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



WEIBULL++®

Version 9



ALTA®

Version 9

ReliaSoft®

ReliaSoft Publishing
A Division of ReliaSoft Corporation

Tucson São Paulo Warsaw Chennai Singapore

Weibull++/ALTA 9 User's Guide
Part Identification: RGUGWA9

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

© 1992-2014 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 7 6 5 4

Contents

1 Introduction 1
1.1 Technical Support 2
1.1.1 When Requesting Support 2
1.2 ReliaSoft Consulting 2
1.3 Minimum System Requirements 3
1.3.1 A Note About Enterprise Database Servers 4
1.4 Installing Synthesis 5
1.5 License Registration & Activation 6
1.5.1 Activating Synthesis Applications 6
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 12
1.7 Region and Language Settings 13
1.7.1 Viewing/Changing the Region and Language Settings for Your Computer 13
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 Weibull++ Features Overview 18
1.10 ALTA Features Overview 21
1.11 Features Common to Weibull++ and ALTA 23
1.12 What's New in the Synthesis Version? 25
2 Working in Synthesis 29
2.1 Databases vs. Standalone Files 30
2.2 Standard vs. Enterprise Repositories 30
2.2.1 Standard Databases 31
2.2.1.1 Creating a New Standard Database 31
2.2.1.2 Opening an Existing Standard Database 32
2.2.2 Enterprise Databases 32
2.2.2.1 Creating a New Enterprise Database 32
2.2.2.2 Connecting to an Existing Enterprise Database 33
2.2.2.3 Upgrade from Version 8 Enterprise Database 35

2.2.3 Repository Settings ... 36
2.2.3.1 E-mail Settings ... 37
2.2.3.1.1 Notification Groups ... 38
2.2.3.2 Manage Units ... 39
2.2.3.3 Default Name Formats ... 40
2.2.3.4 Global Identifiers ... 41
2.2.3.5 Project Categories ... 42
2.2.3.6 XFRACAS Connection ... 43
2.3 Projects ... 43
2.3.1 Project Properties ... 45
2.3.1.1 General Tab ... 45
2.3.1.2 Global Identifiers Tab ... 46
2.3.1.3 Security Tab ... 46
2.3.1.4 Configurable Settings Tab ... 47
2.3.1.5 MPC's Project Properties ... 47
2.3.2 Project Item Wizard ... 48
2.3.3 Manage Projects ... 48
2.3.4 Recycle Bin ... 49
2.4 Security ... 50
2.4.1 Public and Private Projects ... 51
2.4.1.1 New Projects ... 51
2.4.1.2 Existing Projects ... 52
2.4.2 Project Owner ... 52
2.4.3 Repository-Level vs. Project-Level Security ... 53
2.4.3.1 Repository-Level Security ... 53
2.4.3.2 Project-Level Security ... 54
2.4.4 Manage Repository Security ... 56
2.4.4.1 Creating User Accounts ... 56
2.4.4.1.1 User Account Requirements ... 56
2.4.4.2 Creating Security Groups ... 57
2.4.4.3 Table of Database Permissions ... 58
2.4.5 User Login and Contact Information ... 61
2.4.5.1 Display User Information ... 61
2.4.5.2 Create and Edit User Accounts ... 62
2.4.5.3 Create Alternative Login ... 62
2.4.6 Import Users from Active Directory ... 63
2.4.7 SQL Server Logins or Using Impersonation ... 64
2.4.7.1 Creating Individual SQL Server Logins ... 64
2.4.7.2 Using a Login Group ... 65
2.4.7.3 Using Windows Impersonation for the Connection File ... 66
2.4.8 Lock and Unlock Projects ... 67
2.4.8.1 Locking a Project ... 67
2.4.8.2 Unlocking a Project ... 67
2.4.9 Check In or Check Out Projects ... 67
2.4.9.1 Check Out a Project ... 68
2.4.9.2 Working with Checked Out Projects ... 68
2.4.9.3 Check In a Project ... 68
2.4.9.4 Undo Check Out ... 69
2.4.10 Item Permissions ... 69

2.4.11 Simultaneous Access By Multiple Users ... 70
2.4.11.1 Status Indicators ... 71
2.4.12 Prior Logins ... 72
2.4.13 Reset “In Use” Flags ... 72
2.5 Managing and Restoring Data ... 73
2.5.1 Backups and Database Maintenance: Protecting Your Data ... 73
2.5.2 Restore Points ... 75
2.5.2.1 Create a Restore Point ... 75
2.5.2.2 Restore a Project ... 75
2.5.2.3 Manage all Restore Points that Exist for the Database ... 76
2.6 Import, Export and Data Conversion ... 76
2.6.1 Importing from an Existing Repository ... 77
2.6.2 Importing from Prior Version Files ... 77
2.6.2.1 Importing to a New Standard Database (File > Open Repository) ... 77
2.6.2.2 Importing to an Existing Database and Project (Import > Other file) ... 78
2.6.2.3 Converting from BlockSim 6 or 7 Files ... 79
2.6.2.4 Converting Xfmea/RCM++ 5 Databases ... 81
2.6.2.5 Converting MPC 3 Databases ... 83
2.6.3 Importing/Exporting Projects ... 86
2.6.4 Importing/Exporting Project Items or Resources ... 87
2.6.5 Importing from Excel or Delimited Text Files ... 88
2.6.5.1 Selecting Which Data Sheets Will Be Imported ... 89
2.6.5.2 Using the Control Panel to Map the Columns That Will Be Imported ... 89
2.6.5.3 Setting the Import Template Directory ... 90
2.6.5.4 Creating a New Import Template ... 90
2.6.5.5 Applying a Saved Import Template ... 90
2.6.5.6 Importing the Data ... 91
2.6.6 Using XML in BlockSim and Lambda Predict ... 91
2.7 Importing XFRACAS Data to Weibull++ or RGA ... 92
2.7.1 Transferring Data from XFRACAS to the Reliability Data Warehouse (RDW) ... 92
2.7.2 Transferring Data from the Reliability Data Warehouse (RDW) to a Standard Folio ... 93
2.7.3 RDW Interfaces: Configuration and Data Filtering Tools ... 95
2.7.4 Dashboard Viewer ... 96

3 Interface ... 97

3.1 Show, Tile, Dock and Pin Panels ... 98
3.2 Project Manager ... 101
3.2.1 Filter Properties Window ... 103
3.2.2 Current Project Explorer ... 104
3.3 My Portal ... 105
3.3.1 Message Window ... 107
3.3.2 Actions ... 107
3.3.2.1 Actions as Resources ... 107
3.3.2.2 Action Statuses ... 108
3.3.2.3 Action Properties ... 108
3.3.2.4 Approve, E-mail and Outlook Commands ... 110
3.3.3 Action Notifications Window ... 110
3.4 Data Sheets ... 111
3.5 Change Units ... 113
3.6 General Spreadsheets ... 113

3.7 Control Panels ... 114
3.7.1 Switching Between Pages ... 114
3.7.2 Control Panel Identifiers Page ... 115
3.7.3 Control Panel Publishing Page ... 116
3.7.4 Control Panel Notifications Page ... 117
3.7.5 Control Panel Utilized Resources Page ... 118
3.8 Results Window ... 118
3.9 System Panel and Analysis Panel ... 119
3.10 Select Existing Window ... 120
3.11 Check Spelling ... 122
3.12 Find Tool ... 123
3.12.1 Searching FMEA Records in Xfmea, RCM++ and RBI ... 124
3.13 History Logs ... 125
3.13.1 Project History Log ... 126
3.13.2 Record History Log ... 126
3.13.3 Creating and Managing Archives ... 126
3.14 Attachments ... 127
3.14.1 Attachments Window ... 127
3.14.2 Add/Edit Attachment Window ... 128
3.14.3 Attachment Locations ... 129
3.14.3.1 Project Attachments ... 129
3.14.3.2 Synthesis Resource Attachments ... 129
3.14.3.3 Hierarchy Attachments ... 129
3.14.3.4 Block Attachments ... 130

4 Ribbon ... 131

4.1 Quick Access Toolbar ... 131
4.2 Backstage View (File Tab) ... 132
4.2.1 Help Center (File > Help) ... 135
4.2.2 License Manager ... 136
4.3 Home Tab ... 137
4.4 My Portal Tab ... 140
4.5 Project Tab ... 141
4.6 Insert Tab ... 143
4.7 View Tab ... 146
4.8 Help Tab ... 147
4.9 Active Tabs ... 148
4.9.1 ALTA Tab ... 148
4.9.2 Degradation Tab ... 150
4.9.3 Diagram Tab ... 151
4.9.4 Equation Fit Solver Tab ... 153
4.9.5 Event Log Tab ... 154
4.9.6 Life Comparison Tab ... 154
4.9.7 Life Data Tab ... 155
4.9.8 Maintenance Planning Tab ... 157
4.9.9 Non-Parametric LDA Tab ... 157
4.9.10 Plot Tab ... 158
4.9.11 Recurrent Event Data Tab ... 159
4.9.12 Sheet Tab ... 159

4.9.13 SimuMatic (A) Tab 161
4.9.14 SimuMatic (W) Tab 162
4.9.15 Stress Profile Tab 162
4.9.16 Stress-Strength Tab 162
4.9.17 Test Design Tab 163
4.9.18 Test Plans Tab 163
4.9.19 Warranty Tab 164
4.9.20 Workbook Tab 165

5 Resources 167

5.1 Types of Resources 168
5.1.1 Local and Global Resources 169
5.2 Resource Wizards 170
5.2.1 Selecting an Existing Resource 171
5.2.2 Creating a Resource 172
5.2.3 Editing a Resource 172
5.2.4 Removing a Resource 172
5.2.5 Specifying the Resources Shown in the Wizard 172
5.3 Resource Manager 173
5.3.1 Basic Operations 173
5.3.2 Advanced Operations 174
5.3.3 Exporting Information from the Resource Manager 175
5.3.4 Resource Manager Display Characteristics 176
5.3.5 Column Chooser 177
5.3.6 Advanced Filters 177
5.3.7 Select Resource Window 179
5.4 Dependency Viewer 180
5.4.1 Graphical Dependency Viewer 181
5.5 Universal Reliability Definitions (URDs) 184
5.5.1 Hierarchical View 184
5.5.2 Filtered View 185
5.6 Models 185
5.6.1 Defining and Viewing/Editing Models 185
5.6.2 Model Categories 186
5.6.3 Model Types 186
5.6.4 Model Window 189
5.6.5 Model Wizard 190
5.6.5.1 Selecting an Existing Model 192
5.6.5.2 Creating a Model 192
5.6.5.3 Editing a Model 194
5.6.5.4 Removing a Model 194
5.6.5.5 Using the Settings Page 194
5.6.6 Quick Parameter Estimator (QPE) 194
5.6.6.1 Wizard View 195
5.6.6.2 Expert View 196
5.6.6.3 ALTA Version of the QPE 197
5.6.7 Published Model Properties 200
5.7 Variables 201
5.7.1 Using Variables to Store Output Values 202
5.7.1.1 Example 202

6 Weibull++ Standard Folios 205
6.1 Weibull++ Standard Folio Setup 206
6.1.1 Time Units 206
6.2 Data Types 207
6.2.1 Times-to-Failure Data 207
6.2.2 Times-to-Failure with Suspensions (Right Censored Data) 208
6.2.3 Times-to-Failure with Interval and Left Censored Data 208
6.2.4 Times-to-Failure with All Types of Data 209
6.2.5 Free-Form (Probit) Data 210
6.3 Weibull++ Standard Folio Control Panel 210
6.3.1 Folio Tools 213
6.3.2 Weibull++ Standard Folio Analysis Settings 215
6.4 Life Data Analysis Plots 217
6.5 Quick Calculation Pad (QCP) 219
6.5.1 How to Use the QCP 219
6.5.2 QCP Calculations for Life Data Analysis 221
6.6 Life Data Analysis Example 223
6.7 Additional Tools 226
6.7.1 Weibull++ Distribution Wizard 226
6.7.2 Batch Auto Run 229
6.7.3 Goodness of Fit Results 231
6.7.4 Auto Group Data 232
6.8 Special Analysis Methods 233
6.8.1 Mixed Weibull Analysis 233
6.8.2 Bayesian-Weibull Analysis 236
6.8.3 Competing Failure Modes (CFM) Analysis 240
6.8.3.1 Using Reliability Block Diagrams (RBDs) for Failure Modes Analysis 242
6.8.4 Alter Parameters 244
6.8.5 Specify Points 245
6.8.6 Calculations with No Data Entered 246
6.8.7 Fill Median Ranks 248
7 ALTA Standard Folios 249
7.1 ALTA Standard Folio Setup 250
7.2 ALTA Standard Folio Control Panel 253
7.2.1 Folio Tools 256
7.3 ALTA Plots 257
7.4 Select Stress Columns 259
7.5 Stress Transformation 260
7.6 Use Stress Level 260
7.7 QCP Calculations for ALTA 260
7.8 ALTA Single-Stress Example 263
7.9 ALTA Two-Stress Example 265
7.10 Time-Dependent Stress Profiles 267
7.10.1 Creating Stress Profiles 267
7.10.2 Using Stress Profiles in the ALTA Standard Folio 270
7.10.3 Example 271

7.11 Additional Tools ... 274
7.11.1 ALTA Distribution Wizard ... 274
7.11.2 Add or Remove Columns ... 274
7.11.3 Likelihood Ratio Test ... 275
7.11.4 Transfer Data to Weibull++ ... 277
7.11.5 Convert Stress Values ... 278
7.11.6 Calculations with Insufficient Data Entered ... 279
8 Weibull++ Degradation Analysis ... 281
8.1 Weibull++ Degradation Analysis Folio ... 282
8.2 Weibull++ Degradation Plot Sheet ... 285
8.3 Weibull++ Degradation Analysis Example ... 285
8.4 Degradation Model Wizard ... 288
9 ALTA Degradation Analysis ... 291
9.1 ALTA Degradation Analysis Folio ... 292
9.2 ALTA Degradation Plot Sheet ... 296
9.3 ALTA Degradation Analysis Example ... 297
10 Non-Parametric LDA Folio ... 301
10.1 Non-Parametric LDA Folio Data Sheet ... 302
10.2 Control Panel Settings ... 303
10.3 Transferring Data to a Weibull++ Standard Folio ... 306
10.4 Non-Parametric Life Data Analysis Example ... 306
11 Warranty Analysis ... 313
11.1 Warranty Analysis Folio Setup ... 314
11.1.1 Warranty Analysis Folio Control Panel ... 314
11.2 Nevada Chart Format ... 317
11.2.1 Setting Up the Chart and Entering Data ... 317
11.2.2 Zero-Time Failures ... 319
11.2.3 Statistical Process Control ... 322
11.2.4 Example ... 324
11.3 Times-to-Failure Format ... 328
11.3.1 Entering Data ... 328
11.3.2 Example ... 330
11.4 Dates of Failure Format ... 333
11.4.1 Entering Data ... 333
11.4.2 Example ... 335
11.5 Usage Format ... 338
11.5.1 Entering Data ... 338
11.5.2 Estimating Usage ... 341
11.5.3 Interval Width Estimator ... 346
11.5.4 Example ... 348
11.6 Forecasting Warranty Returns ... 350
11.7 Warranty Analysis Plots ... 352
11.8 Manage Warranty Units in Weibull++ ... 352

12 Event Log Folios 359
12.1 Setting Up the Event Log Folio 360
12.2 Event Log Control Panel Settings 361
12.2.1 Control Panel Main Page 361
12.2.2 Control Panel Analysis Page 363
12.2.3 Control Panel Other Page 364
12.3 Event Log Data Analysis Example 364
12.4 Folio Tools 368
12.4.1 Set Shift Pattern 368
12.4.2 Systems Setup 369
12.4.3 Transfer Life Data 370
12.4.4 Transfer to BlockSim 372
13 Recurrent Event Data Analysis (RDA) Folios 373
13.1 Non-Parametric RDA Folio 374
13.1.1 Setting Up the Folio and Evaluating Results 374
13.1.2 Non-Parametric RDA Example 377
13.2 Parametric RDA Folio 379
13.2.1 Setting up the Folio 379
13.2.2 Parametric RDA Folio Control Panel Settings 380
13.2.2.1 Control Panel Main Page 380
13.2.2.2 Control Panel Analysis Page 381
13.2.3 QCP Calculations for Parametric RDA 383
13.2.4 Parametric RDA Folio Plots 384
13.2.5 Parametric RDA Example 385
14 Plots 387
14.1 Plot Sheet Control Panel 388
14.2 Setting Confidence Bounds 392
14.3 Show/Hide Plot Items Window 393
14.4 Contour Plots 394
14.5 Specifying Life Lines 395
14.6 Overlay Plots 396
14.6.1 Overlay Plot Control Panel 397
14.7 Side-by-Side Plots 397
14.7.1 Side-by-Side Plots Control Panel 398
14.8 3D Plots 399
14.8.1 3D Plot Control Panel 400
14.8.2 Rotating the Chart 401
14.8.3 Alter Parameters - 3D Plot 401
14.8.4 Saving and Opening Saved 3D Plots 402
14.8.5 3D Plot Ribbon 402
14.9 Tips on Working with Plots 404
14.10 Override Plot Color 405
15 Plot Setup 407
15.1 Plot Titles Page 408
15.2 Axis Titles/Labels Page 409
15.3 Plot Labels Page 410

15.4 Legend Page 411
15.5 Canvas Page 412
15.6 Grid Page 413
15.7 Bars Page 414
15.8 Slices Page 415
15.9 Plot Items Page 416
15.10 Offsets Page 417
15.11 Plot Defaults Window 418
15.11.1 Plot Defaults: Titles Page 419
15.11.2 Plot Defaults: Plot Item Display Page 420

16 ReliaSoft Draw 421

16.1 Linking RS Draw 421
16.2 RS Draw Menu Bar 422
16.2.1 File Menu 422
16.2.2 Edit Menu 422
16.2.2.1 Preferences Window 423
16.2.3 Canvas Menu 423
16.2.4 Shape Menu 423
16.2.5 Help Menu 424
16.3 RS Draw Toolbars 424
16.3.1 Standard Toolbar 424
16.3.2 Edit Toolbar 424
16.3.3 Zoom Toolbar 424
16.3.4 Draw Toolbar 425
16.3.5 Customizing Bars in the RS Draw Interface 426
16.3.5.1 Moving Toolbars 426
16.3.5.2 Hiding and Showing Toolbars 426
16.4 Additional Tools 426
16.5 Object Properties — The Tool Options Area 427
16.5.1 Lines Properties 427
16.5.2 Fill Properties 427
16.5.3 Points Properties 427
16.5.4 Angles Properties 428
16.5.5 Text Properties 428
16.5.6 Color Properties 428
16.6 Working with Objects: Examples 429
16.6.1 Drawing a Line 429
16.6.2 Editing an Object 429
16.6.2.1 Example: Editing a Line 429
16.6.2.2 Example: Resizing a Rectangle 429
16.7 Saving an Image 430

17 Monte Carlo and SimuMatic 431

17.1 Weibull++ Monte Carlo Utility 432
17.1.1 User-Defined Equations 434
17.1.2 User-Defined Equation Example 435
17.2 ALTA Monte Carlo Utility 438

17.3 SimuMatic ... 442
17.3.1 Generating Data with Weibull++ SimuMatic ... 442
17.3.2 Generating Data with ALTA SimuMatic ... 443
17.3.3 SimuMatic Folio ... 444
17.3.3.1 Simulation Sheet ... 445
17.3.3.2 Sorted Sheet ... 445
17.3.3.3 SimuMatic Control Panel ... 446
17.3.3.4 Plot Sheet ... 447
17.3.4 Transferring Data to a Weibull++ Standard Folio ... 447
17.3.4.1 Transferring Data to a New Folio ... 448
17.3.4.2 Transferring Data to an Existing Folio ... 448
17.3.5 Designing Reliability Tests with SimuMatic ... 449
17.3.6 Weibull++ SimuMatic Example ... 451
17.3.7 ALTA SimuMatic Example ... 453

18 Diagrams ... 455

18.1 Building a Diagram ... 456
18.1.1 Standard Blocks ... 456
18.1.2 Node Blocks ... 457
18.1.3 Junction Blocks ... 458
18.1.4 Annotations ... 459
18.1.5 Connecting Blocks in Diagrams ... 460
18.1.6 Diagram Analysis Constraints ... 461
18.1.7 Keyboard and Mouse Combinations ... 462
18.2 Diagram Control Panel ... 464
18.3 Equation Viewer ... 465
18.4 Diagram Plot Sheet and Control Panel ... 468
18.5 Diagram Appearance ... 468

19 Test Design Folios ... 469

19.1 Reliability Demonstration Test Design ... 470
19.1.1 Using the RDT Tool ... 470
19.1.2 Edit Bayesian Subsystems ... 472
19.1.3 Parametric Binomial Table and Plot ... 473
19.1.4 Example ... 474
19.2 Expected Failure Time Plot ... 476
19.2.1 Using the Expected Failure Time Plot ... 476
19.2.2 Example ... 478
19.3 Difference Detection Matrix ... 480
19.3.1 Using the Difference Detection Matrix ... 480
19.3.2 Example ... 482

20 Target Reliability Tool ... 485

20.1 Generating a Plot ... 486
20.2 Control Panel ... 487

21 Tests of Comparison ... 489

21.1 Folio Setup ... 490
21.1.1 Probability Equation ... 490

21.2 Control Panel Settings 491
21.2.1 Control Panel Analysis Page 492
21.2.2 Control Panel Setup Page 494
21.3 Life Comparison Example 496
21.4 Stress-Strength Test Example 498
21.5 Target Reliability Parameter Estimator 502
22 Maintenance Planning 505
22.1 Generating a Cost vs. Time Plot 505
22.1.1 Integration with BlockSim 506
22.2 Maintenance Planning Control Panel Settings 507
23 Equation Fit Solver 509
23.1 Using the Equation Fit Solver 509
23.2 Control Panel Settings 512
24 ALTA Test Plan Utility 515
24.1 Generating a Test Plan 515
24.2 Test Plan Types 516
24.3 Test Plan Results Sheet 518
24.4 Test Plan Control Panel 518
24.5 Test Plan Utility Example 519
25 Non-Linear Equation Root Finder 523
26 Quick Statistical Reference (QSR) 525
26.1 Calculating Statistical Values 525
26.1.1 Median Ranks 526
26.1.2 Other Ranks 526
26.1.3 Standard Normal Values 527
26.1.4 Inverse Standard Normal Values 527
26.1.5 Cumulative Poisson 527
26.1.6 Cumulative Binomial Probability 528
26.1.7 F-Distribution Values 528
26.1.8 Inverse F-Distribution Values 529
26.1.9 Chi-Squared Values 529
26.1.10 Incomplete Beta Function 530
26.1.11 Gamma Function 530
26.1.12 Student's t Values 530
26.2 Interpolating a Value from a Data Set 530
27 Analysis Workbooks 533
27.1 Analysis Workbook Wizard/Word Report Template Wizard 534
27.2 Associated Data Sources 534
27.3 Saved Workbook/Report Templates 536
27.4 Analysis Workbook Control Panel 537
27.5 Function Wizard 539
27.5.1 Using the Function Wizard in Any of the Reporting Tools 539
27.5.2 Data Entry Tips for Functions 542
27.5.3 Using Functions to Return Confidence Bounds 545
27.6 Chart Wizard and Designer 546

27.7 Tools for Working with Spreadsheets 547
27.7.1 Spreadsheet Components 547
27.7.2 Referencing Cells in the Spreadsheet 548
27.7.3 Defined Names 548
27.7.4 Format Cells - Number Window 550
27.7.5 Sheet Options Window 551
27.7.6 Calculation Options 552
27.7.7 Protecting Cells 554
27.7.8 Sort 555
27.7.9 Exporting the Spreadsheet 557
27.8 Analysis Workbook Example 557

28 Word Report Templates 561

28.1 Word Report Template Control Panel 562
28.2 Plot Wizard 563
28.3 Tools for Working with Word Report Templates 564
28.3.1 Formatting Paragraphs 565
28.3.2 Find and Replace Text or Special Characters 566
28.3.3 Inserting Pictures and Symbols 567
28.3.4 Inserting Bookmarks and Hyperlinks 568
28.3.5 Inserting Tables and Setting Properties 569
28.4 Word Report Template Ribbon 571
28.4.1 Home Tab 572
28.4.2 Document Tab 573
28.4.3 Insert Tab 575
28.4.4 View Tab 575
28.4.5 Table Tab 576
28.5 Word Report Example 577

29 Printing 581

29.1 Page Setup 581
29.1.1 Page Tab 581
29.1.2 Margins Tab 582
29.1.3 Header/Footer Tab 583
29.1.4 Header/Footer Format Codes 585
29.2 Print Preview 585
29.3 Print Window 587

30 Setup Windows 589

30.1 Synthesis Setup 589
30.2 Application Setup 592
30.2.1 Application Setup User Page 592
30.2.2 Application Setup Weibull++ Folios Page 593
30.2.3 Application Setup ALTA Folios Page 594
30.2.4 Application Setup Calculations Page 596
30.2.5 Application Setup Plot Page 598
30.2.6 Application Setup Other Page 600
30.2.7 Application Setup Customize Folio Tools Page 601
30.2.8 Application Setup Reset Settings Page 602

Appendix A: Minimum Data Requirements 605
31.1 Requirements for Weibull++ 605
31.1.1 Standard Folios 605
31.1.1.1 Case 1: No Data 606
31.1.1.2 Case 2: No Failures and 1+ Suspensions 606
31.1.1.3 Case 3: 1 Failure and No Suspensions 607
31.1.1.4 Case 4: 1 Failure and 1+ Suspensions 609
31.1.1.5 Case 5: 2+ Failures 610
31.1.2 Degradation Analysis Folios 610
31.1.3 Non-Parametric Life Data Analysis (LDA) Folios 610
31.1.4 Warranty Analysis Folios 610
31.1.5 Event Log Folios 611
31.1.6 Parametric Recurrent Event Data Analysis (RDA) Folios 611
31.1.7 Non-Parametric Recurrent Event Data Analysis (RDA) Folios 612
31.2 Requirements for ALTA 612
31.2.1 Standard Folios 612
31.2.2 Degradation Analysis Folios 613

Introduction 1

With the release of the Synthesis version, the capabilities of ReliaSoft's Weibull++ and ALTA 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. Within the user documentation:

- This graphic indicates that a feature is available only for users with a Weibull++ license. 

- This graphic indicates that a feature is available only for users with an ALTA license. 

Weibull++ and ALTA 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. 2)
- 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. 12)
- Region and Language Settings - Section 1.7 (p. 13)
- Product Features
  - Introduction to the Synthesis Platform - Section 1.8 (p. 16)
  - Weibull++ Features Overview - Section 1.9 (p. 18)
  - ALTA Features Overview - Section 1.10 (p. 21)
  - Features Common to Weibull++ and ALTA - Section 1.11 (p. 23)
  - What's New in the Synthesis Version? - Section 1.12 (p. 25)

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. 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.

1.2 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**.

[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.

- 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 desktop 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 12*.
Operating Systems Supported	
Microsoft Windows	**XP** - 32-bit (Service Pack 3 or higher). *The 64-bit version of Windows XP is not supported*.
	Vista - 32-bit and 64-bit (Service Pack 1 or higher)
	Windows 7 - 32-bit and 64-bit
	Windows 8 - 32-bit and 64-bit
Microsoft Windows Server (Requires a license that specifically allows server installations/deployments.)	**2003 R2**
	2008 - 32-bit and 64-bit (Service Pack 2 or higher)
	2008 R2
	SBS (Small Business Server) 2011

Microsoft Office Integration	
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.	
IMPORTANT NOTE: 32-bit and 64-bit Versions	
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.	

