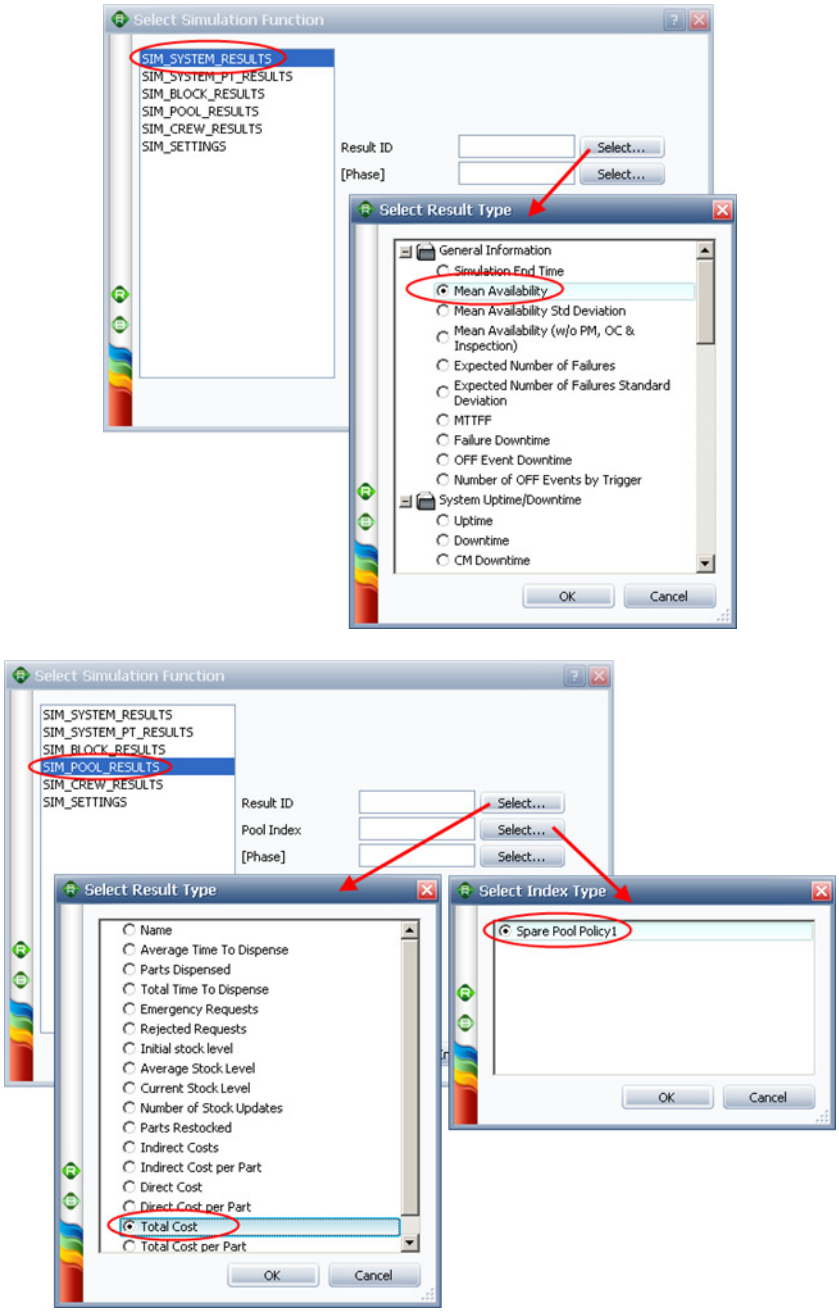
- In the **Source of Variable Inputs** area, assign the variables representing the factors of interest to the appropriate input columns in the worksheet. Available variables will be shown in the drop-down list for each column. If a factor of interest is represented by a dynamic model, choose the variable upon which the dynamic model is based.
- In the **Simulation Output** area, for each result column, specify the result to be entered:
  - If the associated diagram is a BlockSim simulation RBD, define the function to calculate from each simulation. This is done by clicking the cell associated with the result column to open a limited version of the Function Wizard, where only the simulation functions are available.
  - If the associated diagram is a RENO flowchart, select the variable to read for the result.

Example

The Worksheet Simulation window used in our example is shown next.



The result columns were specified as follows:



17.4 Simulate the Worksheet in BlockSim/RENO

With all columns defined, you can perform the simulation in BlockSim/RENO.

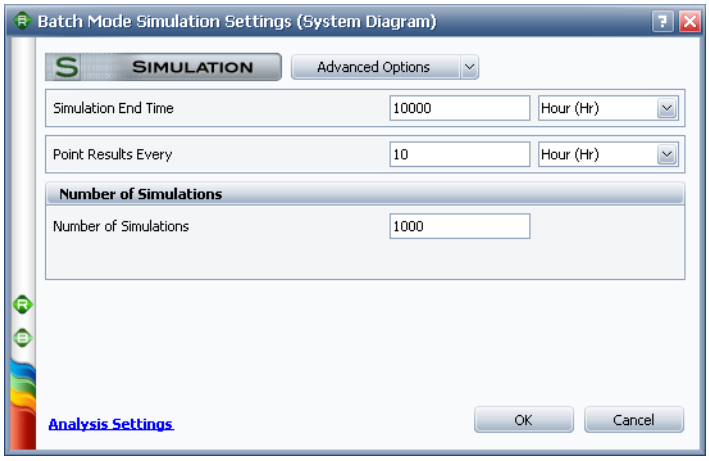
- Click the **Edit Simulation Settings** button to open the Batch Mode Simulation Settings window. This is an abbreviated version of BlockSim's Maintainability/Availability Simulation window or of the RENO simulation window, depending on whether the associated diagram is a simulation RBD or a RENO flowchart. The settings

that you specify will apply to the simulation performed for each row (i.e., each factor setting combination). Specify the desired simulation settings and click **OK** to return to the Worksheet Simulation window.

- Click **Run** to perform the simulations. The results will be entered in the simulation worksheet.
- Commit and lock the simulation worksheet to make it available for analysis in DOE++.

Example

The simulation settings used in our example are shown next.



The first fifteen rows of the simulation worksheet, with results, are shown next.

	Initial Stock Level	Restock Time	Number Added per Restock	Mean Availability	Total Cost
1	1	500	4	0.9023542272	1388.75053
2	2	500	3	0.9030720215	1363.074631
3	3	2000	1	0.2684987514	159.8430884
4	1	2000	4	0.5268783099	284.7889089
5	5	1000	2	0.6764819262	408.3389279
6	1	2000	1	0.2101938712	76.59627093
7	1	1000	5	0.8930326011	1155.376282
8	5	2000	3	0.5256511636	336.6203036
9	4	500	3	0.9031430388	1418.818016
10	3	500	1	0.6515371131	307.0716965
11	1	500	1	0.6024898174	223.760133
12	5	2000	4	0.6245792068	408.2404636
13	4	500	5	0.9031430388	1449.721292
14	4	2000	2		
15	5	500	3		

Note: Complete data set is not shown.

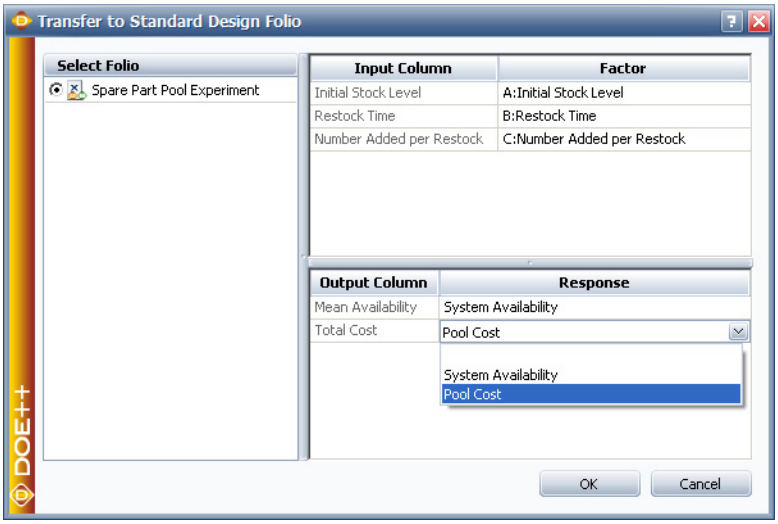
17.5 Analyze the Results in DOE++

The results that were derived from the simulation can now be analyzed in DOE++. To do this:

- Open the simulation worksheet in DOE++. (If it was already open, click the **Refresh** icon on the control panel to load the results and unlock the simulation worksheet for use in DOE++.)
- Transfer the results to a design folio for analysis. You can do either of the following:
  - Click the **Transfer to Standard Design Folio** icon on the control panel to transfer the defined input and output columns to the factor and response columns in an existing standard design folio.

In the window that appears, select the design folio to transfer to and match up the columns in the design folio (factor/response columns) with the columns in the simulation worksheet (input/output columns), as shown in

the following picture.



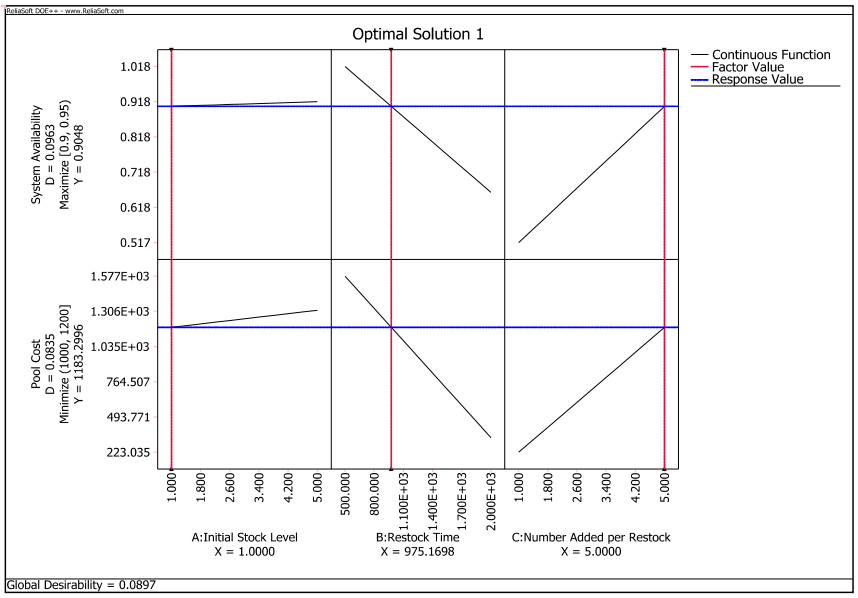
Note that all data will be transferred from the simulation worksheet, so if you have made changes to the input/factor columns in the simulation worksheet, those changes will be transferred to the design folio.

- Click the **Transfer to Free Form Folio** icon on the control panel to transfer the defined input and results columns to a new free form folio.

You can then perform any analysis you choose in DOE++, including optimization.

Example

The results of optimization in DOE++ for our example are shown next.



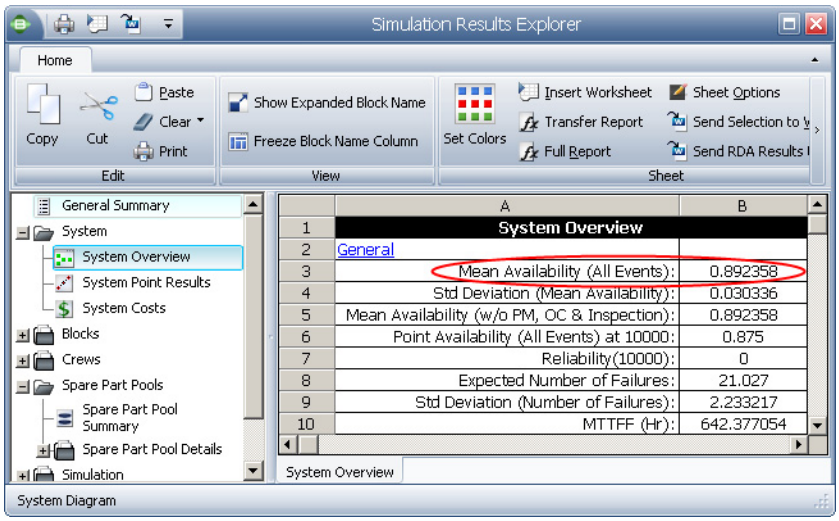
A single optimal solution is found with Initial Stock Level = 1, Restock Time = 975.1698 and Number Added per Restock = 5. This gives a system availability of 0.9048 and pool cost of 1183.30.

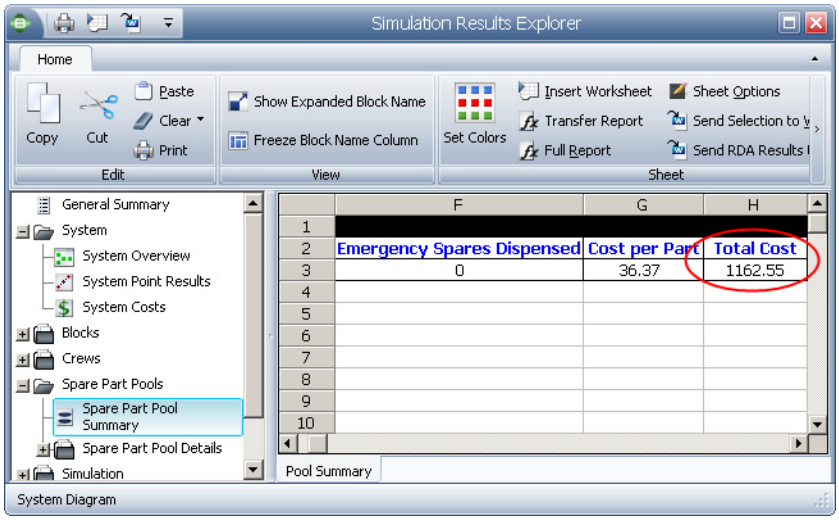
17.6 Verify the Results in BlockSim/RENO

Because BlockSim/RENO and DOE++ rely on different methodologies to arrive at results, predictions made by DOE++ will vary from the simulated results generated by BlockSim/RENO. Therefore, the optimal settings selected by DOE++ must be simulated in BlockSim/RENO to ensure that the predicted results are reliable. To do this, return to BlockSim/RENO and, for each of the variables, set its initial value to the value given in the optimal solution and then simulate the diagram directly (i.e., not via the simulation worksheet) to make sure that the results are close to those estimated in DOE++.

Example

The simulation results from BlockSim are shown next.





The results show a mean system availability of 0.8924 and a total cost for the pool of $1162.55. These values are very close to the estimated values in DOE++.

Analysis Workbooks 18

The analysis workbook is a custom analysis and reporting tool that is built directly into many Synthesis applications (including Weibull++/ALTA, RGA and BlockSim).

These flexible workbooks provide functionality similar to Microsoft Excel (with over 150 built-in functions and complete in-cell formula support), and they can be used to integrate data and/or results from up to four analyses at the same time. You also have the option to save any existing workbook as a template that can be reused with any data set.

To add a new analysis workbook in an existing project, choose **Insert > Reports and Plots > Analysis Workbook**.

For more information about using these flexible workbooks, see:

- Using the Analysis Workbook Wizard when you create a new workbook - Section 18.1 (p. 472)
- Managing the associated data sources - Section 18.2 (p. 472)
- Creating and using saved templates - Section 18.3 (p. 474)
- Using the analysis workbook's control panel - Section 18.4 (p. 475)
- Using the Function Wizard to insert data/results - Section 18.5 (p. 477)
- Tools for working with spreadsheets - Section 18.7 (p. 484)
- Using the Chart Wizard and Chart Designer to create custom plots/charts - Section 18.6 (p. 483)
- An analysis workbook example - Section 18.8 (p. 495)

Other Options for Custom Analysis and Reports

Word Report Templates offer custom reporting functionality that is similar to a Microsoft Word document. You may prefer to use this tool if you want to have a more polished, professional looking report. The Word reports also allow you to use the Plot Wizard to insert the same types of plots that are generated in folio or diagram plot sheets.

In Weibull++/ALTA and RGA, you also have the option to insert general spreadsheets into any data folio. These flexible spreadsheets provide functionality that is very similar to analysis workbooks.

18.1 Analysis Workbook Wizard/Word Report Template Wizard

When you create a new analysis workbook or Word report template, a wizard gives you the opportunity to assign the first associated data source and/or select to base the new report on a saved template, if desired. (Note that the examples shown here are from Weibull++ but similar functionality is available in ALTA, RGA and BlockSim.)

If you prefer to start with a blank report:

Simply click **OK** and then click **Yes** when prompted to confirm that you want to create the report without associating a data source. (You will be able to associate data source(s) later if you wish.)

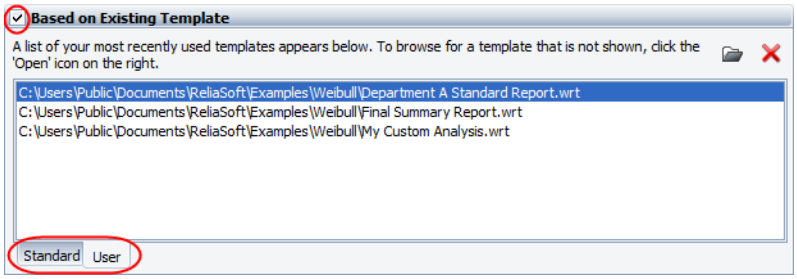
If you want to assign the first data source:

Click **Select** and choose one of the available analyses (i.e., a data sheet in Weibull++/ALTA and RGA, or a diagram in BlockSim). When you return to the wizard, click **OK** to create the report.

Note that you will be able to associate up to four data sources after the report is created, and change any of those assignments at any time. For more information, see Associated Data Sources.

If you want to use a saved template:

Select the **Based on Existing Template** check box and then choose a template from either the Standard tab (templates that are installed with the software) or the User tab (templates you have saved). Then click **OK** to create the report.



For more information about creating and using templates, see Saved Workbook/Report Templates (Section 18.3 on page 474).

18.2 Associated Data Sources

In both the analysis workbooks and the Word report templates, you have the option to associate up to four default data sources that can be used for any function that obtains data or results from an existing analysis.

When you create this type of function, you will have the option to use an index to specify which of the currently assigned default data sources will be used. For example, if you want to compare the reliability values calculated from two different analyses, you can use `=RELIABILITY(Default1,1000)` to get the result from the first default data source, and `=RELIABILITY(Default2,1000)` to get the result from the second default data source.

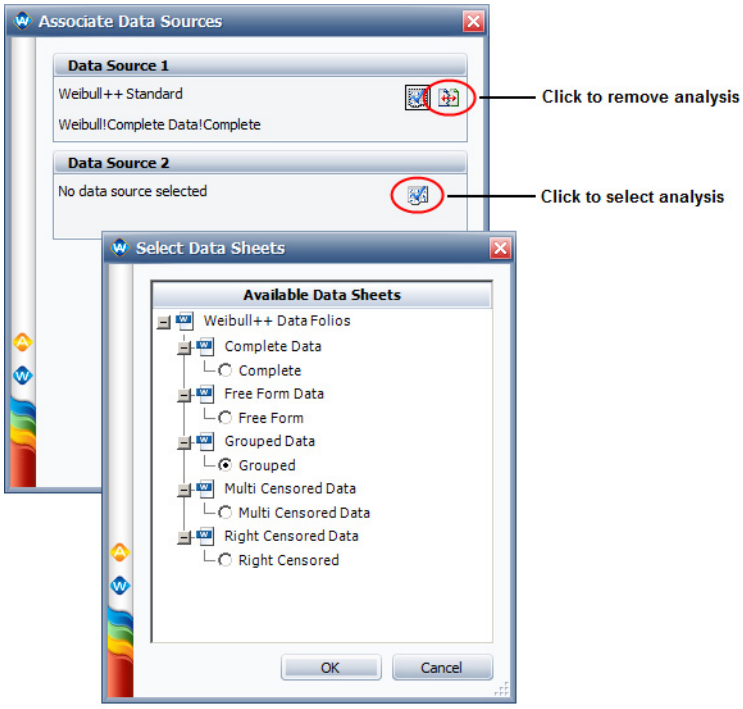
Although you will be prompted to associate the first data source when you create the report, you can add or change associated data sources for an existing report at any time.

Associating Data Sources

The **Associated Data Sources** area in the control panel shows which analyses, if any, are currently assigned as default data sources. To add or remove analyses, click the **Associate Data Sources** icon.

(This command can also be accessed from the Ribbon. In Analysis Workbooks, choose **Workbook > Actions > Associate Data Sources**. In Word Report Templates, choose **Home > Report > Associated Data Sources.**)

The following picture shows the windows for Weibull++. Similar functionality is available in ALTA, RGA and BlockSim.



Note that:

- In **Weibull++/ALTA**, you can select any data sheet from a standard folio, or a specialized folio that contains a life data analysis (i.e., non-parametric LDA folio, degradation analysis folio or warranty analysis folio). For the specialized folios, the results shown in the report will be based on the life data analysis of the extrapolated failure/suspension times.
- In **RGA**, you can select any data sheet in a standard folio.
- In **BlockSim**, you can select any analytical diagram, simulation diagram or phase diagram. The function results can either be based directly on the analyzed/simulated diagram, or based on a model that has been fitted to the diagram. You specify this preference on the diagram's control panel, using the **Report folio model** drop-down list on the Analysis Settings page.

Linking Multiple Data Sources with Different Time Units

In BlockSim, function results that are time-based (e.g., downtime, time to event, etc.) will be returned in the System Base Unit (SBU) for the repository. However, in Weibull++/ALTA or RGA, each function result is returned in the

units used by its data source. If you wish to compare the results from multiple data sources that use different time units, there are two possible scenarios:

- For functions used in analysis workbooks, general spreadsheets and Word reports, convert the data in the original data source. For example, if one data sheet uses "week" and another uses "workweek," you could use the Change Units feature to automatically convert the data in the first analysis from "week" to "workweek."

  Tip: If you want to keep the original analysis unchanged, you could create a copy of the data sheet, then convert the duplicate data sheet to use the new units.

- For functions used in analysis workbooks and general spreadsheets, manually adjust the functions so they return results in the same units. For example, if you want the results to display the B10 life in "workweek" units, you could adjust the Weibull++ function that returns the B10 life for the "week" data sheet by the ratio of the week/workweek SBU values. For example, if 168 is the SBU value for "week" and 120 is the SBU value for "workweek," you could adjust the B10 life function for the "week" data as shown next: `=(TIMEATPF("Weibull!Week!Data1",0.1))*(168/120).`

18.3 Saved Workbook/Report Templates

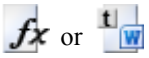
In both analysis workbooks and Word reports, saved templates make it easy to reuse the same report multiple times — in different analysis projects and with different data sets.

In addition to the Standard templates that are installed with the software, you can also create your own User templates that are saved from any existing report.

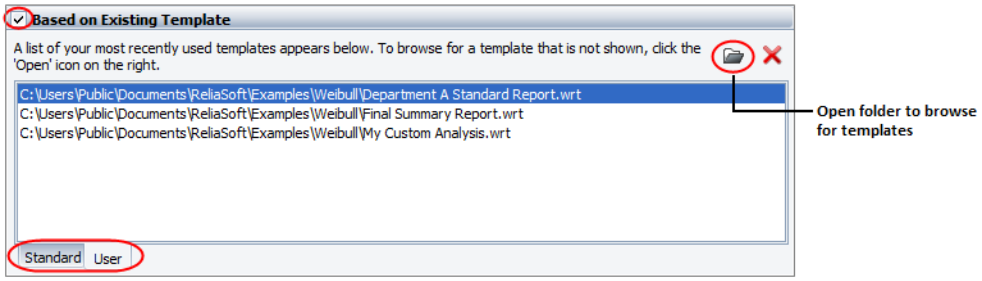
Using a Template to Create a Report

The template must be applied from the wizard that appears when you create a new workbook or Word report. Do the following:

1. Open the wizard by choosing **Insert > Reports and Plots**, then select either the **Analysis Workbook** or **Word Report Template**.

    or

2. Select the Based on Existing Template check box to enable both of the template tabs:
   - The **Standard** tab displays the templates that are installed with the software and stored in the applicable sub-folder under Documents\ReliaSoft\Templates.
   - The **User** tab displays custom templates that were saved from an existing report. These can be stored in any location that is convenient for you. If the custom template you need is not displayed, click the **Open** icon to browse for the desired template and add it to the list.

3. Select a template from either tab then click **OK** to create the report.

Saving Your Own Templates

It is easy to create a template from any existing analysis workbook or Word report template that is currently open.

From an analysis workbook, choose **Workbook > Actions > Save as Template**. From a Word report template, choose **Document > Template > Save as Template**.

You can name and store the custom templates however you wish. To share a template with other users, you can simply send them a copy of the file, or you can save the file in a shared network location that multiple users can access.

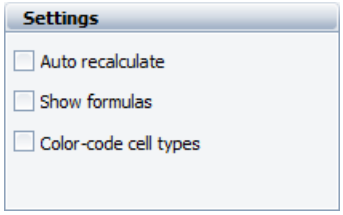
Note that the software automatically appends the appropriate file extension, and each template works only in the application that was used to create it.

Application	File Extension
Weibull++/ALTA	.wrt (workbooks) .rtt (Word reports)
RGA	.rgart (workbooks) .rtt (Word reports)
BlockSim	.brt (workbooks) .rtt (Word reports)

18.4 Analysis Workbook Control Panel

The analysis workbook's control panel provides the following options. (These pictures are from Weibull++. Similar functionality is available in ALTA, RGA and BlockSim.)

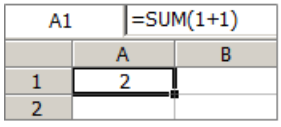
Settings



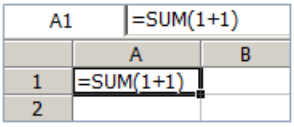
- **Auto recalculate** automatically recalculates formulas when relevant values change in any of the referenced recalculated data sources. This option is also available from the Calculation Options window (**Sheet > Format and View > More Settings > Calculation Options**).

  In BlockSim, it takes more time to recalculate the formulas. Therefore, the automatic option is not available in the control panel. We recommend to manually click the **Recalculate** icon only if you need it.

- **Show formulas** displays the formulas instead of the calculated results in the worksheet cells. This option is also available from the Sheet Options window (**Sheet > Format and View > More Settings > Sheet Options**).



Result displayed in A1

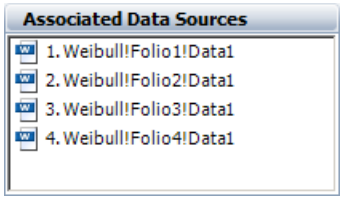


Formula displayed in A1

- **Color-code cell types** displays color-coded borders around the cells. This option is also available from the Sheet Options window.
  - Red: the cell contains a formula
  - Green: the cell contains a value (i.e., number or text)
  - Blue: the cell is blank but formatted
  - No color: the cell is blank and unformatted

Associate Data Sources

The **Associated Data Sources** area displays the list of up to four data sources that are currently associated with the analysis workbook.