		Microsoft Office Version	
		32-Bit	64-Bit
Windows Version	32-Bit	32	
	64-Bit	32 or 64†*	32 or 64*

[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).*

Enterprise Database Platforms	
Microsoft SQL Server	2005, 2008 and 2012 32-bit and 64-bit versions
Microsoft SQL Server Express Edition	2005, 2008 and 2012 32-bit and 64-bit versions
Oracle	10g and 11g 32-bit and 64-bit versions
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 applications and then you can choose which application(s) to activate on your computer. Please note the following:

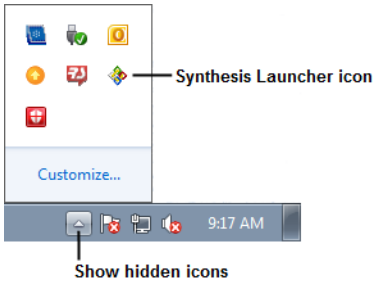
- Before running the installation program, make sure your computer meets the **minimum system requirements** for Synthesis. *See Section 1.3 on page 3*.
- To perform the installation, you must be logged in with a user account that has **administrative rights**.
- It is strongly recommended that you **close all other applications** during the installation.
- After the installation, all Synthesis 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:

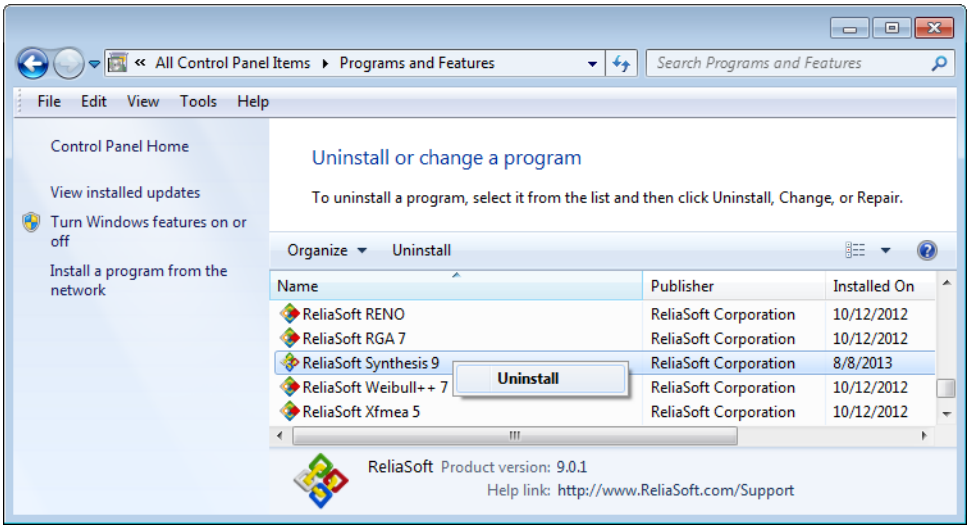
- **Deactivate Without Uninstall**. This will free up the specified activation(s) for your product license key so you can activate on another computer or for another user. However, you can continue to use any other Synthesis applications that remain activated on this computer. If you decide to start 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 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 6*.

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]

[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.

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, 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

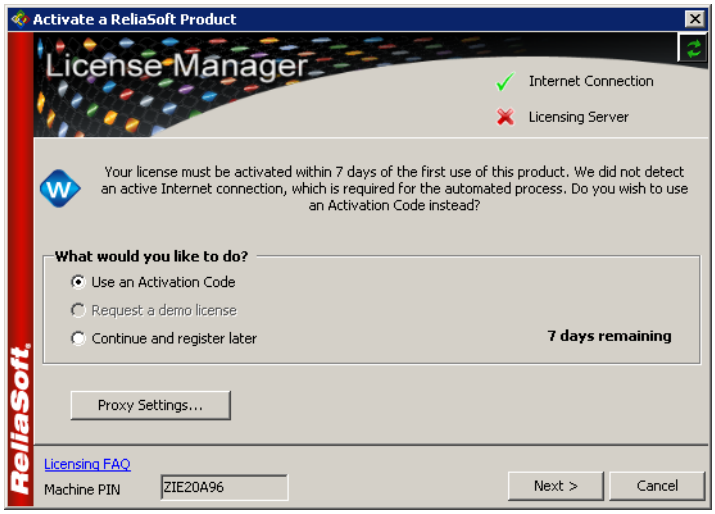
The software must communicate with the ReliaSoft License Server at **https://Validate.ReliaSoft.org** each time you activate a particular application on a particular computer. For Concurrent Network Licenses, the software must also connect each time you attempt to access and use the software. Personal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

Possible Solutions:

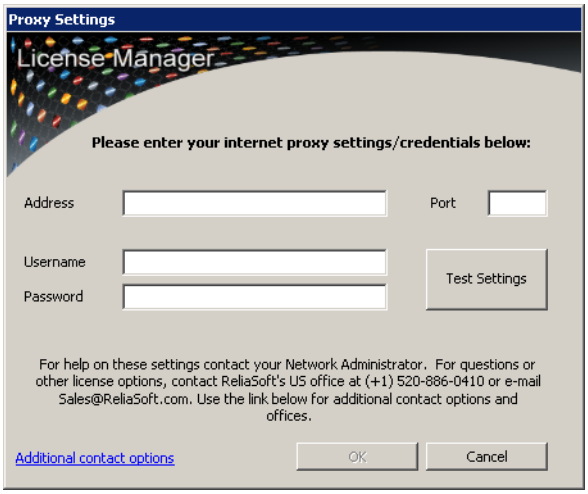
- Personal firewalls (Norton Firewall, ZoneAlarm, BlackIce, Outpost, McAfee Firewall, etc.) need to be configured to allow the software to communicate with the server. Generally, these products will ask for your authorization when the software first tries to establish communication with the server (e.g., "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.)

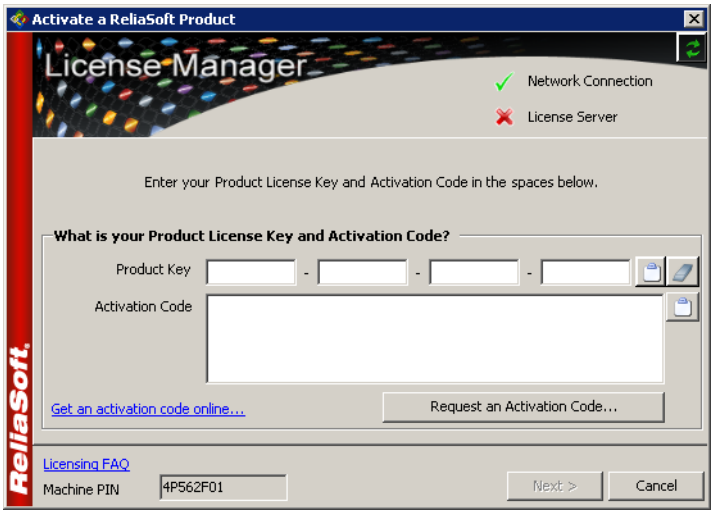


2. In the case that the server connection fails but the software detects network access, the wizard will first prompt you to indicate whether a proxy server is being used. If you choose to configure the software to communicate through a proxy, the Proxy Settings window looks like the one shown next. 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.

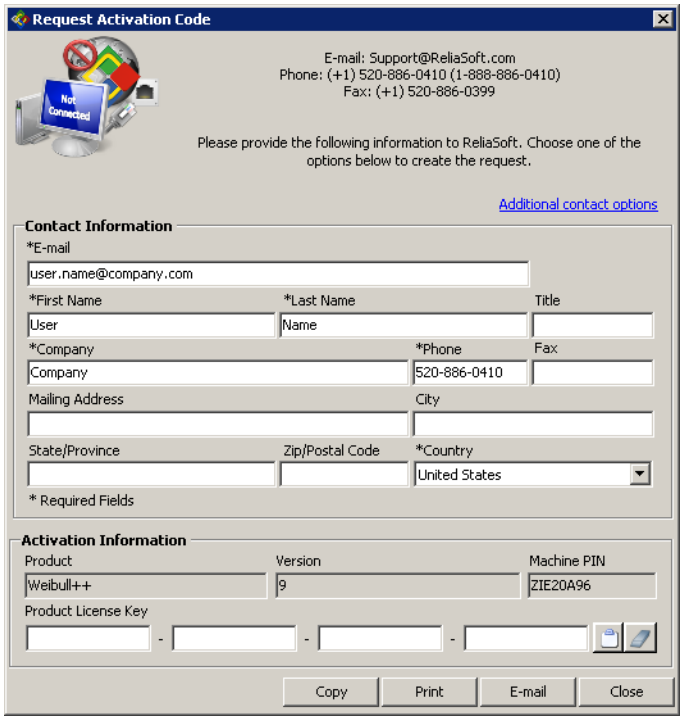


Once a proxy is successfully set, you can continue with the automated process. If the proxy fails, you may want to try again and confirm that the entries are valid. If the test is still not successful, you can exit the window using the **Cancel** button and then try one of the two methods for requesting 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.



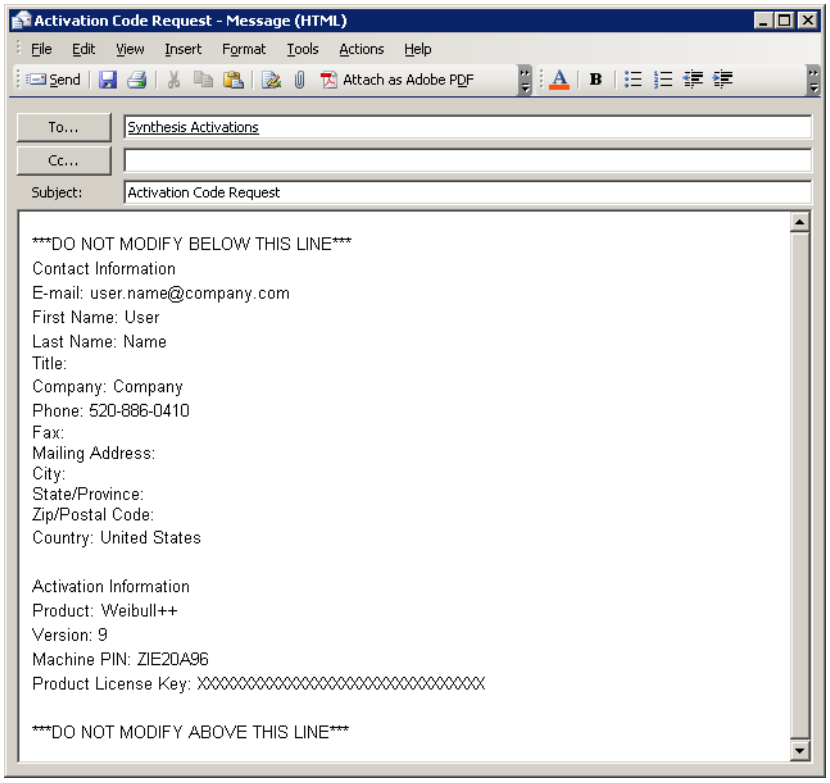
Once you enter the required information, you have the following options:

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.

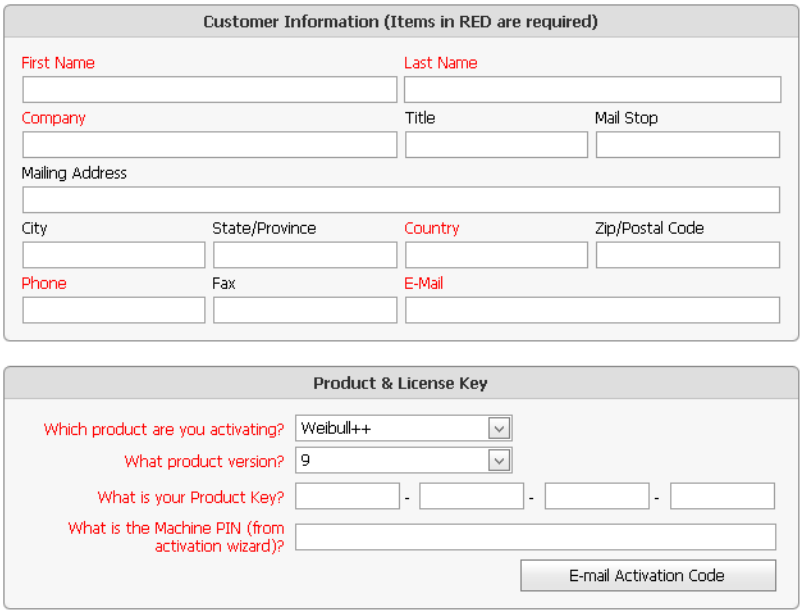
C. Copy the information to a text file so you can submit it to ReliaSoft at a later time via e-mail, mail or phone.

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.”

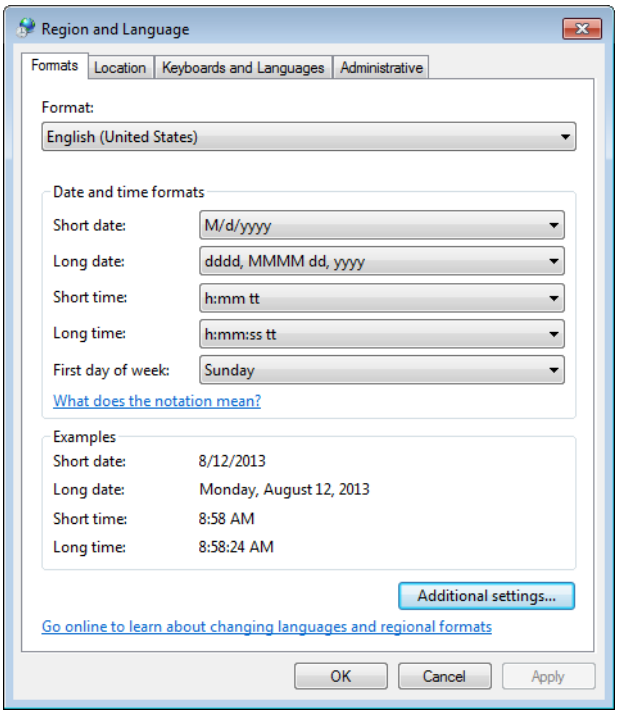
This section first explains how to view or change the Region and Language settings on your computer and then discusses some specific considerations for Synthesis applications.

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 cursor 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.

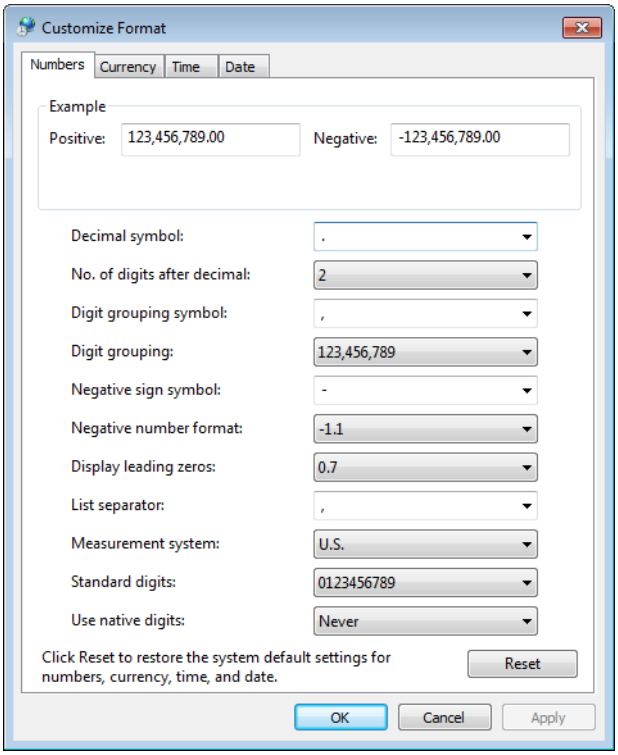
As an example, the Windows 7 interface is shown here. The Windows 8, Windows Vista and Windows XP versions look different but function similarly.



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 displayed 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 Weibull++ and ALTA.

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 "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 versions of Weibull++ and ALTA 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 Weibull++). 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. *For specific recommendations, see page 73 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 "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 Weibull++ and ALTA, the Synthesis Platform gives you the ability to:

- Publish models that are based on data analyses performed in Weibull++ and ALTA folios. This serves to make this information available to other analyses performed in BlockSim, Xfmea, RCM++ or RBI.
- Extract incident report data from XFRACAS for analysis in Weibull++.

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 applications offer the **My Portal** interface (*see page 105 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. Now in the Synthesis version, 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 Weibull++ Features Overview

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

ReliaSoft's Weibull++ provides the most comprehensive toolset available for reliability life data analysis, calculated results, plots and reporting. The software is also packed with tools for related analyses, such as warranty data analysis, degradation data analysis, non-parametric data analysis, recurrent event data analysis (for repairable systems) and reliability test planning.

Standard Life Data Analysis (LDA)

The **Weibull++ standard folio** supports all life data types and all major lifetime distributions. You can analyze time-to-failure (complete), right censored (suspension), left censored, interval censored or free-form data, entered individually or in groups. Available lifetime distributions and analysis methods include:

- 1, 2 and 3 parameter Weibull
- 1 and 2 parameter Exponential
- Normal and Lognormal
- Gamma and Generalized Gamma
- Logistic and Loglogistic
- Gumbel
- Bayesian-Weibull (which considers prior knowledge of the Weibull shape parameter)
- 2, 3 and 4 subpopulation Mixed Weibull (for situations when there are different trends in the data but you cannot identify a distinct failure mode for each data point)
- Competing Failure Modes (CFM) analysis (which analyzes failure modes separately and then assumes a series reliability model in which each failure mode "competes" to cause the failure)

If you are not sure which model is appropriate for a given data set, the convenient **Distribution Wizard** automatically performs several types of goodness-of-fit tests in order to rank the available distributions.

Parameter Estimation options for standard life data analysis include Maximum Likelihood Estimation (MLE), Rank Regression on X (RRX) or Rank Regression on Y (RRY).

Weibull++ also provides **Confidence Bounds** for parameters, calculated results and plots. Depending on the specific analysis method used, the confidence bounds may be calculated using the Fisher Matrix, Likelihood Ratio, Beta Binomial or Bayesian approach.

The software provides a complete array of **Calculated Results and Plots** based on the analysis. For life data analysis, this includes:

- Reliability or probability of failure
- Reliable life (i.e., time for a given reliability, also called "warranty time")
- BX% life (i.e., time for a given unreliability)
- Mean life
- Failure rate
- Probability plots and pdf plots
- Contour plots
- Failure/suspension histograms, pie charts or timelines

Degradation Data Analysis

The Weibull++ degradation analysis folio allows you to extrapolate the expected failure times of a product based on measurements that reflect how some performance measure (e.g., increase in crack propagation, decrease in tread depth, increase in vibration, etc.) has degraded for sample units over a period of time. The software offers a choice of the Linear, Exponential, Power, Logarithmic, Gompertz or Lloyd-Lipow models to analyze the degradation data, and generates Degradation vs. Time plots on either a linear or logarithmic scale.

Now in the Synthesis version, Weibull++ also automatically calculates the life data model based on the extrapolated failure times and allows you to obtain calculated results and plots directly within the same folio — no need to transfer or link to another analysis folio!

Non-Parametric Life Data Analysis

Weibull++'s non-parametric LDA folio offers a choice of three methods for analyzing life data without assuming an underlying life distribution: Kaplan-Meier, Simple Actuarial and Standard Actuarial. This folio may be useful when dealing with unknown failure modes, when there is not enough data to assume a life distribution or when the data set does not fit any life distribution in a satisfactory way. Now in the Synthesis version, Weibull++ also performs a parametric analysis directly within the same folio using the unreliability estimates that are generated by the non-parametric analysis.

Warranty Data Analysis

Weibull++'s popular warranty analysis folio converts warranty claims data (sales and returns) that are readily available in many organizations into failure/suspension data sets that can be analyzed with traditional life data analysis methods. You can use this analysis to better understand the failure behavior of products in the field and to generate forecasts of future returns that will be covered under warranty. The software provides a choice of data entry formats to fit your particular needs: Nevada Chart, Times-to-Failure, Dates of Failure or Usage. The folio provides all of the special options you need to analyze the data in a way that's appropriate for the available data and your organization's warranty fulfillment practices. For example:

- The **Use Subsets** option allows you to deal with non-homogeneous populations by analyzing data from different design iterations simultaneously and performing forecasts based on mixed sales data.
- The **Suspend After** option allows you to take into account the possibility that failure data were not collected beyond the specified warranty period and/or to exclude predicted failures that will not be covered under warranty.

- The **Statistical Process Control** feature (available for Nevada chart folios) can automatically detect abnormal sales or return periods and color-code the results to highlight specific data points you may wish to investigate further.
- The **Usage** format allows you to enter returns data in terms of the amount of usage accumulated (e.g., mileage, cycles, etc.) rather than time in service. In the Synthesis version, we have enhanced this feature by providing additional tools to help you configure the analysis to appropriately estimate the likely usage for units that are still operating in the field (i.e., suspensions).

Convert Maintenance Event Logs to Failure and Repair Data

Weibull++ provides a specialized folio designed specifically to capture system failure and repair data in an event log format like those commonly used in the machine tools and other industries. If you have a log that records the date/time when a system downing event occurred and the date/time when the system was restored to operation, the software converts this information to time-to-failure and time-to-repair data that can be analyzed with life data analysis techniques.

Some useful options in this folio include the ability to:

- Define the shift pattern that describes the time periods when the systems are operating each day.
- Analyze data for a single system or multiple systems simultaneously.
- Categorize events as “failures” or “events,” and then specify whether non-failure events will be considered in the life data analysis.
- Specify which component/assembly is responsible for each event, and then perform the analysis for any level of the system configuration.

Recurrent Event Data Analysis (RDA) for Repairable Systems

The failure behavior of repairable systems is dependent on the history of repairs, and therefore traditional life data analysis methods (which treat failure data as independent and identically distributed) are usually not applicable. To provide the appropriate analysis treatment for such data, Weibull++ offers two analysis folios that can be used to analyze recurrent event data from repairable systems:

- The **Non-Parametric RDA** folio uses the well-known Mean Cumulative Function (MCF) to plot the average number of recurring failures over a given period of time. The plot can be used to evaluate whether the number of failures is increasing or decreasing over time, to predict the future number of failures and to compare data sets from different designs, operating conditions or production periods.
- The **Parametric RDA** folio uses the General Renewal Process (GRP) model, which takes into account the effectiveness of repairs on the condition of the system and models the cumulative number of failures over time. You can use the analysis to generate a variety of plots and calculated results, including the number of failures, failure intensity, mean time between failures (MTBF) and conditional reliability.

Reliability Test Design

The Synthesis version offers a new **Test Design Assistant** that helps you select which reliability test design tools will meet your specific needs.

- The **Reliability Demonstration Test Design** tool has been completely redesigned and expanded in the Synthesis version. You can use the Parametric Binomial, Non-Parametric Binomial, Exponential Chi-Squared or Non-Parametric Bayesian methods to help choose the right test time/sample size for a reliability demonstration test. These methods can be used for designing a zero-failure test (where the reliability target is demonstrated if you don’t observe any failures during the test) and for tests with other quantities of “allowable failures” (e.g., one-failure test, two-failure test, etc.)

- The **Expected Failure Times Plot** provides a visual depiction of the failure times you can expect to observe when you implement a particular test plan. If you perform the test and enter the actual failures as they are observed, you can use the plot to monitor whether the test is proceeding as expected or receive an early warning that adjustments may be needed.
- The **Difference Detection Matrix** calculates how much test time is required before it is possible to detect (and demonstrate) a statistically significant difference in the life of two product designs.

Target Reliability Estimator

Choosing an optimal reliability goal involves deciding on important trade-offs. For example, higher reliability typically requires higher production costs, but higher reliability will typically also lead to lower warranty costs and higher market share. The new **Target Reliability Tool** generates multiple plots that will help you select a target reliability that will minimize cost, maximize profit and maximize the return on an investment that affects reliability.

1.10 ALTA Features Overview

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

Quantitative accelerated life testing techniques, used in conjunction with powerful accelerated life data analysis methodologies, give design and reliability engineers the power to significantly reduce test times and produce more robust designs. This can, in turn, provide faster time to market, lower product development costs, lower warranty costs and a host of other tangible and intangible benefits. ReliaSoft's ALTA software provides a comprehensive toolset for accelerated life test planning and quantitative accelerated life testing data analysis. ALTA is available in two versions:

- **ALTA Standard** provides the life-stress relationship models required to analyze accelerated life test data with 1 or 2 constant stresses.
- **ALTA PRO** offers advanced capabilities for analyzing accelerated life test data with up to 8 simultaneous stress types where stress is constant or varies with time.

Accelerated Life Testing Data Analysis

ALTA provides all the tools and options you will need for accelerated life testing data analysis. It is designed for use with complete (time-to-failure), right censored (suspended), interval or left censored data. Data can be entered individually or in groups.

ALTA Standard gives you a choice of the following life-stress relationships:

- **Arrhenius**: a single stress model typically used when temperature is the accelerated stress.
- **Eyring**: a single stress model typically used when temperature or humidity is the accelerated stress.
- **Inverse Power Law (IPL)**: a single stress model typically used with a non-thermal stress, such as vibration, voltage or temperature cycling.
- **Temperature-Humidity**: a double-Arrhenius model that is typically used when temperature and humidity are the acceleration variables.
- **Temperature-Nonthermal**: a combination of the Arrhenius and IPL relationships that is typically used when one stress is temperature and the other is non-thermal (e.g., voltage).

In addition, ALTA PRO offers four other models for more advanced applications:

- **Generalized Eyring**: a variation of the Eyring relationship that is typically used when both temperature and humidity are accelerated.
- **Proportional Hazards**: analyzes data with up to 8 stress types using the Exponential relationship for each stress. It also allows the use of zero as a stress value, which enables the analysis of data with indicator variables (e.g., 0 = on/off and 1 = continuous operation).
- **General Log-Linear**: supports the analysis of data with up to 8 stress types and provides the flexibility of specifying the life-stress relationship (Arrhenius, Inverse Power Law or Exponential) for each stress.
- **Cumulative Damage**: analyzes data with up to 8 stress types and/or situations where the stress varies with time.

You can select to combine these life-stress relationships with an underlying Weibull, Exponential or Lognormal life distribution. If you are not sure which is appropriate for a given data set, the convenient **Distribution Wizard** automatically performs multiple goodness-of-fit tests in order to rank the available distributions.

Maximum Likelihood Estimation (MLE) is used for all parameter calculations. ALTA also provides Fisher Matrix confidence bounds for parameters, calculated results and plots.

The software provides a complete array of **Calculated Results and Plots** based on the analysis. For accelerated life testing data analysis, this includes:

- Reliability or probability of failure
- Reliable life (i.e., time for a given reliability, also called “warranty time”)
- BX% life (i.e., time for a given unreliability)
- Mean life
- Failure rate
- Acceleration factor
- Probability plots and pdf plots
- Life vs. stress, acceleration factor vs. stress, and standard deviation vs. stress plots
- Residuals plots
- Three-dimensional surface plots such as reliability vs. time vs. stress

Time-Dependent Stress Profiles

As described in the “ALTA Standard Folio” chapter, the cumulative damage life-stress model (available only in ALTA PRO) can handle data from tests where the stress varies with time (e.g., a step-stress or ramp-stress profile). You can define and store any number of stress profiles, then easily assign the appropriate profile to each point in an ALTA data sheet. Each segment in a profile can be associated with a specific stress value (e.g., temperature is 310 K from 0 to 100 hours and 320 K from 101 to 200 hours) or a time-dependent function (e.g., start from a temperature of 310 K and then increase by 10 K with each hour of testing).

Accelerated Degradation Analysis

The ALTA degradation analysis folio allows you to analyze degradation data obtained under accelerated stress levels in order to estimate when failure would occur under the accelerated conditions. Now in the Synthesis version, the software also automatically performs accelerated life data analysis on the extrapolated failure times and allows you to

obtain calculated results and plots for the use stress conditions directly within the same folio — no need to transfer or link to another analysis folio!

Accelerated Life Test Planning

The ALTA Test Plan utility implements the complex mathematical models required to design an effective accelerated life test plan. The tool offers a choice of one-stress or two-stress test planning methods. Based on your inputs about the expected failure behavior and available test time, the software recommends the stress levels to be tested and the most effective allocation of available test units to each level.

1.11 Features Common to Weibull++ and ALTA

Many of your favorite features are available for both Weibull++ and ALTA analyses. Some of these features include:

Plots and Charts to Visualize Your Analysis Results

Weibull++ and ALTA make it easy to create a complete array of plots and charts to present analysis information 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 applications, such as comparing different data sets or analysis methods (e.g., Design A vs. Design B or MLE vs. Rank Regression) or demonstrating the effects of a design change (e.g., Before vs. After).

The **Side-by-Side Plots** utility allows you to view (and print) multiple plots for a given data set side-by-side. For example, you may want to show the probability, reliability, pdf and failure rate plots for a given analysis together in the same window. Alternatively, you may wish to compare the probability or *pdf* plots for a given data set when analyzed with different distributions/models.

Monte Carlo Simulation

As described in the "Monte Carlo and SimuMatic" chapter, both Weibull++ and ALTA continue to use **Monte Carlo** simulation for generating data sets that can be analyzed directly in a standard folio. You can also use the **SimuMatic®** utilities to automatically perform a large number of reliability analyses on data sets that have been created via simulation. These simulated data sets and calculated results can be used to perform a wide variety of reliability tasks, such as:

- Experimenting with the influence of sample sizes and censoring schemes on analysis methods.
- Constructing simulation-based confidence bounds.
- Experimenting with confidence intervals for analyses performed with different distributions.
- Developing and evaluating test plans.
- Utilizing user-defined equations for risk analysis and probabilistic design.

Stress-Strength Calculator and Life Comparison Test

As described in the "Tests of Comparison" chapter, Weibull++ and ALTA provide two tools designed for statistical comparison of data sets. The **Life Comparison** tool allows you to compare two data sets to determine whether items from the first set will outlast those of the second. The **Stress-Strength** comparison tool uses the same statistical approach to determine the probability of failure based on the probability of a specified "stress" data set exceeding a specified "strength" data set. Both tools have been redesigned and enhanced in the Synthesis version.

Reliability Block Diagrams (RBDs) for Failure Modes Analysis

A simple Competing Failure Modes (CFM) analysis can be performed directly within the Weibull++ standard folio by assuming a series reliability model and using only one type of distribution to analyze all the modes. When a more complex analysis is required, you can use the **reliability block diagram** (RBD) feature that is directly integrated into Weibull++ and ALTA projects.

The blocks in the diagram are linked directly to analyzed standard folios and you can use the RBD in any or all of the following situations:

- The analysis involves more than four failure modes.
- The failure modes are described by different life distributions.
- The relationship between the failure modes does not follow a series configuration. For example, if more than one failure mode must occur together in order for the component to fail, a parallel configuration must be used for the analysis and this is referred to as "complex failure modes analysis" rather than "competing failure modes analysis."

Workbooks and Reports for Custom Analysis

As described in the "Reports" chapter, all Synthesis applications offer several powerful tools for custom analysis and reporting.

- If you need a quick way to perform additional calculations that are associated with a particular analysis, you can insert any number of **General Spreadsheets** into any Weibull++ or ALTA standard folio.
- If you want to create a custom analysis or report that integrates results from different analysis folios, the Synthesis version 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®).

All three tools allow you to use the **Function Wizard** to automatically insert calculated results based on selected data sheets. With the workbooks and report templates, you also have the option to configure the functions to use generically numbered "data sources" instead of named data sheets. This makes it easy to use the same template again for different data sets.

Quick Parameter Estimator (QPE)

The **Quick Parameter Estimator** (formerly called the "Parameter Experimenter") allows you to estimate the parameters of a statistical model based on what you know about the behavior over time. With the Weibull++ utility, you can solve for one parameter of the distribution if you provide one data point and the other parameter(s), or solve for all parameters if you provide two data points. The ALTA version allows you to incorporate information about the life at the accelerated stress levels in order to estimate the missing parameters of the entire model (i.e., life-stress relationship and use-level life distribution).

Quick Statistical Reference (QSR)

The **Quick Statistical Reference** frees you from tedious lookups in tables by quickly returning results for commonly used statistical functions. Results include Median Ranks, Chi-Squared Values, Cumulative Binomial Probability, and many more. There is also a Polynomial Interpolation Function that allows you to enter known data points and then calculate Y for any given X value.

Non-Linear Equation Fit Solver and Root Finder

Weibull++ and ALTA continue to offer two familiar statistical tools for working with non-linear equations.

- The **Non-Linear Equation Fit Solver** allows you to estimate the parameters of any user-defined non-linear equation. This gives you the flexibility to perform simple parameter estimation on statistical models other than the life distributions and life-stress relationship models available in standard folios. After you have solved the equation, the tool provides a plot to visualize how the data fit the function, and also makes it easy to calculate Y for any given X value.
- The **Non-Linear Equation Root Finder** allows you to quickly solve for the root of any user-defined non-linear equation. This helps you eliminate some of the guesswork of solving for the value of the unknown variable that makes the function be as close to zero as possible.

1.12 What's New in the Synthesis Version?

With the release of the Synthesis versions, Weibull++ and ALTA 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

Weibull++ and ALTA 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 one simple example of the many integration opportunities, you can now seamlessly use Weibull++ and/or ALTA analyses to set the properties for a reliability block diagram or fault tree in BlockSim, and the software can automatically update the diagram if the underlying data analysis changes.

New and Enhanced Test Design Tools

Weibull++'s reliability demonstration test design tool has been completely redesigned and expanded in the Synthesis version. The new interface uses terminology that is more familiar to real-world practice and stores the analysis in a folio for future reference. We have also added support for a fourth test design method: **Non-Parametric Bayesian**. *See Chapter 19*.

The Synthesis version also offers two completely new tools that were inspired by feedback from Weibull++ users:

- The **Expected Failure Times Plot** can help you to anticipate what will occur during a reliability test and also provide an early warning if the test is not proceeding as expected.
- The **Difference Detection Matrix** helps to determine how much test time may be required before the captured data will make it possible to detect a difference in the reliability of two competing designs.

New Target Reliability Estimator

The new **Target Reliability Tool** generates multiple plots designed to help you select a target reliability that will minimize cost and maximize profit/ return on investment. *See Chapter 20*.

New Maintenance Planning Tool

The ability to calculate the optimum replacement time provides a powerful opportunity to reduce a system's maintenance costs while maximizing uptime. The new **Maintenance Planning Tool** generates a cost vs. time plot designed to help you determine the most cost-effective time to replace a system's worn or failed components. *See Chapter 22*.

The tool also offers an option to create preventive and/or inspection tasks for use in BlockSim simulation diagrams.

Improved Usability for the Stress-Strength and Life Comparison Tools

Both the Life Comparison and Stress-Strength Calculator tools have been improved in the Synthesis version. Now you can save individual analyses in the project for future reference, and each folio automatically generates a *pdf* plot to visualize the comparison. The new version also allows you to obtain the confidence bounds on the calculated probability.

Obtain the Reliability Model Directly Within Degradation and Non-Parametric LDA Folios

When you use a degradation analysis or non-parametric LDA folio to extrapolate failure/suspension times, it's no longer necessary to transfer the data to a separate folio for life data analysis or ALTA analysis. The Synthesis version now automatically calculates the LDA or ALTA model directly within the same folio and gives you immediate access to the calculated results and plots.

Redesigned QCP

The Quick Calculation Pad (QCP) has been updated and redesigned in the Synthesis version. The new interface provides a "Calculation Log" that works like a 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.

Additional Confidence Bounds Calculations

The Likelihood Ratio confidence bounds method is now available for logistic, loglogistic, Gumbel and gamma distributions.

Bayesian confidence bounds calculations are now also available for normal, lognormal, logistic, loglogistic, Gumbel and gamma distributions.

Redesigned Quick Parameter Estimator

The Quick Parameter Estimator has proven to be a useful tool whenever you need to define a model without being able to calculate the parameters of a data set. Both of the QPE tools have been improved in the Synthesis version.

In the Weibull++ utility (for life distributions such as Weibull or lognormal), we've added a new wizard view that bypasses the statistical terminology and guides you through a series of questions that will "translate" what you know from practical experience with the product's behavior into the information required to estimate the distribution and parameters.

In the ALTA version, you can now estimate the parameters of the entire model, including both the life-stress relationship and the use-level life distribution (e.g., Arrhenius-Weibull or Eyring-Lognormal). We have also added support for models with more than one accelerated stress type (e.g., Temperature-Humidity or General Log-Linear).

New Interval Width Estimator for Warranty Folio Usage Format

The Usage format in Weibull++'s warranty analysis folios supports the analysis of warranty returns data in terms of usage (e.g., miles, cycles, etc.) rather than time in service. When you define the usage as a distribution, the Synthesis version now offers a new tool to help you estimate appropriate usage rate intervals.This tool allows you to describe the typical usage (per day, month or year) in terms of either an average rate or a statistical distribution. *See page 346 in Chapter 11*.

Reintroducing Word Report Templates

The software continues to offer built-in spreadsheets (similar to Microsoft Excel) that can be used for custom analysis and reporting (called "general spreadsheets" and "analysis workbooks"). Now in the Synthesis version we have reintroduced the "Word Report Template," a custom reporting utility that has functionality similar to Microsoft Word. All three tools allow you to automatically insert calculated results from selected data sheets. With the Word Report Templates, you can also insert tables and plots that are built automatically based on your specifications and the currently selected data source.

Automatic Conversion for Time Values Entered in Different Units

With the release of the Synthesis version, all Synthesis applications now allow you to define conversion factors for entering time values in different units. To give just one simple example, you can now enter test data into a standard folio as *hours* but then use the QCP to calculate the reliability for a *year* of operation — the software performs the conversion automatically based on the multipliers specified for the Synthesis repository.

Color-Coding to Facilitate Data Entry

With the Synthesis version, we have introduced new color-coding aids to assist with data entry and management. There is now an immediate color change if you enter data in an incorrect format. For Weibull++ standard folios, you also have the option to color-code data points based on their data type (e.g., failure time vs. suspension) or subset ID.

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

In all Synthesis applications, you will work with repositories, which are relational databases that can contain one or many *projects*. Each project can contain one or more analyses; in addition, the project can contain *resources*, which represent various types of information available for use throughout the project. The databases and all of their projects can be accessed by any Synthesis application that is installed on your computer. In each application, you will see only the analyses and resources that are applicable to the current application. Certain types of resources (such as models, maintenance tasks, etc.) are applicable in, and accessible from, multiple applications.

The sections in this chapter address the following:

- Some general tips about transitioning from a document/file format (as in Weibull++7 or BlockSim 7) to a multi-user, database driven approach in Synthesis - Section 2.1 (p. 30).
- The differences between standard and enterprise databases (repositories), and how to create new databases and open existing databases - Section 2.2 (p. 30).
- Repository settings - Section 2.2.3 (p. 36), 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.
- Creating and managing projects - Section 2.3 (p. 43).
- Implementing repository security and managing workflow in a multi-user environment - Section 2.4 (p. 50).
- Managing and restoring data via backups/maintenance and restore points - Section 2.5 (p. 73).
- Storing modification information in the history log - Section 2.6 (p. 76).
- Importing and exporting information - Section 2.6 (p. 76).
- 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. 92)

2.1 Databases vs. Standalone Files

With integration into the Synthesis Platform, many of 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.

At the same time, we recognize that the experience of working with database-driven, multi-user applications will be a bit different from other types of "standalone" applications that you may be more familiar with (such as prior versions of Weibull++, BlockSim and other ReliaSoft applications).

To help ease the transition, we have compiled some targeted tips and FAQs on the ReliaSoft.com website that are designed to help you take full advantage of the new database-driven Synthesis Platform.
See http://Synthesis.ReliaSoft.com/tips.

Some of the topics that may be addressed in these tips include:

- The importance of protecting against data loss and corruption by establishing adequate backups and database maintenance. *See Section 2.5.1 on page 73*.
- Why Synthesis applications no longer use a Save command, and how you can use the Restore Points feature if you want to be able to discard recent changes. *See Section 2.5.2 on page 75*.
- Using features such as Project Categories (*see Section 2.2.3.5 on page 42*) and the Manage Projects window (*see Section 2.3.3 on page 48*) to effectively manage many different analysis projects in the same centralized database.
- Establishing the appropriate project-level or repository-level security, and understanding how Synthesis applications facilitate simultaneous access by multiple users. *See Section 2.4.3 on page 53*.

This user's guide provides a supplement to the information that's already available here in the application's *User's Guide* about these and other features.

2.2 Standard vs. Enterprise Repositories

Synthesis applications offer the choice to store analysis data in a standard database (*.rsr9) or enterprise database (Oracle or Microsoft SQL Server).

- Standard databases are easy to create and maintain without any special IT infrastructure or support but there are limitations to the amount of data they can store and the number of users who can access the database simultaneously.
- An enterprise database requires implementation and support of Oracle or Microsoft SQL Server 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. 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).

Note: For each instance of the application running on your computer, you can open only one database at a time. If a database is currently open, it will be closed automatically when you create or open a different one.

Note About Establishing a Database Server with Oracle or SQL Server

When using Oracle or SQL Server for data storage, most organizations will choose to establish a separate database server with the appropriate IT maintenance and support. It also is necessary to purchase the appropriate license

package for the database platform. This 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). If you have a reasonably powerful computer and administrative rights to install and configure software, you can establish a functioning enterprise database on your own. If your organization's needs grow beyond the capabilities of the Express edition, you can upgrade to a more robust version with the appropriate IT infrastructure and support.

Note: For power users and/or more robust performance, SQL Server Express or Oracle Express can be installed on the local computer, allowing the user to create enterprise repositories without the need for a database server.

Although it is important to note that ReliaSoft cannot provide full documentation and support for third-party database platforms that provide back-end data storage for Synthesis applications, we do provide a limited number of resources 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.

To access these resources, please visit:

- SQL Server: http://Synthesis.ReliaSoft.com/sql_server.htm (includes instructions for how to configure SQL Server Express for use with Synthesis application)
- Oracle: http://Synthesis.ReliaSoft.com/oracle.htm

2.2.1 Standard Databases

2.2.1.1 Creating a New Standard Database

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. *See Section 2.4 on page 50*. You can then select the **Immediately go to security window** option if you want to add user accounts as part of the database creation process. If you do not select this option, you will be the only authorized user until you add more users. *See Section 2.4.4 on page 56*.

  Note: If you create a non-secure database, you can enable security later by choosing **File > Manage Repository > Authorized Users** and then clicking the **Apply Login Security** button. 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. *See Section 2.6.1 on page 77*.

An additional option is also available:

- Select the **Always start by connecting with the last repository** check box if you want the application to always connect to the last database you used. The database will open automatically if the connection information is valid and your user account is active.

When you create a new database, at least one project is automatically created as part of the process. Specifically:

- If security is used, then two projects will be created: a private project and a public project. *See Section 2.4.1 on page 51*.
- If security is not used, then a single public project will be created.

2.2.1.2 Opening an Existing Standard Database

To open an existing database, choose **File > Recent** and then select the *.rsr9 file in the **Recent Repositories** list. You can also choose **File > Open Repository** and browse for the *.rsr9 file.

If the database has login security enabled, the software will get the domain and username that you used to log in to your computer and check to see if they match an active user account in the database. (*See Section 2.4.4 on page 56*.) If there is a match, the database will open automatically with the permissions that have been defined for that user account.

Tip: The domain is part of your database login information. For this reason, if you anticipate that you will need to open a standard database from a computer that is not on the same domain, you should make arrangements in advance to set up an alternative login that you can use to access the database from other domains. This is required regardless of where the database file is currently located. *See Section 2.4.5.3 on page 62*.

Connection Issues

If you are unable to connect with an existing standard database, you may be encountering any of the following issues:

- **No access to the database**. If you have not been granted access to the database, you will receive a message stating that your account in the database is not active or not assigned to at least one security group. To correct this problem, you will need assistance from a user with the "Manage users and logins" permission to edit your account settings. *For details on activating/deactivating accounts and assigned security groups, see Section 2.4.4 on page 56*.
- **The database does not recognize your current Windows login**. Synthesis repositories use Windows authentication to verify the user. If your current Windows login (domain/username) is different from what was specified for your user account, then you will receive a message to connect using an alternative login. You will need assistance from a user with the "Manage users and logins" permission to enable the use of an alternative login for your user account. *See Section 2.4.5.3 on page 62*.

2.2.2 Enterprise Databases

2.2.2.1 Creating a New Enterprise Database

If your organization already has established a database server with Oracle or Microsoft SQL Server and you have the permissions necessary to create a new database on the server, you can choose **File > Manage Repository > Repository Creation** to open a window that you will use to create a new database.



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.
- **Schema** of the new Enterprise database.
- **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. *See Section 2.6.1 on page 77*.

Click **OK** to create the database. The database will not open automatically; you must connect to it. *See Section 2.2.2.2 on page 33*.

IMPORTANT for SQL Server Databases: To create a functioning user account in a SQL Server database, the username must be associated with a SQL Server Login, which allows SQL Server to recognize the username via Windows Authentication. There are three ways to ensure that a user account created via a Synthesis application will be associated with a SQL Server Login. If you have not employed one of these three methods, then the user account will not be able to log in to the SQL Server database. *See Section 2.4.7 on page 64*.

2.2.2.2 Connecting to an Existing Enterprise Database

If your organization already has created an enterprise database on Oracle or Microsoft SQL Server and you have an active user account for the database, you can connect to the database as explained in this section.

Connecting for the First Time

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. Once this file is created, it can be used to connect to the database and to import from or export to the database.

Specify the 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 2.4.7 on page 64*.
- 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.

Select the **Always start by connecting with the last repository** check box if you want to connect to the last database you used the next time you launch the application. The database will open automatically if the connection information is valid and your user account is active. If you are using an Oracle database and the current Windows login credentials cannot be validated, the software will display a window that allows you to type a different domain, username and password, if appropriate. You may need to use this option, for example, if you are logged in to your personal home computer but you are trying to access a database on your company's network via VPN.

Once you have connected to the database, you can manage user accounts by choosing **File > Manage Repository > Authorized Users**.



Reconnecting

To connect to an enterprise database that you have already connected to, choose **File > Recent** and in the **Recent Repositories** list, choose the *.rserp file. You can also choose **File > Open Repository** and then browse for the *.rserp file.

Tip: It is important to note that the domain is part of your database login information. For this reason, if you anticipate that you will need to connect to an enterprise database from a computer that is not on the same domain, you should make arrangements in advance to set up an alternative login that you can use to access the database from other domains. *See Section 2.4.5.3 on page 62*.

Connection Speed

When you are connected to an enterprise database, a connection speed indicator will be displayed in the MDI status bar.

- < 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)

Connection Issues

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

- **No access to the database**. If you have not been granted access to the database, you will receive a message stating that your account in the database is not active or not assigned to at least one security group. To correct this problem, you will need assistance from a user with the "Manage users and logins" permission to edit your account settings. *See Section 2.4.4 on page 56*.

- **Cannot connect to the server or login failed**. Server-related issues may occur for several reasons, and you may receive various messages pertaining to the situation. Common issues are:
  - You do not have a network connection or you may have entered the incorrect database or server name in the connection file. Check if you have a network connection and enter the correct name of the database or server.
  - 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 2.4.7 on page 64*.
- **The database does not recognize your current Windows login**. Synthesis repositories use Windows authentication to verify the user. If your current Windows login (domain/username) is different from what was specified for your user account, then you will receive a message to connect using an alternative login. (*See Section 2.4.5.3 on page 62*.) You will need assistance from a user with the "Manage users and logins" permission to enable the use of an alternative login for your user account. *See Section 2.4.4.3 on page 58*.

2.2.2.3 Upgrade from Version 8 Enterprise Database

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). *See Section 2.2.2.1 on page 32*.

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.
- **Schema** of the new Enterprise database.
- **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: To create a functioning user account in a SQL Server database, the username must be associated with a SQL Server Login, which allows SQL Server to recognize the username via Windows Authentication. There are three ways to ensure that a user account created via a Synthesis application will be associated with a SQL Server Login. If you have not employed one of these three methods, then the user account will not be able to log in to the SQL Server database. *See Section 2.4.7 on page 64*.

2.2.3 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 sections 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 2.2.3.1 (p. 37).
- **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 2.2.3.2 (p. 39).
- **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 2.2.3.3 (p. 40).
  - 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 2.2.3.4 (p. 41).
  - Project Categories can be used to filter the projects displayed in the project manager - Section 2.2.3.5 (p. 42).
- **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 2.2.3.6 (p. 43)

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.

2.2.3.1 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**.

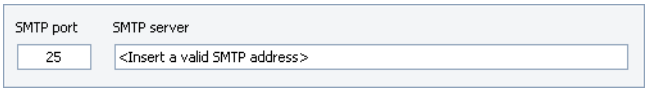


(If you do not have the "Manage e-mail notifications" permission, this command will not be visible in the Backstage view.)

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.

2.2.3.1.1 Notification Groups

Notification groups provide an easy way to make sure that all users in a certain group receive information. E-mail notifications for actions can be directed to notification groups. In addition, messages in My Portal can be directed to notification groups.

Users with the "Manage e-mail notifications" permission can manage notification groups by choosing **File > Manage Repository > Notification Groups**.



The table displays a list of the notification groups that have been defined in the current database. To add a new group, click **Add**. To edit an existing group, select a row and click Edit or double-click the row. To delete an existing group, select a row and click **Delete**. ***There is no undo for delete***.

Note: Upon deletion, the notification group will no longer be available for future notifications. Any existing actions/messages assigned to the group's members will remain in the Message page of each member's My Portal window but the name of the group will no longer be visible.

When you choose to add or edit a notification group, the Notification Group window will be displayed.

A unique name is required and a short description of the intended purpose for the group is optional. In addition, you must specify at least one user account for each notification group. The **Available Users** area displays a list of all user accounts that have been defined in the current database and have not yet been added to the current notification group. The **Selected Users** area displays a list of all user accounts that have been added to the current notification group. If you double-click an account in one list, it will be moved to the other list automatically. In addition, you can use the >> and << buttons to move all user accounts from one list to the other or use the > and < buttons to move only selected user accounts.

Tip: To quickly add multiple user accounts to the current notification group, press the **CTRL** key and click the user accounts in the **Available Users** list, then click the > button to move them to the **Selected Users** list.

2.2.3.2 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.



2.2.3.3 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 2.2.3.4 on page 41*.
  - \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.).

2.2.3.4 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 2.3.1 on page 45*.

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.

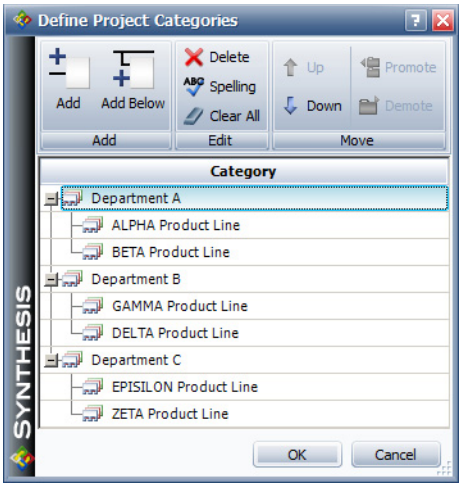
Tip: You can change identifiers at the project level on the Global Identifiers tab of the Project Properties window, which is accessed by choosing **Project > Management > Edit Project Properties**.

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.

2.2.3.5 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***.



2.2.3.6 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 2.4.7 on page 64*.
- 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.

2.3 Projects

Synthesis repositories can contain one or many projects, which serve as a way to keep related analyses together. Any project can be opened in any Synthesis application. Shared data (such as attachments and resources) can be accessed from any application. All other analyses (e.g., folios in Weibull++, diagrams in BlockSim, hierarchies in Xfmea, etc.) are visible only in the application(s) that can edit them.

Projects can be public or private (*see Section 2.4.1 on page 51*). In a secure database, a private project is accessible only to the project owner and to users with the “Manage all projects” permission. For public projects, access depends on the security settings that have been implemented for the project and the user's account. *See Section 2.4.3 on page 53*.

There are two interfaces in which you can view/manage the projects in a database.

- The 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).
- If you have the "Manage all projects" permission, you can use the Manage Projects window to view and edit the properties and security settings of all the projects in the database.

Creating a New Project

When you create a new database, at least one project is automatically created as part of the process. To add a project to an existing database, you can:

- Choose **Project > Management > Create Project**.



- Select an existing project in the project list and choose **Project > Management > Duplicate Project**.



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

Setting the Project Properties and Adding Project Items

Depending on the circumstances, creating a new project may open one or both of the following windows:

- The Project Properties window contains information that can be used to organize and manage multiple projects in the database (e.g., name, category, etc.). For a secure database, it also provides the option to specify which users will be able to access the project. Finally, in applications such as Xfmea and RCM++, it allows authorized users to set configurable properties that are the same for all analyses in the current project. You can access this window at any time by choosing **Project > Management > Edit Project Properties**.



- In some Synthesis applications, the Project Item Wizard provides information about all of the analyses that you could add to the project from the current application. (*See Section 2.3.2 on page 48*.) When applicable, you can access this window at any time by choosing **Insert > Wizard > Project Item Wizard**.



Deleting a Project

To delete a project, select it from the project list 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. If you decide to proceed, the **Send project to recycle bin** check box in the confirmation window determines whether the project will be permanently deleted (no way to undo unless you have a backup or restore point), or just moved to the Recycle Bin where it can be quickly restored, if needed. *For more information on the Recycle Bin, see Section 2.3.4 on page 49*.

2.3.1 Project Properties

The Project Properties window will be displayed when you add a new project to an existing database 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. This may include:

- General Tab - Section 2.3.1.1 (p. 45)
- Global Identifiers Tab - Section 2.3.1.2 (p. 46)
- Security Tab - Section 2.3.1.3 (p. 46)
- Configurable Systems Tab (in Xfmea/RCM++/RBI Only) - Section 2.3.1.4 (p. 47)
- MPC's Project Properties - Section 2.3.1.5 (p. 47)

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, which provides an easy way to determine which projects have been updated recently. *See Section 2.3.3 on page 48*.

2.3.1.1 General Tab

The General tab will be displayed 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, Project Properties and many other windows and reports throughout the software. This field is required and cannot contain any of the following characters: \ / : " * ? < > |.
- **Description** is an additional identifier for the project that appears only in the Project Properties window.
- **Remarks** contains additional remarks about the project that appear only in the Project Properties window.
- **Proprietary Label** allows you to enter a statement regarding the proprietary nature of the project.

  In Xfmea, RCM++ and RBI, this label appears the footer of the reports created via the Reports window. To define a default statement that will be assigned to each new project that you create, choose **Home > Reporting > Reports** then click the **Settings** button in the Reports window.
- **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. *See Section 2.2.3.5 on page 42*.

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. For more information, see "Choosing the FMEA Structure" in the Xfmea, RCM++ or RBI User's Guide.
- **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.



The **Add** icon allows you to add a new profile and the **Edit** icon allows you to edit the profile that is currently selected in the list. These features will create or change a profile in the active library and then apply the same settings to the current project.

2.3.1.2 Global Identifiers Tab

The Global Identifiers tab will be displayed 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 names of the identifiers are defined at the database level via the Global Identifiers window; these identifiers will be used in all projects throughout the database unless different settings are specified at the project level. This tab works in the same way as the Global Identifiers window. *See Section 2.2.3.4 on page 41*.

2.3.1.3 Security Tab

The Security tab will be displayed in the Project Properties window if you are working with a public project in a secure database. It allows you to specify the users and groups that can access the project. The ability to access this tab is restricted to a) the project owner, b) users with the "Edit project security" permission for this project and c) users with the "Manage all projects" permission.

- The **Use repository-level security** option is the default setting for all projects created in the database. The project will be accessible to any user who has an active user account that is assigned to at least one security group. The user will have all of the permissions within this project that have been granted via any of the security groups that are assigned to his/her user account.
- The **Use specific security groups and users** option allows you to implement "project-level security" limiting access to the project only to selected security group(s) and/or user(s).
  - In the **Limit by Groups** area, select the check box for each security group you want to give access to the project.
  - In the **Limit by Users** area, you can grant access to individual users by clicking the **Add** button. This opens a list of users who currently have access to the database. Select the check box for each user you want to give access to the project and click **OK**. You can then click the + beside each individual user account to view a list of permissions that have been granted to the user via any of the security groups assigned to his/her account. Use the check boxes to select which of those permissions will be in effect for that user in this particular project.