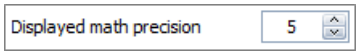


- To open a folio or diagram, click the name.
- To change one or more of the data sources, click the **Associate Data Sources** icon.

Tip: If you use default data source references in your functions (instead of named data sheets or diagrams), you can quickly display the results from different analyses simply by changing the associated data sources.

Displayed Math Precision

The **Displayed math precision** area is available for Weibull++/ALTA and RGA. It sets the number of decimal places shown in values returned from functions using a data source (up to 15 digits, as limited by Windows). By default, this will be the same as your preference in the Application Setup (**File > Application Setup > Calculations**). If you make a change, it will be saved with the current report, but will not change the Application Setup or any other reports.



If the report already contains formulas, the new setting will not take effect until you force a recalculation of all formulas by choosing **Sheet > Format and View > More Settings > Recalculate Formulas** or click the **Recalculate** icon.

Note: If you apply a custom number format (*see page 488*) to the cell (e.g., 0, 0.00, #,##0, etc.), the selected format has priority over the precision setting.

Workbook Tools

The folio tools are arranged on the left side of the control panel:

Function Wizard opens the Function Wizard, which helps you to build functions in analysis workbooks. These functions can return results ranging from mathematical values (e.g., sine, pi or averages) to reliability metrics based on associated data sheets (e.g., reliability, probability of failure, expected number of failures, etc.).

Spreadsheet View sets the analysis workbook to Spreadsheet view, which displays the grid and the column/row headings.

Report View sets the analysis workbook to Report View, which hides the grid and the column/row headings, and displays the analysis workbook in the format in which it will be printed.

Recalculate recalculates formulas when relevant values change in any of the referenced recalculated data sources.

Defined Names opens the Defined Names window (*Section 18.7.3 on page 486*), which allows you to create and manage variable names that can be referenced in any function. You can define a name for any spreadsheet cell or data source.

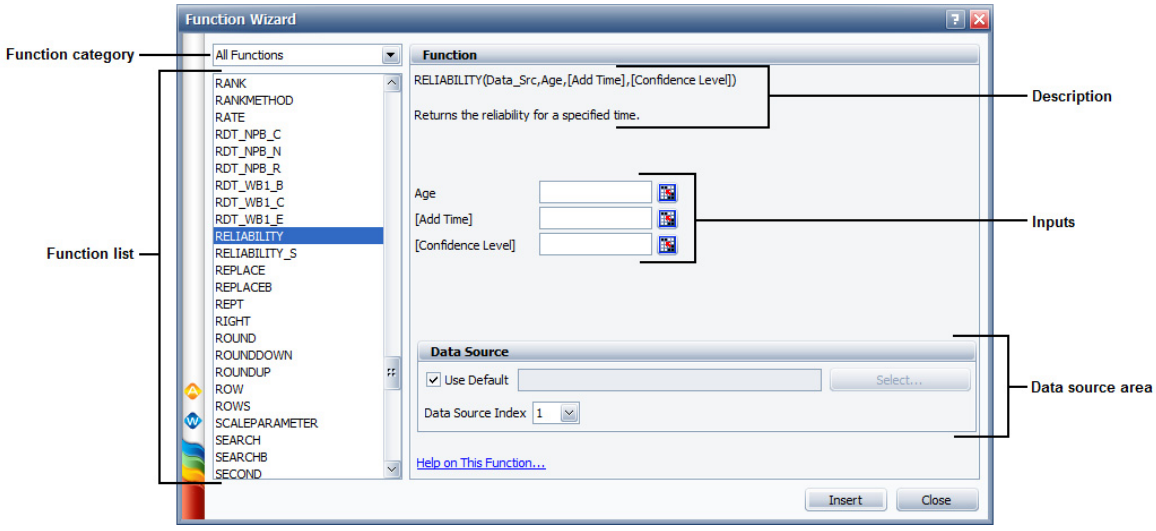
Associate Data Sources allows you to define or change up to four default data sources that can be referenced by functions in this analysis workbook.

18.5 Function Wizard

The Function Wizard generates a variety of results based on your inputs and, when applicable, a referenced analysis.

To open the Function Wizard from an analysis workbook or a general spreadsheet, choose **Sheet > Sheet Actions > Function Wizard**. From a Word report template, choose **Home > Report > Function Wizard**. The wizard can also be opened by clicking the icon in the control panel.

The contents of the Function Wizard window will vary depending on where you are using it. The following picture shows the full functionality of the wizard in Weibull++/ALTA, which is available from analysis workbooks. Similar functionality is available in the wizards for RGA, BlockSim and DOE++.



There are three steps to use this tool:

1. Select the function and enter the inputs
2. Select the data source (if applicable)
3. Insert the function

Select the Function and Enter the Inputs

First select one of the available functions from the panel on the left side. The **Function List** displays the functions in alphabetical order based on your selection in the function category drop-down list. The list will vary depending on where you're using the wizard. Each function that is available in the Function Wizard is described in the online help at: http://help.SynthesisPlatform.net/blocksim_reno9/index.htm#function_wizard_functions.htm.

The right side of the wizard displays some information about the function that is currently selected and allows you to make relevant inputs, if any. Note that:

- The **Help on this Function** link provides a quick reference with additional information about any inputs that may be used in the function.
- For analysis workbook functions, when applicable, brackets indicate that the input is optional. For example, the Add Time and Confidence Level parameters are optional in this Weibull++ analysis workbook function:

  ```
  RELIABILITY(Data_Src,Age,[Add Time],[Confidence Level])
  ```

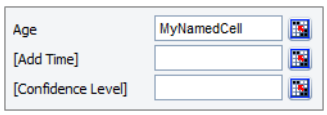
- For Word report templates functions, the brackets are part of the function field and are not optional.

  ```
  [RELIABILITY(Source Number)(Time)]
  ```

- When you are working in an analysis workbook, you can use cell references as inputs, if desired. For example, instead of entering 1000 for a time input, you could specify to use whatever time is currently entered into cell A10, using either the relative reference (A10) or the absolute reference ($A$10). (See Cell References.)

  The **Insert Workbook Reference** icon provides a quick way to insert a reference to the cell that is currently selected in the sheet. Note that you can move the cursor in the sheet while the Function Wizard is open. Pressing **CTRL** while clicking this icon inserts an absolute reference to the currently selected cell.

- When you are working in an analysis workbook, you can also use variable names as inputs. See Defined Names in Spreadsheets. Specifically:
  - Variable names representing specific spreadsheet cells can be used in the input fields.



  - Variable names representing specific data sheets or diagrams can be used in the Data Source field by entering a caret then the name (e.g., ^MyNamedSheet).

For more information on data entry tips for analysis workbooks, see Data Entry Tips for Functions (Section 18.5.1 on page 479).

Select the Data Source (If Applicable)

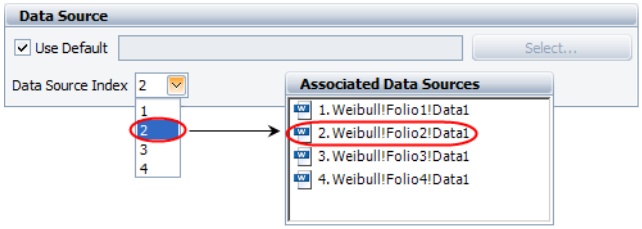
When you are creating a function that obtains data or results from an existing analysis in Weibull++, ALTA, RGA or BlockSim, there are two ways to specify the data source. Both options are available in analysis workbooks. The data source name option is used with general spreadsheets and the data source index option is used with Word reports. (This is not applicable for the Function Wizard in DOE++.)

- Using the **Data Source Name** (analysis workbooks and general spreadsheets only): If you want to ensure that the function always returns the result from a specific data sheet or diagram, you can place the name directly in the function expression. For example, `=MODEL("Weibull!Folio1!Data1")` will always return the distribution that was used to calculate the data sheet called "Data1" in the Weibull++ standard folio called "Folio1."

  To do this in the analysis workbook's function wizard, clear the Use Default check box and use the Select button to choose the data source, as shown next. (In the general spreadsheet's function wizard, you will always need to choose a specific data source.)

- Using a **Data Source Index** (analysis workbooks and Word reports only): If you anticipate that you will be reusing the same template with a variety of different data sets, you can configure the function expression to use a data source index number. For example, `=MODEL(Default2)` will return the distribution or model that was used to calculate whatever data source is currently second in the list of associated data sources.

  To do this in the analysis workbook's Function Wizard, select the **Use Default** check box and choose a number from the **Data Source Index** drop-down list, as shown next. (In the Word report's function wizard, you will always need to choose the appropriate index from the drop-down list.)



Insert the Function

When the function is fully defined, click Insert to place it into the report at the current cursor location. You can move and/or modify the defined function expression after it has been inserted. Note that the functions inserted in a Word report template will not return any results until the report is generated.

For analysis workbooks and general spreadsheets, you can also type the function expressions directly in the cell once you are familiar with the syntax. For more information, see Data Entry Tips for Functions. For Word report templates, you must use the Function Wizard to initially add a function to a report as the functions are inserted as fields. After doing so, you may copy the function and change its inputs.

18.5.1 Data Entry Tips for Functions

When entering functions in any of the spreadsheet utilities (i.e., analysis workbooks, general spreadsheets, etc.), it may be helpful to keep the following tips in mind. If you create the function using the Function Wizard, most of the

syntax and formatting issues will be handled automatically. However, you have the option to create or modify function expressions directly in spreadsheet cells.

Note: For DOE++, you can disregard any tips related to data source functions (i.e., functions that obtain data or results from a specific data sheet or diagram). The math functions available in the DOE++ wizard do not utilize a data source.

Case Sensitivity

The functions are not case sensitive.

Entering Text as an Input

When entering text as an input to a function, you must enclose it in quotation marks. This includes situations where you need to specify the data source – `DISTR("Folio1!Data1")` – and situations where you need enter a time or date value in one of the accepted text formats – `DAY("22-Aug-2014")`.

Regional Settings

If your regional settings use a comma as the decimal separator, you must use a semicolon to separate function arguments (e.g., `=RELIABILITY("Folio1!Data1";A4)`).

Referencing a Cell in a Spreadsheet

If you want to use another cell in the spreadsheet/analysis workbook to provide the input for a function, enter the cell reference with a letter to identify the column and a number to identify the row. The cell references can be relative (e.g., B2) or absolute (e.g. $B$2). For example, if you want to obtain the probability of failure for the time that has been entered in cell B2, the function could be either `=PROBFAIL(B2)` or `=PROBFAIL($B$2)`. You can type the cell location directly into the field or click the Function Wizard's Insert Workbook Reference icon to insert the reference to the cell currently selected in the sheet. If you want to insert an absolute reference, press **CTRL** while you click the icon.

Another option is to use the **Defined Names** tool to assign a name to the cell and use the name in all of the function expressions that require that input. For more information, see Defined Names.

Referencing a Cell in a Data Source

Some functions (e.g., `DATAENTRY` and `FMATRIX`) require you to reference a particular cell in a data source. This must be defined differently than references to a cell in a spreadsheet. For data source cell references, you must identify first the row and then the column, and use a number rather than a letter to represent the column (e.g., A=1, B=2, C=3 and so on). For example:

- `=DATAENTRY(Default1,2,1)` returns the value that was entered into cell A2 in the Weibull++, ALTA or RGA standard folio that is the data source for this function.
- `=FMATRIX(Default1,2,1)` returns the value from the second row in the first column of the Fisher variance/covariance matrix that was calculated for that data source.

Creating Composite Functions

It is possible to combine different types of data sources and/or functions to create a composite function. For example, in the following formula, two different data sources are used to return the difference between the reliability at 100 hours calculated from the specific Weibull++ standard folio data sheet called "Weibull!Target!Data1" and the reliability at 100 hours calculated from any given Weibull++ data sheet that is currently first in the list of associated data sources for the workbook or general spreadsheet.

```
=(RELIABILITY("Weibull!Target!Data1",100))-(RELIABILITY(Default1,100)
```

In the next example, nested functions are used to round up the returned reliability result to the nearest two decimals.

```
=ROUNDUP((RELIABILITY(Default1,1000)),2)
```

Omitting Optional Inputs in the Middle of a Function

If you do not use an optional input in the middle of the function, the function expression must specifically indicate that the input is being omitted. For example, when using the Weibull++ reliability function (`RELIABILITY(Data_Src,Age,[Add Time],[Confidence Level]`), if you want to get the confidence bound on the reliability, you must use two commas (,,) to indicate that the [Add Time] input is intentionally blank, before entering the [Confidence Level] in its usual fourth position (e.g., `=RELIABILITY(Default1,1000,,0.95)`).

Note that this is handled automatically if you use the Function Wizard to build and insert the function expression.

For information about the different results that can be returned from the same function by using the optional inputs, see the individual function description in Spreadsheet Functions at: http://help.SynthesisPlatform.net/blocksim_reno9/index.htm#spreadsheet_functions.htm).

Working with Date Functions

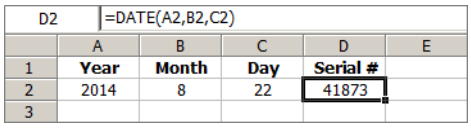
When using one of the spreadsheet date functions (`DAY`, `DAYS360`, `MONTH`, `WEEKDAY` and `YEAR`) to enter a date, you can use one of the following accepted text formats:

- Month/Day/Year ("8/22/2014"). For example, `=DAY("8/22/2014")` returns 22.
- Day-Month-Year ("22-Aug-2014"). For example, `=MONTH("22-Aug-2014")` returns 8 (because August is the 8th month).

If you do not include the year (e.g., "8/22" or "22-Aug"), the current year is assumed.

Alternatively, you can use the date's serial number (which is the number of elapsed days since January 1, 1900). For example, `=YEAR(41873)` returns 2014.

- You can obtain a date's serial number using either of the following two functions. This may be helpful in cases where you want to filter, sort or use the date(s) in calculations.
  - The `DATE` function uses the inputs of other cells to obtain the serial number. For example, if you have dates specified in three cells where A2=Year, B2=Month and C2=Day, `=DATE(A2,B2,C2)` returns the serial number for that date.



  - The `DATEVALUE` function requires you to enter the date in an accepted text format. For example, `=DATEVALUE("8/22/2014")` returns 41873.

Finally, you can also use the results of other functions within a date function. For example:

- To return the month from today's current date, use: `=MONTH(TODAY())`

- To return the day of the week for a date that is specified in three separate cells (A2=Year, B2=Month and C2=Day), use: `=WEEKDAY(DATE(A2,B2,C2))`

D2	=WEEKDAY(DATE(A2,B2,C2))				
	A	B	C	D	E
1	**Year**	**Month**	**Day**	**Weekday**	
2	2014	8	22	6	
3					
4	(Friday is the 6th day of the week.)				
5					

Working with Time Functions

When using one of the spreadsheet time functions (`HOUR`, `MINUTE` and `SECOND`) to enter a time, you can use one of the following valid text formats:

- Hour:Minute[:Second] [AM/PM]. For example, `=HOUR("4:48:10 PM")` returns 16 (the hour using the 24 hour system).
- Month/Day/Year Hour:Minute[:Second] [AM/PM]. For example, `=MINUTE("8/22/2014 4:48:10 PM")` returns 48.

Alternatively, you can use the hour, minute or second's serial number (which is the fractional portion of a 24 hour day). For example, `=MINUTE(0.70011574)` returns 48 (as the specified serial number represents 4:48 PM).

- You can calculate a time's serial number using either of the following two functions. This may be helpful in cases where you want to filter, sort or use the time(s) in calculations
  - The `TIME` function uses the inputs of other cells to obtain the serial number. For example, if you have dates specified in three cells where A2=Hour, B2=Minute and C2=Second, `=Time(A2,B2,C2)` returns the serial number for that time.

D2	=TIME(A2,B2,C2)				
	A	B	C	D	E
1	**Hour**	**Minute**	**Second**	**Serial #**	
2	16	48	10	0.70011574	
3					

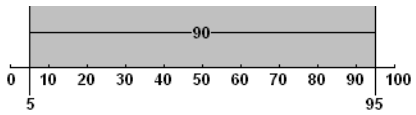
  - The `TIMEVALUE` function requires you to enter the time as text in one of the accepted text formats. For example, `=TIMEVALUE("4:48:10 PM")` returns 0.70011574.

Finally, you can also use the results of other functions within a time function. For example:

- To generate current values, you can use the `NOW` function. If the current time is 4:48 PM, then `=HOUR(NOW())` returns 16.

18.5.2 Using Functions to Return Confidence Bounds

For functions in Weibull++/ALTA and RGA that return confidence bounds, note that the Function Wizard only inserts one-sided bounds. If you want to show two-sided bounds, you can insert the same function twice — once at the lower confidence bound and once at the upper confidence bound. For example, for a 90% confidence level, you would set the lower confidence bound at 5% and the upper confidence bound at 95%.



- In an **analysis workbook**, you can use the same function (with different input parameters) to obtain any of the three values. For example, in Weibull++:

  `=RELIABILITY(Default1,1000)` returns the estimated value

  `=RELIABILITY(Default1,1000,,0.05)` returns the lower one-sided bound at 5%

  `=RELIABILITY(Default1,1000,,0.95)` returns the upper one-sided bound at 95%

  Note that the above function expressions for the confidence bounds uses two commas (,,) to indicate that another optional input ([Add Time]) was intentionally left blank. For more information about the syntax for functions containing multiple optional inputs, see Data Entry Tips for Functions.

- In a **Word report template**, the wizard provides separate functions for confidence bounds. For example, use the Reliability function to get the estimated value and use the Bound on Reliability function to get each one-sided confidence bound.

For more information about confidence bounds, refer to:
http://www.ReliaWiki.org/index.php/Confidence_Bounds.

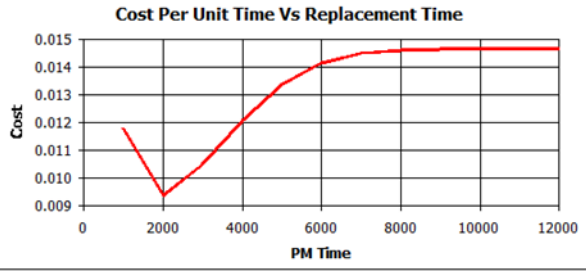
18.6 Chart Wizard and Designer

Charts often make it easier to understand large quantities of data and the relationship between different series of data. You can use the Chart Wizard and Chart Designer tools to insert and customize charts in spreadsheet utilities (e.g., analysis workbooks and general spreadsheets). These charts can be placed anywhere in the spreadsheet, and are dynamic (automatically update when the data used to create the chart is changed). In the example below, the report contains an X Y (scatter) plot showing the best time for preventive maintenance.

Chart Start Time:	1000
Increment By:	1000

Cost for Planned Replacement:	$10
Cost for Unplanned Replacement:	$50

Time Units	Cost/Unit Time
1000	0.01177
2000	0.00940
3000	0.01050
4000	0.01210
5000	0.01339
6000	0.01416
7000	0.01451
8000	0.01462
9000	0.01465
10000	0.01466
11000	0.01466
12000	0.01466
Minimum at:	0.00939671



For more information about the Chart Wizard and Designer, refer to:
http://help.SynthesisPlatform.net/chartdesigner/vcfi5.htm.

Inserting a New Chart

To insert a chart, such as a bar graph or XY (scatter) plot, you will use the Chart Wizard.

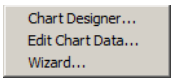
1. Select the cells that contain the data of interest and choose **Sheet > Sheet Actions > Chart Wizard** to activate the Chart Wizard.

2. In the worksheet, click inside the worksheet and drag to select the area where you want to insert the chart.
3. In the Gallery window, select a chart type. Click **Next** and follow the wizard to add chart and axis titles.
4. Click **Finish** to insert the report.

See the analysis workbook example (Section 18.8 on page 495) for an example of a bar chart being added to a reliability report.

Modifying a Chart

Both the Chart Wizard and Chart Designer can be used to modify an existing chart. The Chart Wizard allows you to change only high level settings (e.g., chart type and titles). The Chart Designer allows you to change the high level settings as well as specific settings (e.g., axis label fonts, location of the legend and series labels). To open either one, double-click inside the chart to select it, then right-click to open the Format Chart shortcut menu.

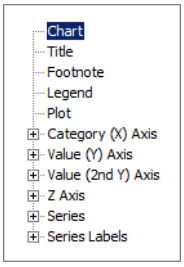


To Use the Chart Wizard:

1. Choose **Wizard** on the shortcut menu.
2. In the Gallery window, you can change the chart type, if desired. Click **Next** and follow the wizard to modify chart and/or axis titles.
3. Click **Finish** to update the chart.

To Use the Chart Designer:

1. Choose **Chart Designer** on the shortcut menu.
2. Use the Element Tree, located on the left side of the window, to select the setting you want to modify. If you are making more than one change, you can click **Apply** to apply changes while keeping the Chart Designer window open.



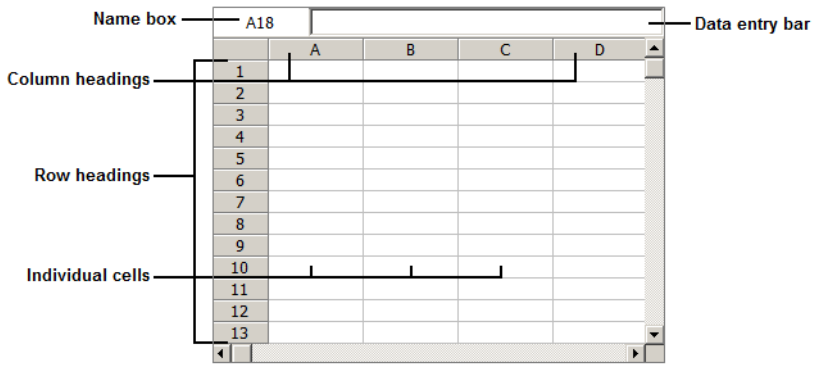
3. Click **OK** when done.

18.7 Tools for Working with Spreadsheets

There are a number of tools that can be used with spreadsheet utilities (e.g., analysis workbooks, general spreadsheets, etc.). For example, cells can be formatted, data can be sorted and spreadsheet attributes can be enabled or disabled.

18.7.1 Spreadsheet Components

The following components are part of any general spreadsheet, analysis workbook or data folio.



Components

- **Name box** contains the column and row indicators that identify the currently selected cell.
- **Data entry bar** is where you enter the data or formula. Pressing an arrow key or ENTER enters the current contents of the data entry bar into the current cell in the spreadsheet. You can also type data directly into individual cells by pressing F2 or double-clicking the selected cell.

  Note that you can also enter formulas manually in the data entry bar (see Data Entry Tips for Functions). However, using the Function Wizard ensures correct syntax.
- **Column headings** and **Row headings** identify the entries in the individual cells. You can also perform the following functions:
  - Enlarge and reduce the height/width of an individual row/column by dragging the borders of the row/column heading cell.
  - Select an entire row/column by clicking the row/column heading cell.
  - Change the name of a row/column heading by double-clicking the heading. Type a new name for the row/column in the window that appears.
- **Individual cells** display each piece of information that you enter into the spreadsheet. The contents of the cell currently selected are automatically displayed in the data entry bar. To edit the contents of a cell in the data entry bar, double-click the cell or press F2.

Formatting

You can customize the appearance of the text (e.g., font style and color) and cells (e.g., borders and fill patterns) by choosing **Sheet > Format and View**. See Format and View in the “Ribbon” chapter for the full list of formatting options.

18.7.2 Referencing Cells in the Spreadsheet

Two types of cell references are possible when entering a formula: absolute and relative.

- A **relative reference** points to a cell based on its relative position to the current cell (e.g., A1). When the cell containing the reference is copied, the reference is adjusted to point to a new cell with the same relative offset as the original cell.
- An **absolute reference** points to a cell at an exact location (e.g., $A$1). When the cell containing the formula is copied, the reference does not change.

Absolute references are designated by placing a dollar sign ($) in front of the row and/or column that is to be absolute. For instance, $A$1 is an absolute reference that points to the cell located in Column A, Row 1 regardless of the position of the cell containing the reference.

Referencing a Cell in a Different Sheet

To reference a cell in a different sheet from the one in which the formula is entered, use an exclamation mark (!) after the sheet name. For instance, `=Sheet1!$A$1` is a reference to the cell located in Column A, Row 1 in Sheet 1. When referencing a cell in a different sheet from the one in which the formula is entered, the reference must be absolute. If the reference is not absolute, the calculations will not be carried out properly.

You can only reference sheets in the same analysis workbook, standard folio or DOE++ spreadsheet.

18.7.3 Defined Names

In any of the spreadsheet utilities (i.e., analysis workbooks, general spreadsheets, etc.), you can use the Defined Names window to create variable names that reference specific spreadsheet cells or data sources. These names can then be used in any formula or function within the same workbook or folio. (Note that the following examples are for Weibull++. Similar functionality is available in ALTA, RGA, BlockSim and DOE++.)

For example, the following formula will return the reliability obtained from a specific Weibull++ standard folio data sheet for the time value that is currently entered in cell B5 of the spreadsheet:

```
=RELIABILITY("Weibull!Bulb!Data1!"$B$5))
```

Alternatively, if you want to create a shorter function expression that may be easier to build and interpret, you can use defined variable names instead. For example, if you use the name “Bulb_A” for the data sheet and the name “OpTime” for the cell that contains the operating time, the function will be:

```
=RELIABILITY(Bulb_A,OpTime)
```

Defining a Variable Name

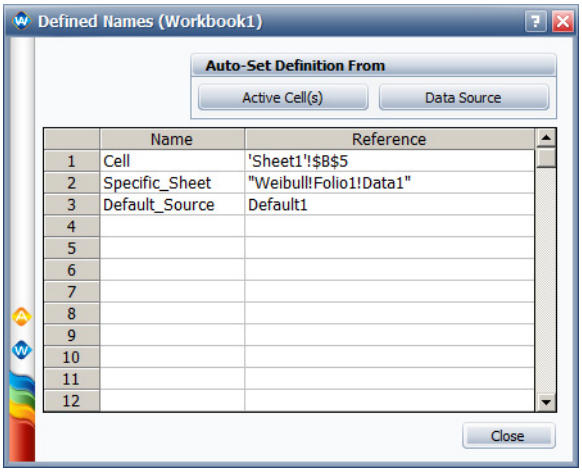
To define a name, do the following:

1. Choose **Sheet > Format and View > More Settings > Defined Names**. In analysis workbooks, you can also click the icon on the control panel.