For a more detailed discussion of how you might want to configure the security for the projects in your database, see Section 2.4.3 on page 53.

Tip: For quick access to the Security tab of the Project Properties window, select the project in the project list and choose **Project > Security > Project Security**.

2.3.1.4 Configurable Settings Tab

In Xfmea/RCM++/RBI only, the Configurable Settings tab 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. The ability to access this tab is restricted to a) the project owner, b) users with the "Edit project properties" permission for this project and c) users with the "Manage all projects" permission.

This tab allow 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.

2.3.1.5 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. This is applicable only in MPC.

Note: 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. This is applicable only in MPC.

In addition, note that any information that you enter into the **Proprietary Label** field will be shown in the footer of MPC's Standard report template, but is not applicable for the Dassault or Sukhoi templates.

2.3.2 Project Item Wizard

For some Synthesis applications, the Project Item Wizard provides information about all the analyses that you could add to the project from the Synthesis application that you are currently working with, and guides you through any steps required to configure the analysis that you have selected. In Weibull++, ALTA, BlockSim, RENO, DOE++ and RGA, it appears automatically upon creation of a new project. You can also open it by choosing **Insert > Wizard > Project Item Wizard**.



The navigation panel on the left side of the wizard displays the categories of folios, tools and/or diagrams that you can create with the current application. The right side of the window displays the options available in the category that's currently selected. You can click any item in the list to see a more detailed description. For more information, click the **Learn More…** link.

If you've selected an option that requires further input (e.g., to select the data type for a standard folio), click **Next** to proceed. Once all required inputs have been made, click **OK** to create the new project item.

2.3.3 Manage Projects

If you have the "Manage all projects" permission, you can use the Manage Projects window to view and edit the properties and security settings of all the projects in the database. To do this, choose **Project > Management > Manage Projects**.



This displays a list of all the projects that currently exist for the database. Note that in this window, you can edit the properties and security settings of a project or delete a project *even when the project is open or in use*. The user who has the project open will see the changes 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.).

The Ribbon in the Manage Projects 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 2.3.1 on page 45*.



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 *(see Section 2.5.1 on page 73)* or restore point *(see Section 2.5.2 on page 75)*.



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 2.4.8 on page 67*.

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 2.4.1 on page 51*.

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).

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 2.4.3 on page 53*.

Check Out

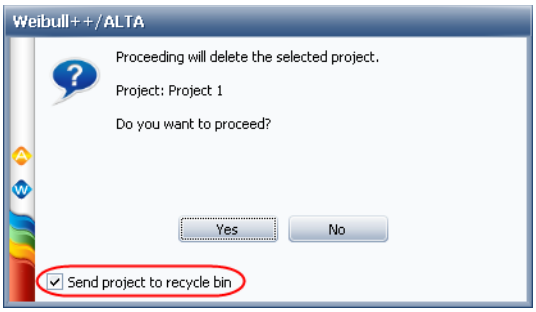
Undo Check Out discards all changes made to a checked out project and restores it to the state it was at the time it was checked out. *See Section 2.4.9 on page 67*.

Excel

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

2.3.4 Recycle Bin

When you delete a project from a database, you have the option to move the project to the Recycle Bin, as shown next.



To immediately delete the project entirely, clear the **Send project to recycle bin** check box. If you leave the check box selected, the project will be stored in the Recycle Bin until you either empty the Recycle Bin or delete the project from the Recycle Bin. This gives you an opportunity to recover accidentally deleted projects and restore them to their original locations. Any deletion that skips the Recycle Bin or removes the project from the Recycle Bin cannot be undone unless you have a backup or restore point.

In order to restore or delete a project from the Recycle Bin, you must have access to the project and have the same 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).

Restoring a Project

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

Deleting All Projects from the Recycle Bin

To permanently delete all projects currently in the Recycle Bin from the database, select the Recycle Bin in the Project Manager and choose **Project > Management > Recycle Bin > Empty Recycle Bin**.



There is no undo for emptying the Recycle Bin.

Deleting a Selected Project from the Recycle Bin

To permanently delete the currently select projects from the database, select the project in the Recycle Bin and choose **Project > Management > Recycle Bin > Delete Project**.



There is no undo for delete.

2.4 Security

All enterprise databases use login security (i.e., they are secure). A standard database can be configured with or without login security (i.e., it may be *secure* or *non-secure*).

In a **non-secure standard database**, any user who has read/write access to the database file (*.rsr9) will have full administrative and user permissions throughout the database. Security-related features (such as the ability to create a private project or to lock a project) will still be visible in the interface to help you prepare for the transition from a non-secure to secure database, if desired. But the only factor that will prevent any user from performing any function in the database is whether another user who is simultaneously logged in to the same database has the record currently in use. If you wish to enable security for a standard database, choose **File > Manage Repository > Authorized Users** and then click the **Apply Login Security** button.

In a **secure database (both standard and enterprise)**, the actions that a user can perform in any given project within the database depend on multiple factors:

- Whether the project is **public or private** - Section 2.4.1 (p. 51).
- Whether the user is assigned as the **project owner** for the project - Section 2.4.2 (p. 52).
- For a public project, whether the project uses **repository-level or project-level security** - Section 2.4.3 (p. 53).

  In general, we recommend to:

  - Use **repository-level security** if you want all projects to be accessible to any user account that is assigned to at least one security group.
  - Use **project-level security** if you want different users to have different levels of access for different projects.
- The security group(s) that have been assigned to the user account (via the **Manage Repository Security** window), and the specific permissions associated with each of those groups. *See Section 2.4.4 on page 56*. (When managing user accounts, an administrator may also need to view/edit the **login and contact info** [*see Section 2.4.5 on page 61*] for each user, **import user account information** from Microsoft Active Directory® [*see Section 2.4.6 on page 63*] and/or take steps to make sure that users will be able to **connect with SQL Server databases** [*see Section 2.4.7 on page 64*].)
- Whether the project is **locked** - Section 2.4.8 (p. 67).

- Whether the project has been **checked out** - Section 2.4.9 (p. 67).
- Whether the project item (e.g., a diagram or allocation analysis in BlockSim, a folio or tool in Weibull++/ALTA, or a system hierarchy item in Xfmea/RCM++) inherits its **item permissions** from the project, or can only be edited by a specific subset of the user(s) who are able to edit the rest of the project - Section 2.4.10 (p. 69).

Note: In addition to the factors listed above, the security groups can also be used to assign certain permissions that apply throughout the entire database (such as the ability to manage user accounts or manage database time units). If one of these permissions is granted via any of the security groups assigned to the user's account, he/she will have the capability throughout the entire database.

In both secure and non-secure databases, the centralized data storage allows multiple users to work collaboratively on analysis projects. Therefore, access to a particular record at any given time will also depend on whether it is currently in use by **another user who is simultaneously logged in** to the same database. *See Section 2.4.11 on page 70*.

Also note that all Synthesis applications keep track of **prior logins** (*see Section 2.4.12 on page 72*) to the database, and it may sometimes be necessary to **reset the "in use" flags** (*see Section 2.4.13 on page 72*) that are used to determine which user(s) are currently editing some portion of the analysis data.

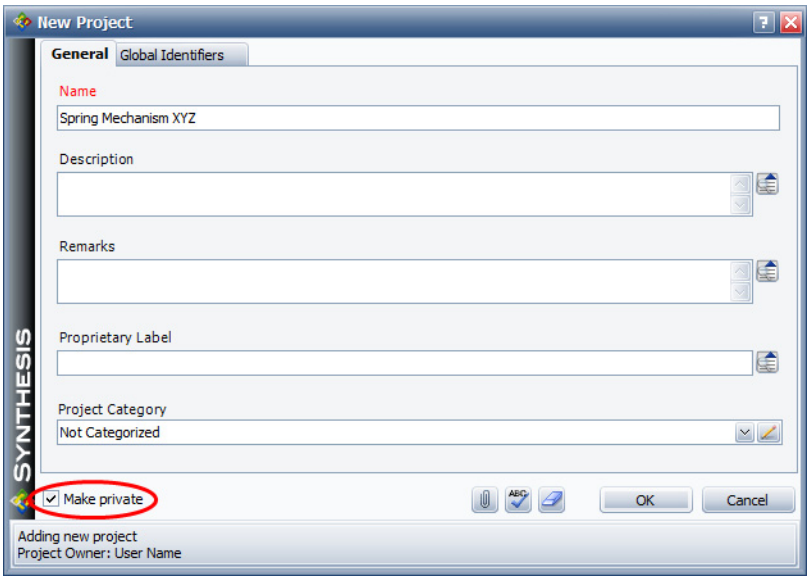
2.4.1 Public and Private Projects

In Synthesis repositories, there are two types of projects that you can create: a public or a private 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). *See Section 2.4 on page 50*.

The following sections describe how to set the public or private status of a project.

2.4.1.1 New Projects

If you have the "Create and own private projects" permission, the **Make Private** check box will be displayed in the Project Properties window whenever you create a new project, as shown next. If you select the option, the project will be created under the **Private** heading of the project list. If you clear the option, the project will be created under the **Public** heading.



2.4.1.2 Existing Projects

To change the private status of an existing project, select the project in the Project Manager and choose **Project > Security > Make Public**.



To change the public status of an existing project, select the project in the Project Manager and choose **Project > Security > Make Private**.



Tip: If you have the "Manage all projects" permission, you can select multiple projects and change their private/public status all at the same time via the Manage Projects window. *See Section 2.3.3 on page 48*.

2.4.2 Project Owner

In a secure database, the user who is assigned as the project owner for a particular project has full permissions within that project. This includes the "Basic permissions at project level" and the "Advanced permissions at project level," even if the user does not have those permissions for any of the security groups assigned to his/her account.

By default, the project owner will be the user who created the project, but users with the "Manage all projects" permission can change the owner at any time. *See Section 2.4.4.3 on page 58*.

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

- You can use the filters in the project list to sort projects based on the project owner. Each project will be displayed under the name of its corresponding owner. *See page 101 in Chapter 3*.
- If you have the "Edit project properties" or "Manage all projects" permission, you can identify the owner by selecting the project in the project list and then choosing **Project > Management > Edit Project Properties**. This opens the Project Properties window where the name of the current owner is displayed in the status bar at the bottom of the window.

To assign a different database user to be the owner for a selected project, select the project in the project list and choose **Project > Security > Change Owner**.



Tip: If you have the "Manage all projects" permission, you can select multiple projects and assign one database user to be the owner of those projects via the Manage Projects window. *See Section 2.3.3 on page 48*.

2.4.3 Repository-Level vs. Project-Level Security

For every project in a secure database, there is a choice of which type of security will control how users can access the project. In general, we recommend to:

- Use repository-level security if you want all projects to be accessible to any user account that is assigned to at least one security group - Section 2.4.3.1 (p. 53).
- Use project-level security if you want different users to have different levels of access for different projects - Section 2.4.3.2 (p. 54).

Tip: Although it is possible for some projects in the database to use repository-level security and other projects to use project-level security, it is generally recommended to configure all projects with one type or the other.

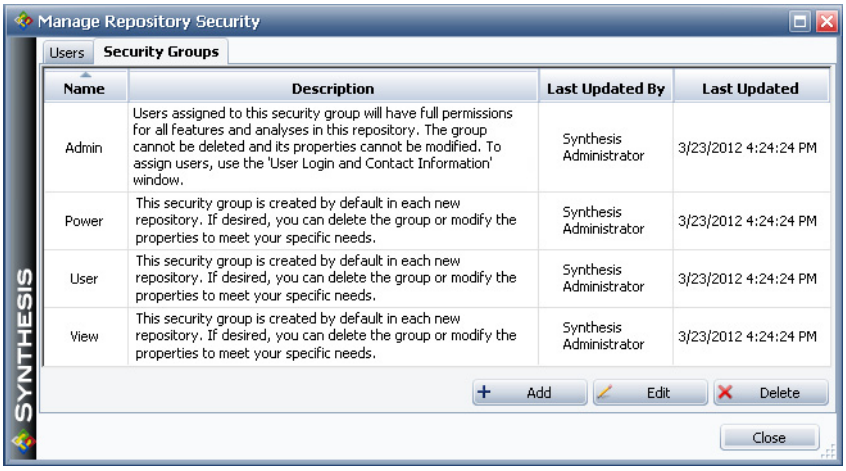
Both security methods use security groups to determine the permissions that users will have within projects. A security group is a set of permissions that can be granted to user accounts, with permissions ranging from read-only access to full administrative authority. Users with the "Manage users and logins" permission can use the Manage Repository Security window to set up security groups and users. *See Section 2.4.4 on page 56*.

To set or change a project's security, use the Security tab of the Project Properties window. (*See Section 2.3.1 on page 45*.) You will need at least one of the following roles/permissions to access this tab:

- You can configure the security settings of any project for which you are assigned as the project owner. *See Section 2.4.2 on page 52*.
- You can configure the security settings of any project for which you have the "Set project security" permission. *See Section 2.4.4.3 on page 58*.
- If you have the "Manage all projects" permission, you can also use the Manage Projects window to view and edit the properties of all projects in the database. *See Section 2.3.3 on page 48*.

2.4.3.1 Repository-Level Security

The following example shows the security groups that are created by default when you create a new database. The Admin group, which provides full permissions for all features and all analyses in the database, cannot be deleted and the properties cannot be modified. For the other predefined groups, you can edit their permissions or replace them with new groups that fit the specific way that the database will be used.



If all projects in the database use only repository-level security, then all users will have the specified level of access for all projects. For example, with the basic set of security groups that are created by default in a new database, user

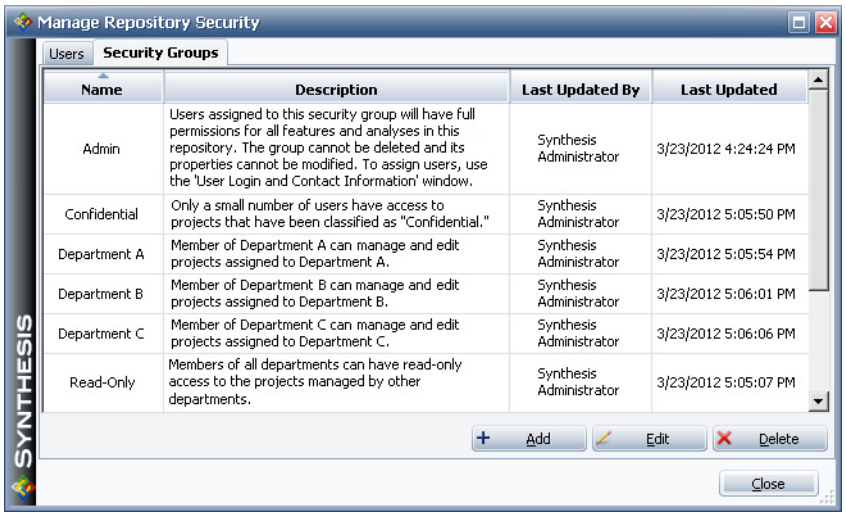
accounts assigned to the “View” security group will have read-only access to all projects in the database, whereas user accounts assigned to the “User” security group will have read and write permissions to all projects.

If you need the database to provide users with different levels of access for different types of projects, then you will need to implement project-level security, as described in the next section.

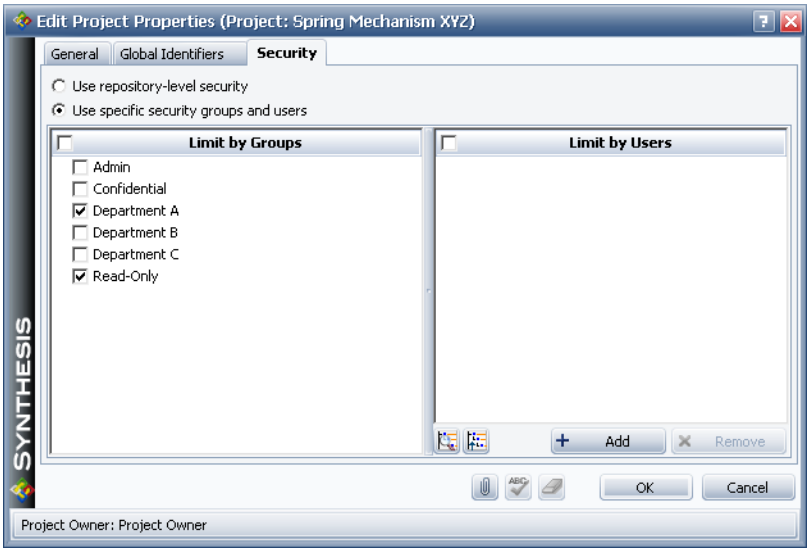
2.4.3.2 Project-Level Security

Project-level security allows you to control which security group(s) or individual user(s) can view or edit each individual project.

For example, suppose you want to grant a user read/write access to all projects that belong to her own organizational group (Department A), read-only access to all projects that belong to another group (Department B and Department C), and no access at all to projects that have been classified as “Confidential.” In this case, you would create a security group for each type of access, and then assign the appropriate security groups to the user's account. The settings for this scenario are shown next.

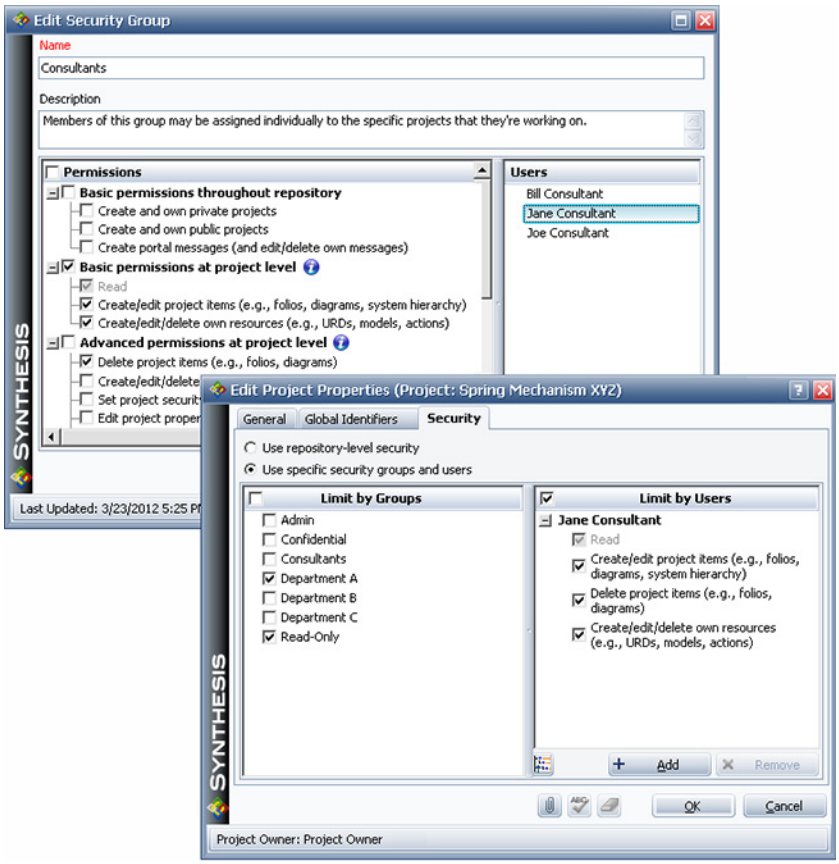


Next, you must limit access to each project in the database based on specific security groups and/or users (i.e., select **Use specific security groups and users** on the Security tab of the Project Properties window). The following picture shows the security settings for one of the projects maintained by Department A. This configuration shows how access may be limited by security group(s).



In this example, users assigned to the "Department A" security group will be able to access the project with the full set of read/write permissions that have been granted to that group, while users from other departments who are also assigned to the "Read-Only" security group will have the limited read-only permissions that have been granted to that group.

Now assume that Department A is working with an outside consultant for one of their many different analysis projects. To configure the database to allow this consultant to have individual access to only one specific project, you assign her account to a new security group called "Consultants." This new group defines the permissions that will be available to a consultant for any particular project that he/she might be individually assigned to. Then you use the **Limit by Users** option to give her these permissions for only one of the Department A projects, as shown next.



Note that if a particular user is assigned in the **Limit by Users** area and is also a member of one (or more) of the groups assigned in the **Limit by Groups** area, the user will have the least restrictive set of permissions. For example, suppose that Jane Consultant later becomes a member of the Department A group. Even if you clear the "Create project items" check box under her name in the **Limit by Users** area, Jane will still have the permission for this project because she belongs to the Department A group. Likewise, if the Department A group does not have the "Delete project items" permission, but Jane is eligible for that permission from another security group that is not assigned to the current project, then if you select the "Delete project items" check box under her name, Jane will have an additional permission in this project that is not available to the other members of the Department A group.

For additional information, see http://Synthesis.ReliaSoft.com/tips/security_groups.htm.

2.4.4 Manage Repository Security

In order for a user to access a Synthesis repository, he/she must have a valid user account in the database. This section describes how users with the "Manage users and logins" permission can create and manage user accounts in the database via the Manage Repository Security window.

The Manage Repository Security window may be displayed when you create a new enterprise database, when you enable login security for a standard database, or when you choose **File > Manage Repository > Authorized Users**.



The following sections describe how to use the Manage Repository Security window:

- Creating user accounts - Section 2.4.4.1 (p. 56)
- Creating security groups - Section 2.4.4.2 (p. 57)
- Table of database permissions - Section 2.4.4.3 (p. 58)

2.4.4.1 Creating User Accounts

In the Manage Repository Security window, the Users tab displays a list of the user accounts that have been created in the current database. Note that:

- In non-secure databases, the software automatically creates an account for anyone who opens the database file. All accounts have full administrative permissions in the database, and the accounts cannot be deactivated or deleted. In addition, any user can create new accounts for users who have not yet accessed the database. This is useful when you need to send action notification e-mails to those users from within the database.
  - To enable the security settings for this type of database, click the **Apply Login Security** button that will be displayed in the lower left corner of the window.
- In secure databases, you will need to create a user account for everyone who needs access to the database. Use the **Add**, **Edit** or **Delete** buttons below the table to manage the user accounts. When you create or edit a user account, the User Login and Contact Information window will appear, allowing you to enter/edit contact details, add the user to security groups and activate/deactivate the account. *See Section 2.4.5 on page 61*.

To export the data currently displayed in the table to an Excel spreadsheet, click the **Send to Excel** button.

You can also import user accounts from Microsoft Active Directory® by clicking the **Active Directory** button. This opens the Import Users from Active Directory window, which allows you to search for user accounts that have been defined in the Active Directory and choose one or more to import as new user accounts in the database. *See Section 2.4.6 on page 63*.

2.4.4.1.1 User Account Requirements

In secure databases, the user account must meet the following requirements in order to have access to the database:

- The user account must be in active status. An inactive user account cannot be used to access the database unless an authorized user makes the account active again. To display only the user accounts that are currently active, select the **Show active users only** check box, which is at the lower left side of the window. If the check box is cleared, the table will include both active and inactive user accounts. Note that if the account has been used to log in to the database at least once, deleting the account will make it inactive and the same username cannot be used

again in the database. This is because the username may be associated with existing analysis information. Alternatively, if the user account has never been used, you can delete the record from the database and you will be able to create another user account with the same username in the future if desired.

- The user account must be assigned to at least one security group. (The next section describes how to use security groups.)
- 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. If you create a user account that is not recognized by SQL Server, the user will not be able to log in to the database. *See Section 2.4.7 on page 64*.
- Synthesis applications use Windows authentication to identify the user; therefore, the user must be logged in to a computer that uses the same domain/username that is defined in the user account. If the user needs to connect to a database from a different domain, you can set up an alternative login that will allow the user to access the database without Windows authentication. *See Section 2.4.5.3 on page 62*.

2.4.4.2 Creating Security Groups

Throughout this user's guide, we use the phrase *authorized users* to indicate functionality that may be restricted based on specific security permissions. Security permissions allow you to control how users access and use the database. In Synthesis repositories, a set of permissions is known as a **security group**. Note that:

- In non-secure databases, all users with access to the *.rsr9 file have full permissions to the database; therefore, the concept of security groups does not apply to this type of database.
- In secure databases, each user may have different permissions based on the security group(s) that have been assigned to the user account.

In the Manage Repository Security window, the Security Groups tab displays a list of all the security groups that have been created in the database. The user who created the database automatically belongs to the Admin group, and has full user and administrative permissions. The Admin group cannot be deleted or have its permissions modified; therefore, its window cannot be opened from the Security Groups tab. To assign or remove a user account from this group, open the User Login and Contact Information window and select or clear the "Admin" check box. *See Section 2.4.5 on page 61*.

By default, the software also includes three additional predefined security groups. Depending on your organization's specific needs, you may choose to make adjustments to the default security groups or replace them with new groups *See Section 2.4.3 on page 53*.

Use the **Add**, **Edit** or **Delete** buttons below the table to manage the security groups. When you add or edit a security group, you can assign the permissions for the security group as well as select the database users that belong to each group. In addition:

- When you do not assign a user account to a security group, the account will have no access to the database; however, users with access to the database can send action notification e-mails to that account from within the database. *See page 110 in Chapter 3*.
- You can assign a user account to more than one security group.
  - For repository-level permissions (e.g., the ability to manage users and logins), if the permission is granted in any of the security groups assigned to the user's account, he/she will have that permission throughout the database.

- For project-level permissions (e.g., the ability to create/edit project items), it depends on the type of security assigned to the project.
  - If the project has repository-level security, the user will have *all* of the project-level permissions that have been granted from *any* of the security groups that are assigned to the user's account.
  - If the project has project-level security, the user will have only the project-level permissions that have been granted via the security group(s) that are assigned to the project and also assigned to that user's account.

> **Note:** To avoid unintended consequences, if you choose to implement different levels of access for different projects, it is generally recommended to assign project-level security for all projects in the database.

2.4.4.3 Table of Database Permissions

The following table provides a summary of the permissions that can be granted in the database. If the permission is application-specific, the affected application(s) are identified in the name.

You can edit the permissions of a security group even when a user associated with that security group is currently logged in to the database; however, the changes will not take effect until the user closes the database and reconnects to it.

> **Note:** Project owners *(see Section 2.4.2 on page 52)* have certain permissions that are equivalent to having the "Manage all projects" permission, but only for the projects that they own. For example, project owners can view private projects and lock/unlock projects, but only if they own those particular projects.

Basic permissions throughout repository	
Create and own private projects	Allows you to create projects that, in a secure database, are accessible only to you and to users with the "Manage all projects" permission. In a non-secure database, the project will be designated as "private" but it will still be accessible to all database users. Note that if you remove this permission from a security group, the users in that group who own private projects in the database will no longer have "owner" permissions for the existing projects. You can assign a new owner to each affected project via the Manage Projects window.
Create and own public projects	Allows you to create projects that may be accessible to any database user (depending on the project security setting). Note that if you remove this permission from a security group, the users in that group who own public projects in the database will no longer have "owner" permissions for the existing projects. You can assign a new owner to each affected project via the Manage Projects window. *See Section 2.3.3 on page 48*.
Create portal messages	Allows you to create and send messages to other users via the Messages page in My Portal.

Basic permissions at project level	
Read	Allows you to view but not edit any of the analyses in a given project. You can perform tasks that do not modify the data (e.g., calculate metrics in a Quick Calculation Pad, export data, etc.).
Create/edit project items	Allows you to create and edit items in a given project, such as folios in Weibull++/ALTA and RGA, diagrams in BlockSim, system hierarchy items in Xfmea/RCM++/RBI, and the like.
Create/edit/delete own resources	Allows you to use the Resource Manager to manage the resources (e.g., models, URDs, etc.) that *you* have created in a given project. This permission will be assigned automatically when the "Create/edit project items" permission is assigned.
Advanced permissions at project level	
Delete project items	Allows you to delete any item in a given project (e.g., folios in Weibull++/ ALTA and RGA, diagrams in BlockSim, system hierarchy items in Xfmea/ RCM++/RBI, and the like). This permission cannot be assigned unless the "Create/edit project items" permission is also assigned.
Create/edit/delete all resources	Allows you to use the Resource Manager to manage all resources available in a given project.
Set project security	Allows you to control user access within a given project. This includes the ability to access the Security page in the Project Properties window and to set Item Permissions *(see Section 2.4.10 on page 69)* for any item in the project.
Edit project properties	Allows you to use the Project Properties window to edit the project name, description, category and other settings of a given project.
Lock or check out project	Allows you to: ● Lock and unlock a given project. *See Section 2.4.8 on page 67*. ● Check in and check out a given project. *See Section 2.4.9 on page 67*.
Create restore points	Allows you to 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 2.5.2 on page 75*.
Delete project	Allows you to delete a given project.
Manage change log s in Xfmea/RCM++	Allows you to 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++	Allows you to 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	
Manage all projects	Grants you all of the basic and advanced project-level permissions for all public and private projects in the database. It also allows you to use the Manage Projects window to view and edit settings of all projects in the database.
Manage users and logins	Allows you to: ● Use the Manage Repository Security window to add user accounts to the database and define security groups. ● Use the Prior Logins window to view and export a history of database logins. *See Section 2.4.12 on page 72*. ● Use the Reset "In Use" Flags window to clear the "in use" status for selected database users. *See Section 2.4.13 on page 72*.
Manage e-mail notifications	Allows you to: ● Use the E-mail Notifications window to enable e-mail notifications sent by the software and control when they are sent. ● Use the Notification Groups window to add and manage the distribution groups for the e-mail notifications. *For more information about e-mail settings and notification groups, see Section 2.2.3.1 on page 37*.
Manage profiles and templates in Xfmea/ RCM++	This permission is available only in enterprise databases. It allows you to: ● 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	Allows you to: ● Use the Manage Units window to define the time units available for use in any project within the database. ● 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. *See page 352 in Chapter 11*. ● Use the Define Default Names window to specify how default names for resources and blocks are created. ● 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 Define Task Type Abbreviations window in MPC to change the abbreviations used for tasks. ● 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.

	• Use the Project Categories window to define the categories that can be used for grouping and filtering the projects in the database. • 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. *Most of the repository settings are discussed in Section 2.2.3. The restore points are discussed in Section 2.5.2*
Approve actions	Allows you to review and approve actions, which are Synthesis resources that allow you to track progress made in a project.
Manage all portal messages	Allows you to edit or delete all messages you sent and messages addressed to you via the Messages page in My Portal.
Create/edit/delete global resources	Allows you to use the Resource Manager to manage all global resources available in the database.
Manage Lambda Predict repository settings	Allows you to: • 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	Allows you to create and save layouts for use in the Dashboard Viewer in Weibull++, BlockSim and RGA. *See Section 2.7.4 on page 96*.

2.4.5 User Login and Contact Information

The User Login and Contact Information window allows you to either display information about a user account, create a new user account or create an alternative login for a user account. The function varies depending on where the window is opened from. The following sections provide a description of all three functions.

2.4.5.1 Display User Information

If you have an existing account in the database, you can display the information for your own user account by choosing **My Portal > Users > My Profile**.



You can edit the contact details and personal user image in your own user account; however, only users with the "Manage users and logins" permission *(see Section 2.4.4.3 on page 58)* can edit the security group that has been assigned to your account.

IMPORTANT: The domain and username fields provide the authentication that allows you to access the database. For existing user accounts in Oracle and SQL Server databases, these two fields cannot be modified because the information is associated with the login that allows the database to recognize the user.

In standard databases with login security, you have the option to edit the username, but you must have a user with the "Manage users and logins" permission inactivate the old username that you used to access the database and create a new account with the desired username; otherwise, simply changing your username may cause you to be inadvertently locked out of the database.

Click the **Active Directory** button if you wish to update your contact details based on information stored in the Microsoft Active Directory® for the current domain and username.

2.4.5.2 Create and Edit User Accounts

If you have the "Manage users and logins" permission, you can create new accounts, as well as display or edit the information for any existing user account in the database. To do this, click the **Add** or **Edit** button in the Manage Repository Security window. *See Section 2.4.4 on page 56*.

There are two ways to create a new account:

- Directly enter the username, contact information and security group(s) for the new user account.
- Use the **Active Directory** button to import user information stored in the Microsoft Active Directory® for the current domain.

For SQL Server databases, the **Create SQL Server** login check box will be displayed. Select the check box if you want to create an individual SQL Server login when you create each new Synthesis user account. The SQL Server login will be created if a login does not already exist for the username and if you have the appropriate authority in SQL Server to create logins. If you clear this check box, the user can still be recognized by SQL Server if a) an individual SQL Server login was created in advance in SQL Server, b) the user belongs to an Active Directory group that has a SQL Server login shared by all members of the group or c) the user connects to the database with an enterprise connection file (*.rserp) that impersonates another Windows user account that has a SQL Server login. *See Section 2.4.7 on page 64*.

To allow the user account to log in to the current database, select the **Active** check box at the bottom of the window. You can temporarily or permanently prevent the account from accessing the database by clearing the check box.

2.4.5.3 Create Alternative Login

A secure database uses Windows authentication to identify the user; therefore, the user must be logged in to a computer that uses the same domain/username that is defined in that user's account. If the user needs to connect to the database from a different domain, you can set up an alternative login that will allow the user to access the database without domain authentication. This is not necessary for non-secure databases.

If you have the "Manage users and logins" permission, you can set up an alternative login for an account by selecting the account in the Manage Repository Security window *(see Section 2.4.4 on page 56)* and then clicking the **Edit** button. This opens a User Login and Contact Information window that has two tabs: a User Info tab and an Alternative Login tab.

In the Alternative Login tab, select the check box to enable the use of an alternative login for the user account. When the setting is enabled, you can create a username and password combination that will allow the user to access the database. The username and password combination must be unique, and the password is CaSe SeNsiTiVe. You can temporarily or permanently disable the use of an alternative login for the account by clearing the check box.

The alternative login must be set up in advance. If you know, for example, that the user will need to make a copy of a standard database and open it on a computer that is not on your company's network or that the user will need to connect to an enterprise database from a computer that is not on your company's network, you must set up the alternative login and provide the user with the login information beforehand.

Saving or Clearing Alternative Logins

When a user connects to a database using an alternative login, the database automatically remembers that user's login on the computer. To clear a saved login, the user must click the **Forget Alternative Login** button in the Synthesis Setup window (**File > Synthesis Setup**). The database will prompt for the alternative login the next time the user attempts to connect to the database on that computer.

2.4.6 Import Users from Active Directory

When you click the **Active Directory** button in the Manage Repository Security window, the Import Users from Active Directory window will be displayed. This window allows you to search for user contact information from Microsoft Active Directory® and then use the selected records to create new user accounts in the database.

- Use the **Search Options** on the right side of the window to select which user information will be displayed in the table on the left side of the window. Click **Search** to update the table based on the criteria that you have specified. While the query is in progress, the Search button changes to a Cancel button. Depending on the number of records in the Active Directory, a full search may take some time. If you need to end the search before it is complete, click **Cancel**. The utility will return the results of the search up to the point of cancellation.

  The table will display records that:

  - Match your selections on the right side of the window.
  - Have the minimum information required to create a user account in a Synthesis repository (i.e., username, first name, last name and e-mail address).
  - Do not already have a user account in the current database.

  Use the check boxes to choose the rows that will be used to create new user accounts. If you select or clear the check box in the table header, it will select or clear the check boxes for all rows in the table.

  Tip: You can click the table header to sort the data in ascending or descending order.

- Use the **Authentication domain** field to identify the authentication source so that users can access the database using their domain usernames and passwords.
- Use the **Import as Members of** option to set the security group(s) that will be assigned to all the selected accounts. Security groups control the permissions of each user in the database.

For SQL Server databases, the **Create SQL Server login** check box will be displayed. Select the check box if you want to create an individual SQL Server login when you create each new Synthesis user account. The SQL Server login will be created if a login does not already exist for the username and if you have the appropriate authority in SQL Server to create logins. If you clear this check box, the user can still be recognized by SQL Server if a) an individual SQL Server login was created in advance in SQL Server, b) the user belongs to an Active Directory group that has a SQL Server login shared by all members of the group or c) the user connects to the database with an

enterprise connection file (*.rserp) that impersonates another Windows user account that has a SQL Server login. *See Section 2.4.7 on page 64*.

Click **Import** to create a user account for each row that is currently selected. You will receive a message to acknowledge that the accounts have been created but the window will remain open to allow you to continue importing data, if desired.

2.4.7 SQL Server Logins or Using 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 application database. 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. *See Section 2.4.7.1 on page 64*.
- **Group Login**: The user belongs to an Active Directory group that has a SQL Server login shared by all members of the group. *See Section 2.4.7.2 on page 65*.
- **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. *See Section 2.4.7.3 on page 66*.

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). This document provides an overview of all three options.

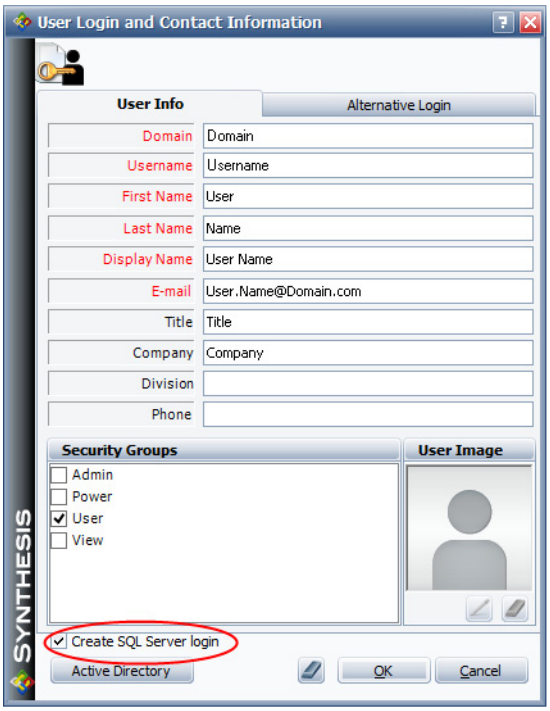
2.4.7.1 Creating Individual SQL Server Logins

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.

2.4.7.2 Using a Login Group

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.

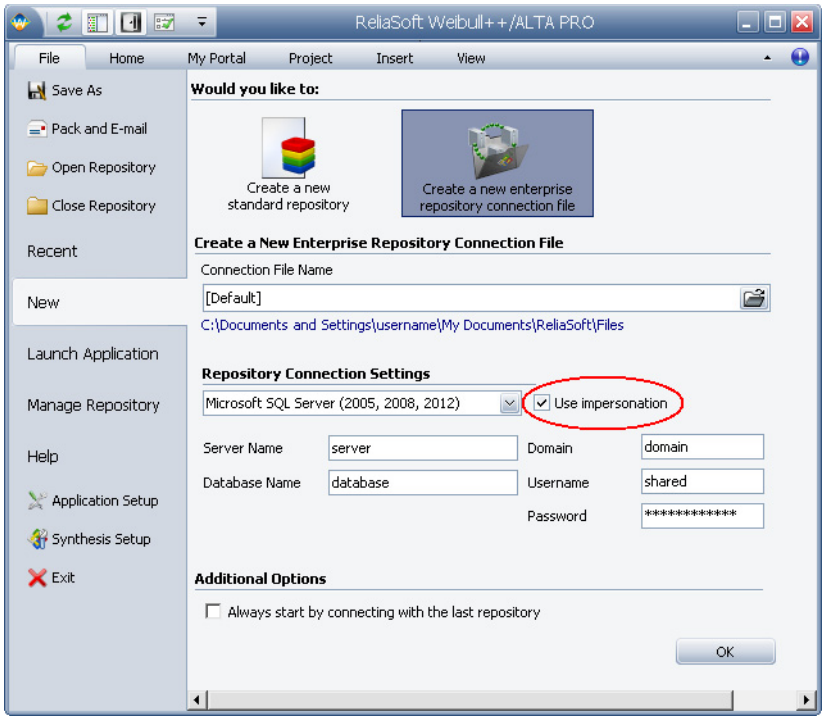
For example, base installations of Microsoft SQL Server Express 2005 and 2008 include the "Builtin\Users" Active Directory group as a SQL Server login by default. This means all users with a Windows account for that domain will be able to log in to the enterprise database with no need to create individual 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).

2.4.7.3 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 desktop 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 repository 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.

If the user is going to connect to the database with a connection file that uses Windows impersonation, you can clear the **Create SQL Server** login check box that is displayed when you are adding or importing the account.

2.4.8 Lock and Unlock Projects

The Lock and Unlock Project commands are available in all types of Synthesis repositories, and they provide an additional layer of control on how users access projects. Locked projects are displayed under the **Locked** heading of the project list, while unlocked projects are displayed under the **Public** heading.

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. In a non-secure database, any user can lock or unlock a project, and the commands may be useful only when preparing to convert the non-secure database into a secure one.

2.4.8.1 Locking a Project

If you want to temporarily (or permanently) prevent all users from editing a public project, you can lock the project by selecting the project in the project list and choosing **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.

2.4.8.2 Unlocking a Project

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



Tip: If you have the "Manage all projects" permission, you can select multiple projects and lock/unlock them all at the same time via the Manage Projects window. *See Section 2.3.3 on page 48*.

2.4.9 Check In or Check Out Projects

Synthesis applications allow multiple users to simultaneously access the database and cooperatively work on an analysis. However, on some occasions, you may need to exclusively work on one of the projects for an extended period of time, or work on the project on a computer that is not connected to the database network. In these cases, it is recommended that you use the Check In/Out feature.

When you *check out* a project, only you can modify the project while all other users will have read-only access to a copy of the project that shows the state of the project at the time it was checked out. The project cannot be edited by any other user unless you *check in* the project or *undo the check out*.

2.4.9.1 Check Out a Project

In a non-secure database, any user can check 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 the project in the project list then choose **Project > Management > Check In/Out > Check Out**.



A copy of the project will be displayed in the project list under the **Checked Out** heading. All users will have read-only access to that copy, which shows the state of the project at the time it was checked out. The purpose of this copy is to make other users aware that the project is checked out and give them limited access to the project information (e.g., to query, copy data, etc.) while you are editing the project "offline." At the same time, an editable copy of the project will be saved in a "Checked Out" folder in your local drive.

Tip: You can edit the save location of the "Checked Out" folder via the Synthesis Setup window. 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.

To identify the user who currently has the project checked out, open the Project Properties window by choosing **Project > Management > Edit Project Properties**. The lower right side of the window will display a "Locked by" status, with the name of the user who has the project checked out.

2.4.9.2 Working with Checked Out Projects

A checked out project is saved in the designated "Checked Out" folder as a standard database file (*.rsr9) that has the same name as the project. To work on a file, open it by choosing **File > Recent** and then selecting the *.rsr9 file on the list. You can also choose **File > Open Repository** and browse for the *.rsr9 file.

If you want to open the file on a different computer, make a copy of the file and save it on the computer that you will be working on. When you are ready to check in the file from the original computer, overwrite the file in the "Checked Out" folder with the more recent version of the file. This will ensure that any changes you made to the project will be reflected in the database when you check in the project.

2.4.9.3 Check In a Project

To check in a project, select the project 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. *See Section 2.5.2 on page 75*.

If you rename, move or delete the file in the "Checked Out" folder, you will no longer be able to check in the project. You can, however, undo the check out.

2.4.9.4 Undo Check Out

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



When you undo a check out, all changes made to the project will be discarded, and the copy of the project before it was checked out will be restored. In a non-secure database, any user can undo a 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.

2.4.10 Item Permissions

For public projects in a secure database, any user who has read/write permissions for the project will have read/write permissions for all of the items within the project by default. However, if you desire, each item in the project can have its own security settings. To set the permissions for an item, you must a) have the "Set project security" permission for the project, b) be the project owner or c) have the "Manage all projects" permission.

Setting Item Permissions from the Project Explorer

In Weibull++, ALTA, BlockSim, RENO, DOE++, RGA or Lambda Predict, select the item in the current project explorer (i.e., folio, diagram, workbook, etc.) and choose **Project > Current Item > Item Permissions**.



In the Item Permissions window for items in the current project explorer, there are two ways the permissions can be set:

- It can have its own permissions defined at the item level. To do this, select **Restrict editing to selected project users** and then use the **Select Project Users** area to select which specific users will be able to edit this particular item. Other project users will have read-only access.
- It can inherit permissions from the project. If you select **Inherit from project**, any user who has edit permissions for the project will have edit permission for this item.

Setting Item Permissions from a Hierarchy

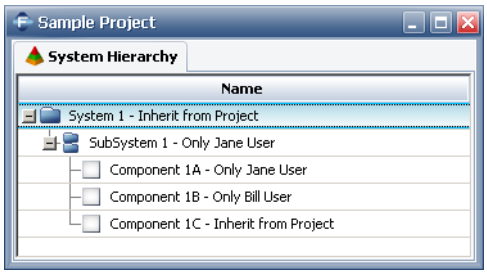
In Xfmea, RCM++ or RBI, select an item and choose **System Hierarchy > Security > Item Permissions**; in MPC, select an item and choose **[Systems/Structures/Zones] > Security > Item Permissions**.



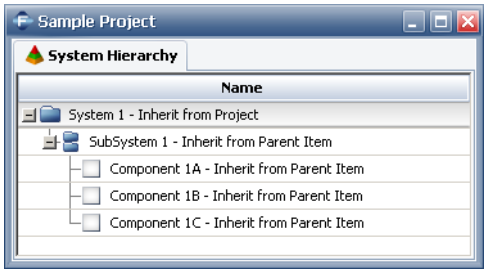
In the Item Permissions window for items in a system hierarchy, there are two ways the permissions can be set:

- It can have its own permissions defined at the item level. To do this, select **Restrict editing to selected project users** and then use the **Select Users** area to select which specific users will be able to edit this particular item. Other project users will have read-only access.
- It can inherit permissions from the project or the parent item.
  - If an item is set to **Inherit from project**, any user who has edit permissions for the project will have edit permission for this item.
  - If an item is set to **Inherit from parent item**, the item will have the same permissions as the next item above in the hierarchy. This option is not available for top-level items.

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.



2.4.11 Simultaneous Access By Multiple Users

For any type of database (standard or enterprise), multiple users can log in to the same database simultaneously. The following is a general summary of the actions that are limited when multiple users are logged in to the same database or project. These rules apply to all Synthesis applications.

- A standard database cannot be deleted if another user is currently logged in to that database. Before deleting a database, check the Users page of the My Portal window to see if you are the only user logged in.
- A project cannot be deleted nor have its properties and public/private status edited when another user has the same project open. The same rule applies when the project is open in more than one Synthesis application (e.g., the project is open in both Weibull++ and BlockSim), even if it is the same user who has the project open.
- Projects contain analysis folios, plots, reports and other items that apply to the software you are using. Multiple users can view the same item simultaneously; however, note that:
  - In projects that use a current project explorer to manage analysis data, an item is automatically flagged as "In Use" to the first user who *opens* the item. It will not be available for editing until the first user closes the item. Items that are in use cannot be deleted.
  - In projects that use hierarchical trees to manage analysis data (as seen in the Xfmea, RCM++, RBI and MPC applications), an item is automatically flagged as "In Use" to the first user who *selects* the item. Its sub-items, however, will remain available for editing by other users. The selected item and all the analyses related to that particular item will not be available for editing until the first user selects a different item or closes the project. Furthermore, items cannot be deleted when another user has the project open.

  To check the status of a project or its items, see Section 2.4.11.1 on page 71.

- Items may use project resources (e.g., models, tasks, actions, etc.). When you edit a resource, the change will be reflected in all items that use that particular resource. For example, you could use a particular model to define the reliability for blocks in several different BlockSim diagrams. When you edit the model, the change will be reflected automatically in all blocks that rely upon that model. To check all the items that may be dependent on a particular resource, use the Dependency Viewer.

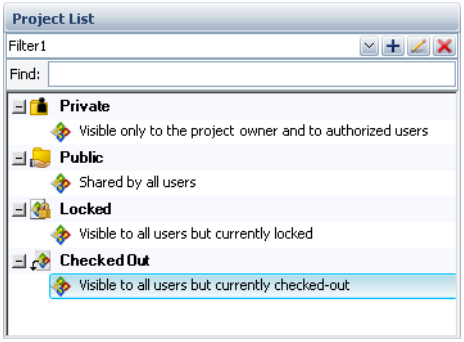
Secure databases have additional rules:

- You can edit the permissions of a security group even when a user associated with that security group is currently logged in to the database; however, the changes will not take effect until the user closes the database and reconnects to it.
- Projects and items cannot be locked or have their security permissions modified when the project or item is in use.

2.4.11.1 Status Indicators

When you encounter a project or item that is read-only (e.g., if the **OK** button is disabled in the properties window or if you are unable to type in the cells of a worksheet or table), you can check the status of the project or item to see if it is locked, checked-out, restricted to selected users or currently in use. There are two places in the software interface that display the status:

- In the project list, projects that are displayed under the **Locked** and **Checked Out** headings are read only. *See Section 2.4.8 on page 67 and Section 2.4.9 on page 67*.



- Within a project, items may be read-only if access to the project is restricted to selected users or if the item is currently in use by another user. The statuses are indicated by the following icons:

  Restricted Access. An item that is restricted to selected users will not be available for editing until you are granted the appropriate item permissions. *See Section 2.4.10 on page 69*.

  In-Use. An item that is in-use will not be available for editing until the other user closes the item or selects a different item. You can display the username of the account that is editing the item by pointing your mouse cursor over the icon whenever you see it.

  The location of these icons varies depending on the software 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. In addition, a read-only item will display its status in the

caption bar of its window, 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.

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 2.4.13 on page 72 for instructions on how to reset the flags throughout the database if this problem occurs.

2.4.12 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 in order 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.

2.4.13 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.

2.5 Managing and Restoring Data

2.5.1 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.

2.5.2 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.

2.5.2.1 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.

2.5.2.2 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.

2.5.2.3 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 repositories. 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. 77)
- Convert and import data from prior version files - Section 2.6.2 (p. 77)
- Import/export selected projects - Section 2.6.3 (p. 86)

When applicable, you can also:

- Import/export selected project items or Synthesis resources - Section 2.6.4 (p. 87)
- Import from an Excel spreadsheet or delimited text file into a data folio - Section 2.6.5 (p. 88)
- Use XML files to import/export system configuration information in BlockSim or Lambda Predict - Section 2.6.6 (p. 91)

These features can be accessed either from the Import/Export Wizard (**Project > Management > Import/Export**) or from the Backstage View (*see page 132 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. *For details, see Section 2.4.7 on page 64*.

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 81*.

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 79.
   - To convert from MPC 3 Databases, see Section 2.6.2.5 on page 83.

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 81.

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 86*.

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 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 81.

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. *For details, see Section 2.4.7 on page 64.*

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 2.4 on page 50 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**.

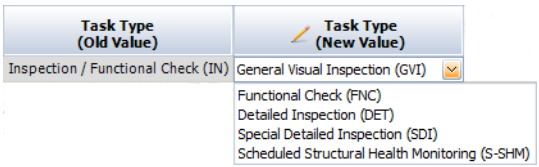
The following subsections describe the default conversion logic for Task Type, Interval and Zone.

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**.

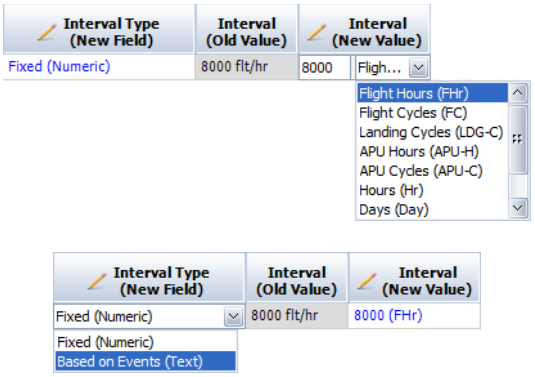
Interval Type and Interval

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:

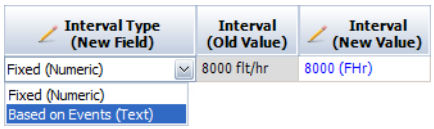
- Edit the text in the **Interval (New Value)** column so it will match either the name or the abbreviation of a predefined time unit. For example, if the Version 3 task record contains a misspelling for "2000 fliight hours," you can click inside the cell and remove the extra i.
- Click the **Time Units** 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."

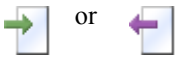
If you don't want the task to be considered for zonal analysis even though it meets the requirements, you can click inside the Zonal Candidate (New Value) column and choose No from the drop-down list. If this column displays No in italics, this indicates that at least one of these conditions is not met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

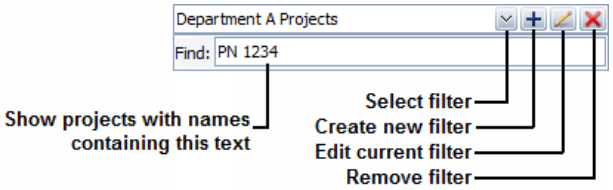
First, choose **Project > Management > Import/Export**.



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 101 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.

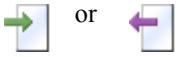
Tip: Project names must be unique within each database. If you attempt to import/export a project with a name that already exists in the destination database, the application will automatically increment the name. For example, if "Project1" already exists, the new project might be renamed to "Project1_1,"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

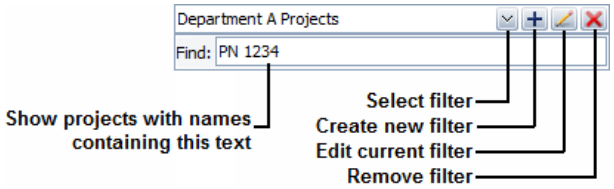
First, open the project that you want to import to or export from and choose **Project > Management > Import/ Export**.



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 101 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**.

 or 

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

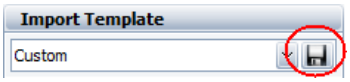
If you will be importing data from multiple files that have the same structure, you can use saved template files (*.waim in Weibull++/ALTA or *.rgaim in RGA) to automatically map the columns for all other similar files that you need to import from.

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

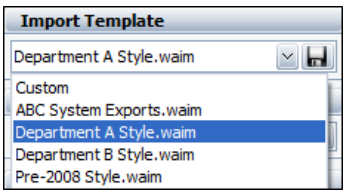
1. Open a data file and use the control panel buttons to map the columns.
2. Click the **Save** icon in the Import Template area on the control panel.



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.



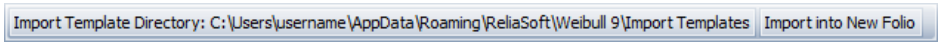
Tip: If you make adjustments after applying the template, you have the option to click **Save** in the **Import Template** area to either replace the existing template, or create a new template with a different name.

2.6.5.6 Importing the Data

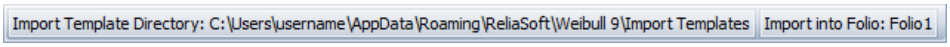
When you are ready to import the data, click the **Import** button.



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 2.2.3.6 on page 43*).

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. 92)
- 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. 96)
  - 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. 93)

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 2.2.3.6 on page 43*).

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 95.

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 95.

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 95for 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 101*.

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 hover over the tab to make the entire panel visible.) *See Section 3.3 on page 105*.

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 98*.

Tip: If desired, you can change the overall color scheme used in the MDIs for all Synthesis 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 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.