2. Type an appropriate identifier into the Name column. The names cannot contain spaces or special symbols (e.g., ! < > = ; / \ ~ @ # $ % ^ & * ( ) + - [ ] or quotes or commas); however, the underscore can be used.

3. Enter a valid location into the Reference column. The option to reference a data source is not available in DOE++.

To reference a...	Do the following...
Cell	Identify the location by the sheet name and the absolute cell reference (e.g., Sheet1!$B$5). You can enter this manually or select the cell in the spreadsheet and then click **Active Cell(s)**.
Specific data source	Identify the location by the folio type, folio name and data sheet name (e.g., "Weibull!Folio1!Data!) or by the diagram type and diagram name (e.g., "RBD!RBD1"). The quotation marks are required. You can enter this manually or click **Data Source** to choose from a list of available data sources.
Default data source	Enter Default1, Default2, Default3 or Default4 with no spaces and no quotation marks.

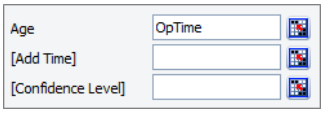


4. Click **Close** to save the changes and return to the spreadsheet.

Using a Defined Name in a Formula or Function

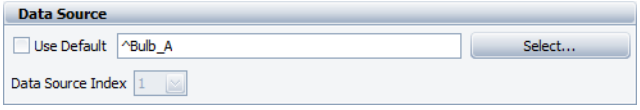
For a Referenced Cell

Type the name in a formula expression in the spreadsheet (e.g., `=RELIABILITY(Default1,OpTime)`) or in the input field of the Function Wizard.



For a Referenced Data Source

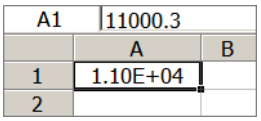
Type the name in a formula expression in the spreadsheet (e.g., `=RELIABILITY(Bulb_A,$B$5)`) or in the **Data Source** field of the Function Wizard. In the Function Wizard, the name must be preceded by a caret (e.g., ^Bulb_A). This option is not available in DOE++.



Note that quotation marks around the name in the function expression (e.g., `=RELIABILITY("Bulb_A,"$B$5)`) will result in a "Sheet not found" error.

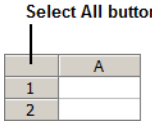
18.7.4 Format Cells - Number Formats

By applying different number formats (e.g., percent, scientific, fixed, date, etc.) you can change the appearance of a number in the current spreadsheet without changing the number itself.[1] For example, if you enter **11,000.3** in cell A1 and select **0.00E+00** (scientific notation) from the **Type** list, the value in cell will then be displayed as 1.10E+04. The value displayed in the data entry bar (11000.3) will be used for all calculations.



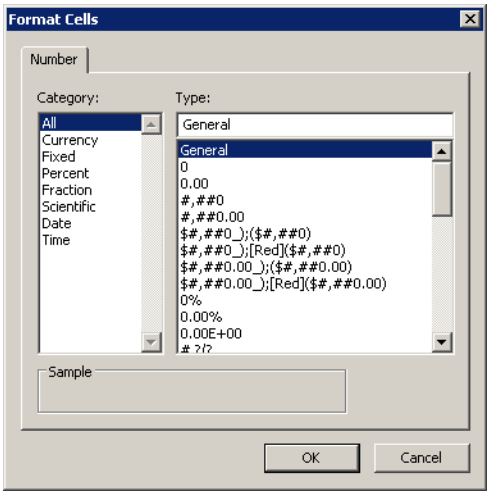
To set a number format:

- Select the cell(s) you want to configure. Note that if you want to select the whole sheet, you can click the **Select All** button.



- Choose **Sheet > Format and View > Custom Number**.

- In the Format Cells window, select the format you want to use from the **Type** list.



Cells in the same spreadsheet can use different formats. To revert selected cells to the default number format, select **All** and **General**.

Note that if you apply a custom number format to the cell (e.g., 0, 0.00, #, ##0, etc.), the selected format has priority over the displayed math precision setting in both the control panel and Calculation page of the Application Setup (**File > Application Setup > Calculations**).

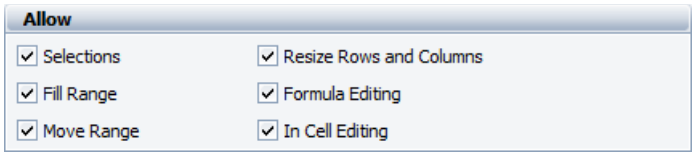
1. If you have selected the **Precision as Displayed** option in the Calculation Options window, the appearance of the number may affect the calculated results (see Section 18.7.6 on page 490).

18.7.5 Sheet Options Window

You can determine the appearance of the spreadsheet and which options are allowed within it by choosing **Sheet Options > Format and View > More Settings > Sheet Options**.

Allow

By default, users are able to select and edit cells, work with cell ranges, resize rows and columns, create and edit formulas, and edit the cell contents either directly within the cell or in the data entry bar.

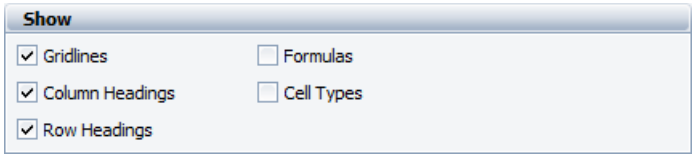


To restrict access:

If you clear...	Results
Selections	Prevents users from selecting cells in the spreadsheet.
Fill Range	Prevents users from filling a range of cells using the cross-hair that appears on the bottom right corner of the selected cell(s).
Move Range	Prevents users from moving a selected range of cells.
Resize Rows and/or **Columns**	Prevents users from resizing the rows and/or columns.
Formula Editing	Prevents users from entering or altering any of the formulas in the spreadsheet.
In Cell Editing	Prevents users from editing the cell contents directly within the cell. All edits must be applied from the data entry bar.

Show

By default, the gridlines and row/column headings are visible in a spreadsheet, the individual cells display the formula results (rather than the formulas themselves) and the cell borders are not color-coded based on the contents of the cell.



To change any of these attributes

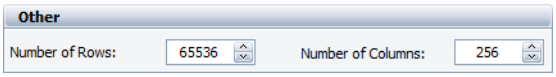
If you clear...	Results
Gridlines	Hides the gridlines on the spreadsheet.
Column and/or **Row Headings**	Hides the column and/or row headings

If you select...	Results
Formulas	Displays the cell formulas instead of the results in the individual cells. For example, you might choose to use this option for convenience while building the formulas and then turn it off again later when you are ready to view the results.
Cell Types	Colors the borders of the cells depending on the contents of the cell using the following scheme: • Red: the cell contains a formula • Green: the cell contains a value (i.e., number or text) • Blue: the cell is blank but formatted • No color: the cell is blank and unformatted

Tip: When working with an analysis workbook in Weibull++, ALTA, RGA, BlockSim or RENO, the last two options can also be turned on and off using the control panel.

Other

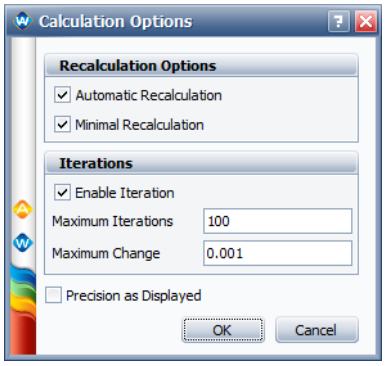
To set the number of rows and/or columns in a spreadsheet, enter a value in the appropriate field. The maximum values are shown in the following picture.



18.7.6 Calculation Options

You can set the options for performing calculations in the spreadsheet by choosing **Sheet > Format and View > More Settings > Calculation Options**.

The Calculation Options window will appear.



After you change any of these options, click **OK** to save the changes and then choose **Sheet > Format and View > More Settings > Recalculate Formulas** to recalculate the formulas within the spreadsheet based on the changes that have been made.

The available options are:

- If you select **Automatic Recalculation**, the spreadsheet will automatically recalculate formulas when relevant values change in referenced recalculated data sources. If you clear this option, you must choose **Sheet > Format and View > More Settings > Recalculate Formulas** to recalculate the formulas and display the new results.

  Note that in BlockSim, it takes time to recalculate the formulas. We recommend to keep this option cleared, and manually recalculate the formulas as needed.

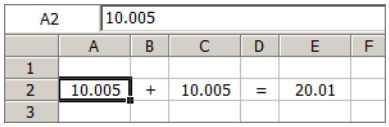
- If you select **Minimal Recalculation**, the spreadsheet will recalculate only the formulas containing references to a changed cell. This will prevent other cells from changing and displaying new results.

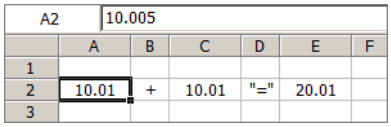
- If you select **Enable Iteration**, the spreadsheet will calculate until it iterates the number of times specified in the Maximum Iterations field or until all cells change by less than the amount specified in the Maximum Change field. This can be used to solve circular references.

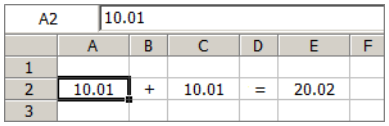
- If you select **Precision as Displayed**, any cell values in the spreadsheet that use a custom number format (e.g., 0, 0.00, #, ##0, etc.) will be rounded to the same number of decimals that are displayed in the cell. The custom number format is set for each individual cell in the Custom Number window. Note that for values entered directly into the cell (i.e., not returned by a function), the original values cannot be restored after they have been rounded.

The behavior will differ based on the combination of options you select. For example, consider a simple scenario in which values entered directly into two cells are being added together:

<table>
<tr><th>If the cell uses the default number format (General)…</th><th>If the cell uses a custom number format (e.g., 0.00)…</th></tr>
<tr><td>The Precision as Displayed option has no effect:

• Both the cell and the data entry bar show the non-rounded values.
• The calculated result is based on the non-rounded values (10.005 + 10.005 = 20.01).
 
</td><td>And if the Precision as Displayed option is cleared:

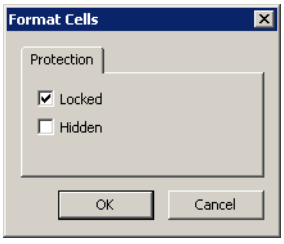
• The cell shows the rounded value, but the data entry bar shows the original non-rounded value.
• The calculated result is based on the non-rounded values (10.005 + 10.005 = 20.01).
 

And if the Precision as Displayed option is selected:
• Both the cell and the data entry bar show the rounded value.
• The calculated results are based on the rounded values (10.01 + 10.01 = 20.02).
 
</td></tr>
</table>

18.7.7 Protecting Cells

You can protect the contents of a cell from being modified and/or viewed. This protection can be applied to all cells in a sheet or just a few.

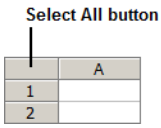
The locked and hidden attributes for any cell can be configured in advance and then will be applied only if cell protection is enabled for the spreadsheet. By default, every cell attribute is set to locked and not hidden.



To lock/hide and protect cells, do the following:

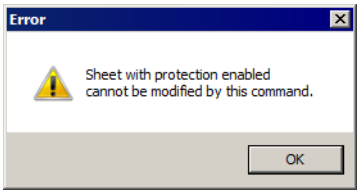
1. If cell protection is already enabled, toggle it off temporarily by choosing **Sheet > Format and View > More Settings > Enable Protection**.

2. Select the cell(s) you want to configure. Note that if you want to select the whole sheet, click the **Select All** button.



3. Choose **Sheet > Format and View > More Settings > Cell Protection**.

4. Specify whether the selected cell(s) will be locked and/or hidden when protection is enabled.
   - **Locked**: The cell's contents are visible in both the cell and data entry bar but cannot be changed. If sheet protection has been enabled and you try to make a change, you will get the error message shown below.



   - **Hidden**: The cell's contents are visible in the cell but not in the data entry bar. For example, you might choose to use this option if you want to hide a formula. The result is displayed in the cell but the formula in the data entry bar is hidden from view.

     While you can use this option by itself, it is more practical to combine it with the locked option to keep the cell's contents from being changed.

5. Once you have set the type of protection, you can toggle cell protection on and off in the spreadsheet by choosing or clearing **Sheet > Format and View > More Settings > Enable Protection**.

18.7.8 Sort

The Sort tool allows you to quickly sort columns or rows alphabetically or numerically in ascending or descending order, or you can create your own custom sort to sort multiple columns in a user-defined order.

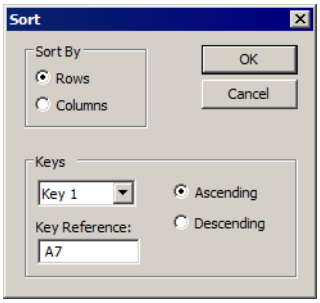
To sort the data, select a range of cells in the spreadsheet and choose **Sheet > Sheet Actions > Sort**.

Sort Ascending or Descending

Choose **Sort Ascending** or **Sort Descending**. The data in the selected cells will automatically be sorted in the appropriate order.

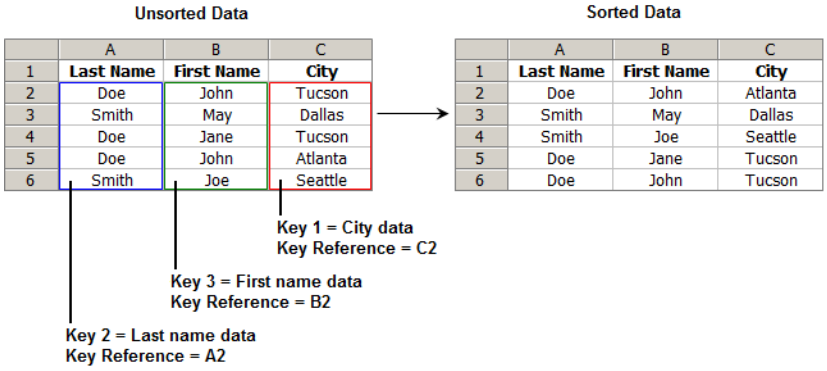
Sort with Multiple Criteria

1. Select **Custom Sort**. The Sort window will appear.



2. In the **Sort By** area, choose how you want to sort the data: either by rows or by columns.
3. In the **Keys** area, choose a key from the drop-down list then use the **Key Reference** field to specify a cell in the column you want to sort by.
   - The columns or rows can be sorted in any order you wish. You can use up to 10 different keys.
   - You can select any cell in the selected columns and the sort will return the same results (e.g., if you have selected cells A1 - A20, you could use cell A13 as the Key Reference and the sorted results would be the same as if you had used cell A1).
4. Click **OK**. The data in the selected cells will be sorted in the order of the keys. If the data for Key 1 contains identical values, then the data for Key 2 would be checked to sort between the same values. If Key 2 contains identical values, then Key 3 would be checked, and so on.

For example, consider a simple case of sorting the names and cities shown below. To sort by city, you would set Key 1 to use Key Reference C2 (“City” data), set Key 2 to use Key Reference A2 (“Last Name” data) and set Key 3 to use Key Reference B2 (“First Name” data). After clicking OK, the cities are listed alphabetically, and then the names are sorted with in each city.



	A	B	C
1	**Last Name**	**First Name**	**City**
2	Doe	John	Tucson
3	Smith	May	Dallas
4	Doe	Jane	Tucson
5	Doe	John	Atlanta
6	Smith	Joe	Seattle

	A	B	C
1	**Last Name**	**First Name**	**City**
2	Doe	John	Atlanta
3	Smith	May	Dallas
4	Smith	Joe	Seattle
5	Doe	Jane	Tucson
6	Doe	John	Tucson

18.7.9 Exporting the Spreadsheet

You can export the spreadsheet in either of two formats:

- To export all sheets to an Excel file, choose **Sheet Options > Transfer Data > Send to Excel**.

The Save As window will appear. Provide a name for the export file, then click **Save**.

- To export the current sheet to a Word file, choose **Sheet Options > Transfer Data > Send to Word**. A Word document opens with the data from the current sheet displayed as a Word table.

18.8 Analysis Workbook Example

You can use the Function Wizard to generate many different types of results in an analysis workbook or general spreadsheet. This section provides an example that shows how you can generate custom reports from your data. In this case, you will use Weibull++ and compare the failure time distributions from different tests of a component.

The data sets used in this example are available in the example database installed with the Weibull++ software (called "Webiull9_Examples.rsgz9"). To access this database file, choose ***File > Help,*** *click* ***Open Examples Folder,*** *then browse for the file in the Weibull sub-folder. The name of the project is "Standard - Overlay Plot."*

Note that while this example uses the Weibull++ software but the same process can be used to generate custom analyzes in ALTA, RGA and BlockSim.

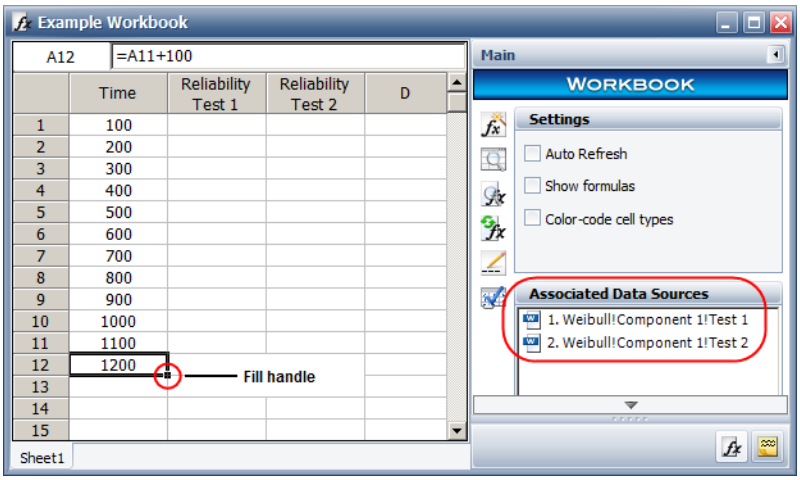
Using the Function Wizard to Create a Table of Results

1. Add an analysis workbook to the project (**Insert > Reports and Plots > Analysis Workbook**) and select the first data source, Test1 for Component 1.

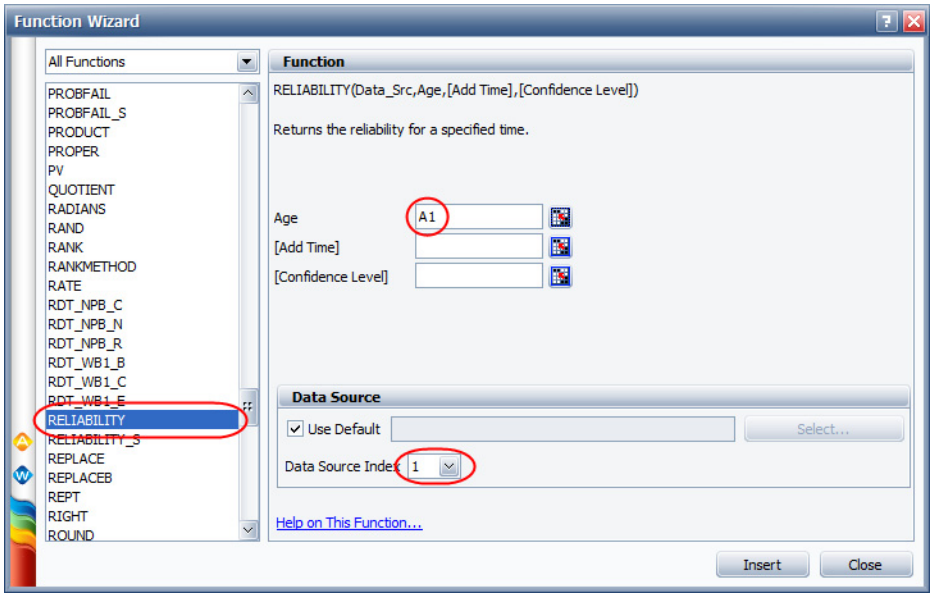
2. Click the Associate Data Source icon and assign the second data source, the "Test 2" data sheet for Component 1.

3. Update the column headings in the report: **Time** in column A, **Reliability Test 1** in column B and **Reliability Test 2** in column C.

4. In cell A1, enter the first time value (100). In cell A2, use the formula `=A1+100` to increment the time value used in that cell. Then select cell A2 and drag the fill handle to cell A12 to copy the same function into the rest of the cells in the column.



5. Select cell B1 and open the Function Wizard. Select the `RELIABILITY` function, use a relative reference to cell **A1** for the Age value and select the first associated data source.



6. Insert the function into cell B1. Then drag the fill handle to cell B12 to copy the same function into the rest of the cells.

B12 =RELIABILITY(Default1,A12)

	Time	Reliability Test 1	Reliability Test 2	D
1	100	0.9941		
2	200	0.9816		
3	300	0.9645		
4	400	0.9436		
5	500	0.9196		
6	600	0.8931		
7	700	0.8644		
8	800	0.834		
9	900	0.8023		
10	1000	0.7695		
11	1100	0.736		
12	1200	0.7021		
13				
14				

7. Select cell C1 and open the Function Wizard. Select the RELIABILITY function, use a relative reference to cell A1 for the Age value and select the second associated data source.
8. Insert the function into cell C1. Then drag the fill handle to cell C12.

C12 =RELIABILITY(Default2,A12)

	Time	Reliability Test 1	Reliability Test 2	D
1	100	0.9941	0.9977	
2	200	0.9816	0.9905	
3	300	0.9645	0.9786	
4	400	0.9436	0.962	
5	500	0.9196	0.9408	
6	600	0.8931	0.9155	
7	700	0.8644	0.8863	
8	800	0.834	0.8536	
9	900	0.8023	0.8179	
10	1000	0.7695	0.7796	
11	1100	0.736	0.7393	
12	1200	0.7021	0.6974	
13				
14				

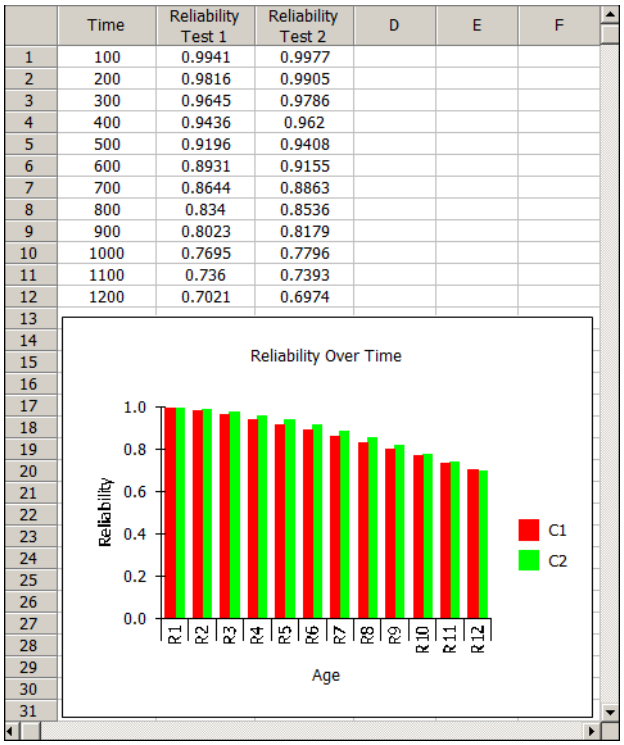
By looking at the data in the table you can see which design has the better reliability at a given time.

Using the Chart Wizard to Create a Bar Chart

You can also add custom charts to the report using the Chart Wizard. For example, to add a bar chart that shows a comparison of the reliability values over time, do the following:

1. Select the cells in the “Reliability Test 1" and “Reliability Test 2" columns and then choose **Sheet > Sheet Actions > Chart Wizard**.
2. Click inside the worksheet and drag to select the area under the data to insert the chart.

3. In the **Gallery** window, select the **Bar** option, then click **Next** and follow the wizard to add titles. When you are done, the report will look like this:



	Time	Reliability Test 1	Reliability Test 2	D	E	F
1	100	0.9941	0.9977			
2	200	0.9816	0.9905			
3	300	0.9645	0.9786			
4	400	0.9436	0.962			
5	500	0.9196	0.9408			
6	600	0.8931	0.9155			
7	700	0.8644	0.8863			
8	800	0.834	0.8536			
9	900	0.8023	0.8179			
10	1000	0.7695	0.7796			
11	1100	0.736	0.7393			
12	1200	0.7021	0.6974			

Word Report Templates 19

The Word report template is a custom analysis and reporting tool that is built directly into many Synthesis applications (including Weibull++/ALTA, RGA and BlockSim).

This utility offers custom reporting functionality that is similar to a Microsoft Word document. Simply design the template – which can include formatted text as well as inserted functions, plots and other graphics – and then generate the report directly in Word. Each report can utilize data and/or results from up to four analyses at the same time, and you can save any existing report template so that it can be reused with any data set.

You may prefer to use this reporting tool if you want to have a more polished, professional looking report, and/or if you want to use the Plot Wizard to insert the same type of plots that are generated in folio or diagram plot sheets.

To add a new report template in an existing project, choose **Insert > Reports and Plots > Work Report Template**.

To generate a report in Microsoft Word, open the template and choose **Home > Report > Create Report**.

Many of the techniques for working with Word report templates are the same as those discussed in Chapter 18 for analysis workbooks, including:

- Using the wizard to create a new template.
- Managing the associated data sources.
- Creating and using saved templates.
- Using the function wizard to insert data/results.

In addition, this chapter discusses the following topics that are specific to Word reports:

- Using the Word report template's control panel - Section 19.1 (p. 500)
- Using the plot wizard to insert plots - Section 19.2 (p. 501)
- Tools for working with report templates - Section 19.3 (p. 502)
- Word report template Ribbon - Section 19.4 (p. 509)
- A Word report example - Section 19.5 (p. 515)

Other Options for Custom Analysis and Reports

Analysis workbooks offer custom reporting functionality that is similar to Microsoft Excel (with over 150 built-in functions and complete in-cell formula support). The analysis workbooks also allow you to use the Chart Wizard to insert your own custom pie, bar and other types of charts.

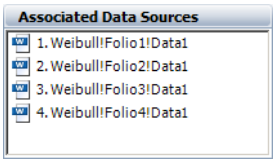
In Weibull++/ALTA and RGA, you also have the option to insert general spreadsheets into any data folio. These flexible spreadsheets provide functionality that is very similar to analysis workbooks.

19.1 Word Report Template Control Panel

The Word Report Template's control panel provides the following options. (These pictures are from Weibull++. Similar functionality is available in ALTA, RGA and BlockSim.).
The control panel tools include the following options:

Associated Data Sources

The **Associated Data Sources** area displays the list of up to four data sources that are currently associated with the Word report template.

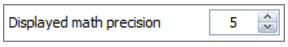


- To open a folio or diagram, click the name.
- To change one or more of the data sources, click the **Associate Data Sources** icon.

For more information about Associated Data Sources, see the "Analysis Workbooks" chapter.

Displayed Math Precision

The **Displayed math precision** area is available for Weibull++/ALTA and RGA. It sets the number of decimal places shown in returned values from functions using a data source (up to 15 digits, as limited by Windows). By default, this will be the same as your preference in the Application Setup. If you make a change, it will be saved with the current template, but will not change the Application Setup or any other reports.



Word Report Template Tools

The folio tools are arranged on the left side of the control panel:

Function Wizard opens the Function Wizard, which allows you to build functions in the report template that return results (e.g., reliability values, failure rate, etc.) and other information from the associated data sources. Note that the functions will not return any results until the report is generated.

Tip: After the function has been inserted by the wizard, you can copy and/or edit its settings directly in the template.

For more information about using the Function Wizard, see the "Analysis Workbooks" chapter.

Plot Wizard opens the Plot Wizard, which allows you to insert plot holders for the same types of plots that are generated in folio or diagram plot sheets. Although you also have the option to copy/paste a static plot graphic from any folio or diagram plot sheet into the report template, using the Plot Wizard instead makes it easy to change the associated data source when you reuse the template and ensures that the plot in the generated report will always show the latest analysis. Note that the plot holders will not return any results until the report is generated.

Associate Data Sources allows you to define or change up to four default data sources that can be referenced by functions and plots in this report template.

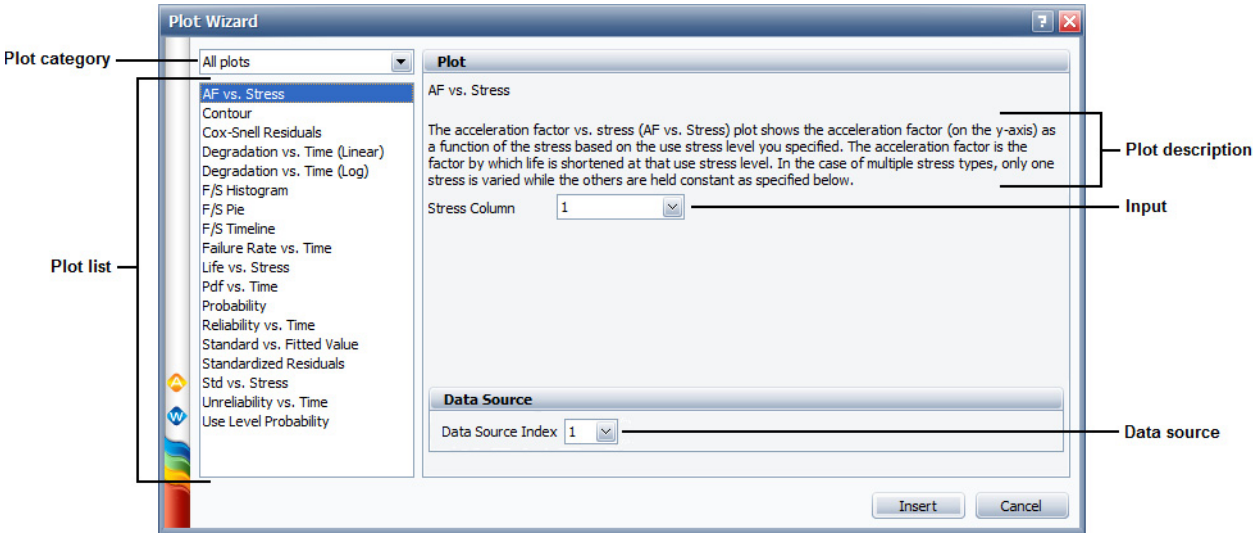
Create Report generates the report with calculated results and plots in Microsoft Word.

19.2 Plot Wizard

The Plot Wizard generates a variety of plots based on a referenced analysis. These are the same types of plots that are generated in folio or diagram plot sheets. Although you also have the option to copy/paste a static plot graphic from any folio or diagram plot sheet into the report template, using the Plot Wizard instead makes it easy to change the associated data source when you reuse the template and ensures that the plot in the generated report will always show the latest analysis.

To open the Plot Wizard, choose **Home > Report > Plot Wizard**. You can also click the icon on the Word report template's control panel.

The following picture shows the full functionality of the wizard in Weibull++/ALTA. Similar functionality is available in the wizards for RGA and BlockSim.



There are three steps to use this tool:

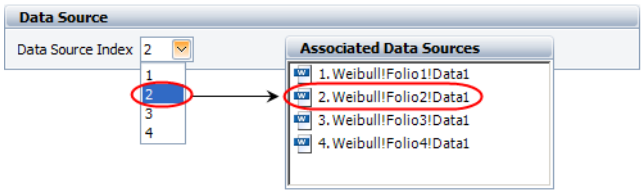
1. Select the plot type (and enter the inputs, if applicable)
2. Select the data source
3. Insert the plot holder

Select the Plot Type

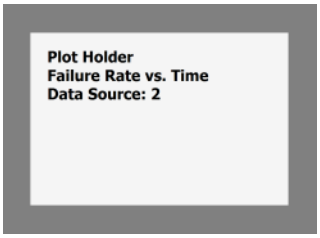
First, select one of the available plot types from the panel on the left side. The right side of the wizard displays some information about the plot that is currently selected. If applicable, this area also allows you to make relevant inputs (e.g., for ALTA plots that require you to select a specific stress column).

Select the Data Source

To specify the data source that the plot will be based on, choose a number from the Data Source Index drop-down list, as shown next. Using the index makes it easy to reuse the same template with a variety of different data sets.



For example, when the report is generated, the plot holder shown next will return the failure rate vs. time plot for whatever data source is currently second in the list of associated data sources.



Insert the Plot Holder

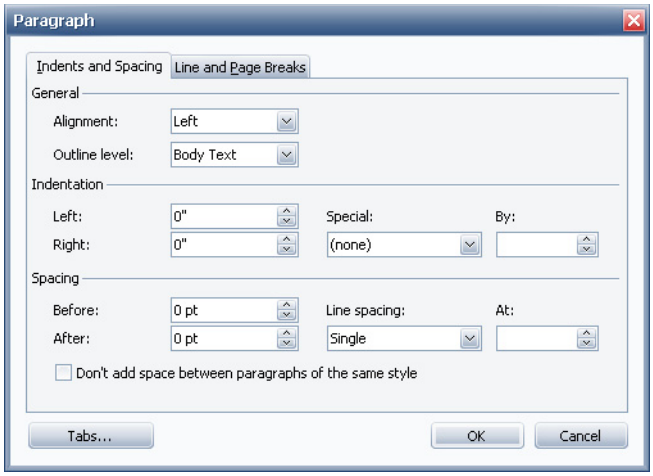
When the plot is fully defined, click **Insert** to place it into the report template at the current cursor location. You can move and/or resize the plot holder after it has been inserted. Note that the plot holders inserted in a Word report template will not return any results until the report is generated (**Home > Report > Create Report**).

19.3 Tools for Working with Word Report Templates

There are a number of tools that can be used with Word report templates. For example, paragraphs can be formatted, hyperlinks and tables can be inserted, and text can be searched for and replaced.

19.3.1 Formatting Paragraphs

You can change a paragraph's formatting options by choosing **Home > Paragraph > Paragraph** to open the Paragraph window.



The following properties can be changed:

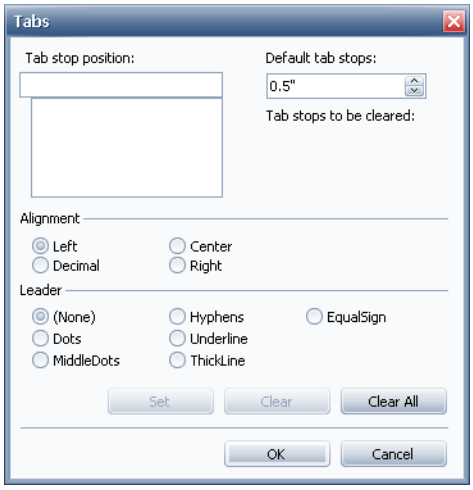
On the **Indents and Spacing** tab:

- In the **General** area, set the paragraph's alignment and outline level.
- In the **Indentation** area, set the amount of space you want to indent from the left and/or right margins.
- In the **Spacing** area, set the amount of space before and after each paragraph and the amount of vertical spacing between lines of text.

On the **Line and Page Breaks** tab:

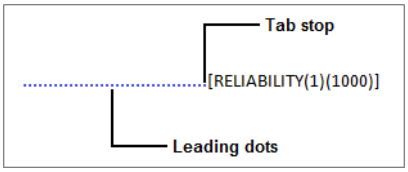
- Select **Keep lines together** to prevent the paragraph lines from being on different pages. The option can be applied to selected paragraph(s) or the entire document.
- Select **Page break before** to force a page break before the paragraph.

To define the tab stops, click **Tabs** to open the Tabs window. To set a tab stop:



- Under **Tab stop position**, type the location (in inches) where you want to set the tab stop.

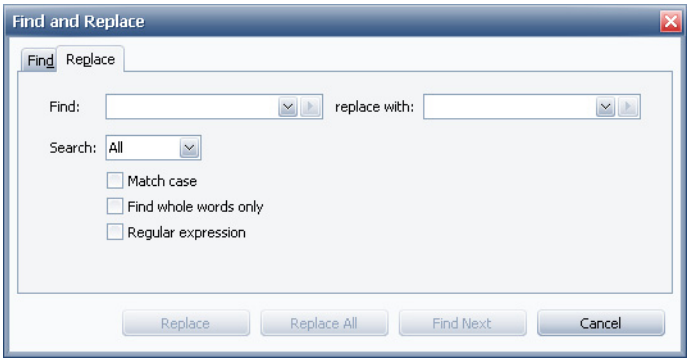
- Under **Alignment**, click the type of tab stop that you want.
- If you want to add dots with your tab stop, or to add another type of leader, click the option that you want under Leader.



- Click **Set** to create the tab stop. You can add additional tab stops, or click **OK** to add the tab stops to the paragraph.

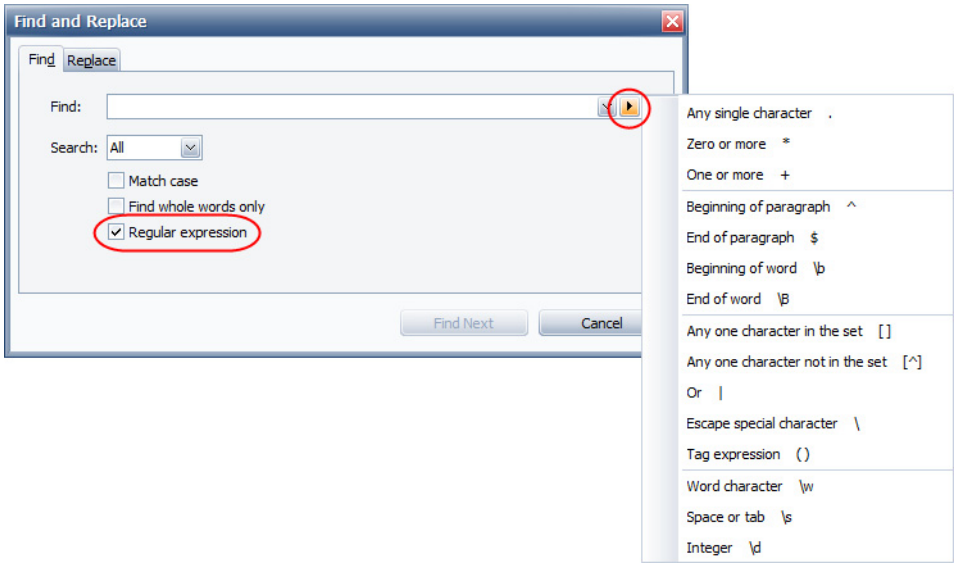
19.3.2 Find and Replace Text or Special Characters

You can find text and/or numbers in the Word report template by choosing **Home > Edit > [Find/Replace]** to open the Find and Replace window.



- Use the **Find** field to type the text and/or number that you want to find in the current Word report template.
- Use the **replace with** field to type the text or numbers that will replace any text or number that matches your search.
- Use the **Search** field to select the order in which to search. If you select **All**, the search will look through the entire Word report template. If you select **Down**, the search will start from the cursor location and move down through the Word report template. If you select **Up**, the search will start from the cursor location and move up through the Word report template.
- Select **Match case** if you want to find only the text in the current Word report template with the same case. For example, the search term “teXt” will not find “Text” because the case does not match.
- Select **Find whole words only** if you want to find only the text where the entire word matches the text and/or number you specified in the **Find** field. If your search term makes up only part of a word, that word will not be found.

- To search for special characters, such as a tab stop or a paragraph marker, or to limit the search in special ways, such as using a wild card character, select **Regular expression**. Then click ▶ in the **Find** and/or **replace with** fields to display a drop-down list of options. Using the available options, build the desired expression.



After making your selections, do the following:

- To replace all instances without review, click **Replace All**.
- To review the instances before replacing the material, click **Find Next** to locate the first place the material occurs, then either click **Find Next** to leave the text as-is and find the next occurrence or click **Replace** to replace the text and find the next occurrence.

19.3.3 Inserting Pictures and Symbols

Insert a Picture

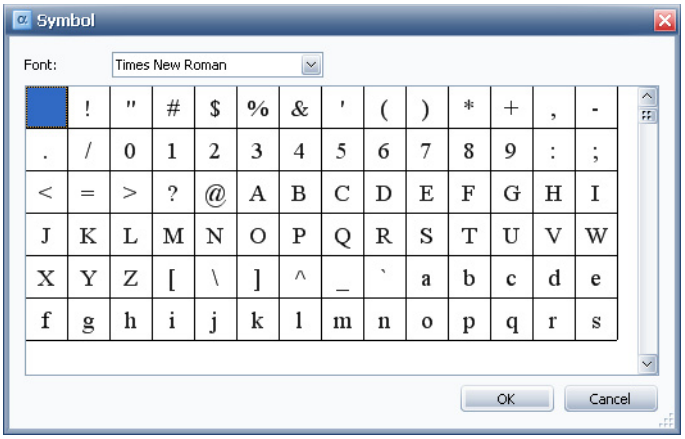
- Place the cursor into the desired position and choose **Insert > Items > Picture**.

- Select a picture and click **Open**. Resize the picture, if desired.

Insert a Symbol Character

- Place the cursor into the desired position and choose **Insert > Items > Symbol**.

- Select the character and click **OK**. Note that different symbols may be available depending on the font you selected.



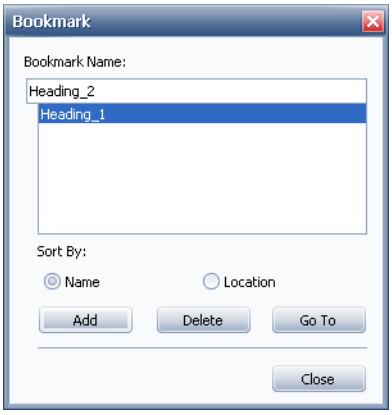
19.3.4 Inserting Bookmarks and Hyperlinks

A bookmark identifies a location or a selection of text that you name and identify for future reference. For example, you might use a bookmark to identify text that you want to revise at a later time. Whereas, a hyperlink can be text or graphics that you click to go to a specific location in a document, other file, web site or intranet site.

Insert a Bookmark

1. Place the cursor into the desired position and choose **Insert > Items > Bookmark**.

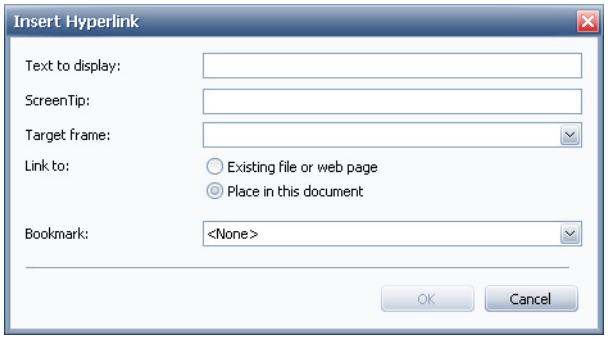
2. In the **Bookmark Name** field, type the bookmark's name, which must begin with a letter and can contain numbers. While you cannot include spaces in a bookmark name, you can use the underscore character to separate words (e.g., "Analysis_Section").



Insert a Hyperlink

1. Select the text or picture to be displayed as a hyperlink.
2. Choose **Insert > Items > Hyperlink**.

3. In the **Insert Hyperlink** window:



- Customize the tip that displays when a reader moves the pointer over the anchor text of the link, type the desired text in the ScreenTip field. If you do not specify a ScreenTip, the Word report template uses the path or address of the file as the tip. (This is similar to using the ALT or Title tags in HTML.)
- In the **Target frame** field, specify how you want the destination of the hyperlink to appear. You can open a page in the same frame, in the parent frame's page, in a new window or as a whole page.
- In the **Link to area**, do one of the following:
  - To link to an external file, type the URL (e.g., http://www.ReliaSoft.com) in the Address field.
  - To link to another location in the current document, select an existing bookmark in the Bookmark field.

The link will be active after the report is generated.

19.3.5 Inserting Tables and Setting Properties

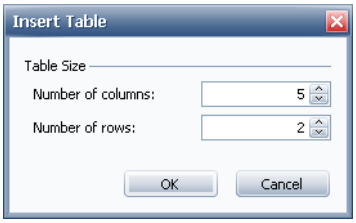
You have the option to insert a table to contain your data and results. The table properties can then be changed to suit your needs.

Note that these user-created tables are different from the table functions that return results in table form (e.g., functions with parameter bounds in Weibull++ or component failure in BlockSim). Table functions are not used in RGA.

Insert a Table

1. Place the cursor where you want to insert the table.
2. Choose T**able > Design > Insert Table** to open the **Insert Table** window.

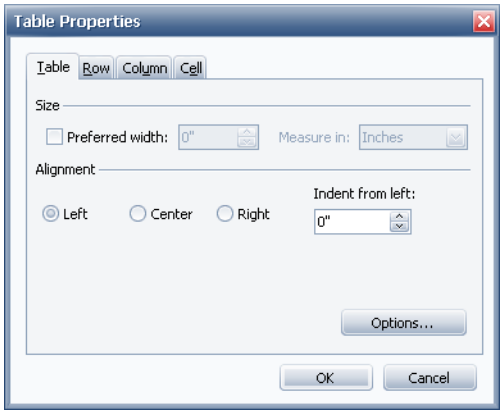
3. Enter the desired values in the **Number of columns** and **Number of rows** fields, then click **OK**.



Customize the Table Properties

To open the Table Properties window, choose **Table > Design > Properties**.

The following properties can be changed:



On the **Table** tab:

- By default, the table width varies depending on its contents. To force a particular size, select the **Preferred width** check box, then specify the width in either **Inches** (if you want to use a fixed measurement), or choose **Percent** (if you want to use a specific percentage of the available area).
- In the **Alignment** area, select how you want the table aligned on the page. Use the **Indent from left** field to enter the distance that you want to indent the table from the left margin.
- To apply custom borders and/or shading to selected cell(s) or entire table, click **Borders and Shading**. On the Borders tab, select the line color, style and width. Click the Shading tab to select a color.
- Click **Options** to open the Table Options window, where you can set: 1) default cell margins; 2) whether or not an additional space between cells is added; and 3) whether or not the columns will expand to accommodate unbroken text or a graphic.

On the **Row** tab:

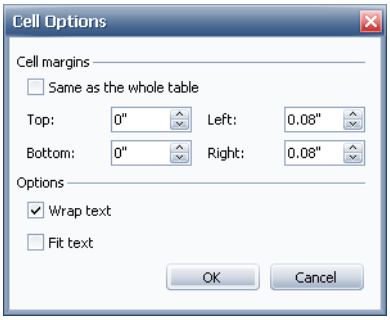
- To fix the height of a row, select the **Specify height** check box, then enter a value. The row height options are: **At Least** (the minimum height the cell can be) or **Exactly** (will not allow the row expand to accommodate the content).
- To allow the row to split across a page break, select the **Allow row to break across pages** check box.
- To have the row repeat at the top of each page, select the **Repeat as header row at the top of each page** check box.

On the **Column** tab:

- To fix the width of a column, select the **Preferred width** check box, then enter a value. Use the **Measure in** field to specify the width in either **Inches** (if you want to use a fixed measurement), or **Percent** (if you want to use a specific percentage of the available area).

On the **Cell** tab:

- To fix the width of the selected cells, select the **Preferred width** check box, then enter a value. Use the Measure in field to specify the width in either **Inches** (if you want to use a fixed measurement), or **Percent** (if you want to use a specific percentage of the available area). Note that this setting may be overridden if the Automatically Resize checkbox in the Table Options box is checked.)
- Use the **Vertical alignment** area to specify how you want the text to be aligned vertically in the cell.
- Click **Options** to open the Cell Options window, where you can set: 1) default cell margins; 2) whether or not the contents will be wrapped within the cell; and 3) whether or not the width of characters and spaces will be reduced to force the text to fit within the specified cell or column width.



19.4 Word Report Template Ribbon

The following is a list of all the tabs on the Word report template Ribbon. All commands available in the shortcut menus are also available on the Ribbon.

- **Home tab** contains commands related to using the Clipboard and using the Function Wizard and Plot Wizard.
- **Document tab** contains commands related to printing the report and laying out the page elements, such as column size, line spacing, text alignment and the like.
- **Insert tab** contains commands related to inserting items (e.g., symbol characters, hyperlinks), and working with page headers and footers.
- **View tab** contains commands related to how the Word report template pages are displayed.
- **Table tab** contains commands related to inserting and using Word report template tables.

19.4.1 Home Tab

The following commands are available in the Home tab of the Word report template's Ribbon.

Report

Create Report generates the report with calculated results and plots in Microsoft Word.

Function Wizard opens the Function Wizard, which allows you to build functions in the report template that return results (e.g., reliability values, failure rate, etc.) and other information from the associated data sources. Note that the functions will not return any results until the report is generated.

> **Tip:** After the function has been inserted by the wizard, you can copy and/or edit its settings directly in the template.

Plot Wizard opens the Plot Wizard, which allows you to insert plot holders for the same types of plots that are generated in folio or diagram plot sheets. Although you also have the option to copy/paste a static plot graphic from any folio or diagram plot sheet into the report template, using the Plot Wizard instead makes it easy to change the associated data source when you reuse the template and ensures that the plot in the generated report will always show the latest analysis. Note that the plot holders will not return any results until the report is generated.

Associate Data Sources allows you to define or change up to four default data sources that can be referenced by functions and plots in this report template.

Clipboard

- **Paste Special** opens a window where you can choose to paste the copied text as either RTF format, XTML format or as unformatted text.
- **Cut** cuts the selected objects. You can then paste the objects into another location within the same Word report template.
- **Copy** copies the selected objects to the Clipboard. You can then paste the objects into another location within the same Word report template.
- **Paste** pastes the contents of the Clipboard onto the report area.

Edit

- **Spelling** activates the Spell Check utility, which allows you to check the spelling within the template.
- **Find** and **Replace** open the Find and Replace window, which allows you to search through the current template for text and/or numbers and replace the matching text or numbers with other values.
- **Undo** cancels the last editing change you made to the document. You can cancel multiple actions by choosing **Undo** as many times as necessary.
- **Redo** reapplies the previously canceled action. You can redo multiple actions by choosing **Redo** again.

Font

- Use the options in this area to format text. You can specify the font, font size and font color. You can also apply special formatting (e.g., bold, italics, underline and/or highlight color) to the text.
- Click **Clear Formatting** to remove all formatting from the selected items and reset the formatting to the default values.

Paragraph

- **Paragraph** opens the Paragraph window, which allows you to specify the paragraph settings.
- Use the **Line Spacing** options to change the spacing between lines or the spacing before or after each paragraph.
- **¶ Show/Hide** displays paragraph markers and dots in place of the spaces between words.
- Use the alignment options to set how the paragraph should be aligned.

- Use the list options to change the paragraph into a list.

- To increase and decrease the paragraph indent, click **Increase Indent** or **Decrease Indent**.

19.4.2 Document Tab

The following commands are available in the Document tab of the Word report template's Ribbon.

Template

- **Open Template** opens a saved Word report template (*.rtt) file.
- **Save Template** opens the Save As window where you can save the current Word report template as a *.rtt file.

Note that each template works only in the application that was used to create it.

Print

Note that these print commands apply to the template only. To print the report, you first create the report (**Home > Report > Create Report**) then print it from Microsoft Word.

- **Print** sends the current template containing the function and plot placeholders to the printer.
- **Print Preview** opens the Print Preview window, which allows you to view how the current template will appear on the printed page.
- **Quick Print** prints the page.

Page Setup

- **Margins** displays a submenu where you can select the page margins either for the entire document or just the current selection.
- **Orientations** allows you to switch the report pages between landscape and portrait mode.
- **Size** allows you to select the paper size for the current selection.
- **Columns** allows you to split the text into two or more columns.
- **Breaks**
  - **Page** adds a page break.
  - **Column** adds a column break.
  - **Section (Next Page)** inserts a section break and forces the next section to start at the top of the next page.
  - **Section (Even Page)** inserts a section break and starts the next section on the next even-numbered page.
  - **Section (Odd Page)** inserts a section break and starts the next section on the next odd-numbered page.

- **Line Numbers**

  Add line number to entire page:

  - To number consecutively throughout the document, click **Continuous**.
  - To start with number 1 on each page, click **Restart Each Page**.
  - To start with number 1 after each section break, click **Restart Each Section**.

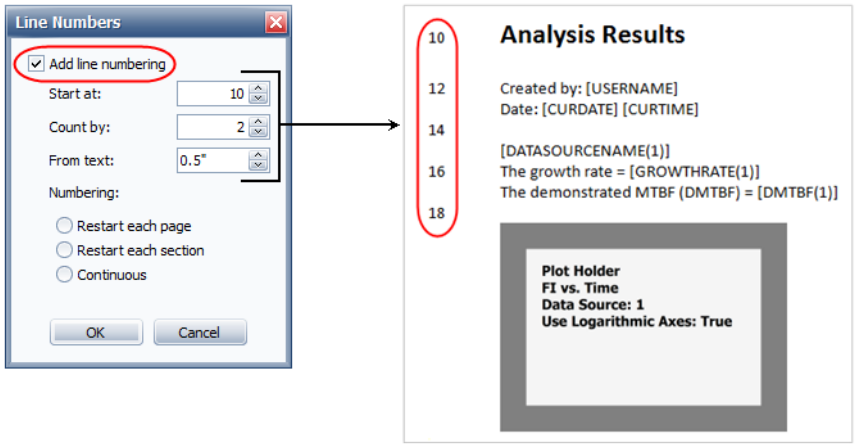
  Remove line numbers:

  - To remove line numbers from the entire document, click **None**.
  - To remove line numbers from selected text, click **Suppress for Current Paragraph**.