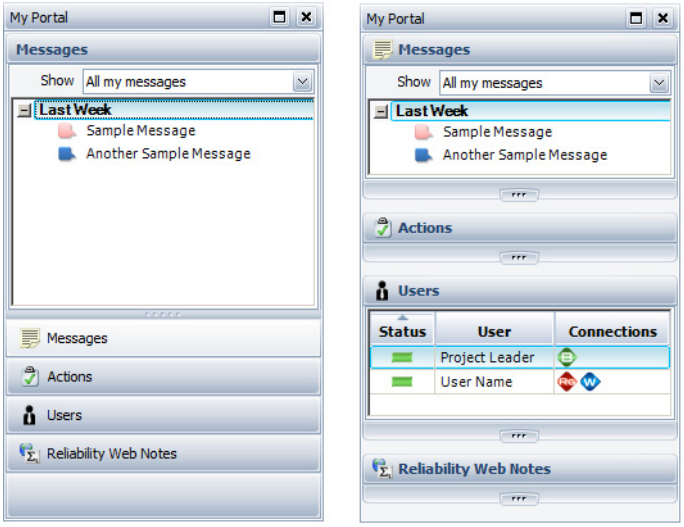


Tile or View One Page at a Time

In all Synthesis 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.



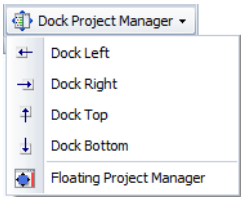
Dock or Float

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.

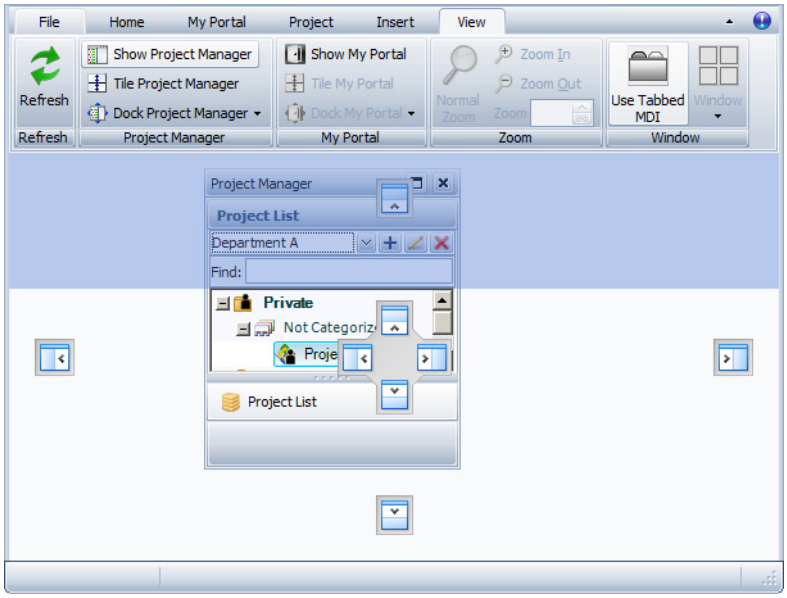


The same options are available when you choose **View > My Portal > Dock My Portal**.

- 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

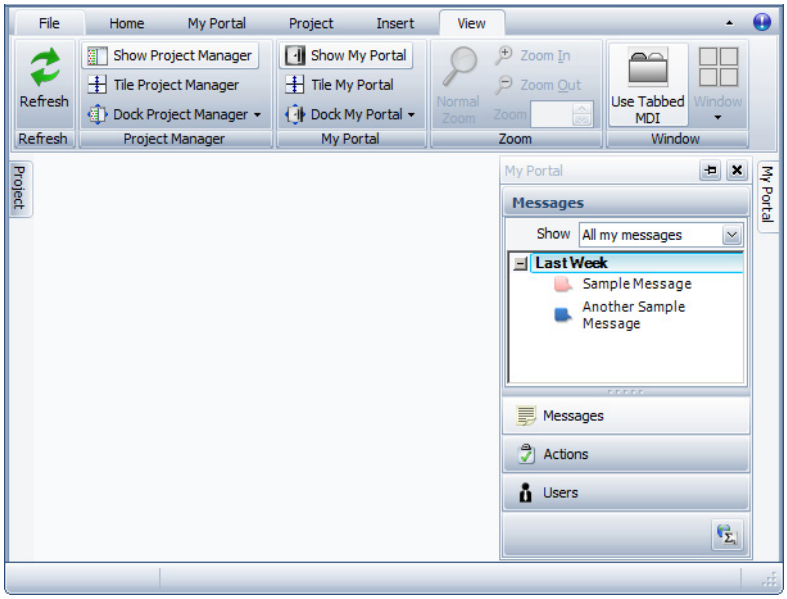
When docked, both panels can also be toggled between pinned and unpinned states by clicking the pushpin icon in the panel's title bar. When the pushpin in the icon is vertical, the panel is pinned and will be displayed at all times.

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 hovered 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 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 104*.

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

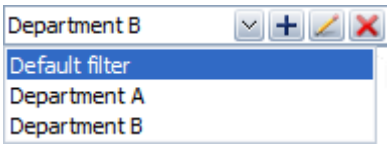
To open an existing project, double-click the project name or select the project and choose **Project > Management > Open Project**.



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.

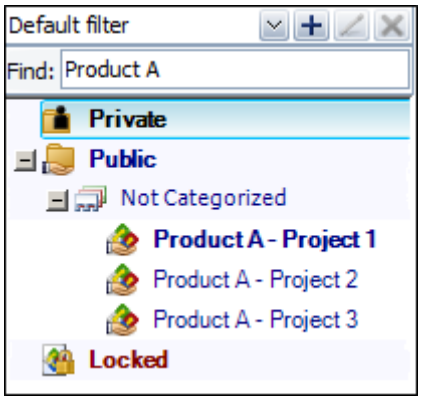
Click the **Create new filter** icon + to create a new filter or click the **Edit current filter** icon to edit the filter that is currently selected. Both commands open the Filter Properties window so you can view or modify the filter criteria.

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 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

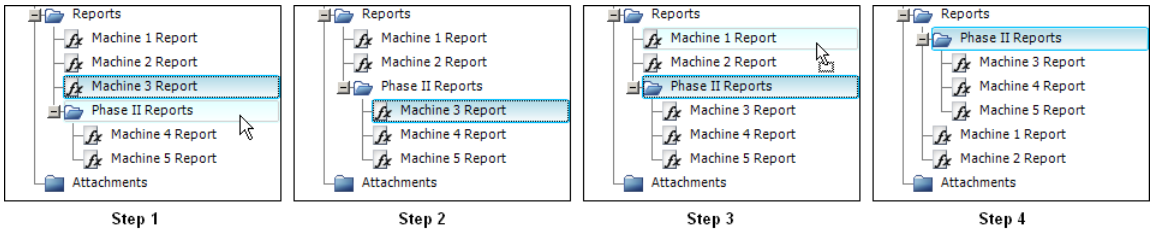
To manage existing items within the project (i.e., **Edit Item**, **Rename Item**, **Duplicate Item** and **Delete Item**), right-click the item and choose a command from the shortcut menu or select the item and use the commands in the Current Item group on the Project tab of the Ribbon.

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 98*.

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. (Refer to the "Actions" section in the "Resources" chapter for more information.) 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. 107)
- Action statuses - Section 3.3.2.2 (p. 108)
- Action properties - Section 3.3.2.3 (p. 108)
- Using the Approve, E-mail and Outlook commands - Section 3.3.2.4 (p. 110)

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.

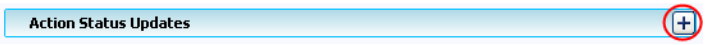
3.3.2.3 Action Properties

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.

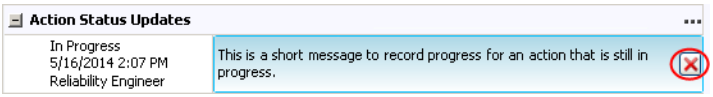


- In the **Action** area, note the following:
  - The **Action Description** is required.
  - The **Short Description** is a shortened version of the record description that will be used in locations where the full description would be too long to display (e.g., diagrams). If the field is blank when 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 3.3.3 on page 110*. 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

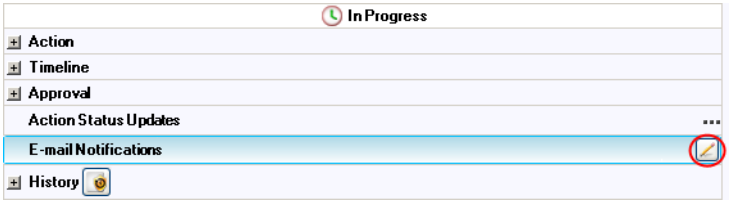
Three commands will be enabled only when applicable. For most applications, they will be available as buttons at the bottom of the window. If you are using Xfmea/RCM++/RBI and working with an action that's used in an FMEA or DFR plan, they will be available from the Ribbon at the top of the window.

- 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.

3.3.3 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.

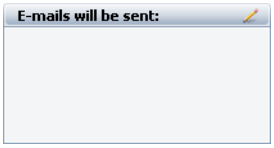


E-mails Will Be Sent

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.

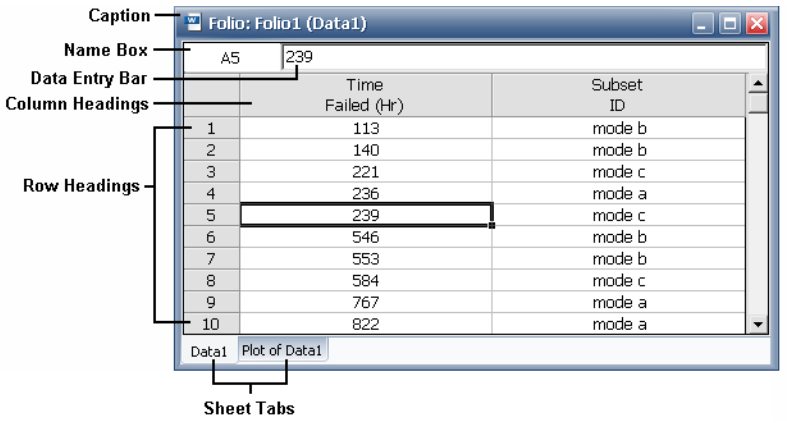
- The list of notification groups can be defined for the current database by user(s) with the "Manage e-mail notifications" permission.
- The list of other users contains all of the user accounts that are defined in the current database.

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:

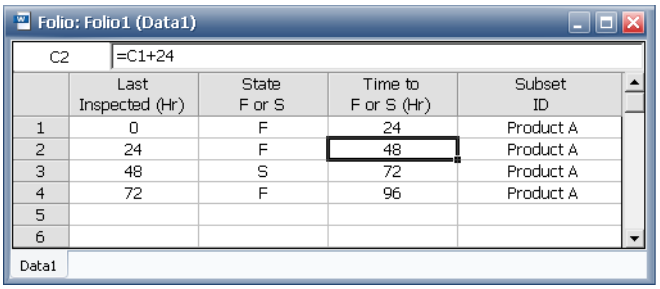
- To resize a single column, point the mouse pointer on the line that borders the column in the column header. The mouse pointer will change to a double-headed arrow. Click and then drag the double-headed arrow to the left or right to resize the column. To save the changes, choose **Home > Edit > Format 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, point the mouse pointer on the line that borders the row in the row header. The mouse 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

Except for Xfmea, RCM++, RBI, MPC and Lambda Predict (which use the Analysis Panel; see Section 3.9) 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. This section describes some features that are common to most control panels, including:

- Switching between pages in a control panel, and hiding/displaying the entire panel - Section 3.7.1 (p. 114)
- Using the Identifiers page to record information about the particular analysis - Section 3.7.2 (p. 115)
- Using the Publishing page to publish or update a model based on the current analysis - Section 3.7.3 (p. 116)
- 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. 117)
- 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. 118)

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

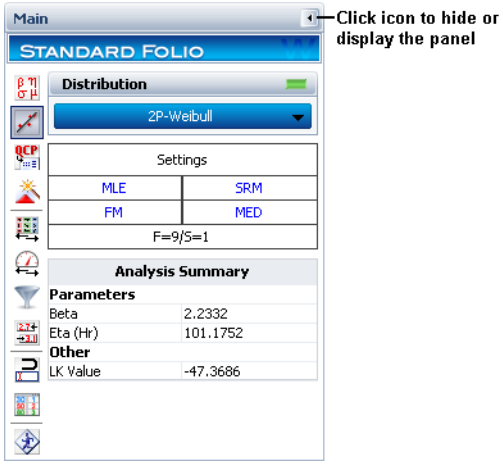
Many control panels and navigation panels (e.g., the navigation panel on the left side of the Resource Manager) contain multiple pages that vary depending on the type of analysis. You can switch between pages by clicking either the large buttons or small icons at the bottom of the control panel.

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.

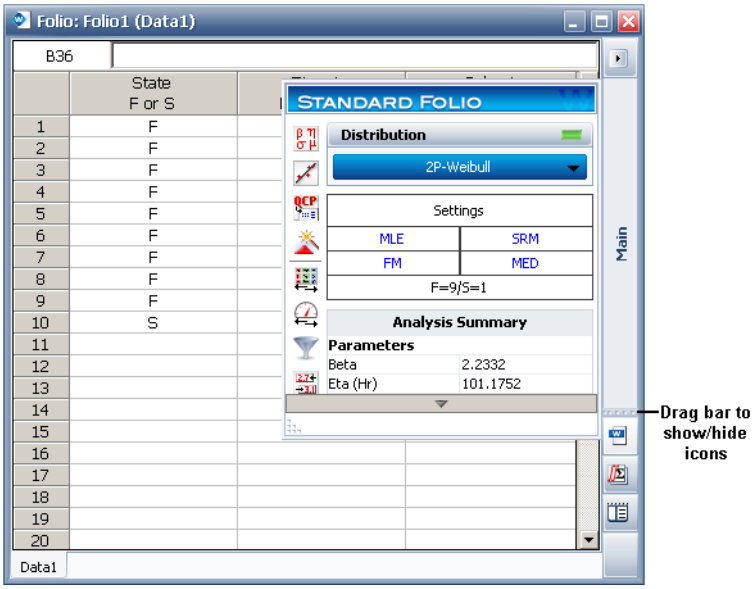


Hiding or Displaying a Control Panel

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.



3.7.2 Control Panel Identifiers Page

As described in the Control Panel Publishing Page section (see Section 3.7.3), 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.

3.7.3 Control Panel Publishing Page

For many types of data folios and diagrams in Synthesis, all tools associated with publishing a model resource are found on the Publishing page of the control panel. The exception is Lambda Predict where the tools are located on the Model tab of the Properties panel.

Note that BlockSim, DOE++ and Lambda Predict have additional options/settings and requirements for publishing models. For more information see:

- Publishing Fitted and Analytical Models in the BlockSim User's Guide.
- Publishing Models from R-DOE Results in the DOE++ User's Guide.
- Publishing from Failure Rate Predictions in the Lambda Predict User's Guide.

Background

As discussed in "Resources" chapter, one of the many ways in which the Synthesis Platform facilitates effective collaboration between different reliability activities is by making it possible to publish a shared model resource from one analysis that can then be used in other analyses where it is relevant. To give just one very simple example, you can use Weibull++ to analyze the available time-to-failure data for a particular component, publish the distribution/parameters as a shared model and then use the published model to define the reliability of a block in a BlockSim RBD. For a summary of which Synthesis applications can create and/or use model resources, see http://www.ReliaSoft.com/synthesis/integration.htm.

It is important to note that 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.

Tools for Working with Published Models

Publish to Model publishes a shared model resource based on the current analysis.

- 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 that 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.

For example, you might start by relying on engineering judgment to estimate the life distribution for a particular component and using a manually defined model to represent that component in other Synthesis analyses. When you later obtain actual life data for the component, you might want to replace this model with the more realistic estimate obtained from the data analysis in a Weibull++ standard folio. The Synthesis Platform makes it easy to achieve this substitution without having to update every other analysis where the original model was being used.

 **Trace Usage** opens the Dependency Viewer, 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.

Summary Details for Published Models

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 Published Model 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.
- **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."
- 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.

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 "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**.

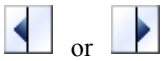


Resize a Panel

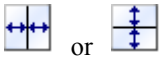
You can resize the panels by dragging and dropping the vertical (or horizontal) separator into the desired position.

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

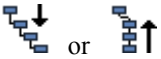
When you wish to return to the two-panel layout, choose the **Split Panels** command.


or

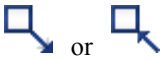
Expand or Collapse Nodes

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**.


or

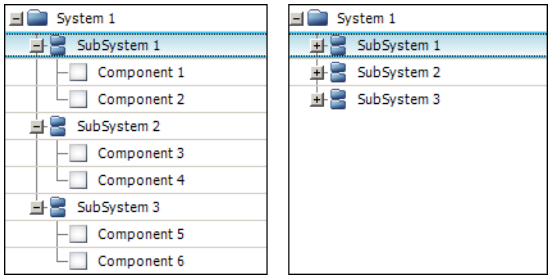
To expand or collapse a specific branch, you can click the + or - icons, or select the item and choose **Expand Node** or **Collapse Node**.



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.

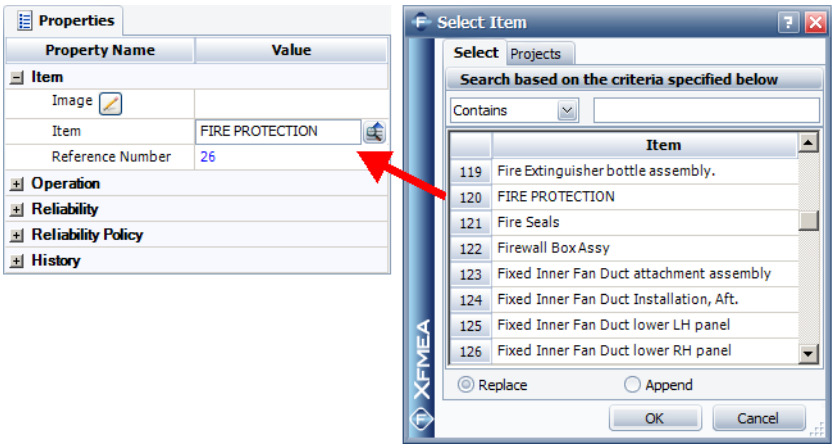
3.10 Select Existing Window

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.

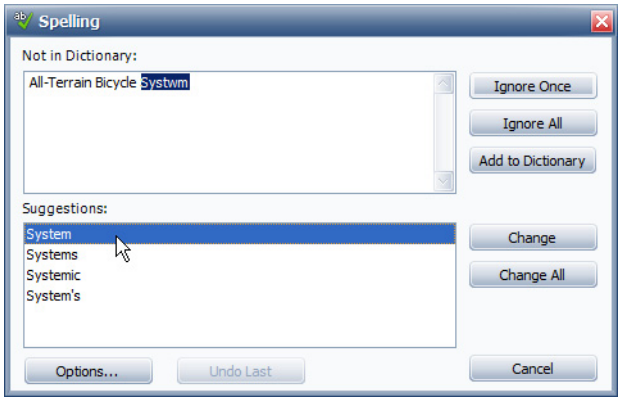
For both the Project and Phrase Set tabs, you can select either the **All** option, which displays the relevant descriptions from all projects/phrase sets in the current database, or the **Selected** option, which allows you to specify which projects/phrase sets to draw the descriptions from. When this 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. **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**

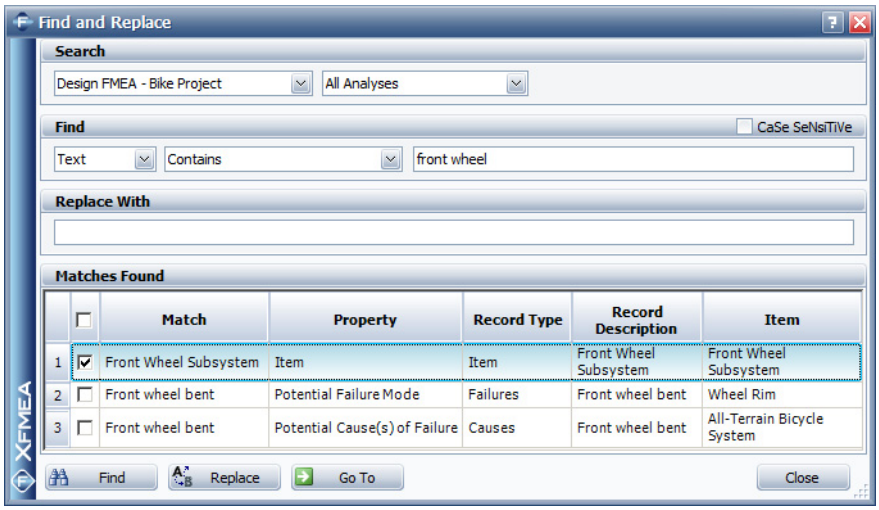
or **By Columns**; and the option to limit the search to within equations (**Formulas**) or within the results of an equation (**Values**).

- 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 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 Manager 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 3.13.1 on page 126*.
- The **Record History Log** displays changes that were performed for a specific folio, system hierarchy item, FMEA record, resource, etc. *See Section 3.13.2 on page 126*.

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 3.13.3 on page 126*.

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.

3.13.1 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.

3.13.2 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.

3.13.3 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 75 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.

3.14 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.

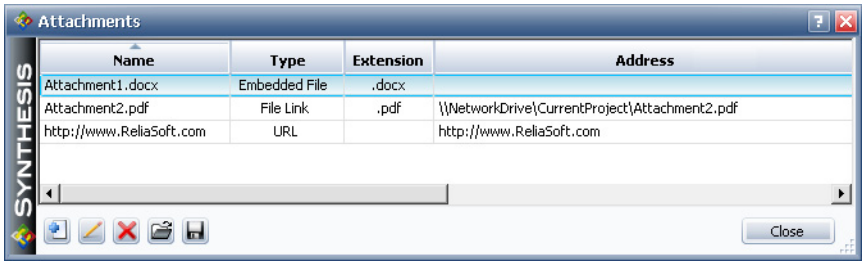
- **Linked** attachments may be URLs or files. For these attachments, the software stores the path to the URL or file. This allows you to open the resource in its original location (e.g., Internet, intranet or network directory), provided that the necessary software is installed on your computer and 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.

3.14.1 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.

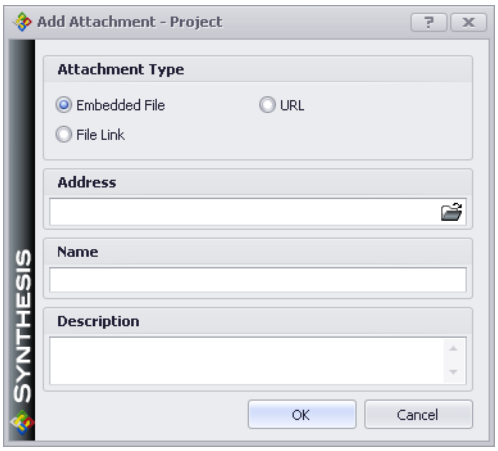
The icons at the bottom of the window allow you to **Add** a new attachment (), **Edit** the properties of the selected attachment (), **Delete** the selected attachment (), **Open** the selected attachment () or **Save** a copy of the selected attachment ().

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 73 in Chapter 2*) or restore point (*see page 75 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.

3.14.2 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.

3.14.3 Attachment Locations

As mentioned above, the locations where you can attach files will vary depending on which Synthesis application you are using.

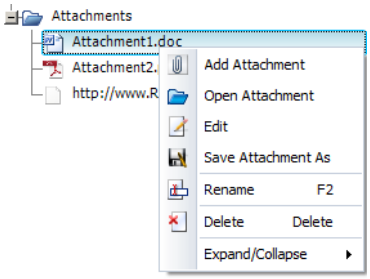
Tip: In most cases, the caption bar in the Attachments window will provide an indication of which type of attachment you are working with. For example, when you are working with project-level attachments, the caption bar will indicate "Project:" followed by the specific name of the project.

3.14.3.1 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**.

3.14.3.2 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.

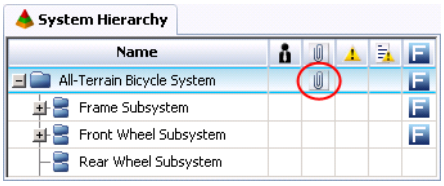


3.14.3.3 Hierarchy Attachments

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.



3.14.3.4 Block Attachments

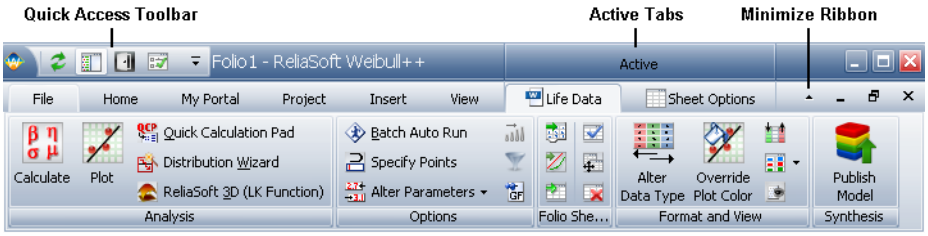
BlockSim supports attachments for individual blocks in a diagram. To access attachments for a block, you can click the **Attachments** icon in the Block Properties window.



Ribbon 4

The application now 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 chapter 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 131*.



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 above). 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 earlier versions of Weibull++ or ALTA, choose the appropriate option from the drop-down list in the Open Repository 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 56 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 72 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 72 in Chapter 2*.

- **Notifications**

  E-mail Settings controls whether e-mail notifications are enabled and when they are sent. *See page 37 in Chapter 2*.

  Notification Groups allows you to define distribution groups to receive e-mail notifications. *See page 38 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 39 in Chapter 2*.

  Manage Warranty Units allows you to set equivalencies between the time units used in the repository and the units used in the warranty calculations. *See page 352 in Chapter 11*.

  Define Default Names allows you to specify how default names for resources and blocks are created. *See page 40 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 41 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 42 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 75 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 73 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 43 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 96 in Chapter 2*.

- **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.

- 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. *See Section 4.2.1 on page 135*.



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. The settings apply only to the current computer/username, and they do not affect the results of the analyses. *See Section 30.2 on page 592*.



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. The settings apply only to the current computer/username, and they do not affect the results of the analyses. *See Section 30.1 on page 589*.

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. Note that an active Internet connection is required to see the information.



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.

4.2.1 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. *See Section 4.8 on page 147*.

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 the next section).
- **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.

4.2.2 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.

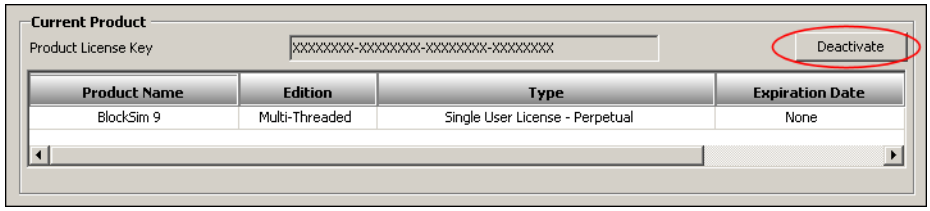
- If you need to download the latest information from the license server, click **Synchronize**.
- If you want to change your password or any of the current contact information, type the new information into the fields on this page and then click **Update Information on Server**.

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.

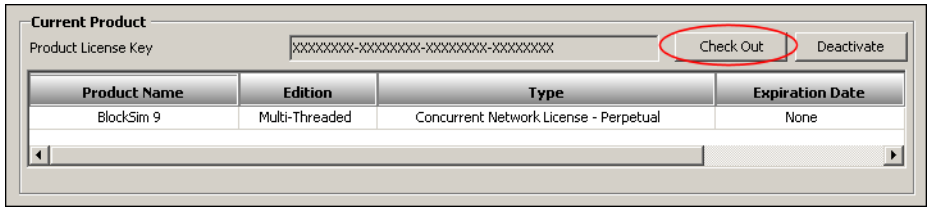


The application will shut down immediately after you deactivate, so you will be prompted to confirm that you're ready to continue. If you later try to access this application again for this computer/user, you will be prompted to repeat the activation process.

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 data sheets. It also contains statistical tools such as the Monte Carlo data simulation tool and the Quick Parameter Estimator.

Clipboard



Paste

- **Paste All** pastes the formats, formulas and values of the contents of the Clipboard to the current folio or spreadsheet.
- **Paste Values** pastes only the values from the Clipboard to the current folio or spreadsheet.
- **Paste Formats** pastes only the formats from the Clipboard to the current folio or spreadsheet.
- **Paste Formulas** pastes only the formulas from the Clipboard to the current folio or spreadsheet.

[1] For more information on license types, please visit http://www.ReliaSoft.com/license.htm.



Cut cuts the selected text to the Clipboard. Data stored in the Clipboard can be pasted into any data sheet within the application or in other applications.



Copy copies the selected text to the Clipboard. Data stored in the Clipboard can be pasted into any data sheet within the application or in other documents. 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.



Format Painter allows you to copy the format properties of text in a sheet and apply it to other text. To use the Format Painter, select the text with the format properties that are to be copied and then choose 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 to the sheet.



Delete deletes the selected text.



Clear

- **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, which contains the following commands.

- **Alignment a**llows 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).
- **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 Style** allows you to set the column width of the selected cells.
  - **Span Evenly** spans the columns across the spreadsheet with equal width.
  - **Span Proportionally** keeps the ratio of the column widths to span across the spreadsheet after they have been resized.
  - **Keep As Set By User** keeps the column width settings defined by the user.
- **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 cells in the currently active sheet.



Find opens the Find window, which allows you to search through the active sheet for text and/or numbers in values or formulas and, if desired, replace the found material. *See page 123 in Chapter 3*.



Spelling activates the Spell Check utility, which allows you to check the spelling within the current sheet. *See page 122 in Chapter 3*.

Print



Print sends the current data sheet or spreadsheet to the printer.



Print Preview opens the Print Preview window, which allows you to view how the current data sheet or spreadsheet will appear on the printed page. *See page 585 in Chapter 29*.



Page Setup opens the Page Setup window, which allows you to specify printing options. *See page 581 in Chapter 29*.



Print Layout displays the page boundary lines on the reliability block diagram (RBD). 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.

Tools



Weibull++ Monte Carlo opens the Weibull++ Monte Carlo utility, which allows you to generate a data set that fits a specified life distribution or user-defined model. The generated data set can then be analyzed like any other data set and used in a variety of applications such as in the analysis of probabilistic design models or to compare the data generated from simulation with actual field data. *See page 432 in Chapter 17*.



ALTA Monte Carlo opens the ALTA Monte Carlo utility, which allows you to generate a data set that fits a specified life distribution and life-stress relationship at specified stress conditions. The generated data set can then be analyzed like any other data set and used in a variety of applications such as in the analysis of probabilistic design models or to compare the data generated from simulation with actual field data. *See page 438 in Chapter 17*.



Non-Linear Equation Root Finder opens the Non-Linear Equation Root Finder utility, which allows you to solve for the root of any user-defined non-linear equation. *See Chapter 25*.



Quick Statistical Reference opens the Quick Statistical Reference utility, which allows you to calculate common statistical values such as median ranks, chi-squared values and student's t values. It also includes a

polynomial interpolation/extrapolation function, which allows you to obtain new data points from a set of known data points that you provide. *See Chapter 26*.



Quick Parameter Estimator (W++) opens the Weibull++ version of the QPE, which 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. *See page 194 in Chapter 5*.



Quick Parameter Estimator (ALTA) opens the ALTA version of the QPE, which 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. *See page 197 in Chapter 5*.

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 105 in Chapter 3*.

Messages



Create Message opens the Message window, which allows you to compose and send messages within the database. *See page 107 in Chapter 3*.



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. *See page 107 in Chapter 3*.



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. *See page 61 in Chapter 2*.

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 (*see page 101 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.

Note: For secure databases, access to the commands on this tab may be restricted based on the permissions assigned to the user account. *See page 58 in Chapter 2*.

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 45 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. *See page 49 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 48 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 76 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 Restore Points. *See page 75 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 67 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 (*see Section 2.5.1 on page 73*) or restore point (*see Section 2.5.2 on page 75*).



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 125 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 67 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 51 in Chapter 2*.



Change Owner allows you to assign a different database user to be the owner for the selected project. *See page 52 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 53 in Chapter 2*.

Current Item

The following commands are available when you select a specific item (a folio, 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., Prediction1_1, Prediction1_2, etc.).



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 and 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 173 in Chapter 5*.



XFRACAS to RDW opens the XFRACAS to RDW interface, which allows you to specify the components of interest in an XFRACAS entity and pull the data from XFRACAS into the Reliability Data Warehouse (RDW). *See page 92 in Chapter 2*.



RDW to Folio opens the RDW to Folio window, which allows you to specify which data you want to extract and how the values in the data should be entered in the resulting standard folio. *See page 93 in Chapter 2*.

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.

Folios



Weibull++ Standard Folio adds a Weibull++ standard folio to the project for life data analysis. The folio allows you to capture life data (e.g., failure times) from a sample of units and analyze the data based on an underlying life distribution. From the analysis, you can measure the reliability performance of a product over time or make projections about future behavior. *See Chapter 6*.



ALTA Standard Folio adds an ALTA standard folio to the project for accelerated life testing data analysis. The folio allows you to capture life data (e.g., failure times) from a sample of units that are subjected to elevated stress conditions (the elevated stress conditions cause the units to fail more quickly, reducing the test time required to obtain data for a product). You can then analyze the data based on a life-stress relationship model in order to extrapolate a use level pdf, which allows you to obtain reliability predictions about the product at normal operating conditions. *See Chapter 7*.



Warranty adds a warranty data analysis folio to the project. The folio allows you to capture and analyze failure data from fielded products in order to predict future warranty returns (product failures) and plan for warranty fulfillment needs, such as the number of spares to stock. *See Chapter 11*.



Weibull++ Degradation adds a Weibull++ degradation analysis folio to the project. The folio allows you to record the amount or level of degradation of a product over time in order to estimate the point at which the product is expected to fail. The folio uses basic mathematical models to extrapolate failure times from the measurements. The software then analyzes the extrapolated failure times using standard life data analysis in order to obtain reliability predictions about the product. *See Chapter 8*.



ALTA Degradation adds an ALTA degradation analysis folio to the project for accelerated degradation analysis. The folio allows you to record the amount or level of degradation over time for a product that is subjected to elevated stress conditions in order to estimate the point at which the product is expected to fail (the elevated stress conditions accelerate the degradation of the product, reducing the test time required to obtain the measurements). The folio uses basic mathematical models to extrapolate failure times from the measurements. The software then analyzes the extrapolated failure times using accelerated life testing data analysis in order to obtain reliability predictions about the product at normal operating conditions. *See Chapter 9*.



Non-Parametric LDA adds a non-parametric life data analysis (LDA) folio to the project. The folio allows you to capture the life data from a sample of units and analyze the data without assuming an underlying life distribution. This analysis is useful when dealing with unknown failure modes, when there is not enough data to assume a life distribution or when the data set does not fit any life distribution in a satisfactory way. The folio also allows you to perform a standard (parametric) life data analysis on the data set in order to compare the results obtained from both methods and perform further analysis. *See Chapter 10*.



Event Log adds an event log folio to the project. The folio allows you to capture data in an event log format (commonly used in the Machine Tools and other industries) and analyze the information in order to create a reliability model for a piece of equipment. The reliability model can be used to help you understand the performance of the equipment, such as the total system uptime/downtime, or obtain reliability predictions about its future behavior. *See Chapter 12*.



Parametric RDA adds a parametric recurrent event data analysis (RDA) folio to the project. The folio allows you to capture and analyze data obtained from recurring events, as in the case of a repairable system that may fail or be repaired multiple times during its operation life. The folio uses the General Renewal Process (GRP) model to describe the failure behavior of a system over time and obtain estimates such as the cumulative number of failures, mean time between failures (MTBF) and failure intensity. *See page 379 in Chapter 13*.



Non-Parametric RDA adds a non-parametric recurrent event data analysis (RDA) folio to the project. The folio allows you to capture and analyze data obtained from recurring events, as in the case of a repairable system that may fail or be repaired multiple times during its operation life. The folio uses the mean cumulative function (MCF) to model the reliability performance of a system over time and obtain estimates such as the cumulative number of failures. *See page 374 in Chapter 13*.

Tools



Diagram adds a reliability block diagram (RBD) to the project. In Weibull++ and ALTA, RBD diagrams are used to graphically represent the failure modes of a product and analyze how each mode contributes to the overall reliability performance of the product. *See Chapter 18*.



Weibull++ SimuMatic opens the Weibull++ SimuMatic Setup window, which allows you to generate several data sets all at once (via the Weibull++ Monte Carlo utility) and automatically perform basic life data analysis on each of the generated data sets. The simulated data could be used in a number of applications such as in the analysis of probabilistic models, risk analysis and in designing reliability tests. *See page 442 in Chapter 17*.



ALTA SimuMatic opens the ALTA SimuMatic Setup window, which allows you to generate several data sets all at once (via the ALTA Monte Carlo utility) and automatically perform accelerated life testing data analysis on each of the generated data sets. The simulated data could be used in a number of applications such as in the analysis of probabilistic models, risk analysis and in designing reliability tests. *See page 442 in Chapter 17*.



Test Design opens the Test Design Assistant, which provides tools that help you design, evaluate and compare reliability demonstration tests based on the number of failures (pass/fail tests) or the time of failure. *See Chapter 19*.



Target Reliability adds a target reliability estimator folio to the project. The folio generates plots to help you select a reliability goal that will minimize cost, maximize profit and maximize the return on an investment. *See Chapter 20*.



Life Comparison adds a life comparison folio to the project. The folio allows you to compare two data sets to determine whether the lives of the units in the first data set will outlast the lives of the units in the second. *See Chapter 21*.



Stress-Strength adds a stress-strength comparison folio to the project. The folio allows you to compare two data sets to determine the probability of failure based on the probability of stress exceeding strength. *See Chapter 21*.



Equation Fit Solver adds a non-linear equation fit solver folio to the project. The folio allows you to estimate the parameters of any user-defined non-linear equation. This gives you the flexibility to perform simple parameter estimation on statistical models other than the life distributions and life-stress relationship models available in Weibull++ and ALTA. *See Chapter 23*.



ALTA Test Plan adds an ALTA test plan folio to the project. The folio helps you design a test plan for accelerated life data testing analysis. Based on information you provide, the test plan will include recommendations for stress levels and the number of units to be tested at each stress level. *See Chapter 24*.



ALTA Stress Profile adds an ALTA stress profile folio to the project. Stress profiles describe how stress levels will vary with time and can be used to describe the stress conditions during testing when you analyze the data using the cumulative damage model in the ALTA standard folio. *See page 267 in Chapter 7*.



Maintenance Planning adds a maintenance planning folio to the project. The folio generates a cost vs. time plot that helps you determine the most cost-effective time to replace aged components. *See Chapter 22*.

Reports and Plots



Analysis Workbook opens a wizard to add a 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 page 533 in Chapter 27*.



Word Report Template opens a wizard to add a 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 page 561 in Chapter 28*.



Overlay Plot adds an overlay plot to the project allowing you to display in a single plot results from multiple data sheets. 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 or compare a simulation-based data set with actual data obtained from fielded products. *See page 396 in Chapter 14*.



Side-By-Side Plot adds a side-by-side plot to the project allowing you to display different plots of a single data set all in a single window for easy comparison. In Weibull++, you can select to plot the data set using multiple distributions or plot types. In ALTA, you get a third option to plot the data set using multiple life-stress relationship models. *See page 397 in Chapter 14*.

Attachment



Attachment adds linked or attached files to the project. *See page 127 in Chapter 3*.

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 101 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 105 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 (multiple document interface) 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** drop-down 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 Ribbon.
- **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).

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, 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)

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. *See page 2 in Chapter 1*.



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 4.2.2 on page 136*.

4.9 Active Tabs

Active (contextual) tabs contain commands related to a type of folio or analysis. These tabs are displayed only when the related item is active. For example, the Life Data tab contains commands related to working with a Weibull standard folio, while the Warranty tab contains commands related to working on a warranty analysis.

4.9.1 ALTA Tab

The ALTA tab is visible when you view an ALTA standard folio data sheet for accelerated life testing analysis. *See Chapter 7*.

Analysis



Calculate estimates the parameters of the chosen model, based on the current data set and the specified analysis settings.



Plot creates a new sheet in the folio that provides a choice of applicable plot types. For standard folios, this includes plots such as use level probability, reliability vs. time, life vs. stress, etc. *See page 257 in Chapter 7*.



Quick Calculation Pad opens the ALTA Quick Calculation Pad, which allows you to calculate results, such as the mean life and the probability of failure, based on the currently active data sheet. *See page 260 in Chapter 7*.



Distribution Wizard opens the Distribution Wizard, which helps you select the life distribution that will best fit your data. *See page 274 in Chapter 7*.



3D Plot creates a 3D graph of the surface plot of the likelihood function. It plots the likelihood values (LK values) with respect to the estimated parameters of a distribution. The shape of the plot may help you to evaluate the amount of variability in the parameter estimates. A small surface (tall, tower-like peak) suggests that there is less uncertainty in the parameter estimates at the maximum LK value, while a large surface suggests that there is a large uncertainty in the parameter estimates. *See page 399 in Chapter 14*.

In addition, the graph provides a choice of several functions that you can plot with respect to time and stress. This includes the *pdf*, failure rate, reliability and unreliability functions. The shape of the plot may help you to evaluate how the function is influenced by time and stress.

Folio Sheets



Insert Data Sheet inserts a new blank data sheet into the currently active folio and opens the ALTA Folio Data Sheet Setup window, which allows you to select what types of data to include in the new data sheet.



Insert Additional Plot inserts an overlay plot in the currently active folio. This allows you to display in a single plot results from multiple data sheets within the folio. 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.



Insert General Spreadsheet inserts a new blank general spreadsheet into the currently active folio.



Select Sheet opens the Select Sheet window, which allows you to select which sheet in the current folio you want to be active.



Move or Copy Sheet opens the Move/Copy window, which allows you to move or copy any of the sheets within the currently active folio. Select the sheet that you wish to move and then click **(move to end)** to move the selected sheet to the end of all the sheets in the folio. Select **Create a Copy** to create a copy of the selected sheet.



Delete Sheet deletes the current sheet within the folio.

Options



Set Use Stress opens the Use Stress Level window, which allows you to define the use stress values for the selected stress type(s) in the current folio. *See page 260 in Chapter 7.*



Select Stress Columns opens the Select Stress Columns window, which allows you to select which stress type(s) in the current folio to consider when calculating the parameters of the model (i.e., the life-stress relationship and distribution combination). *See page 259 in Chapter 7.*



Alter Parameters gives you two options for manually editing the value of the calculated parameters. Alter **Parameters (w/o Recalculation)** allows you to alter the value of one (or more) parameters, while keeping the original values of the rest of the parameters and the variance/covariance matrix the same. **Alter One Parameters (and Recalculate)** allows you to alter one of the parameter values, and then have the software estimate the other parameter values that would result in a good fit for the model. For both options, the LK value, solution line on plots and all subsequent analyses will be based on the modified set of parameters.



Stress Transformation opens the Stress Transformation window, which allows you to specify one transformation equation for each stress that will be used in the analysis. *See page 260 in Chapter 7.*



Likelihood Ratio Test opens the Likelihood Ratio Test window, which allows you evaluate the discrepancy between the estimates of the shape parameter at the various accelerated stress levels. The test helps you to determine whether the units in the sample will fail in the same manner across different stress levels. *See page 275 in Chapter 7.*



Convert Stress Values allows you to convert all the values entered in one of the stress columns to another value or to another stress unit (e.g., Celsius to Kelvin).

Format and View



Alter Data Type opens the ALTA Folio Data Sheet Setup window, which allows you to change the data entry columns in the current data sheet. Note that when you change the setup, you may lose information already entered in the data sheet. For example, switching the data sheet from one that supports interval data to one that supports only right censored data will remove all the inspection times. *See page 250 in Chapter 7*.



Override Plot Color opens the Override Plot Color window, which allows you to ignore the default colors of the plotted line and points, and apply a color that you specify. This setting is activated in the source data sheet of the plot. This means that when you enable the setting for a particular data sheet, it applies only to the plots that you create for that data sheet and not the plots for all other data sheets that may be within the same folio. Note that only the colors of the lines and points are affected. Other settings such as the line style and thickness, or the point size and shape are based on the settings that you have specified in the Plot Items page of the Plot Setup window. *See page 405 in Chapter 14*.



Set Headers as Default sets the column headers in the currently active data sheet as the default headers for all ALTA standard folio data sheets with the same data type (i.e., times-to-failure or free-form data).



Restore Default Column Headers uses the default column headers for the columns of the currently active data sheet.



Add/Remove Columns allows you to add or remove stress columns or subset ID columns in the currently active data sheet.



Auto Group Data opens the Auto Group Data window, which allows you to group together data points with the same failure/suspension times. Grouping data significantly speeds up calculations when you have a very large data set. *See page 232 in Chapter 6*.



Hide Control Panel allows you to hide the control panel in the current window.

Synthesis



Publish Model allows you to use the results of analysis (life-stress relationship and parameters) as a resource object and make it available for unlimited use anywhere within the project, in any Synthesis application that has a need for that type of object. *For more information on resource objects, see Chapter 5*.

4.9.2 Degradation Tab

The Degradation tab is visible when you view either a Weibull++ or ALTA degradation folio data sheet for degradation analysis. *See Chapter 8 and Chapter 9*.

Analysis



Calculate extrapolates the times at which the degradation will reach the critical degradation level—i.e., the times at which the degradation is serious enough that the unit is considered to be failed.

- In a Weibull++ degradation folio, the software performs life data analysis on the extrapolated times.
- In an ALTA degradation folio, the software performs accelerated life testing analysis on the extrapolated times at the accelerated stress level to extrapolate failure times to normal use conditions.



Plot creates a new sheet in the folio that provides a choice of applicable plot types. This includes the degradation vs. time plot, which displays the degradation of each unit over time.

- In a Weibull++ degradation folio, the plot sheet includes the life data analysis plots.
- In an ALTA degradation folio, the plot sheet includes accelerated life data analysis plots that are based on the failure times at normal use conditions.



Quick Calculation Pad opens the Quick Calculation Pad, which allows you to calculate results identical to the calculations you can perform in a Weibull++ standard folio QCP or an ALTA standard folio QCP.



Model Wizard opens the Degradation Model Wizard, which helps you select the degradation model that best fits your data.



Distribution Wizard opens the Distribution Wizard, which helps you select the distribution that best fits the data set based on the chosen parameter estimation method (i.e., RRX or RRY). In the degradation folio, the ranking in the Distribution Wizard is based on the life data model.

Transfer Life Data



Transfer Life Data to New Folio allows you to transfer the extrapolated failure and suspension times to a new standard folio.



Transfer Life Data to Selected Folio allows you to transfer the extrapolated failure and suspension times to an existing standard folio.

Note: If you are using an ALTA degradation folio, the icons show the ALTA version.

Synthesis



Publish Model allows you to use the results of the degradation analysis as a resource object and make it available for unlimited use anywhere within the project, in any Synthesis application that has a need for that type of object. *For more information on resource objects, see Chapter 5*.

Format and View



Override Plot Color opens the Override Plot Color window, which allows you to ignore the default colors of the plotted line and points, and apply a color that you specify. This setting is activated in the source data sheet of the plot. This means that when you enable the setting for a particular data sheet, it applies only to the plots that you create for that data sheet and not the plots for all other data sheets that may be within the same folio. Note that only the colors of the lines and points are affected. Other settings such as the line style and thickness, or the point size and shape are based on the settings that you have specified in the Plot Items page of the Plot Setup window. *See page 405 in Chapter 14*.

4.9.3 Diagram Tab

The Diagram tab is visible when you view a reliability block diagram (RBD). *See Chapter 18*.

Analysis



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 Weibull++ Quick Calculation Pad (QCP), which allows you to obtain reliability results based on the algebraic solution for the diagram. Clicking the **QCP** icon before the diagram has been analyzed will automatically perform the analysis and then open the QCP.

Diagram Tools



Add Block adds a new block representing a data sheet in a Weibull++ or ALTA standard folio to the current 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 (*k*-out-of-*n* switch) to the current diagram.



Add Junction adds a non-failing block to the diagram. Because diagrams must flow from a single starting point to a single ending point, you may need to use junction blocks as starting and/or ending blocks (see page 461 in Chapter 18). They can also be used anywhere within the diagram that you need a block that does not fail (i.e., does not affect the system reliability). This type of block does not have properties.



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.

Format and Style



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 Selected Graphic copies the current block to the Clipboard. The block can then be pasted into other documents as an *.emf graphic.



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.



Auto Arrange automatically arranges the blocks in the diagram so that all blocks are evenly spaced and centered.



Make Same Size allows you to specify how you want the blocks to the resized.

- **Height** resizes the selected blocks to be the same height as the first selected block.
- **Width** resizes the selected blocks to be the same width as the first selected block.
- **Both** resizes the selected blocks to be the same width and height as the first selected block.



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.



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.

Edit Diagram allows you to open 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 468 in Chapter 18*.

Edit Block

- **Block Properties** opens the data sheet that is associated with the block.
- **Block Style** opens the Block Style window, which allows you to customize the appearance of the selected block. *See page 468 in Chapter 18*.
- **Bring Block to Front** moves the currently select blocks in front of other objects on the diagram.
- **Send Block to Back** moves the currently select blocks behind other objects on the diagram.

Edit Connector

- **Connector Style** opens the Connector Style window where you can customize the appearance of the selected connectors. *See page 468 in Chapter 18*.
- **Line Bend** allows you to create a bend in the line.
- **Bring to Front** moves the currently select line in front of other objects on the diagram.
- **Send to Back** moves the currently select line behind other objects on the diagram.

4.9.4 Equation Fit Solver Tab

The Equation Fit Solver tab is visible when you view the Non-Linear Equation Fit Solver utility. *See Chapter 23*.

Analysis



Calculate estimates the parameters of the user-defined equation and calculates other results you could use to evaluate how well the function fits the data.



Plot creates a plot demonstrating the function fit for each known data point that you have defined. The scaling, setup, exporting and other features of the plot are similar to the options available for other Weibull++ plot sheets. Features that are not applicable to this plot will be hidden or disabled.



Initialize Parameters automatically copies all the unknown variables in your equation (except the random variable x) to the Function Parameters area. This also resets the minimum, maximum and initial guess values to their default values.

Solver Templates



Select from Templates allows you to use a saved template for use in the analysis.



Add to Templates allows you to use user-defined equations as templates for future use.

4.9.5 Event Log Tab

The Event Log tab is visible when you view an event log data folio data sheet. *See Chapter 12*.

Analysis



Calculate converts the log entries into failure/repair times and then fits distributions to the converted data. In addition, it calculates the total uptime and downtime of the components.

Actions and Settings



Set Shift Pattern opens the Shift Pattern window, which allows you specify the days and times when the system is operating.

Transfer Life Data



Transfer Life Data to New Folio transfers all of the analyzed time-to-failure and time-to-repair data sets to separate data sheets in a new Weibull++ standard folio.



Transfer Life Data to Selected Folio opens the Transfer Life Data window, which allows you to choose specific data set(s) to transfer to either new or existing folio(s).



Transfer to BlockSim is available only if BlockSim is activated on your computer. The Transfer to BlockSim window allows you to export the analysis results as blocks in a new simulation diagram.

4.9.6 Life Comparison Tab

The Life Comparison tab is visible when you view the Life Comparison tool. *See Chapter 21*.

Analysis



Calculate estimates the probability that the times-to-failure of the first population will be better or worse than the times-to-failure of the second population.



Select Data Sheets opens the Select Data Sheets window, which allows you to select the data sheets to compare.



Parameter Estimator opens the Target Reliability Parameter Estimator window, which allows you to explore how you could attempt to change one of the distribution parameters in order to meet a specified target reliability. The plot and reliability estimate will be based on the altered distributions and therefore may be appropriate only within the context of your specific scenario.

4.9.7 Life Data Tab

The Life Data tab is visible when you view a Weibull++ standard folio data sheet for life data analysis. *See Chapter 6*.

Analysis



Calculate estimates the parameters of the chosen distribution, based on the current data set and the specified analysis settings.



Plot creates a new sheet in the folio that provides a choice of applicable plot types. For standard folios, this includes plots such as reliability vs. time, failure rate vs. time, probability plot, etc. *See page 217 in Chapter 6*.



Quick Calculation Pad opens the Weibull++ Quick Calculation Pad, which allows you to calculate results, such as the mean life and the probability of failure, based on the analyzed data sheet. *See page 221 in Chapter 6*.



Distribution Wizard opens the Distribution Wizard, which helps you select the distribution that best fits your data based on the selected parameter estimation method (i.e., RRX, RRY or MLE). *See page 226 in Chapter 6*.



3D Plot creates a 3D graph of the surface plot of the likelihood function. It plots the likelihood values (LK values) with respect to the estimated parameters of a distribution. The shape of the plot may help you to evaluate the amount of variability in the parameter estimates. A small surface (tall, tower-like peak) suggests that there is less uncertainty in the parameter estimates at the maximum LK value, while a large surface suggests that there is a large uncertainty in the parameter estimates. You can create a 3D plot of the likelihood function for any of the available distributions in Weibull++ except for 1-parameter Weibull, mixed Weibull, Bayesian-Weibull or Weibull using competing failure modes distribution. *See page 399 in Chapter 14*.

Options



Batch Auto Run opens the Batch Auto Run window, which allows you to quickly extract data from an existing data set based on the subset ID. *See page 229 in Chapter 6*.



Specify Points opens the Specify Points window, which allows you to adjust the way that the line is fitted to the points for a rank regression analysis. The ranks will be calculated based on all of the data points but the regression line will be fitted only to the points you have specified. *See page 245 in Chapter 6*.



Alter Parameters gives you two options for manually editing the value of the calculated parameters. Alter **Parameters (w/o Recalculation)** allows you to alter the value of one (or more) parameters, while keeping the original values of the rest of the parameters and the variance/covariance matrix the same. **Alter One Parameters (and Recalculate)** allows you to alter one of the parameter values, and then have the software estimate the other parameter values that would result in a good fit for the model. For both options, the LK value, solution line on plots and all subsequent analyses will be based on the modified set of parameters.



Fill Median Ranks is available only if the data type is free-form (*see page 248 in Chapter 6*). It allows you to experiment with median ranks and see how they affect the solution of the analysis. Enter the time-to-failure values in the "X-Axis value" column, then choose this command. The "Y-Axis value" column will be filled with the corresponding median ranks.



Auto Group Data opens the Auto Group Data window, which allows you to group together data points with the same failure/suspension times. Grouping data significantly speeds up calculations when you have a very large data set. *See page 232 in Chapter 6*.



Goodness of Fit Results opens the Goodness of Fit Results utility, which uses different tests to determine how well the parameters of the distribution fit the data. *See page 231 in Chapter 6*.

Folio Sheets



Insert Data Sheet inserts a new blank data sheet into the currently active folio and opens the Weibull++ Folio Data Sheet Setup window, which allows you to select what types of data to include in the new data sheet. *See page 206 in Chapter 6*.



Insert Additional Plot inserts an overlay plot in the currently active folio. This allows you to display in a single plot results from multiple data sheets within the folio. 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.



Insert General Spreadsheet inserts a new blank general spreadsheet into the currently active folio.



Select Sheet opens the Select Sheet window, which allows you to select which sheet in the current folio you want to be active.



Move or Copy Sheet opens the Move/Copy window, which allows you to move or copy any of the sheets within the currently active folio. Select the sheet that you wish to move and then click **(move to end)** to move the selected sheet to the end of all the sheets in the folio. Select **Create a Copy** to create a copy of the selected sheet.



Delete Sheet deletes the current sheet within the folio.

Format and View



Alter Data Type opens the Weibull++ Folio Data Sheet Setup window, which allows you to change the data entry columns in the current data sheet. Note that when you change the setup, you may lose information already entered in the data sheet. For example, switching the data sheet from one that supports interval data to one that supports only right censored data will remove all the inspection times. *See page 206 in Chapter 6*.