  Customize starting and increment numbers, and position of line numbers:

  The following options can be used alone or in combination.

  - To change the starting line number (e.g., from 1 to 10 or 100), enter the new value in the **Start at** field.
  - To change the incremental line number (e.g. from 1, 2, 3 to 10, 20, 30 or 50, 100, 150), enter a new value in the **Count** by field.
  - To change the position of the numbers in the left margin, enter a new value (in inches) in the **From text** field. A positive value will move the numbers further to the left. A value of zero will return the numbers to the default location.



Tip: The same Line Numbers options are available on the Page Layout tab in Microsoft Word. You have the option to change or remove the line number settings after the report has been generated.

Protect

- Protect Document opens a window that allows you to prevent the Word report template from being modified. You have the option to enter a password, if desired.
- Range Editing Permissions opens a window that allows you to specify which users and/or groups can make changes to a template. This option is not available when the document is protected.
- Unprotect Document opens a window to enter your password to remove the protection from a protected document and allows changes to be made to that document. This window will not display if the document was protected without a password.

19.4.3 Insert Tab

The following commands are available in the Insert tab of the Word report template's Ribbon.

Items

- **Page Break** inserts a page break at the cursor location.
- **Picture** inserts a picture at the cursor location. The available formats are *.bmp, *.dib, *.jpg, *.jpeg, *.png, *.gif, *.tif, *.tiff, *.emf and *.wmf.
- **Symbol** inserts a symbol character at the cursor location.
- **Hyperlink** inserts a hyperlink at the cursor location.
- **Bookmark** inserts a bookmark at the cursor location.

Header/Footer

- **Header** and **Footer** enable either the page header area or the page footer area.
- To switch to a header while working in a footer, click **Go to Header**.
- To switch to a footer while working in a header, click **Go to Footer.**
- Click **Go to Next Header/Footer** to display the next section's header or footer.
- Click **Go to Previous Header/Footer** to display the previous section's header or footer.
- Click **Link to Previous** to link the current section's header and footer to the previous section's header and footer.
- **Page Number** inserts the page number at the cursor location.
- **Page Count** inserts the number of pages in the document at the cursor location.
- To exit the header or footer and return to the main body of the text, click **Close Header and Footer**.

19.4.4 View Tab

The following commands are available in the View tab of the Word report template's Ribbon.

Document Views

- **Simple View** displays the Word report template with all page layout features disabled (i.e., headers and footers, horizontal and vertical rulers, line numbers and page and column breaks).
- **Draft View** displays the Word report template with certain page layout elements disabled, including the headers and footers, page breaks, line numbers and vertical ruler.
- **Print Layout** displays the Word report template as it will be printed with all page layout elements enabled.

Zoom

- **Zoom In** increases the degree of magnification.
- **Zoom Out** decreases the degree of magnification.

Show

- **Horizontal Ruler** displays a horizontal ruler at the top of the document area.
- **Vertical Ruler** displays a vertical ruler along the left edge of the document area.

Options

- Click **Different First Page** to specify that the first page of the document will have different header and footer than the rest of the document.
- Click **Different Odd Even Pages** to specify that even pages will have different headers and footers than odd pages.

19.4.5 Table Tab

The following commands are available in the Table tab of the Word report template's Ribbon.

Design

- **Insert Table** allows you to define the number of columns and row for the table being inserted. There are two options:
  - Click the icon to enter the number of columns and rows in the appropriate fields of the Insert Table window.
  - Use the **Insert Table** drop-down list to select the number of columns and rows that you want (drag the pointer over the squares). The maximum is 10 columns and 8 rows.
- **Select**
  - Select **Cell**, **Select Column** or **Select Row** selects the cell, column or row that the cursor is currently located in.
  - **Select Table** selects the entire table.
- **Properties** opens the Table Properties window where you can set the properties of a user-created table.
- **View Gridlines** displays blue gridlines on inside and outside borders of the selected table. The gridlines will not display around table cells that currently have borders. The gridlines do not display in the generated report.
- **Merge Cells** merges the currently selected cells.
- **Split Cells** splits the currently selected cells into multiple cells.
- **Split Table** divides the table into two pieces at the current location of the cursor. The currently selected row becomes the first row of the new table.
- **Shading** opens a window where you can pick a background color for the selected cells.
- **Borders** displays a submenu where you can select which table and cell borders to enable/disable.

For more information, see Inserting Tables and Setting Properties. The borders and shading option can also be accessed on the Table tab of the Table Properties window.

Borders

- **Line Style** sets the style of the border lines.
- **Line Weight** sets the thickness of the border lines.
- **Pen Color** sets the color of the border lines.

Alignment

Use the options in this area to specify how the selected table cells will be aligned.

Rows/Columns

- **Insert Rows Above** adds a row above the selected row and **Insert Rows Below** adds a row below the selected row.
- **Insert Columns to the Left** adds a column to the left of the selected column and **Insert Columns to the Right** adds a column to the right of the selected column.
- **Insert Cells** opens the Insert Cells window, where you choose the direction in which you want to shift the surrounding cells after inserting the new cell(s).
- **Delete**
  - **Delete Cells** opens the Delete Cells window, where you choose to delete the cell and shift the other cells to the left or right, or deleted the entire row or column that contains the cell.
  - **Delete Columns** and **Delete Row** deletes the selected column(s) or row(s).
  - **Delete Table** deletes the entire table.

19.5 Word Report Example

This example shows how you can generate a custom report from your data. In this case, you will use Weibull++ to compare selected results from two life data analyses.

*The data sets used in this example are available in the example database installed with the Weibull++ software (called "Weibull_ALTA9_QuickStart_Rev2.rsgz9"). To access this database file, choose **File > Help**, click **Open Examples Folder**, then browse for the file in the Weibull sub-folder. The folios are "Bulb A – In-House Data" and "Bulb A – Supplier Data".*

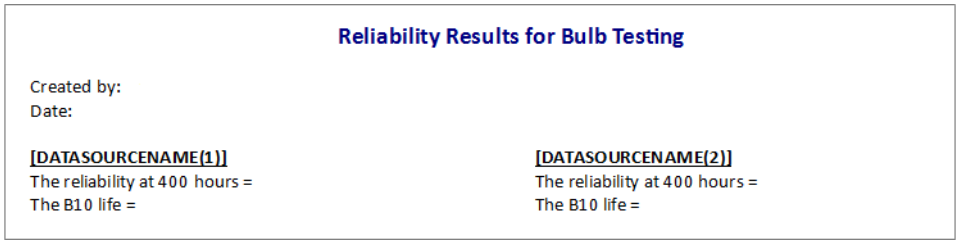
This example uses the Weibull++ software, but a similar process can be used to generate custom analyzes in ALTA, RGA and BlockSim.

Building a Report Template

1. Add a Word report template to the project (**Insert > Reports and Plots > Word Report Template**) and select the first data source, Bulb A – In-House Data.

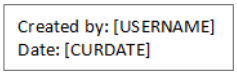
2. Click the **Associate Data Sources** icon and assign the second data source, Bulb A – Supplier Data.

3. Because you want to view the plots side by side, change the template layout to landscape (**Document > Page Setup > Orientation > Landscape**).
4. Type the title for the report and other commentary, as shown next. You can change the font type, size and color (**Home > Font**), if desired.

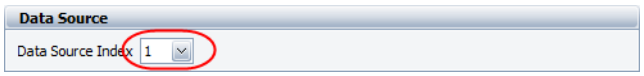


5. Insert the user name and date functions.
   - Place the cursor after the "Created by:" label and then click the **Function Wizard** icon on the template's control panel.

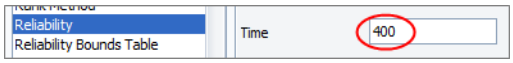
   - Select `User Name` from the function list, then click **Insert**.
   - Repeat the process to insert the Current Date function after the "Date:" label. The inserted functions are shown next.



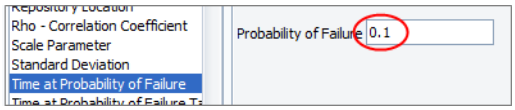
6. Use the Function Wizard to insert the data source name, reliability and B10 functions for the in-house data. For each function, select **1** as the data source index.



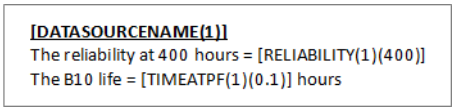
   - **Data Source**: Select `Data Source Name` from the function list.
   - **Reliability at 400 hours**: Select `Reliability` from the function list and enter **400** for the time input.



   - **B10 Life**: Select `Time at Probability of Failure` from the function list and enter **0.1** for the probability of failure input.

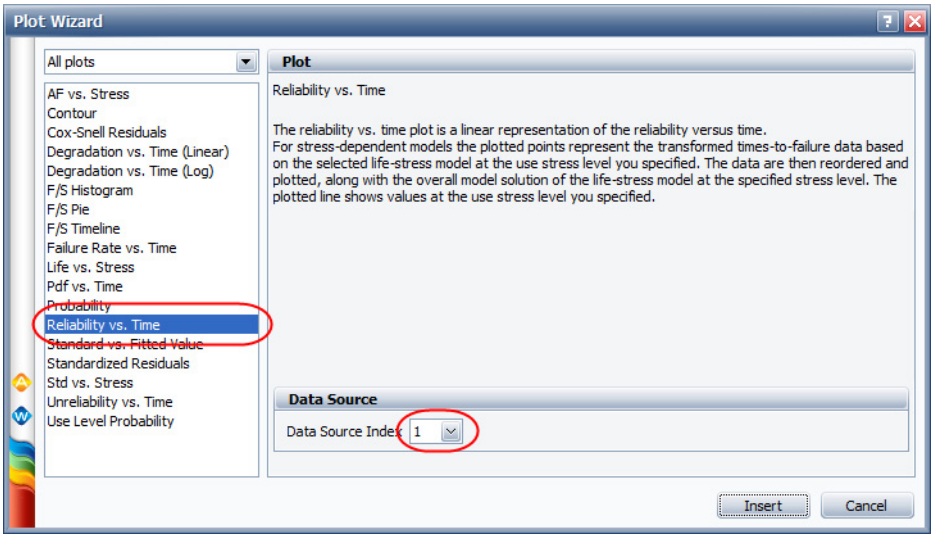


   The inserted functions are shown next.

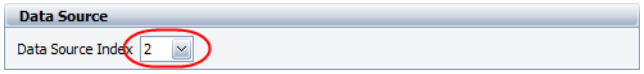


7. Insert the reliability plot for the in-house data (data source 1).
   - Place the cursor where you want to insert the plot and then click the Plot Wizard icon on the template's control panel.

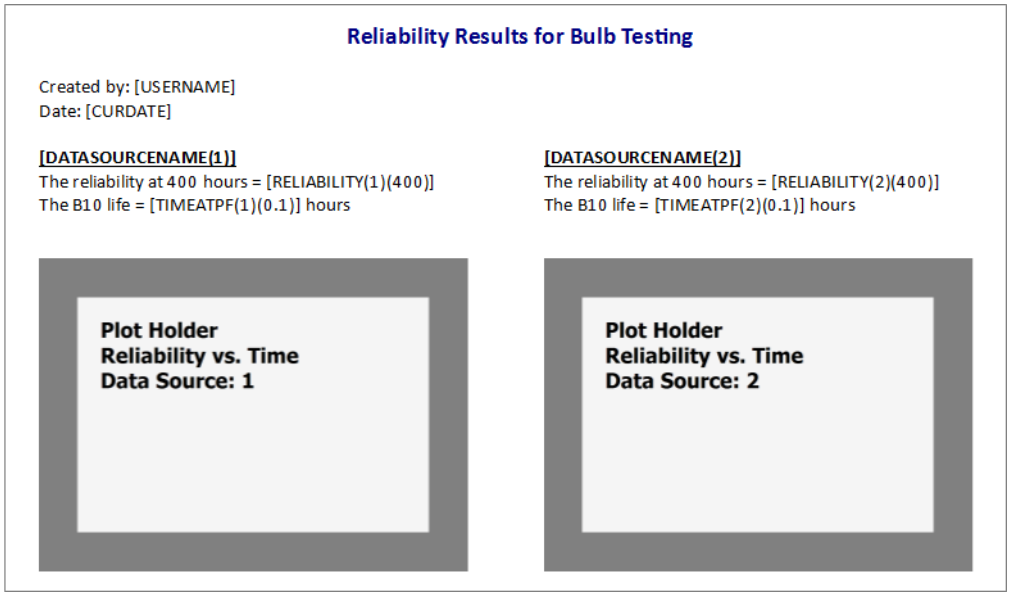
- Select **Reliability vs. Time** from the plot list, select **1** as the data source index and click **Insert**.



8. Resize the plot holder to the desired dimensions.
9. Repeat the same process to insert the data source name, reliability and B10 life functions, and the reliability plot for the supplier data. Be sure to select **2** as the data source index.



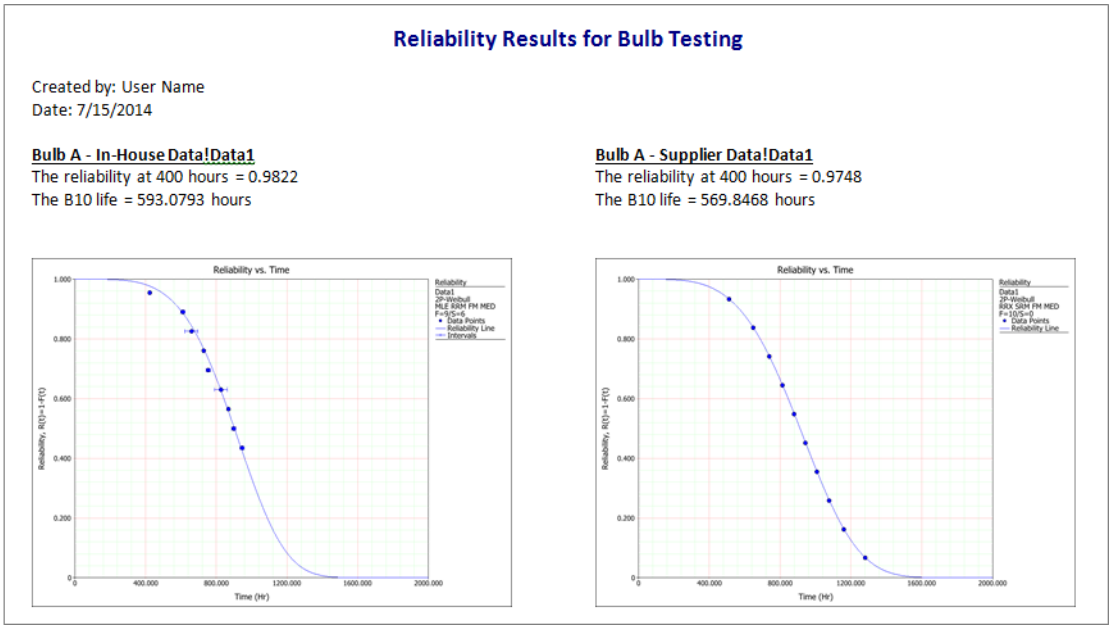
When you are done, the report template will look similar to this:



Generating the Report

1. When you have finished building the report template, generate the report in Microsoft Word by choosing **Home > Report > Create Report**. You can also click the icon on the control panel.

2. You can view and save the report in Microsoft Word.



Reliability Results for Bulb Testing

Created by: User Name
Date: 7/15/2014

Bulb A - In-House Data!Data1
The reliability at 400 hours = 0.9822
The B10 life = 593.0793 hours

Bulb A - Supplier Data!Data1
The reliability at 400 hours = 0.9748
The B10 life = 569.8468 hours

FRED Reports 20

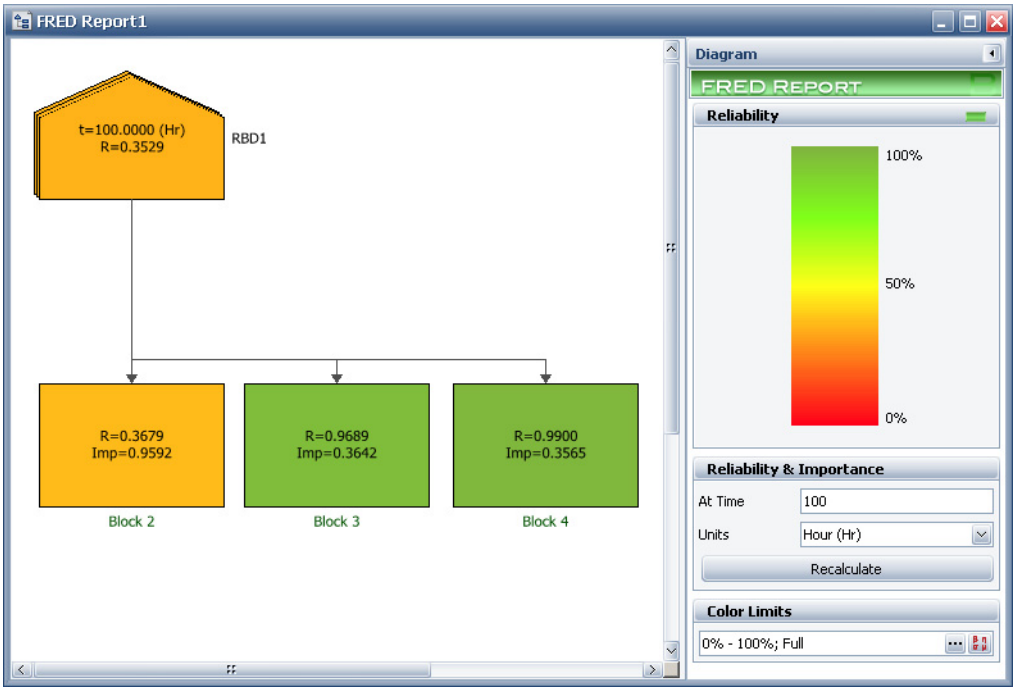
You can use the analytical results and maintainability/availability simulation results for your BlockSim diagrams and fault trees to generate Failure Reporting, Evaluating and Display (FRED) Reports. These reports provide a graphical demonstration of the reliability and maintainability/availability characteristics of the components in a system and help to identify the components that may require improvement.

Creating a FRED Report

To add a FRED report to the project, choose **Insert > Reports and Plots > FRED Report**.

In the Select Diagram window that appears, select the diagram that you want to create the report for. You can select any diagram that is currently in an analyzed state; diagrams that have not been analyzed will be displayed in the list but cannot be selected.

The FRED report will be displayed, as shown next.



For any subdiagram that appears in the FRED report, you can add a row below it that shows the blocks in the subdiagram. To do this, select the subdiagram and choose **FRED > Levels > Append Level Below**.

Note: For simulation diagrams that contain subdiagrams, you must select the **Report subdiagram results** check box on the Analysis Settings page of the diagram control panel in order for the subdiagram results to be available for use in the FRED report. If you have not selected this check box, subdiagram blocks in the FRED report will show "N/A" instead of results. If you attempt to append a level below such a block, you will see a message that explains the problem and offers you the opportunity to resimulate the diagram with the subdiagram results reported.

To remove a single row, select the subdiagram block that the row was appended to and then choose **FRED > Levels > Remove Level(s) Below**.

You can clear all additional rows by choosing **FRED > Levels > Restart FRED Report**. This clears all levels of the FRED report except the original diagram so that you can restart the report creation.

You can print the report by choosing **Home > Print > Print**. You can also import and export FRED reports among projects by choosing **Project > Management > Import/Export**.

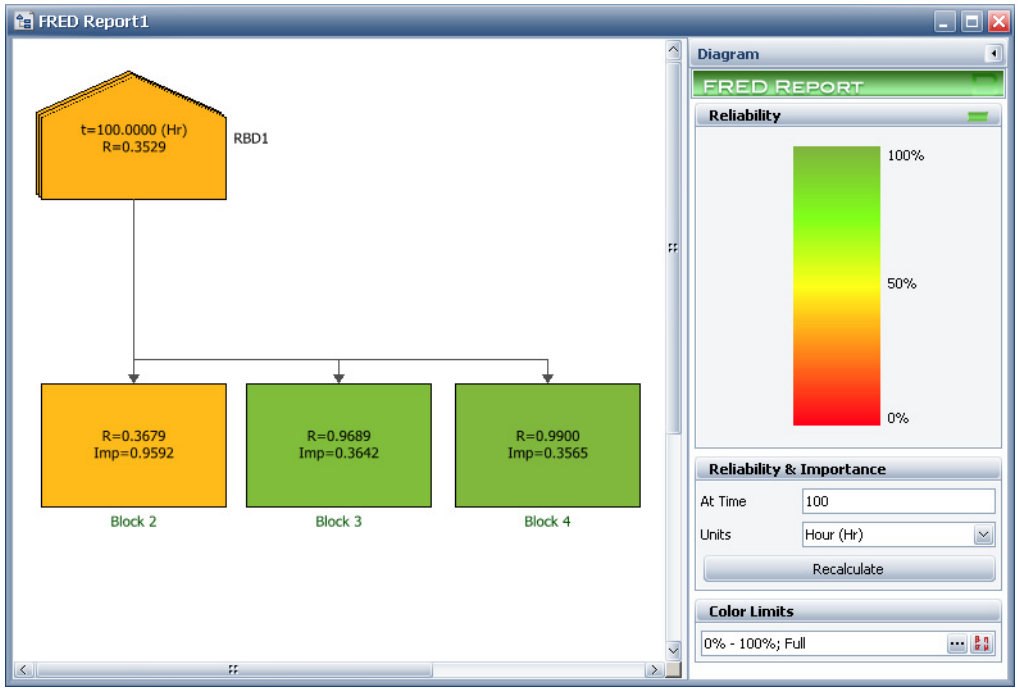
The FRED reports that you create in the current project will be listed under the Reports folder in the current project explorer.

Diagram Style Window

The Diagram Style window for FRED reports is the same as that used with other diagrams. To open it, choose **FRED > Options > Diagram Style**.

20.1 Analytical FRED Reports

Analytical FRED reports illustrate the reliability characteristics of a system's components in a flexible, color-coded format. This report can be used to easily identify the components most in need of improvement in order to achieve the desired system reliability.



The pentagon-shaped block in the first row represents the reliability results for the first diagram included in the analytical FRED report.

- **t** displays the time at which the reliability is reported for the system and its blocks and at which time the importance factor is reported for the blocks. You set the time in the **At Time** field in the control panel.
- **R** displays the reliability of the system.

The boxes in the following row represent the blocks and subdiagrams in the first diagram. Green boxes represent the blocks with the best reliability results, yellow boxes represent the next best blocks and red boxes represent the blocks with the worst reliability results.

- **R** displays the reliability of the block at time **t**.
- **Imp** displays the reliability importance of the block at time **t**. *For details on measuring reliability importance, see page 439 in Chapter 15*.

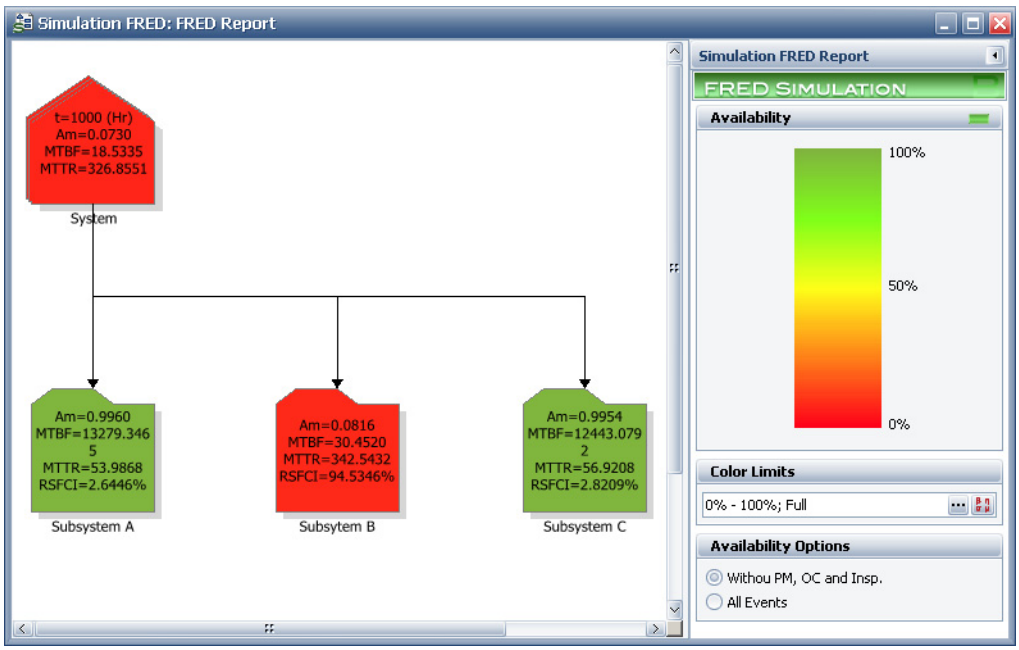
While two blocks may have the same reliability, their importance factors may differ because of their positions within the RBD. For example, a block in a series configuration may have a greater importance factor than a block in a parallel configuration, because any failure of the block in the series configuration will bring the system down.

By default, a spectrum scale is used to rank the blocks according to their reliability, as displayed in the report's legend. You can change the reliability range that is represented by the color scale, if desired, either by choosing **FRED > Options > Set Color Limits** or by clicking the **(...)** button on the control panel to open the Set Color Limits window and set a new lower limit and upper limit. For example, if you want to highlight the blocks with a reliability under a certain percentage, set the lower limit of the spectrum to that value, then refresh the plot. To automatically determine a min/max for the Color Limits based on the blocks in the diagram, click the **Calculate Color Limits** button.

You can recalculate the report to display the blocks' reliabilities and reliability importance at different times by entering the time in the **At Time** field in the **Reliability & Importance** area of the control panel. You can also set the time units by selecting an option from the **Units** drop-down list. When you click **Recalculate**, the analytical FRED report will update all of the blocks to show their reliabilities and reliability importance at the new time and/or units.

20.2 Simulation FRED Reports

Simulation FRED reports illustrate the maintainability/availability characteristics of a system's components in a flexible, color-coded format. This report can be used to easily identify the components most in need of improvement in order to achieve the desired system maintainability/availability.



The pentagon-shaped block in the first row represents the simulation results for the first diagram included in the simulation FRED report.

- **t** displays the simulation time at which the results are reported for the system and its blocks.
- **Am** displays the mean availability of the system at simulation time t.
- **MTBF** displays the mean time between failures for the system at simulation time t.
- **MTTR** displays the mean time to repair for the system at simulation time t.

The boxes in the following row represent the subdiagrams and blocks in this diagram. Green boxes represent the blocks with the best maintainability/availability results, yellow boxes represent the next best blocks in terms of maintainability/availability results and red boxes represent the blocks with the worst maintainability/availability results.

- **Am** displays the mean availability of the block at simulation time t.
- **MTBF** displays the mean time between failures for the block at simulation time t.
- **MTTR** displays the mean time to repair the block at simulation time t.
- **RSFCI** displays the ReliaSoft Failure Criticality Index for the block at simulation time t. This is a relative index showing the percentage of times that a failure of the block caused a system failure (i.e. the number of system

failures caused by the block divided by the total number of system failures). Note that this relates only to failures; it does not include every event that could bring the system down (such as routine maintenance).

Note: For simulation diagrams that contain subdiagrams, you must select the **Report subdiagram results** check box on the Analysis Settings page of the diagram control panel in order for the subdiagram results to be available for use in the FRED report. If you have not selected this check box, subdiagram blocks in the FRED report will show "N/A" instead of results. If you attempt to append a level below such a block, you will see a message that explains the problem and offers you the opportunity to resimulate the diagram with the subdiagram results reported.

By default, a spectrum scale is used to rank the blocks according to their maintainability/availability, as displayed in the report's legend. You can change the maintainability/availability range that is represented by the color scale, if desired, either by choosing **FRED > Options > Set Color Limits** or by clicking the (**...**) button on the control panel to open the Set Color Limits window and set a new lower limit and upper limit. For example, if you want to highlight the blocks with a reliability under a certain percentage, set the lower limit of the spectrum to that value, then refresh the plot. To automatically determine a min/max for the Color Limits based on the blocks in the diagram, click the **Calculate Color Limits** button.

By default, the information is calculated without taking into account preventive maintenance tasks, on-condition tasks and inspection tasks. To include those tasks in the calculated results, in the **Availability Options** area of the control panel, select the **All Events** option. The report window refreshes to show the updated values. To change what the results are based on, select the **Without PM**, **OC and Insp.** option.

You can set the time units by selecting an option from the **Units** drop-down list. The simulation FRED report will automatically update all of the blocks to show results based on the new units.

Printing 21

Three printing options are available for the diagram sheet, fault tree sheet, phase diagram sheet, plot/overlay plot sheets and FRED report. These options, which are available under the **Home** tab, are **Page Setup**, **Print Layout** and **Print**. The printing options available for workbooks, also on the **Home** tab, are **Print Preview** and **Print**. The printing options will vary depending on the type of diagram that currently has focus.

This chapter includes information on these printing options, including

- Using the Page Setup window to make selections for how the pages will be printed - Section 21.1 (p. 525)
- Previewing how the page(s) will be printed - Section 21.2 (p. 529)
- Using the Print Layout option to determine how a diagram or FRED report will appear on the printed page - Section 21.3 (p. 529)
- Using the Print window to specify the printer to send the pages to - Section 21.4 (p. 530)

21.1 Page Setup

Page Setup allows you to make selections for how the page(s) will be printed. You can access the Page Setup window by choosing **Home > Print > Page Setup**.

The Page Setup window is made up of three tabs. The appearance of this window will vary depending on the type of sheet that currently has focus (e.g., diagram sheet, workbook, plot/overlay plot sheet, etc.). Each of the tabs in the Page Setup window and its components are presented next.

The **Properties** button is always available. It opens the Windows Page Setup window, which allows you to set printing properties for the selected printer.

21.1.1 Page Setup: Page Tab

The Page tab of the Page Setup window allows you to set the layout, scaling and options of the page for the printed sheet.

The following option is available for all sheets:

- **Orientation** displays how the sheet will be printed on the page. (The orientation can also be specified in the Windows Page Setup window, accessed by clicking **Properties**.)

For diagrams, fault trees, phase diagrams and FRED Reports the following options apply.

- **Scaling** allows you to specify how large the item is printed on the page.
  - **No Fit (100%)** prints the item at its actual size.
  - **Fit to __ page(s) wide by __ tall** allows you to specify the number of pages you want the diagram printed on.
- **Options** allows you to specify the details of the printing process.
  - **Stick Pages** prints the pages without margins.
  - **Print Empty Pages** prints any blank pages in the item. If not selected, blank pages will not be printed.

For workbooks and reports the following options apply:

- **Options**
  - **Print Row Headings** prints the row headings at the left side of the rows.
  - **Print Gridlines** the grid lines of the sheet will be visible on the printed page.
  - **Print Column Headings** prints the column headings above the columns.

For plot sheets the following option applies:

- **Options**
  - **Keep Aspect Ratio** allows you to keep the ratio of the plot height to width when the plot is printed. If your printed plot goes off the page, checking this option can help.

21.1.2 Page Setup: Margins Page

The Margins tab of the Page Setup window allows you to set the margins for the printed page. You can specify the top, bottom, left and right margins, as well as the margins for the header and footer.

The following option is available for all diagrams, reports and workbooks:

- **Units** allows you to specify the units in which the margins will be printed.
  - **System's defaults** uses the units used by your computer as the measurement unit by which the margins will be printed.
  - **Inches** uses inches as the measurement unit by which the margins will be printed.
  - **Centimeters** uses centimeters as the measurement unit by which the margins will be printed.

21.1.3 Page Setup: Header/Footer Page

The Header/Footer tab of the Page Setup window allows you to edit the header and footer for the printed page.

The Header and Footer sections of this window have similar customization options, as explained next.

- **Print Header** prints the header information.
- **Print Footer** prints the footer information.
- **Header/Footer Preview Display Boxes** displays the header/footer format codes and their alignment on the page (left, center or right).
- **Header/Footer Format** allows you to select a predefined header/footer format from the drop-down lists. With the exception of the Custom option, the available header/footer format options are different for headers and footers.
  - **Custom** allows you to create a custom header/footer format using the available format codes.
- **Set Font** opens the Select Font window, which allows you to specify the font properties to be used in the header or footer.

The toolbar on the Header/Footer page allows you to edit the text and insert fields into a section of the header or footer and contains the following tools:

Cut cuts the selected text to the Clipboard. Data stored in the Clipboard can be pasted into this and other applications.

Copy copies the selected text to the Clipboard. Data stored in the Clipboard can be pasted into this and other applications.

Paste pastes the contents of the Clipboard into the current control.

Date inserts the date.

Time inserts the time.

Page Number inserts the page number.

Number of Pages inserts the total number of pages.

Repository inserts the path to the current repository.

Project Name inserts the project name.

Sheet Name prints the sheet name.

Item Name Inserts the name of the active sheet with which the active plot sheet is associated.

Application Name inserts the program name (i.e., BlockSim or RENO).

Company Name inserts the company name specified in the User Login and Contact Information window (**My Portal > Users > My Profile**).

User Name inserts the user name specified in the User Login and Contact Information window.

21.1.4 Header/Footer Format Codes

The format codes that can be used for printing the headers and footers in diagrams, analytical plot/overlay plot/ simulation plot sheets, workbooks and FRED reports are presented next.

Format Code	Description	Available For		
		Diagrams/Fault Trees	Plot/Overlay Plot sheets	Workbooks
&L	Left aligns the item.	X	X	X
&C	Center aligns the item.	X	X	X
&R	Right aligns the item.	X	X	X
&A	Prints the workbook name.			X
&D	Prints the current date.	X	X	X
&T	Prints the current time.	X	X	X
&P	Prints the page number.	X	X	X
&P+number (e.g., &P+2)	Prints the page number plus the specified number.	X	X	X
&P-number (e.g., &P-2)	Prints the page number minus the specified number.	X	X	X
&&	Prints an ampersand.	X	X	X
&N	Prints the total number of pages.	X	X	X
&ZA	Prints the name of the application (i.e.,BlockSim or RENO)	X	X	X
&ZC	Prints the company name.	X	X	X
&ZU	Prints the user name.	X	X	X
&ZP	Prints the project name.	X	X	X
&ZF	Prints the project file name and location.	X	X	
&ZI	Prints the name of the diagram, fault tree or plot/overlay plot sheet.	X	X	X
&ZL	Print the path to the current repository	X	X	X

By default, text will be centered in the headers/footers unless otherwise specified. Please note that the alignment codes (e.g., &L, &C, &R) restart each format code section. For example, in the format code &L&P &R&ZI, the page number will be left aligned and the sheet name will be right aligned.

21.2 Print Preview

The Print Preview window for diagrams, analysis workbooks or plots can be accessed by choosing **Home > Print > Print Preview** or by clicking **Preview** in the Print window.

Depending on your screen resolution and screen size, the print preview of the workbook or plot may be missing some lines. However, the lines will appear on the printout.

The toolbar in the Print Preview window contains quick access icons that activate different commands. It also allows you to enter the number of the page that you want to appear in the **Preview Area** in the **Current Page** input box. For example, if you enter **2**, then the second page of the spreadsheet/plot will appear in the Preview area. The toolbar also contains the following tools:

Page Setup opens the Page Setup window.

Print sends the current document or sheet to the printer.

Single Page Layout displays one page in the Preview area.

Double Page Layout displays two pages in the Preview area.

Four Page Layout displays four pages in the Preview area.

Nine Page Layout displays nine pages in the Preview area.

Depending on your screen resolution and screen size, the print preview may be missing some lines. However, the lines will appear on the printout. The print preview will display the correct positioning and location of the data on the preview. The page orientation of the print preview depends upon the orientation selected on the first page of the Page Setup window (i.e., portrait or landscape).

21.3 Print Layout

The Print Layout option allows you to view how a diagram or FRED report will appear on the printed page. The page orientation (i.e., portrait or landscape) of the print layout depends on the orientation selected in the Page Setup window. You can view the print layout by choosing **Home > Print > Print Layout**.

When this option is selected, the page boundaries for the diagram or FRED report will be displayed.

To exit the print layout mode, choose **Home > Print > Print Layout**.

21.4 Print Window

The Print window allows you to specify the printer to send the diagram, fault tree, phase diagram, FRED report, plot/overlay plot sheet or workbook to, as well as printing options for the sheet.

Printing Diagrams

If a diagram is in focus, the following options are available:

- **Diagrams to Print**
  - Choose one of the following options:
    - **Current** to print the current diagram only. Select the **Include Subdiagrams** option to include the diagrams subdiagrams when printing.
    - **Choose Multiple** to print multiple diagrams from the same project.
  - **Options**
    - Select **Print Grid** to print the grid with the diagram.
    - Select **Print Background Color** to print the background color with the diagram.

If you choose to print the current diagram, click OK.

If you choose to print multiple diagrams, click **Next**. A list of available diagrams will be displayed; choose which diagrams to print, then click **OK**.

Printing Plots and Reports

If a plot or a report is in focus, the following options are available:

- **Printer**
  - **Name** allows you to use a drop-down menu to select the printer you want to send the item to. Once you select a printer, the **Status**, **Driver**, **Where** and **Comments** information will automatically be updated to reflect the corresponding information about the selected printer.
  - **Print to File** opens the Print to file window when you click **OK** in the Print window. This option allows you to specify a file name and extension for the printed item, which will then be printed to the file instead of sent to the specified printer. If not selected, the item will be sent to the specified printer.
- **Properties** opens the printer's Document Properties window, which allows you to set printing properties for the selected printer.
- **Orientation** displays how the sheet will be printed on the page. The orientation can also be specified in the Document Properties window, accessed by clicking **Properties**.
  - **Portrait** sets the page orientation to portrait (tall and narrow).
  - **Landscape** sets the page orientation to landscape (short and wide).
- **Copies** allows you to specify the number of copies of the item you want to print.
  - **Collate** collates sheets sent to the printer if more than one copy was printed. If not selected, the printer will print multiple copies of each sheet in their original print order.

If a workbook or report is in focus, the Print window will have additional options:

- **Page Range**
  - **All Pages** prints all the pages in the item.
  - **From __ to __** prints only the specified page range for the item. For example, if an item is 10 pages long and you enter From “5” to “10”, only the last 5 pages will be printed.
- **Print What**
  - **Active Sheet** prints only the currently active sheet in the spreadsheet.
  - **Entire Book** prints all the sheets in the workbook.
  - **Selection** prints only the selected cell range in the currently active sheet in the workbook. To select a cell range, place the cursor in the cell that you want to be the uppermost left cell of the range. While holding down the left mouse button, drag the cursor to the cell that you want to be the lowermost right cell of the range. When you release the mouse, the selected cells will be highlighted with a black background. It is this selected range that will be displayed on the printed page.
- **Preview** opens the Print Preview window, which allows you to preview the pages before sending to the printer. Please note that closing the Print Preview window will return you to the current workbook’s window, not the Print window.

Setup Windows 22

All Synthesis desktop applications provide two setup windows that allow you to customize workspace preferences to fit the way you work:

- The **Synthesis Setup** helps you customize the preferences that apply to every Synthesis application on your computer. These preferences can be changed in any application at any time.
- The **Application Setup** helps you customize the preferences that are specific to the Synthesis application that you are currently using.

For both windows, the preferences are stored on your computer and will be the same for any Synthesis repository that you open. Although some settings may affect display preferences and analysis defaults, they do not affect data that have already been saved in a database. In other words, different users can have different preferences and work on the same project in a database without impacting each other's workspace preferences.

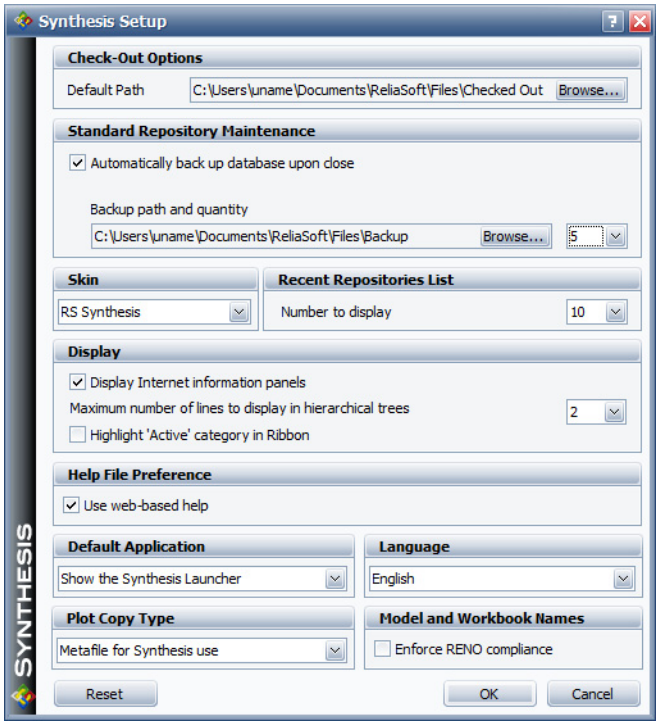
Note: Any configurable settings that must be the same for all users cooperating on the same project will be stored and managed directly in the database, as discussed in Repository Settings (*see page 62 in Chapter 2*) and other relevant sections.

22.1 Synthesis Setup

The Synthesis Setup window contains settings that apply to every Synthesis application on your computer (and can be changed from within any Synthesis application).

To open the Synthesis Setup window, choose **File > Synthesis Setup**.

The Synthesis Setup window is shown next.



The following settings are available:

- **Check-Out Options**
  - **Default Path** allows you to specify the default location for projects that you check out from a database.
- **Standard Repository Maintenance**

  The following settings are applicable for standard database files (*.rsr9) only. See page 69 in Chapter 2 for more tips on keeping both standard and enterprise databases running smoothly.
  - **Automatically back up database upon close** will automatically save a backup of the standard database when you close the application. The backup file will be saved as a repository (*.rsr9) file in the location specified. You can also do this manually at any time by choosing **File > Save As**.
  - **Backup path and quantity** allows you to specify the location for saving backup databases and the number of backups that can be saved. Note that only one backup will be saved per day (e.g., if you open and close the same database three times in one day, only the final closing of the database will be saved in the backup folder). The backup file name will be the name of the database file with the date appended to it in year-month-day format (e.g., RSRepository1_2011-08-15.rsr9). The **Number to display** drop-down list indicates how many days of backups can be automatically saved (up to 10).
- Use the **Skin** drop-down list to change the color scheme of the software interface. Selecting a scheme allows you to preview the changes in the Synthesis Setup window, but the selection will not be applied to the rest of the interface windows until you click **OK**. This setting applies to all Synthesis desktop applications on your computer; the change takes effect the next time you launch each application.

> **Note:** The skins managed here control the overall appearance of the interface. The software also offers diagram skins, which control the default appearance of new diagrams. *See page 203 in Chapter 6*.

- In the **Recent Repositories List** area, use the **Number to display** drop-down list to set the maximum number of recently saved database files to be displayed in the **Recent Repositories** list in the Backstage view.
- **Display**
  - If you would like to hide the ReliaSoft Online pane in the Backstage view and the Reliability Web Notes page in My Portal (e.g., because you do not have an Internet connection or you simply want to save the space allocated to these features), clear the **Display Internet information panels** check box.
  - **Maximum number of lines to display in hierarchical trees** allows you to set the maximum number of lines to display the names and descriptions in any of the hierarchical tree interfaces within any Synthesis application on this computer (such as the system hierarchies in Xfmea, RCM++, RBI, Lambda Predict and MPC).
  - Select the **Highlight 'Active' category in Ribbon** check box to apply the highlight color to the label that shows which tabs in the Ribbon are currently active.
- **Help File Preference**
  - Select the **Use web-based help** check box to always make F1 and other help commands open the most up-to-date help file from the Internet when possible. If you clear this check box, they open the 'local' help file that was installed with the software.
- Use the **Default Application** drop-down list to select which Synthesis application will open when you double-click a database file (*.rsr9) or enterprise connection file (*.rserp). If you want to be prompted to choose which Synthesis application to open each time you double-click a file, select **Show the Synthesis Launcher**.
- Use the **Language** drop-down list to select which language the Synthesis applications will use for the software interface. Note that this option will not be visible/applicable for applications that are currently available only in English. For a list of the applications and their supported languages, see http://www.ReliaSoft.com/languages/index.htm.
- Use the **Plot Copy Type** drop-down list to select which method to use by default when copying a plot to the Clipboard as an image. If you will be pasting copied plots into Synthesis spreadsheets (e.g., analysis workbooks), choose **Metafile for Synthesis Use**. If you will be pasting them into external applications, choose **Bitmap** or **Metafile for External Use**. This default setting is used when you copy a plot using **Home > Copy** or **CTRL+C**.
  - In Weibull++, ALTA, BlockSim, RENO, RGA and DOE++, if you want to choose the graphic type on the spot, you can do so by choosing **Plot > Copy Plot Graphic**, and then selecting the type.
  - In Xfmea, RCM++, Lambda Predict and RBI, you can choose the graphic type on the spot by right-clicking the plot and choosing the **Copy Plot Graphic** command on the shortcut menu.
- In the **Model and Workbook Names** area, you can select the **Enforce RENO compliance** check box to disallow spaces and special characters in names of models and analysis workbooks. This will ensure that all models and analysis workbooks can be used in RENO equations.

To reset all the options in the Synthesis Setup back to their default settings, click the **Reset** button at the bottom of the window. The change will apply to all Synthesis desktop applications on this computer.

To clear any saved alternative logins from this computer, click the **Forget Alternative Login** button. *For more information on alternative logins, see page 52 in Chapter 2*.

22.2 Application Setup

The Application Setup window contains settings that apply specifically to BlockSim and RENO.

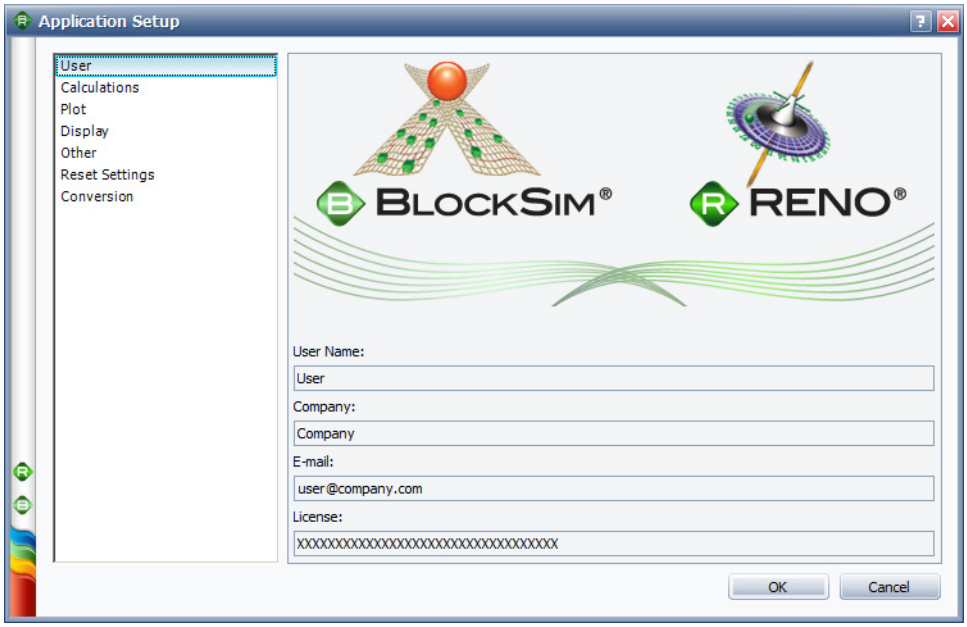
Note that the region and language settings on your particular computer will have some impact on the way information (e.g., dates and times) is displayed in the application.

To open the Application Setup window, choose **File > Application Setup**.

What's Changed? In previous versions of the software, the Application Setup was known as the User Setup.

22.2.1 Application Setup: User Page

In all Synthesis desktop applications, the User page of the Application Setup displays the product registration information for the current registered user of the software. This information, which includes the user's name, company and e-mail address as well as the product license key, is based on the information that you provided to ReliaSoft when you registered and activated the application. If you want to change this information, you can do so via the Contact Information tab of the License Manager (**File > Help > License Manager**).



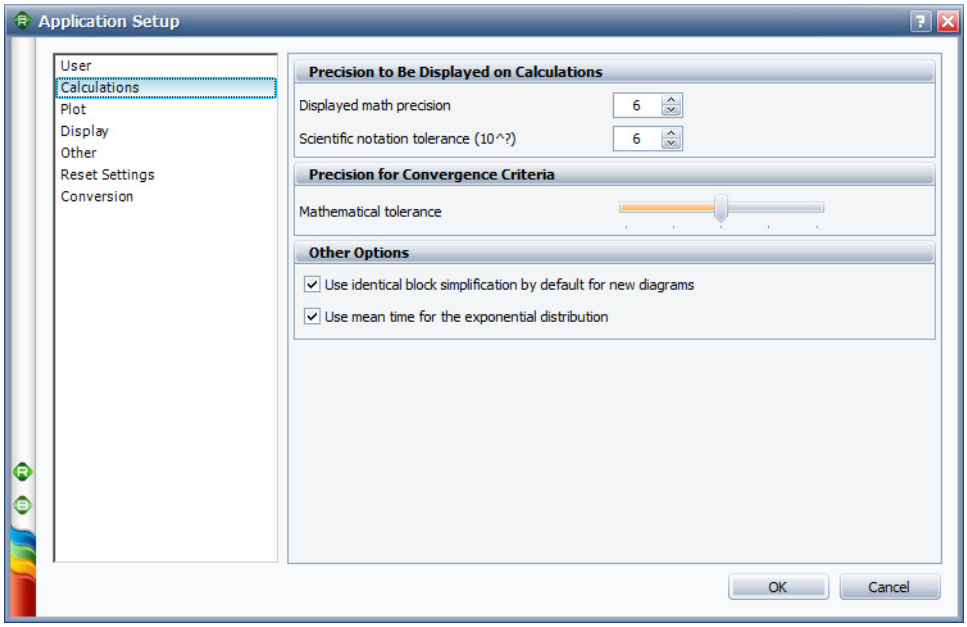
Note that the e-mail address shown here is your ReliaSoft ID. This address, along with the user name and company information associated with it, is used solely for the purpose of activating/deactivating the application on this computer. It is unrelated to personal information that is used elsewhere in the software, specifically:

- In plots and in some report output
- For notification e-mails, if enabled for the database

The contact information used in these locations is based on your user account in the current database, and is managed via the User Login and Contact Information window (**My Portal > Users > My Profile**). This means that any changes you make to this information will apply only to the current database. You will need to make the changes, if they are applicable, for each database that you work with.

22.2.2 Application Setup: Calculations Page

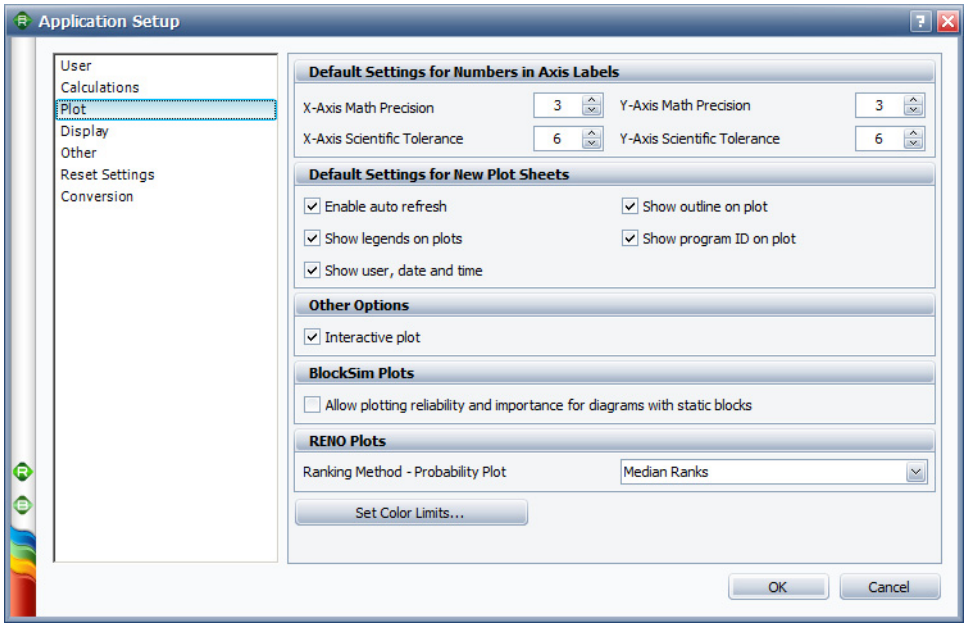
The Calculations page of BlockSim/RENO's Application Setup allows you to set the numerical precision used in the calculated results and plots, and whether to use identical block simplification by default for new diagrams.



- **Precision to Be Displayed on Calculations**
  - Use **Displayed math precision** to set the number of decimal places shown in results and plots. Please note that the tolerance on the precision is limited by Windows to 15 digits.
  - Use **Scientific notation tolerance (10^?)** to set the point at which the software will switch to scientific notation. For example, if the scientific notation tolerance is set to 5, then any number that is larger than 10^5 will be displayed using scientific notation.
- Use the **Precision for Convergence Criteria** slider to set the tolerance that will be used as the convergence limit (degree of accuracy) for mathematical iterations (e.g., a value of 4 indicates 1E-4 tolerance).
- **Other Options**
  - Select the **Use identical block simplification by default for new diagrams** option to use identical block simplification for analytical diagrams. This is useful when analyzing large diagrams that contain a lot of the same blocks. You can also select to use identical block simplification from the analysis settings page for analytical diagrams.
  - By default, lambda is used as the scale parameter of the exponential distribution. Select **Use mean time for the exponential distribution** if you prefer to use MTTF (1/lambda) instead.

22.2.3 Application Setup: Plot Page

The Plot page of BlockSim/RENO's Application Setup allows you to set the default display preferences for each new plot sheet that you create from this computer.

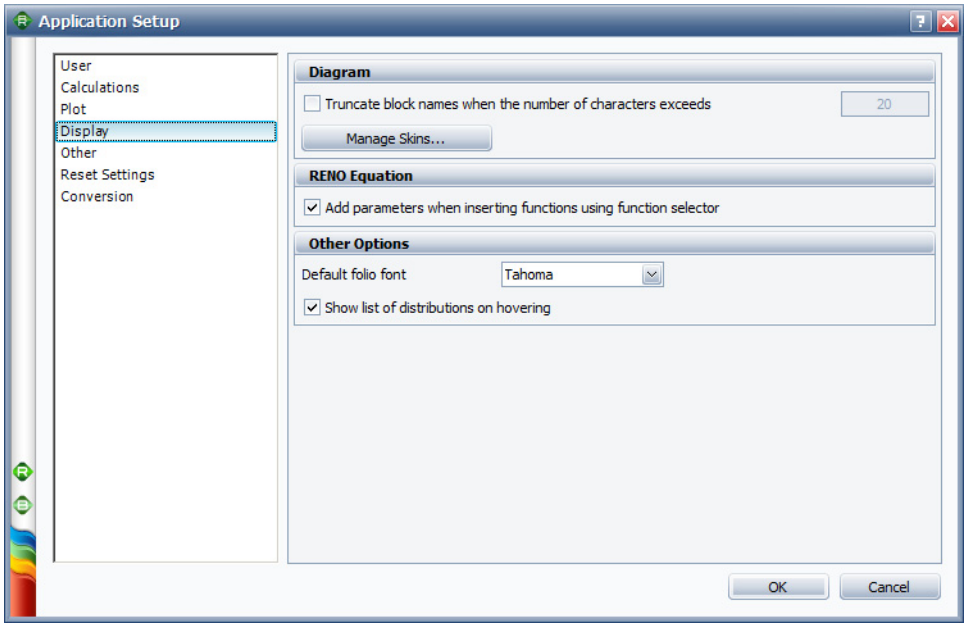


- Use the **Default Settings for Numbers in Axis Labels** area to set the precision displayed on the plots. High precision values can cause numbers to print on top of other numbers in the plots.
  - **X-Axis/Y-axis Math Precision** sets the number of decimal places shown for each axis on the plot. Please note that the tolerance on the precision is limited by Windows to 15 digits.
  - **X-Axis/Y-axis Scientific Tolerance** sets the point at which the software will switch to scientific notation for axis labels.
- Use the **Default Settings for New Plot Sheets** area to set the options for new plots. Note that changing an option does not apply to existing plots.
  - Select **Enable auto refresh** to activate the auto refresh option every time you create a new plot sheet.
  - Select **Show legends on plots** to display descriptions of curves, points and other information shown in plots.
  - Select **Show user, date and time** to display the user name, date and time on plots. The user information comes from you user account in the current database (to view/edit your information, choose **My Portal > Users > My Profile**) and is displayed at the bottom right corner of the plot.
  - Select **Show outline on plots** to display the canvas border on the plot.
  - Select **Show program ID** on plot to display the software name and URL in the Header Title section of the plot (e.g., BlockSim - www.ReliaSoft.com).
- **Other Options**
  - Select **Interactive plot** to highlight each point, line, slice or bar on the plot when you move the pointer over it. Point coordinates/values will be displayed on highlighted objects.
- **BlockSim Plots** BLOCKSIM Only
  - To include the reliability and importance values for diagrams that contain static blocks (i.e., blocks that use a constant model), select the **Allow plotting reliability and importance for diagrams with static blocks** option.

- **RENO Plots** RENO Only
  - Use the **Ranking Method - Probability Plot** drop-down list to specify the ranking method (Median Ranks or Kaplan-Meier) that is used in the flowchart Probability plot.
- **Set Color Limits** opens the Set Color Limits window, which allows you to set the color limits used in bar charts. Choose the metric of interest from the Plot Type drop-down list. You can then set both lower and upper color limits by dragging the markers on the scale between 0% and 100%. For example, in a Block Expected Failures plot, the blocks are displayed according to their RS FCI values, so if you wanted to change the default color limits used in that plot, you would choose RS DECI/RS FCI from the Plot Type drop-down list, then use the sliders to specify the desired color limits.

22.2.4 Application Setup: Display Page

The Display page of BlockSim/RENO's Application Setup allows you to select the MDI background image and customize the fonts used in the software.



- **Diagram**
  - Use the **Truncate block names when the number of characters exceeds** area to set the maximum number of characters displayed for block names.
  - To manage the skins that specify default diagram settings, click **Manage Skins** to open the Manage Skins window.

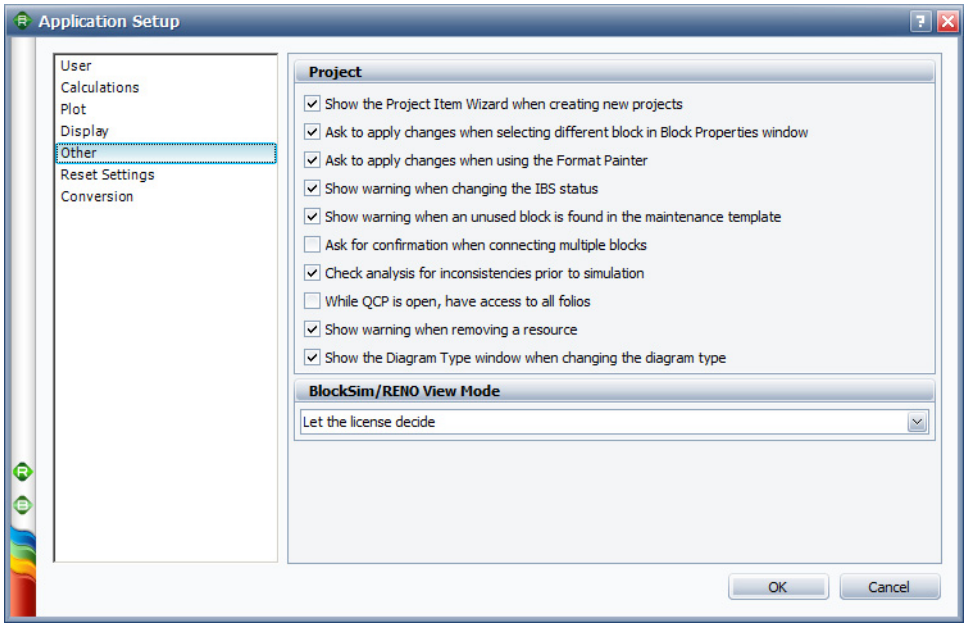
> **Note:** The skins managed here are diagram skins, which control the default appearance of new diagrams. The software also offers skins that control the overall appearance of the interface. These skins can be managed via the Synthesis Setup.

- **RENO Equation** RENO Only
  - Select **Add parameters when inserting functions using function selector** to automatically prompt for parameter inputs when you use the function selector or equation editor to insert functions into equations. *For more information on the function selector and equation editor, see page 390 in Chapter 11.*

- **Other Options**
  - Use the **Default folio font** drop-down list to select the default font for sheets in new analysis workbooks.
  - Select **Show list of distributions on hovering** to display the distribution/model drop-down list in control panels when you move the pointer over the bar. If not selected, you must click the bar in order to display the list.

22.2.5 Application Setup: Other Page

The Other page of BlockSim\RENO's Application Setup allows you to set some other preferences that are applicable only to BlockSim and RENO.



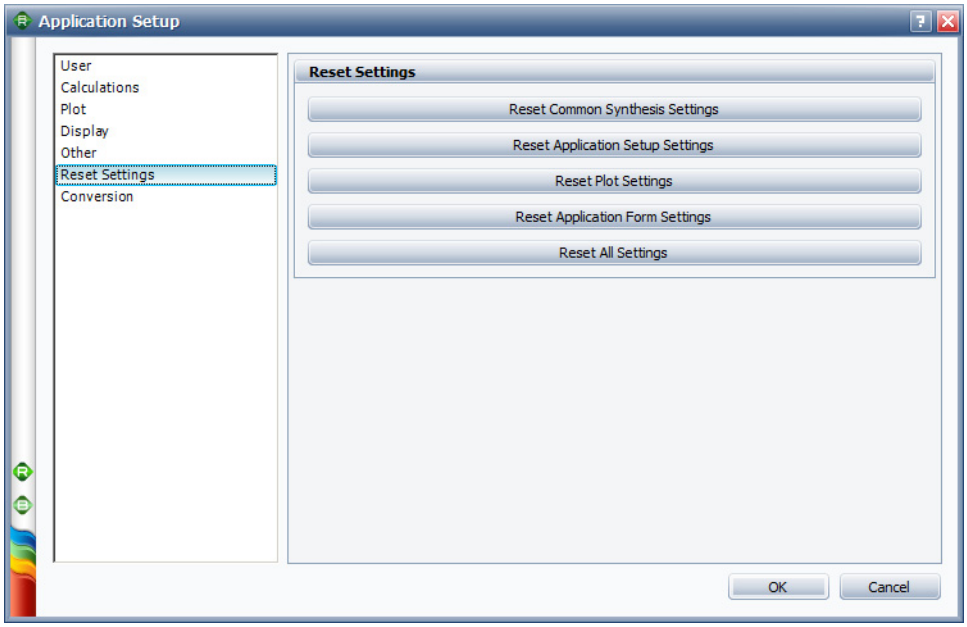
- **Project**
  - Select **Show the Project Wizard when creating new projects** to display a wizard that helps you select the first new diagram to create in a database. If not selected, the new database will be blank.
  - Select **Ask to apply changes when selecting different block in Block Properties window** to specify that a window will appear asking if you want to apply the changes you made to the active block when you select another block from the **Active Block** drop-down menu in the Block Properties window. If not selected, the changes will automatically be saved to the active block and the window will not appear when you select another block from the Active Block drop-down menu. *For more information on the Block Properties window, see page 194 in Chapter 6*.
  - Select **Ask to apply changes when using the Format Painter** to display a window with the Format Painter that allows you to specify which settings will be applied to another block.
  - Select **Show warning when changing the IBS status** to display a warning message when you select or clear the **Use IBS** check box on the analysis settings page for analytical diagrams. If not selected, this warning message will not appear.
  - Select **Show warning when an unused block is found in the maintenance template** to display a warning message when you attempt to run a simulation on a phase diagram with an associated maintenance template that includes a block not used in any of the operational phases. If not selected, this warning message will not appear.

- By default, when you have multiple blocks selected in a diagram and you add a connector to one of the selected blocks, the connection goes only to the block you place the connector on. To be prompted to add connections to all of the selected blocks, select the **Ask for confirmation when connecting multiple blocks** option.
- To check for inconsistencies in a simulation diagram or fault tree prior to simulating the results, select the **Check analysis for inconsistencies prior to simulation** option.
- Select the **While QCP is open, have access to all folios** option to lock the QCP in a top window position (i.e., non-modal) so it can remain open while you have access to all diagrams. The calculations performed in the QCP will be based on the currently active diagram. If not selected, you must close the QCP in order to return to the diagrams. This setting can also be changed from within the QCP by selecting or clearing the **Non-Modal QCP** option in the **Options** drop-down list.
- Select **Show warning when removing a resource** to display a warning message when you attempt to remove a resource that is in use.

- Use the **BlockSim/RENO View Mode** drop-down list to select which product view mode to use when opening a database.

22.2.6 Application Setup: Reset Settings Page

The Reset Settings page of the Application Setup is where you reset particular items to their original configuration. Some of these options apply to all Synthesis desktop applications on this computer; others apply only to the application that you are currently using.



- **Reset Common Synthesis Settings** resets only the settings that are shared by all Synthesis desktop applications (i.e., the settings that are managed via the Synthesis Setup window).
- **Reset Application Setup Settings** resets only the settings that are managed via the Application Setup window for this application.
- **Reset Plot Settings** clears the saved defaults for all pages of the Plot Setup and restores the settings that are shipped with the application.

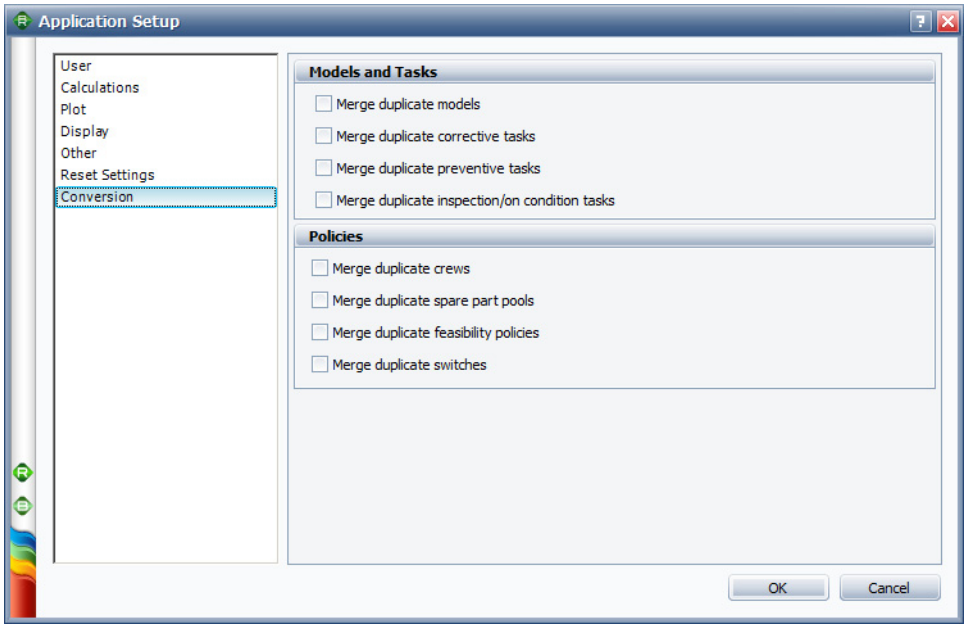
- **Reset Application Form Settings** resets only the application-specific form setting for this application, such as the size, location and last state of the interface windows.
- **Reset All Settings** resets all common Synthesis settings, application settings, plot settings, etc. This is the same as clicking all of the individual "Reset" buttons above.

Note: When selecting **Reset All Settings**, the default settings are not immediately applied to an open project. To see the changes, you must close, then reopen the project.

22.2.7 Application Setup: Conversion Page

BLOCKSIM Only

The Conversion page of BlockSim/RENO's Application Setup allows you to set the default conversion options for importing from previous versions of BlockSim. As described in the "Converting from BlockSim 6 or 7 Files" section (*page 74 in Chapter 2*), many properties were defined for individual blocks instead of being assigned to reusable resources, as they are in the Synthesis version. Because of this, the same information may have been defined in multiple locations in the original project. Using the Conversion page, you can set the default conversion settings that determine how identical records will be converted.



- **Models and Tasks**
  - Select **Merge duplicate models** to merge identical failure distributions and constants into one resource.
  - Select **Merge duplicate corrective tasks** to merge identical corrective tasks into one resource.
  - Select **Merge duplicate prevention tasks** to merge identical prevention tasks into one resource.
  - Select **Merge duplicate inspection/on condition tasks** to merge identical inspection/on condition tasks into one resource.
- **Policies**
  - Select **Merge duplicate crews** to merge identical crews into one resource.
  - Select **Merge duplicate spare part pools** to merge identical spare part pools into one resource.
  - Select **Merge duplicate feasibility policies** to merge identical feasibility policies into one resource.
  - Select **Merge duplicate switches** to merge identical switches into one resource.

Container Algorithm Setup 23

BLOCKSIM Only

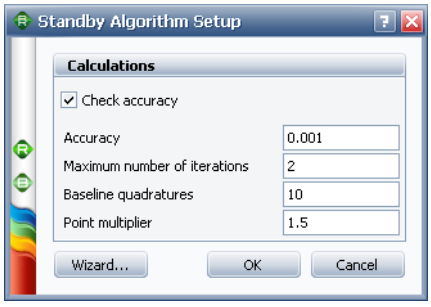
Load sharing and standby redundancy configurations in BlockSim are analyzed using numerical integration routines based on the Gaussian quadratures method. As with any numerical analysis routine, the solution error depends on the number of iterations performed, the step size chosen and related factors, plus the behavior of the underlying function. By default, BlockSim has a certain set of preset factors that it uses. In general, these defaults are sufficient for most problems. If a higher precision is required or if you want to verify the precision for the specific problem, you can modify the preset options and/or check for the integration error via the Algorithm Setup window. From this window you can also access the Container Integration Parameters Wizard, which will calculate the optimum number of quadratures to achieve the accuracy you specify.

23.1 Algorithm Setup Window

The Algorithm Setup window can be accessed by clicking the icon in the Adjust numerical convergence settings field in the Block Properties window when you are working with a container block in an analytical diagram.

The appearance of this window will vary depending on whether or not the **Check Accuracy** check box is selected.

If you select the **Check Accuracy** check box, the window will look like the one shown next.

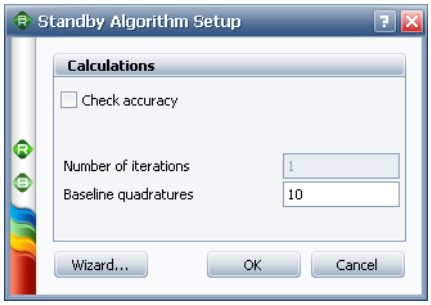


BlockSim will repeat the integration until the desired accuracy is achieved, within the limit you set. Enter the desired accuracy in the **Accuracy** field, then enter the maximum number of iterations that should be performed to achieve that accuracy in the **Maximum number of iterations** field. Next, enter the number of quadratures to be used in the first integration in the **Baseline quadratures** field. The second integration will use a number of quadratures equal to this baseline number multiplied by the value you enter in the **Point multiplier** field and check the result against the desired accuracy. This is then repeated up to the maximum number of iterations or until the desired accuracy is achieved.

If the acceptable accuracy is not reached within the number of defined iterations, you will see a message notifying you of the failure. You can:

- Click **Cancel** to cancel the calculation. You can then change the number of quadratures to meet your specified accuracy, if desired.
- Click **Ignore** to ignore further accuracy checking and continue the calculation.

If you do not select the Check Accuracy check box, the window will look like the one shown next.



Because no desired accuracy is specified, the **Number of iterations** field will be disabled and there will be only one iteration. Enter the number of quadratures to be used in the integration in the **Baseline quadratures** field.

In both cases, you can click the **Wizard** button to open the Container Integration Parameters Wizard, which assists you in computing the optimum number of quadratures.

23.1.1 Container Integration Parameters Wizard

The Container Integration Parameters Wizard assists you in computing the optimum number of quadratures that will meet the specified accuracy. The wizard can be activated by clicking the **Wizard** button in the Algorithm Setup window. You must have at least two blocks in the container (or, in fault trees, at least two dependent events connected to the standby or load sharing gate) to use the Container Integration Parameters Wizard; for standby containers, at least one of the blocks must be in standby mode.

To use the Container Integration Parameters Wizard, enter all of the inputs in the **Input Parameters** area and the **Time** area:

- **Input Parameters**: Enter the desired accuracy in the **Accuracy** field, then enter the maximum number of iterations that should be performed to achieve that accuracy in the **Maximum number of iterations** field. Next, enter the number of quadratures to be used in the first integration in the **Baseline quadratures** field. The second integration will use a number of quadratures equal to this baseline number multiplied by the value you enter in the **Point multiplier** field and check the result against the desired accuracy. If the accuracy is not achieved, subsequent iterations will use the previous iteration's calculation multiplied by the points multiplier until either the desired accuracy or the maximum number of iterations is reached.
- **Time**: Enter the **Start time** and the **End time** for the integration, then specify the length of the intervals into which the integration will be divided in the **Interval** field.

Once you have entered all required inputs, click **Calculate**. The wizard will perform multiple integrations, up to the specified maximum number of iterations. During calculation, the **Calculate** button will become a **Stop** button, allowing you to abort the calculation.

The following results will be returned:

- The **Suggested Parameters** area gives you the number of iterations necessary to meet the required accuracy, and the minimum baseline number of quadrature points that should be used with that number of iterations.
- In the **Results** area, the **Calculated accuracy** displays the maximum calculated accuracy found in all intervals.
- The table displays analysis accuracy at each interval for each iteration. Each row in the table gives results for a single iteration, with the first column displaying the number of quadratures used and each subsequent column displaying the calculated accuracy for the given interval.

You can click **Update** to accept the values in the **Suggested Parameters** area and apply these values to the Algorithm Setup window.

RELIASOFT CORPORATION END-USER LICENSE AGREEMENT

This ReliaSoft Corporation End-User License Agreement is a legal agreement between You and ReliaSoft Corporation for ReliaSoft Corporation's software products, which includes computer software and associated media and printed materials as well as on-line or electronic documentation and any software updates or upgrades, henceforth collectively called the "Application."

By installing, copying or otherwise using the Application, You agree to be bound by the terms of this End-User License Agreement and represent and warrant that You are authorized to do so. If You do not agree to the terms of this End-User License Agreement, do not install or use the Application and promptly return the Application to the vendor from which you obtained it within fourteen (14) calendar days from the date of purchase for a full refund.

This Agreement is effective from the earlier of the date You open the sealed software package, if applicable, or install the Application, or fourteen (14) calendar days from the date the Application was purchased or otherwise obtained from ReliaSoft Corporation, and continues until expressly terminated by ReliaSoft Corporation.

This Application is protected by copyright laws and international copyright treaties, as well as other intellectual property laws and treaties. **This Application is licensed, not sold**.

1 DEFINITIONS

1.1 **Access**. Means connected to or using the Application in any manner, either directly or indirectly through any middle tier application or system.

1.2 **Activate**. Means the act of enabling the Application to operate on a specific computer by registering the Application license with the ReliaSoft License Server and activating the Application for operation.

1.3 **Location**. Means the postal address of the site where the Application shall be installed. Location shall mean a single physical location.

1.4 **Named User(s)**. Means the individual(s) permitted to Access the Application, and it refers to specific individuals.

1.5 **ReliaSoft License Server**. Means the ReliaSoft server(s) that provides license activation, registered end-user management, license management and license compliance enforcement services for the Application.

1.6 **You**. Means the licensee or the individual acquiring the license of this Application and, in either case, the affiliates, employees, contractors and agents of said licensee or individual.

2 LICENSE GRANT

2.1 **Grant**. Subject to the license types defined in Section 3 hereof, You are granted a personal, non-exclusive and limited license to install, Activate and Access the Application under the license type and in the quantities for which You have obtained a license. You may install and Activate the Application onto designated equipment at the specified Location(s), solely as necessary to use the Application in accordance with this Agreement. You may not distribute, sublicense, copy (except as expressly permitted in this Agreement), transfer (except as expressly permitted in this Agreement), nor give the Application to any third party.

2.2 **Ownership**. The Application and the information it contains, including any updates, upgrades, modifications and derivative works, and all copies of them, are the property of ReliaSoft Corporation (or its licensors), and title remains with ReliaSoft Corporation (or ReliaSoft Corporation's licensors). All applicable rights in patents, copyrights and trade secrets in the Application are and will remain with ReliaSoft Corporation (or ReliaSoft Corporation's licensors). No title to, or ownership of, the Application or the information contained therein is transferred to You. ReliaSoft Corporation reserves all rights not explicitly granted in this Agreement.

2.3 **Rights to Developments**. This Agreement also governs your use of any enhancements, data, or information provided by ReliaSoft Corporation while performing any technical, training or consulting services. Any ideas, know-how, techniques and software developed by ReliaSoft Corporation, including any enhancements or modifications made to the Application by any person, are the property of ReliaSoft Corporation.

3 INSTALLATION AND USE

ReliaSoft Corporation licenses any Application under this Agreement on a per user basis. In other words, ReliaSoft Corporation licenses an Application to each user and not to a particular computer. This Agreement grants You the following rights, depending on the license You have obtained from ReliaSoft Corporation:

3.1 **Single User License.** The Single User License is licensed to one individual Named User. This is a perpetual license. Each software package includes the installation media and one copy of the Application documentation per license, either in physical/hard copy format and/or electronic format. The Named User may install, Activate and Access the Application on up to three computers (i.e., multiple Locations), such as a work computer, a home computer and a portable computer, provided that the Named User is the only person who uses this Application, at any time, in any of these Locations. In the event that installation and activation of the Application is performed on three computers, the Named User will be required

to deactivate at least one installation before being allowed to Activate the Application again. There is a limit of 6 activations allowed per year. The Single User License cannot be installed on multi-user capable environments such as Windows, Terminal or Citrix servers.

Use of the Application on any of the computers by any user other than the Named User who has licensed the Application is strictly prohibited. Such other users must obtain a separate license of the Application from ReliaSoft before any use.

3.2 **Standard Network License**. The Standard Network License is licensed to Named Users in an organization on a single network of computers at a specified Location. This is a perpetual license. The software package includes the installation media and one copy of the Application documentation per license, either in physical/hard copy format and/or electronic format. ***Note: This is not a Concurrent Network License***.

This license provides for a limited number of Named Users, dependent on the quantity of seats licensed, to install, Activate and Access the Application, with a single activation allowed for each Named User. For example, if You purchase a ten (10) seat Standard Network License, only ten (10) specific Named Users are allowed Access. If an eleventh (11^{th}) user wishes to have Access to the Application, an additional seat must be licensed. In the event that installation and activation of the Application is required on a new computer for a Named User, the Named User will be required to deactivate the active installation before being allowed to install and Activate the Application on the new computer. The Standard Network License allows for installation on multi-user capable environments such as Windows, Terminal or Citrix servers.

Under this license, no other copies can be installed on non-network computers, such as portable computers, home computers, etc.

3.3 **Concurrent Network License**. The Concurrent Network License is licensed to an organization with multiple regular and occasional Named Users on a single network of computers at a specified Location. This is a perpetual license. The software package includes the installation media and one copy of the Application documentation per license, either in physical/hard copy format and/or electronic format. Concurrent Network Licenses are specified by two attributes:

(i) Number of Named User Activations, which is the maximum number of computers on which a Named User can Activate the Application (i.e., make it available for operation). More than one activation is allowed for each Named User (e.g., to install the Application on both a work computer and a portable computer), but each activation is counted toward the maximum available under the license.

(ii) Number of Concurrent Access Licenses (or CALs), which is the maximum number of computers that can Access the Application at any given time.

For example, for a Concurrent Network License with thirty (30) Named User Activations and ten (10) CALs, the Application can be installed and activated on up to 30 computers (e.g., by 30 Named Users each activating the Application on only one computer, by 15 Named Users each activating the Application on two computers, etc.), and only ten (10) of those computers will be able to Access the Application at the same time. If a user attempts to Access the Application from an eleventh (11th) computer, the user must wait until one of the other ten users ends their session and releases the CAL. The Concurrent Network License allows for installation on multi-user capable environments such as Windows, Terminal or Citrix servers.

Under this license, Named Users are allowed to check out a CAL from the ReliaSoft License Server for a specified period of time, effectively being guaranteed Access to the Application while simultaneously reducing the number of available CALs for the rest of the user pool until the CAL is checked back into the ReliaSoft License Server or otherwise released. This ability to check out a CAL from the ReliaSoft License Server is only available for Applications designated as Major Version 8 and above (e.g., 8.0.1, 8.0.2, etc).

This type of license requires that you provide/allow internet access from the Application to the ReliaSoft License Server at https://Validate.ReliaSoft.org. Concurrency authorization and control is managed and provided by this server. You may need to modify and adjust your internet and/or firewall settings and/or set the aforementioned server as a trusted location in order to use the Application. This requirement for connectivity to the ReliaSoft License Server for concurrency authorization and control only applies for Applications designated as Major Version 8 and above (e.g., 8.0.1, 8.0.2, etc).

3.4 **Asset-based License (only applies to Orion eAPI software)**. The Asset-based License is licensed to all direct employees of a corporate entity within one or more specified Sites, depending on the license obtained. This license excludes subcontractors, suppliers or temporary employees, unless they are using the Application inside an eligible Site on a computer owned by that Site. This is a perpetual license.

Under this license, an unlimited number of Named Users from the licensing Site or Sites is allowed to Access the Application; however, the license restricts the number of physical assets (commonly referred to as Asset TAGs) that can be configured and managed within the Application. For this license, a Site is defined as a distinct physical address/site of a corporate entity, where the physical assets being managed reside (e.g., a refinery or a mine or a power station). The license specifically excludes broader geographical areas, such as cities or states/provinces (where multiple Sites might exist).

3.5 **Unlimited User License.** The Unlimited User License is licensed to all direct employees of a corporate entity within a specified Site, Division, or Corporation, depending on the type of license obtained. This license excludes subcontractors, suppliers or temporary employees, unless they are using the Application inside an eligible facility on a computer owned by that facility. This is a perpetual license.

Site-Wide License: Covers all direct employees working at a distinct physical address/site of a corporate entity. A Site-Wide License specifically excludes broader geographical areas, such as cities or states/provinces (where multiple sites might exist).

Division-Wide License: Covers all direct employees affiliated with a unique business unit of a corporate entity organized in multiple divisions/business units. Examples of divisions are the Aerospace Division of a large conglomerate or the Brakes Division of an automotive corporation. A Division-Wide license might additionally be restricted to a specific geographical area to facilitate licensee requirements (for example, the Aerospace Division of a conglomerate in North America).

Corporate-Wide License: Covers all direct employees of a corporate entity.

This license entitles the specified corporate entity at the specified Site/Division/Corporation to make and distribute copies of the Application in executable code in its entirety, as supplied by ReliaSoft Corporation, OR install and distribute the Application over a network. The software package includes the installation media and one copy of the Application documentation per license, either in physical/hard copy format and/or electronic format. The Unlimited User License allows for installation on multi-user capable environments such as Windows, Terminal or Citrix servers.

3.6 **Rental License.** Rental licenses are available for the Single User, Standard Network and Concurrent Network licenses. The license terms are the same as the corresponding license type being rented except that the license expires according to the term purchased. The rental fee will not be prorated or refunded if license rental is cancelled or not used in full. A standard non-expiring license may be purchased at any time at the current Application list price.

3.7 **Evaluation, Demo, No Save Demo, Beta, or Otherwise Expiring or Non-Expiring Licenses of any Type Provided for Evaluation Purposes.** These are Evaluation/Not for Sale licenses and, notwithstanding any other section of this Agreement, You may not use the Application for commercial purposes, nor sell or otherwise transfer it for value. "Commercial purposes" include the use of the Application in creation of publicly distributed computer software. You are acquiring only the limited right to use a single copy of the Application solely for evaluation purposes and for a limited time. ReliaSoft Corporation can terminate this Agreement at any time. You have no rights under any sections of this Agreement other than those specifically allowed for within this section (Section 3.7). THE APPLICATION IS LICENSED AS IS with no representations or warranties of any kind. Note: As the section title implies, users of any license provided by ReliaSoft Corporation for evaluation purposes (even if it is, for example, a Single User or Network License), shall abide by the rules set forth in this section.

3.8 **Complimentary License.** Complimentary Licenses have the same installation and use rights as their corresponding purchased licenses (e.g., a Complimentary Single User License provides the same installation and use rights as the Single User License). However, You have no rights under any other sections of this Agreement other than those specifically allowed for within this section (Section 3.8). THE APPLICATION IS LICENSED AS IS with no representations or warranties of any kind and license transfers are NOT allowed.

Regardless of the license type You have obtained from ReliaSoft Corporation, You may access shared data storage files or databases relating to the Application across a wide area network (WAN) or a virtual private network (VPN), instead of a local area network (LAN), as long as such access is permitted by your policies and authorized by You via appropriate Application and network configuration settings. Although such access is not prohibited by ReliaSoft Corporation, you acknowledge that WAN and VPN data transfer rates vary widely and could lead to decreased and unpredictable performance (as compared to a LAN) and therefore disclaim any expectation of Application performance should you choose to a) configure any part of the Application, b) use the Application, or c) access shared data storage files in a distributed environment over a WAN or VPN. Such disclaimer does not apply to any Application designated and licensed as a native Web-based Application by ReliaSoft Corporation.

4 SOFTWARE TRANSFERS

4.1 **Single User License.** You may **permanently** transfer all of your rights related to a Single User License under this Agreement, provided that You retain no copies, You transfer all of the Application (including all component parts, media, printed or electronic materials, any upgrades and this Agreement) and the transferee agrees to the terms of this Agreement. If the Application is an upgrade, any transfer must include all prior versions of the Application. Single User License transfers are only allowed between Named Users of the same organization (i.e., transfers are not allowed across different organizations). The Named User transferring the Application must deactivate any and all active installations of the Application before the transferee can Activate the Application. A Single User License may not be transferred more than two times in a year. This section also applies to Single User License Rentals.

4.2 **Standard Network, Concurrent Network, Asset-based and Unlimited User Licenses.** Standard Network, Concurrent Network, Asset-based and Unlimited User licenses are NOT transferable across organizations, and licensing rights

CANNOT be split or transferred between multiple organizations in cases of corporate acquisitions or divestitures, except as allowed in items (a) and (b) of this section (Section 4.2), and with ReliaSoft Corporation's written consent.

In the case of Standard and Concurrent Network Licenses, Named User Activations can be deactivated and replaced by new users on a **permanent** basis. In other words, replacement of users is allowed to accommodate attrition and reassignment of user responsibilities in the ordinary course of doing business. However, replacing users on a daily, weekly, monthly or even quarterly basis and frequently reactivating users that were previously replaced is an indication that the number of Named Users (for Standard Network Licenses) or Named User Activations (for Concurrent Network Licenses) obtained by You is not adequate to serve the pool of users intending to Access the Application and is therefore prohibited. Each Standard Network License seat or Concurrent Network Named User Activation can be reassigned to a different Named User no more than two times in a year.

In cases of corporate acquisitions or divestitures: (a) Standard Network and Concurrent Network licenses remain with the organization that originally obtained the licenses, (e.g., if the organization that holds the license is acquired by another entity, the licensing rights will be transferred to the acquiring entity); (b) Unlimited Site-Wide licenses remain with the original licensing site and therefore pass on to the owner of the specific licensing site as long as the site continues to exist; (c) Unlimited Division-Wide licenses remain with the organization that originally obtained the licenses in cases where the Division is being partially divested, and are terminated when the Division is fully divested. As an example, in a partial divestiture, the portions of the Division divested will not have access to the Division-Wide license whereas the portions of the Division that remain will continue to have access to the Division-Wide license; and (d) Unlimited Corporate-Wide licenses remain with the organization that originally obtained the licenses as long as said organization divested a portion of its business or acquired other organizations, and are terminated in cases where said organization was acquired by another organization. As an example, in a partial divestiture, the portions of the corporation divested will not have access to the Corporate-Wide license whereas the portions of the corporation that remain will continue to have access to the Corporate-Wide license. As a further example, an organization with a Corporate-Wide license having acquired other entities (either in part or in whole) will continue to have the same licensing rights as before the acquisition and can therefore deploy the same licensing rights to the newly acquired entities.

This section also applies to Standard Network and Concurrent Network License Rentals.

5 DESCRIPTION OF OTHER RIGHTS, LIMITATIONS AND MISCELLANEOUS ITEMS

5.1 **Limitations on Reverse Engineering, Decompilation and Disassembly**. You may not reverse engineer, decompile or disassemble the Application. You shall not provide, disclose or otherwise make available the Application, in any form, to any person other than your employees and under your direction and control for purposes specifically related to your permitted use of the Application. You will not: (a) alter, modify or prepare derivative works of the Application; (b) transmit the Application electronically by any means; or (c) cause or permit the translation, reverse engineering, disassembly, or decompilation of the Application to determine any design structure, source code, concepts and methodology behind the Application, whether to incorporate it within any product of your own creation, create a derivative work, create any product that is competitive with the Application or other ReliaSoft Corporation products, or for any other purpose.

5.2 **Copyright**. All title and copyrights to the Application are owned by ReliaSoft Corporation (or its suppliers or licensors). The Application is protected by copyright laws and international treaty provisions. Therefore, You must treat the Application like any other copyrighted material except that You may make one copy of the media solely for backup or archival purposes.

5.3 **Proprietary Notices**. All title, trademarks, copyrights and intellectual property rights in and pertaining to the Application (including but not limited to any copies thereof, software structure and organization, source code, images and new releases) are valuable property of ReliaSoft Corporation and are owned or licensed by ReliaSoft Corporation. You may not intentionally remove, alter or destroy any form of copyright and trademark notices, proprietary markings or confidential legends placed upon or contained within the Application, including but not limited to any such notices contained in physical and/or electronic media or documentation, in the Application interface boxes, or in any of the runtime resources, code or other embodiments originally contained in or dynamically or otherwise created by the Application.

5.4 **Use of ReliaSoft Corporation's Marks**. You may not use the name, logos, trade names or trademarks of ReliaSoft Corporation or any of its affiliates in any manner including, without limitation, in your advertising, promotional literature or any other material, whether in written, electronic, or other form distributed to any third party, except in the form provided by ReliaSoft Corporation and then solely for the purposes of identifying your use of the ReliaSoft Corporation Application.

5.5 **Verification**. You will provide, on ReliaSoft Corporation's reasonable written request, written verification that the Application is being used according to the terms of this Agreement. Upon thirty days prior written notice, if ReliaSoft Corporation has reasonable grounds to believe that this Agreement has been breached, ReliaSoft may audit your use of the Application provided such audit is: (a) limited to records relating solely to the Application necessary to verify compliance with the terms of this Agreement; (b) performed by a reputable independent third party auditor acceptable to You (acting reasonably); (c) the third party auditor shall comply with your standard security policies; (d) the third party

auditor shall execute your standard protective non-disclosure agreement; and (d) the cost of any requested audit will be solely borne by ReliaSoft Corporation if no breach is found as a result of the audit or will be solely borne by You if a breach is found. Such audit shall occur no more than once every twelve months and shall not unreasonably interfere with your normal business operations.

5.6 **Modification**. ReliaSoft Corporation reserves the right to modify or enhance the Application without obligation to notify You of such changes or to furnish them to You, unless otherwise agreed upon with a separate agreement (such as an annual maintenance agreement).

5.7 **Copying**. You may not, under any circumstances, copy the Application, in whole or in part, except as expressly provided under the Copyright section above.

5.8 **Separation of Components**. The Application is licensed as a single product. Its component parts may not be separated for use on more than one computer.

5.9 **Rental or Other Exploitation**. You may not publish, sub-license, re-license, assign, sell, distribute, license, transfer, rent, lease or lend the Application to any party, except transfer the Application as expressly provided under the Transfer section above. If you received any revenues from the unlawful distribution of the Application, such revenues will be forfeited to ReliaSoft Corporation.

5.10 **Fees**. You will pay ReliaSoft Corporation all fees or other amounts due under this Agreement, plus any and all applicable taxes, within the payment term due date specified on the respective invoice. In the event that the respective invoice is not paid on time, or at all, ReliaSoft Corporation reserves the right to terminate this Agreement and revoke the corresponding licenses issued to You within the scope of this Agreement.

5.11 **Termination**. Without prejudice to any other rights, ReliaSoft Corporation may terminate this Agreement if You fail to comply with the terms and conditions of this Agreement and such breach is not cured within thirty days of notice of such breach. In such event, You must destroy all copies of the Application and all of its component parts. Additionally, You may be held liable for any damage or loss of profit caused to ReliaSoft Corporation arising from unauthorized use or duplication of this Application.

5.12 **Supplemental Licenses**. If, after the effective date of this Agreement, You subsequently purchase additional licenses of the Application, these supplemental licenses will be included under this Agreement.

5.13 **Press Releases**. As part of this Agreement, You acknowledge that ReliaSoft Corporation may make reference to You as a customer of ReliaSoft Corporation in press releases, advertising and promotional materials, and You consent to any such reference. ReliaSoft Corporation will NOT disclose any further details beyond referring to You as a customer without prior written consent, not to be unreasonably withheld.

5.14 **Relationship**. You and ReliaSoft Corporation are independent contractors and neither is an agent, joint venture partner, partner or employee of the other, and ReliaSoft Corporation will not be obligated by any agreements or representations made by You to any person, nor with respect to any other action by You, nor will ReliaSoft Corporation be obligated for any damages to any person, whether caused by your actions, failure to act, negligence or willful conduct.

5.15 **Upgrades**. If the Application is an upgrade from another product, whether from ReliaSoft Corporation or another supplier, You may use or transfer the Application only in conjunction with that upgraded product, unless You destroy the upgraded product. If the Application is an upgrade of a ReliaSoft Corporation product, You may use the upgraded product only in accordance with this Agreement. If the Application is an upgrade of a component of a package of software programs that You licensed as a single product, the Application may be used and transferred only as part of that single product package and may not be separated for use on more than one computer.

5.16 **U.S. Government Restricted Rights**. The Application was developed at private expense. No portion of the Application was developed with government funds and the Application is a trade secret of ReliaSoft Corporation for all purposes of the Freedom of Information Act. The Application and documentation are provided with RESTRICTED RIGHTS. Use, duplication or disclosure by the Government is subject to restrictions as set forth in subparagraph (c)(1)(ii) of the Rights in Technical Data and Computer Software clause at DFARS 252.227-7013 (or its successor) or subparagraphs (c)(1) and (2) of the Commercial Computer Software Restricted Rights at 48 CFR 52.227-19 (or its successor), as applicable. Manufacturer is ReliaSoft Corporation, 1450 S. Eastside Loop, Tucson, Arizona 85710, USA.

5.17 **Limited Warranty**. ReliaSoft Corporation warrants that: (a) the Application will perform substantially in accordance with the accompanying written materials, and on machines meeting the published minimum requirements, for a period of sixty (60) days from the date of receipt; and (b) any media on which the Application is furnished will be free from defects for a period of sixty (60) days from the date of receipt. Some states and jurisdictions do not allow limitations on duration of an implied warranty, so the above limitation may not apply to You; in such states and jurisdictions the liability of ReliaSoft Corporation shall be limited to the minimum extent permitted by law. To the extent allowed by applicable law, implied warranties on the Application and media, if any, are limited to sixty (60) days; and (c) at the time of installation, the Application will be free from any mechanism, feature or any other codes or instructions that: (i) cause the Application to remotely transmit information to ReliaSoft or any third party, except to communicate with one of ReliaSoft's servers to

facilitate specific functions of the Application (such as to communicate with the ReliaSoft License Server, access the online help files, etc.); or (ii) may be used to permit Access to, or use of, the Application or computer system on which the Application is loaded, or to which the Application is linked, by ReliaSoft or any third party.

5.18 **Customer Remedies**. ReliaSoft Corporation's and its suppliers' or licensors' entire liability and Your exclusive remedy shall be, at ReliaSoft Corporation's option, either (a) return of the fee paid for the Application, or (b) repair or replacement of the Application or media that does not meet ReliaSoft Corporation's Limited Warranty and which is returned to ReliaSoft Corporation with a copy of your receipt or invoice. Any replacement Application or media will be warranted for the remainder of the original warranty period or thirty (30) days, whichever is longer. None of these remedies nor any product support services offered by ReliaSoft Corporation are available without a valid License Certificate issued by ReliaSoft Corporation.

5.19 **Warranty Exclusions**. The Limited Warranty is void if the damage or defect has resulted from accident, abuse or misapplication. Any modification of the Application by any person other than ReliaSoft Corporation shall void this warranty. Any manipulation of the Application's data storage infrastructure or direct storage of data into the Application's data storage from outside the Application by any person other than ReliaSoft Corporation or ReliaSoft Corporation's authorized representative shall void this warranty. The warranties in this section extend only to You and are contingent upon proper use of the Application. The warranties will not apply to any failure caused by (a) accident, (b) unusual physical, electrical or electro-magnetic stress, (c) negligence, (d) misuse, (e) failure of electrical power, air conditioning or humidity control, (f) use of the Application with any equipment or software not reflected in ReliaSoft Corporation's specifications, (g) installation, alteration or repair of the Application by anyone other than ReliaSoft Corporation or ReliaSoft Corporation's authorized representative, or (h) or installation on equipment on which the original identification marks have been removed or altered.

5.20 **No Other Warranties**. No oral or written information or advice given by ReliaSoft Corporation, its suppliers, dealers, distributors or agents shall create a warranty or in any way increase the scope of the Limited Warranty, and You may not rely on any such information or advice as a warranty.

5.21 **Use of Results Provided By the Application Disclaimer**. You understand that the results provided by the Application cannot replace judgment required for important decisions. Use of the results provided is done completely at your own risk. ReliaSoft Corporation does not warrant that the functions of this Application will meet your requirements or be error free. You assume all risk of the use, quality and performance of the Application, and You are advised to use your own discretion and judgment regarding the use of the Application.

5.22 RELIASOFT CORPORATION, ON BEHALF OF ITSELF AND ITS LICENSORS, DISCLAIMS ALL OTHER WARRANTIES, EITHER EXPRESS OR IMPLIED, INCLUDING, BUT NOT LIMITED TO, IMPLIED WARRANTIES OF MERCHANTABILITY AND FITNESS FOR A PARTICULAR PURPOSE, WITH REGARD TO THE APPLICATION. IN NO EVENT SHALL RELIASOFT CORPORATION OR ITS SUPPLIERS OR LICENSORS BE LIABLE FOR ANY SPECIAL, INCIDENTAL, INDIRECT, OR CONSEQUENTIAL DAMAGES WHATSOEVER (INCLUDING, WITHOUT LIMITATION, DAMAGES FOR LOSS OF BUSINESS PROFITS, BUSINESS INTERRUPTION, LOSS OF BUSINESS INFORMATION OR ANY OTHER PECUNIARY LOSS) ARISING OUT OF THE USE OF OR INABILITY TO USE THE APPLICATION, EVEN IF RELIASOFT CORPORATION HAS BEEN ADVISED OF THE POSSIBILITY OF SUCH DAMAGES. USE OF THIS APPLICATION IS DONE COMPLETELY AT YOUR OWN RISK, INCLUDING THE USE OF RESULTS PROVIDED BY THIS APPLICATION. RELIASOFT CORPORATION'S LIABILITY HEREUNDER SHALL BE LIMITED TO PHYSICAL DAMAGE DIRECTLY CAUSED BY THE SOLE NEGLIGENCE OF RELIASOFT CORPORATION AND SHALL NOT EXCEED THE PRICE PAID FOR THE SOFTWARE PRODUCT. NOTWITHSTANDING ANYTHING HEREIN TO THE CONTRARY, THE FOREGOING DISCLAIMER AND LIMITATION OF LIABLITY SHALL NOT APPLY TO RELIASOFT CORPORATION'S OBLIGATIONS UNDER ANY CLAIM OF INFRINGEMENT AS DESCRIBED IN SECTION 5.28 OF THIS AGREEMENT.

5.23 **Venue**. Venue for any proceedings arising out of or relating to this Agreement shall be in Tucson, Pima County, Arizona. The parties disclaim the application of the United Nations Convention on Contracts for the International Sale of Goods. This Agreement is governed by the laws of the State of Arizona, USA, without reference to conflict to law principles. Each party to this Agreement submits to the exclusive jurisdiction of the state and federal courts in the State of Arizona for the purpose of resolving any disputes arising under or relating to this Agreement. Each party waives any jurisdictional, venue or inconvenient forum objections to such courts.

5.24 **Legal Expenses**. If legal action is taken by either party to enforce this Agreement, all costs and expenses (including reasonable attorney fees) incurred by the prevailing party in exercising any of its rights or remedies or in enforcing any of the terms, conditions, or provisions of this Agreement will be paid by the other party.

5.25 **Force Majeure**. ReliaSoft Corporation will not be responsible for delays or failures in its performance due, in whole or in part, to any cause beyond its reasonable control.

5.26 **Export Control**. Regardless of any disclosure made by You to ReliaSoft Corporation of an ultimate destination of the Application, You will not export, either directly or indirectly any Application without first obtaining all licenses required,

from the U.S. Department of Commerce or any other agency or department of the United States Government, and complying with the applicable laws. Neither the Application nor any direct product thereof may be exported, directly or indirectly, in violation of applicable export laws, or may be used for any purpose prohibited by these laws including, without limitation, nuclear, chemical or biological weapons proliferation. ReliaSoft Corporation will provide You with all reasonable information requested by You in connection to exporting the Application, including providing You with the U.S. Export Controls Classification Number (ECCN) for the Application.

5.27 **Waiver**. The waiver by either party of any breach of this Agreement shall be in writing and shall not constitute a waiver of any other or subsequent breach. No waiver of any of the provisions of this Agreement will be deemed, or will constitute, a waiver of any other provision, whether or not similar, nor will any waiver constitute a continuing waiver. The failure by a party to enforce any provision of this Agreement will not be deemed a waiver of future enforcement of that or any other provision.

5.28 **Indemnification**. You will indemnify and hold ReliaSoft Corporation harmless against any and all claims, damages, losses, costs or other expenses (including reasonable attorney fees) that arise directly or indirectly from your breach of this Agreement. ReliaSoft Corporation shall defend, indemnify and hold harmless, at its own expense, You and your assigns, successors, directors, officers and employees (each an "Indemnified Party") against any and all claims incurred by or made against an Indemnified Party by a third party in connection with a claim, suit or action which is based on an allegation that the Application when used by You as authorized under this Agreement, misappropriates or infringes any third party patent, copyright, trade secret or other intellectual property right (each, a "Claim") provided that ReliaSoft Corporation shall have received from the Indemnified Party: (i) notice of such Claim as soon as possible after You receive notice of the Claim; given that a failure to provide notice shall only relieve ReliaSoft Corporation of its indemnity obligation to the extent ReliaSoft Corporation was prejudiced by such failure; (ii) the exclusive right to control and direct the investigation, defense or settlement of such claim; and (iii) all reasonable necessary cooperation by You. If your use of any of the Application is, or in ReliaSoft Corporation's opinion is likely to be, enjoined due to a Claim, ReliaSoft Corporation may, at its sole discretion: (a) modify the Application so that it becomes non-infringing, provided such modifications result in software with substantially similar functionality and performance; (b) procure for You the right to continue using the Application under substantially the same terms and conditions as provided for hereunder; or (c) if (a) and (b) are commercially impracticable, terminate the Agreement and refund to You the license fee paid by You for the Application which is the subject of the Claim as reduced to reflect a three-year straight-line depreciation from the applicable license purchase date. The foregoing indemnification obligation of ReliaSoft Corporation shall not apply: (1) if the Application is modified by any party other than ReliaSoft Corporation and such modification was not authorized in writing by ReliaSoft Corporation, but solely to the extent the alleged infringement is caused by such modification; or (2) to any release of the Application other than the most current release, provided that: (I) the most current release was either made available at no cost to You and (II) You had a commercially reasonable period of time (not to exceed 60 days) after availability of the current release to implement the current release so as to avoid the infringement claim. This section (Section 5.28) sets forth ReliaSoft Corporation's sole liability and your sole and exclusive remedy with respect to any claim of infringement.

5.29 **Equitable Relief.** You acknowledge and agree that, due to the unique nature of the Application, there can be no adequate remedy at law for any breach of your obligations under this Agreement, that any such breach may allow You or third parties to unfairly compete with ReliaSoft Corporation resulting in irreparable harm and therefore that, upon any such breach or threat thereof, ReliaSoft Corporation shall be entitled to injunctive and other appropriate equitable relief in addition to whatever remedies it may have at law.

5.30 **Entire Agreement; Amendments.** This Agreement is the complete and exclusive statement of the agreement between the parties and supersedes all prior agreements and communications with respect to the subject matter, and there are no oral representations, understandings or agreements that are not fully expressed herein. Any terms appearing on any order or other form used by You which modify or conflict with the terms and conditions set forth herein are expressly rejected. Except for the purpose of negating implied warranties, no course of prior dealings between the parties and no usage of the trade shall be relevant to supplement or explain any term used in this Agreement. No ReliaSoft Corporation employee other than an officer of ReliaSoft Corporation (Vice President and above) shall have any actual or apparent authority to modify the terms of this Agreement in any way. All amendments shall be in writing and signed by the authorized representative of ReliaSoft Corporation.

5.31 **Severability.** If any one or more of the provisions of this Agreement shall for any reason be held to be invalid, illegal or unenforceable in any respect, any such provision shall be severable from this Agreement, in which event this Agreement shall be construed as if such provision had never been contained herein.

5.32 **Electronic Signatures.** ReliaSoft Corporation and You agree that this Agreement may be executed electronically and that electronic copies of this Agreement shall be binding upon the parties to the same extent as manually-executed copies.

ReliaSoft®
EMPOWERING THE RELIABILITY PROFESSIONAL
Tucson • São Paulo • Warsaw • Chennai • Singapore