Override Plot Color opens the Override Plot Color window, which allows you to ignore the default colors of the plotted line and points, and apply a color that you specify. This setting is activated in the source data sheet of the plot. This means that when you enable the setting for a particular data sheet, it applies only to the plots that you create for that data sheet and not the plots for all other data sheets that may be within the same folio. Note that only the colors of the lines and points are affected. Other settings such as the line style and thickness, or the point size and shape are based on the settings that you have specified in the Plot Items page of the Plot Setup window. *See page 405 in Chapter 14*.



Set Headers as Default sets the column headers in the currently active data sheet as the default headers for all Weibull++ standard folio data sheets with the same data type (i.e., times-to-failure or free-form data).



Restore Default Column Headers uses the default column headers for the columns of the currently active data sheet.



Add/Remove Columns allows you to add or remove subset ID columns in the currently active data sheet.



Color Data Values allows you to apply different font colors to the entries in the data sheet for easy visualization. The following commands are available:

- **Default Color** applies a black font color to all the entries in the data sheet
- **Based on Data Type** applies different font colors based on the type of censoring. The colors are: red for failure data (no censoring), green for suspension data, magenta for left-censored data and blue for interval data.

- **Based on Subset ID 1** applies different font colors based on unique subset IDs in the Subset ID 1 column. If a data point has no entry in the Subset ID 1 column, then the font color is black.
- **Based on Subset ID 2** applies different font colors based on unique subset IDs in the Subset ID 2 column. If a data point has no entry in the Subset ID 2 column, then the font color is black.
- **Based on Subset ID Combination** applies font colors based on unique combinations of subset IDs in the Subset ID 1 and Subset ID 2 columns. If a data point has no subset IDs, the font color is black, but if a data point has at least one subset ID, then it is treated as having a unique combination of subset IDs.

Note that whenever you edit or add new data to the data sheet, you must choose one of the Color Data Values commands again in order to apply the correct font colors.



Hide Control Panel allows you to hide the control panel in the current window.

Synthesis



Publish Model allows you to use the results of the life data analysis (life distribution and parameters) as a resource object and make it available for unlimited use anywhere within the project, in any Synthesis application that has a need for that type of object. *For more information on resource objects, see Chapter 5*.

4.9.8 Maintenance Planning Tab

The Maintenance Planning tab is visible when you view the Maintenance Planning folio. *See Chapter 22*.

Analysis



Plot updates the plot to reflect any changes that have been made, and recalculates the estimates for the minimal cost and optimal time.



Select Data Sheet opens a window that allows you to select the data sheet to analyze.



Create Task opens the Maintenance Task window, which allows you to create a preventive and/or inspection task for use in BlockSim simulation diagrams.

4.9.9 Non-Parametric LDA Tab

The Non-Parametric LDA tab is visible when you view a non-parametric life data analysis (LDA) folio data sheet. *See Chapter 10*.

Analysis



Calculate obtains the reliability estimates based on the chosen non-parametric method. It then converts the results to free-form data and performs a parametric analysis based on the chosen life distribution.



Plot creates a Reliability vs. Time plot of the non-parametric analysis. The scaling, setup, exporting and other features are similar to the options available for other Weibull++ plot sheets. Features that are not applicable to this plot will be hidden or disabled.



Quick Calculation Pad opens the Weibull++ Quick Calculation Pad, which allows you to calculate results such as mean life and probability of failure based on the results of the standard life data analysis. It uses the same calculations that you can perform in a Weibull++ standard folio QCP. *See page 221 in Chapter 6*.



Distribution Wizard opens the Distribution Wizard, which helps you select the distribution that best fits the data set based on the chosen parameter estimation method (i.e., RRX or RRY). In the non-parametric LDA folio, the ranking in the Distribution Wizard is based on the life data model. *See page 226 in Chapter 6*.

Transfer Life Data



Transfer Life Data to New Folio allows you to transfer the values from the Life Data Model area of the control panel to a fee-form data sheet in a new Weibull++ standard folio. The selected failure time values (start time, end time or average time) will be copied to the "X-Axis value" column, and the reliability values will be converted to their inverse values (i.e., unreliability) and then transferred to the "Y-Axis value" column.



Transfer Life Data to Selected Folio allows you to transfer the values from the Life Data Model area of the control panel to a free-form data sheet in an existing Weibull++ standard folio.

Synthesis



Publish Model allows you to use the life data model (i.e., life distribution and parameter estimates) as a resource object and make it available for unlimited use anywhere within the project, in any Synthesis application that has a need for that type of object. *For more information on resource objects, see Chapter 5*.

4.9.10 Plot Tab

The Plot tab is visible when you view a plot sheet or the Target Reliability tool (*Chapter 20*).

Plot



Redraw Plot updates the plot to reflect the changes that have been made.



Quick Calculation Pad opens the Quick Calculation Pad (QCP), which allows you to obtain reliability results based on the currently active folio.

Plot Actions



Copy Plot Graphic allows you to copy the plot to the Clipboard as a graphic. You must select an image type. You can copy the plot as a bitmap image or as a metafile image optimized for pasting into a Synthesis spreadsheet (e.g., analysis workbook) or a metafile image optimized for pasting into external applications. 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 including the titles, line styles and point styles. *See Chapter 15*.



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 16 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. *See page 393 in Chapter 14*.

Confidence Bounds



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. *See page 392 in Chapter 14*.



Hide Confidence Bounds removes the line(s) depicting the confidence bounds from the current plot.

4.9.11 Recurrent Event Data Tab

The Recurrent Event Data tab is visible when you view either a parametric or a non-parametric recurrent event data analysis (RDA) folio data sheet. *See Chapter 13*.

Analysis



Calculate estimates the cumulative number of failures (in non-parametric RDA folios) or the parameters of the GRP model (in parametric RDA folios) based on the current data set.



Plot creates a new plot sheet in the folio. In non-parametric RDA folios, there is only one type of plot: the mean cumulative function (MCF). In parametric RDA folios, it provides a choice of applicable plot types such the cumulative failures vs. time, failure intensity vs. time, etc.



Quick Calculation Pad opens the Parametric RDA Quick Calculation Pad, which allows you to calculate results based on the analyzed data sheet, such as the number of failures and mean time between failures (MTBF). This command does not apply to non-parametric RDA folios.

4.9.12 Sheet Tab

The Sheet Options tab is visible when you view standard folios, general spreadsheets or analysis workbooks.

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. Changes cannot be made to a frozen row. This feature only applies to general spreadsheets and analysis workbooks.

  You can also freeze several consecutive rows at once by highlighting a cell on each row to be locked and then clicking **Freeze Panes**.
- **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. *See page 551 in Chapter 27*.
- **Calculation Options** opens the Calculation Options window, which allows you to change formula recalculation, iteration or precision. *See page 552 in Chapter 27*.
- **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. *See page 554 in Chapter 27*.

  The Cell Protection setting is especially useful when your spreadsheet or 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.
- **Define Name** opens the Define Name window, which allows you to specify a name for a particular cell, data sheet or function. Names can help make formulas in the spreadsheet easier to understand and maintain. *See page 548 in Chapter 27*.

Sheet Actions



Function Wizard opens the Function Wizard, which helps you build functions in spreadsheets. The functions can return results from mathematical values (e.g., sine, pi or averages) or reliability values based on data sheets in the p**roject.** *See page 539 in Chapter 27*.



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. *See page 546 in Chapter 27*.



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. Note the following:

- If your analysis takes into account the order of occurrence of a data point (1st, 2nd, 3rd, etc.), then sorting may result to an inaccurate analysis. In Weibull++, analyses that are based on the order of occurrence of failure are: recurrent event data analysis, degradation analysis and non-parametric LDA 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.

4.9.13 SimuMatic (A) Tab

The SimuMatic(A) tab is visible when you view the ALTA SimuMatic results folio. *See page 442 in Chapter 17*.

Simumatic



ALTA SimuMatic opens the ALTA SimuMatic Setup window, which allows you to change your simulation settings and replace your current simulated data sets with new ones.



Plot creates a new sheet in the folio that generates a probability plot of all the data sets.

4.9.14 SimuMatic (W) Tab

The SimuMatic(W) tab is visible when you view the Weibull++ SimuMatic results folio. *See page 442 in Chapter 17*.

Simumatic



Weibull++ SimuMatic opens the Weibull++ SimuMatic Setup window, which allows you to change your simulation settings and replace your current simulated data sets with new ones.



Plot creates a new sheet in the folio that generates a probability plot of all the data sets.



Transfer Life Data to Selected Folio allows you to transfer the calculated parameter values for each simulated data set to an existing Weibull++ standard folio, with one data sheet for each parameter of the model that was used to generate the data. *See page 448 in Chapter 17*.



Transfer Life Data to New Folio creates a new Weibull++ standard folio and then transfers the calculated parameter values for each simulated data set to that folio, with one data sheet for each parameter of the model that was used to generate the data. *See page 448 in Chapter 17*.

4.9.15 Stress Profile Tab

The Stress Profile tab is visible when you view an ALTA stress profile data sheet. *See page 267 in Chapter 7*.

Analysis



Validate Stress Profile validates and then saves any changes you made to the stress profile.



Plot Stress Profile creates a new sheet in the folio that shows a stress vs. time plot. The plot provides a graphical representation of how the stress values vary with time.

4.9.16 Stress-Strength Tab

The Stress-Strength tab is visible when you view the Stress-Strength tool. *See Chapter 21*.

Analysis



Calculate estimates the probability of stress exceeding the strength of a given product.



Select Data Sheets opens the Select Data Sheets window, which allows you to select the data sheets to compare.



Parameter Estimator opens the Target Reliability Parameter Estimator window, which allows you to explore how you could attempt to change one of the distribution parameters in order to meet a specified target reliability. The plot and reliability estimate will be based on the altered distributions and therefore may be appropriate only within the context of your specific scenario.

4.9.17 Test Design Tab

The Test Design tab is visible when you view any of the Test Design tools. Depending on which utility you view, the following commands may be available. *See Chapter 19*.

Reliability Demonstration Test



Calculate implements the selected test design method based on the inputs you provided.



Get Distribution allows you to update the shape parameter and demonstrated time in the RDT sheet based on data from an existing Weibull++ standard folio in the current project.



Show RDT Table is available only for the Parametric Binomial test design method. This command creates a parametric binomial table and plot based on the target metric and life distribution that you specified on the RDT sheet. *See page 473 in Chapter 19*.



Show Bayesian Systems is available when you choose the Non-Parametric Bayesian test design method and the source of prior information is from "Prior tests at the subsystem level." This command opens the Edit Bayesian Subsystems window, which allows you to edit the number of units and number of failures for each subsystem for which you have prior information. *See page 472 in Chapter 19*.

RDT Table



Calculate generates the RDT table based on the sample size and number of failures that you specify on the control panel.



Plot creates a plot in the folio that depicts how the allowed number of failures influences the test time and the required sample size.

Expected Failure Time



Get Failure Model allows you to update the expected failure model (i.e., parameters and time) based on data from an existing Weibull++ standard folio in the current project.

Detection Matrix



Calculate validates your inputs and generates the matrix.



Send to Excel exports the matrix to a new Excel spreadsheet.



Change Matrix Color allows you to choose a new base color for the matrix. The selected color will be used to indicate the shortest test time and darker values of the color will be used to indicate longer test times.

4.9.18 Test Plans Tab

The Test Plans tab is visible when you view the ALTA Test Plans tool. *See Chapter 24*.

Analysis



Calculate generates a test plan recommendations for what stress levels should be used in the test and how many units should be tested at each stress level.

4.9.19 Warranty Tab

The Warranty tab is visible when you view a warranty folio data sheet for warranty analysis. *See Chapter 11*.

Analysis



Calculate converts the warranty claims data to failure/suspension times and then estimates the parameters of the chosen life distribution based on the converted data. If you select the **Use Subsets** option on the Main page of the control panel, the parameters of the distribution of each subset ID will be separately calculated.



Plot creates a new sheet in the folio that provides a choice of applicable plot types. In warranty analysis folios, this includes plots such as expected failures vs. period, reliability vs. time, probability plot, etc. You can plot the data in the warranty analysis folio either with or without generating a forecast. *See page 352 in Chapter 11*.



Quick Calculation Pad opens the Weibull++ Quick Calculation Pad, which allows you to calculate life data analysis metrics. It uses the same calculations that you can perform in a Weibull++ standard folio QCP.



Distribution Wizard opens the Distribution Wizard, which helps you select the distribution that best fits your data based on the selected parameter estimation method (i.e., RRX, RRY or MLE). In warranty analysis folios, the ranking in the Distribution Wizard is based on the converted failure/suspension times. *See page 226 in Chapter 6*.

Tools



Forecast opens the Forecast Setup window, which allows you to specify the time periods at which you wish to estimate the number of expected warranty returns. Clicking **Forecast** before the parameters have been calculated will automatically calculate the parameters and open the Forecast Setup window. *See page 350 in Chapter 11*.



Restore Guide Colors restores the colors of the cells in the data sheet to the default guide colors. This command applies only if you have altered the cell colors of the Returns sheet in a Nevada chart format warranty folio. (The colors can be altered by select the desired cells or row/column header and then choosing **Home > Edit > Format Selection > Fill Color**). Guide colors are used in the Statistical Process Control feature of the Nevada chart format (*see page 322 in Chapter 11*). The default guide colors are:

- **Green** for the number of product returns that are within the expected range.
- **Yellow** for the number of product returns that are between the caution and critical levels.
- **Red** for the number of product returns that are above the critical range.



Data Sheet Setup is available only for the Nevada chart format. It opens the Warranty Setup window, which allows you to change the time periods of the warranty claims data in the current data sheet. *See page 314 in Chapter 11*.



Convert Data converts the format of the currently active warranty analysis folio to a new format. Converting formats creates a new folio and does not overwrite the contents of the existing folio. It is an automatic process, except when converting from the Dates of Failure format to Nevada chart format. In that case, you will be

prompted to define the time unit of each period (e.g., dates, month or years) in order to proceed with the conversion. The following table shows the data conversion options.

From	Convert To
Nevada Chart Format	Dates of Failure Format
	Times-to-Failure Format
Dates of Failure Format	Nevada Chart Format
	Times-to-Failure Format
Times-to-Failure Format	N/A
Usage Format	Times-to-Failure Format

Transfer Life Data



Transfer Life Data to New Folio allows you to transfer the failure/suspension times to a new Weibull++ standard folio in order to perform life data analysis.



Transfer Life Data to Selected Folio allows you to transfer the failure/suspension times to an existing Weibull++ standard folio in order to perform life data analysis.

4.9.20 Workbook Tab

The Workbook tab is visible when you are working with an analysis workbook. *See page 533 in Chapter 27*.

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.



Save Workbook as Template allows you to save the current report as a template so that you can reuse its settings.

Actions



Function Wizard opens the Function Wizard, which helps you to build functions in analysis workbook sheets. These functions can return results ranging from mathematical values (e.g., sine, pi or averages) to reliability values based on associated data sheets (e.g., reliability for a given time, MTTF, 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.



Refresh Report updates the analysis workbook to reflect any changes that have been made to the sheets.



Defined Names opens the Define Names window which allows you to define a variable name for a particular cell, analysis, or function, or to edit existing defined names for the analysis workbook.



Select Data Sheets allows you to define or change the default data sources for the analysis workbook. When you click this button a window will appear, which allows you to select up to four analyses in the project.



Save Workbook as Template allows you to save the current report as a template so that you can reuse its settings.

Resources 5

In Synthesis applications, resources contain various types of information that can be shared between analyses. Synthesis resources may be created in one application (e.g., Weibull++) and then shared with other Synthesis applications where they may be relevant. When the resource is updated with new information, the change is reflected automatically in all analyses that rely upon it. An example of a Synthesis resource is a reliability model that represents a product's probability of operating successfully over a given period of time. The model may be published from an application such as Weibull++/ALTA, RGA and Lambda Predict and then used in applications such as BlockSim, Xfmea, RCM++ or RBI to define the reliability or maintainability characteristics of a component.

For instance, imagine that you have a data set from testing a component called "Component A." You use Weibull++ to analyze the data set. You then take the fitted lifetime distribution and publish that information as a model called "Component A Failure."

Now imagine that you want to analyze the reliability and maintainability of a system that incorporates Component A. In BlockSim, you create a URD called "Component A RAM" to describe the component's failure rate and any activities performed to maintain the component; this includes your failure model and some tasks that describe the maintenance activities. You then assign that URD to the block that represents Component A in your reliability block diagram.

In this example, you have two types of analyses: the test data analysis, which is accessible only via Weibull++, and the system reliability and maintainability analysis, which is accessible only via BlockSim. You also have three types of resources: the model, the URD and the tasks. All of these are accessible from all applications via the Resource Manager, but are used only in BlockSim in this case. Additionally, because the model was created by publishing analysis results, it can be changed only by returning to the original data analysis in Weibull++, changing the analysis and republishing.

The following sections provide information about resources in Synthesis projects, including:

- A description of all available Synthesis resources, including the distinction between local resources, which are available only within the current project, and global resources, which are available in all projects in the current database - Section 5.1 (p. 168)
- How to work with resource wizards to create, assign, edit or delete resources - Section 5.2 (p. 170)
- How to work with the Resource Manager - Section 5.3 (p. 173)
- How to use the Dependency Viewer to track the use of Synthesis resources - Section 5.4 (p. 180)
- A description of the Synthesis resources that can be used in multiple locations, including:
  - Universal Reliability Definitions (URDs) - Section 5.5 (p. 184)
  - Models - Section 5.6 (p. 185)
  - Variables - Section 5.7 (p. 201)

5.1 Types of Resources

Although many resource types can be created and edited via the Resource Manager in any Synthesis application, not all resources can be used in all applications. For example, you can create or edit a universal reliability definition (URD) when you open the Resource Manager from Weibull++, ALTA, RGA or Lambda Predict, but this resource is not used in any of the analyses in those applications. Resources can also be created on the fly in applications where they are used. Each resource can be used unlimited times. The table shown next defines the resource types and shows which applications can create and use each type.

Resource	Used In
Universal reliability definitions (**URDs**) are used to describe a set of reliability and maintenance characteristics. *See Section 5.5 on page 184*.	BlockSim, RCM++, Xfmea, RBI
Models can represent probabilities, durations or costs, either fixed or time-dependent. These resources are used by other resources (e.g., to represent reliability in a URD, to represent the duration of a task, etc.). *See Section 5.6 on page 185*.	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 page 116 in Chapter 3*
Tasks represent maintenance activities that can be assigned to URDs. There are two basic kinds of tasks: Corrective tasks are unplanned maintenance activities that are performed when a failure occurs. Scheduled tasks include preventive maintenance activities, inspections and on condition maintenance activities.	BlockSim, RCM++, RBI
Task packages represent groups of tasks that are performed together at scheduled intervals.	BlockSim, RCM++, RBI
Crews are assigned to tasks. They represent the labor personnel who will perform the maintenance activity.	BlockSim, RCM++, RBI
Spare part pools are assigned to tasks. They are used to describe the conditions that determine whether a spare part will be available when needed and to specify the time and costs associated with obtaining the spare part.	BlockSim, RCM++, RBI
Maintenance groups are sets 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.	BlockSim, RCM++, RBI
Switches are assigned to standby containers in RBDs and standby gates in fault trees. They describe the behavior of the switch that transfers the activity from the active block to a standby block (and possibly back to the original active block, depending on the configuration). Switches are shown only in the BlockSim Resource Manager.	BlockSim

Resource	Used In
Mirror groups are used to represent the same event/component in different locations within your analysis. In Xfmea/RCM++, cause records in a mirror group are treated as one and the same event, allowing you to place the same cause in more than one location in the FMRA hierarchy. In BlockSim, blocks in a mirror group all represent a single component, allowing you to represent a single component more than once in a diagram or in multiple diagrams within a project. With BlockSim's FMRA diagrams, blocks that represent causes can also be mirrored.	BlockSim, RCM++, Xfmea, RBI
Maintenance templates allow you to specify the order in which blocks in a diagram will be maintained when that diagram is used as a maintenance phase in a phase diagram. Maintenance templates are shown only in the BlockSim Resource Manager.	BlockSim
Actions allow you to define and track actions that need to be performed. *See page 107 in Chapter 3*.	All applications via My Portal; RCM++, RBI and Xfmea for analyses
Controls are used in RCM++ or Xfmea to represent methods that are planned or currently in place to reduce or eliminate the risk associated with potential failures. The Controls node will be available in the Resource Manager if the **Enable Controls for Causes** option is selected for the current interface style in Xfmea/RCM++. The Controls node is available by default in projects that have never been opened in Xfmea/RCM++.	RCM++, Xfmea, RBI
Variables each store a fixed numerical value and can be programmatically varied during simulation. *See Section 5.7 on page 201*.	BlockSim, RENO, DOE++
RENO functions store equations that are evaluated based on input values passed to the function during simulation.	RENO
RENO static functions store equations that are evaluated before simulation begins.	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.	RENO
RENO tables store arrays of values in rows and columns.	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
- Tasks

- Crews
- Pools
- 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

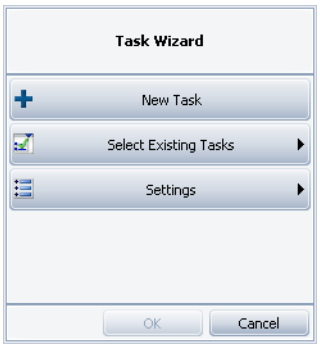
Note: When you create a restore point, any global resources used in the project are converted to local resources and stored with the backup.

5.2 Resource Wizards

Where resources are shown in properties windows (e.g., URDs shown in BlockSim/RENO's Block Properties window, spare part pools shown in the Maintenance Task window, etc.), you can use a resource wizard to work with them. The arrow that provides access to the wizard is shown only for the resource field that is currently selected in the properties window. For all other resource fields, the availability of the tools is indicated by an ellipsis (...) in the field.

All of these wizards work in a similar manner except for the Model Wizard, which provides additional functionality that allows you to create the model directly within the wizard (*see Section 5.6.5 on page 190*). The following images of the Task Wizard serve as the example for how to use all other wizards.

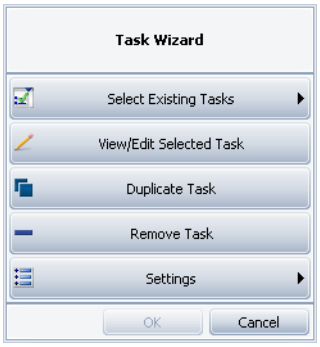
The Main page provides access to all available functions within the wizard. If a resource has not been assigned in the field, you can create a new resource or select an existing one. You can also use the Settings page to specify the resources shown in the wizard.



If a resource has already been assigned, you can:

- Select another existing resource to replace the current resource. *See Section 5.2.1 on page 171*.
- View/edit the current resource. *See Section 5.2.3 on page 172*.

- Create a duplicate of the current resource and assign the new copy in the field (e.g., if you want to use the same/similar properties but remove the association with the original resource that may be used in a different way elsewhere in the project). *See Section 5.2.2 on page 172*.
- Remove the resource from the current location (it will still be available for other records within the current project). *See Section 5.2.4 on page 172*.
- Use the Settings page to specify the resources shown in the wizard. *See Section 5.2.5 on page 172*.

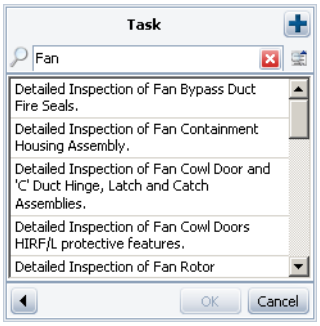


Most functions are also available as icons on the Select page of the wizard, which is the default view (except in the Model wizard, in cases where no model has yet been assigned). You can, however, always return to the Main page of the wizard by clicking the **Back** arrow at the lower left corner of the Select page.

5.2.1 Selecting an Existing Resource

The Select page provides a list of the resources of the specified type that currently exist in the project. This is the default view for all resource wizards except the Model wizard, and can also be accessed by clicking the **Select Existing** button or icon.





You can filter the resources that are displayed in the list by entering text in the **Filter** field (marked with a magnifying glass); clear the filter by clicking the **Clear Filter** icon in the field (red x). Select a resource and click **OK** (or double-click it) to assign the resource to the current location. If a resource was already assigned, the newly selected resource will replace it.

You can click the **Select Resource** icon to open the Select Resource window, which allows you to select an existing resource of the appropriate type. The Select Resource window displays more information about each resource and offers a wider range of filtering tools.



5.2.2 Creating a Resource

To create a new resource from the wizard, click the **Create New** button or icon.



This will open the resource properties window of the appropriate type (e.g., the Maintenance Task window, the Crew window, the Spare Part Pool window, etc.). Once you have specified the properties for the new resource, click **OK** to create the resource and assign it to the current location.

If a resource has already been assigned and you wish to remove the association with the original resource but use the same (or similar) properties for the current location, you can click the **Duplicate** button on the Main page of the wizard. The new copy will then be assigned in place of the original.

5.2.3 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.



IMPORTANT: Any changes you make to the resource will apply everywhere that the resource is used. This includes analyses in other Synthesis applications.

5.2.4 Removing a Resource

If a resource has already been assigned, you can remove the resource from the current location (e.g., remove the task from the URD) by clicking the **Remove** button or icon.

The resource will still be available to be assigned to other records in the current project. If a resource is not assigned to any other locations within the project and you want to completely remove it from the database, choose **Project > Synthesis > Resource Manager**.

5.2.5 Specifying the Resources Shown in the Wizard

On the Settings page of the wizard, you can specify the following:

- Whether you want to see all resources in the wizard, only local resources (i.e., those resources that are available for use only in the current project, which is the default setting for resources) or only global resources (i.e., those resources that are available for use in any project within the database).

  Note: When you are associating resources with a global resource (e.g., associating a task with a global URD), only global resources will be shown in the wizard.

- The maximum number of resources that will be displayed in the wizard. Larger numbers of resources may take a longer time to load. If not all resources are shown in the wizard, you can click the **Select Resource** icon to open the Select Resource window, which displays all existing resources of the current type.



5.3 Resource Manager

The Resource Manager allows you to create, view, edit and delete resources in the project. The types of resources available in the Resource Manager depend on which Synthesis application you are using. *For a list of all available resource types, see Section 5.1 on page 168*.

To open the Resource Manager, choose **Project > Synthesis > Resource Manager**.



To use the Resource Manager, 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. You can limit the number of resources shown in the table using the **Home > Filter > Maximum Display** field. The limit can be adjusted in increments of 100.

You can change the columns that appear in the table for a resource type via the Column Chooser, which is accessed by choosing **Home > Columns > Column Chooser**. (*For details about the Column Chooser, see Section 5.3.5 on page 177*).



You also can click the **Columns** icon (identical to the Column Chooser icon) at the bottom of the navigation panel to display a list of available columns in the navigation panel.

For complete information on configuring the Resource Manager, see Section 5.3.4 on page 176.

What's Changed? In the previous version of RENO, the Definitions Report window allowed you to view all definitions in the project and determine where they were used. This information can now be obtained via the Resource Manager.

5.3.1 Basic Operations

You can:

- Create a new resource of the selected type by choosing **Home > Edit > New**.

  You can also add multiple resources of the selected type by specifying the number of resources you want to add in the **Home > Batch Add > Number to Add** field and then choosing **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.
- Open an existing resource for viewing and/or editing by double-clicking the resource row in the table or by selecting the resource row in the table and choosing **Home > Edit > View**.

  
- Duplicate an existing resource by selecting the resource row in the table and choosing **Home > Edit > Duplicate**.

- Delete an existing resource by selecting the resource row in the table and choosing **Home > Edit > Delete**.



Note: A model cannot be edited or deleted when it is still linked to its data source. You can remove the association between the model and its data source via the control panel Publishing page in most Synthesis applications, or via the Model tab of the Properties panel in Lambda Predict.

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 5.4 on page 180.*



You can make an existing resource into a global resource, available for use in any project in the database, by selecting the resource row in the table and choosing **Home > Actions > Make Global**.



This cannot be undone. Note that 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. In addition, crews, spare part pools, etc. that are associated with the affected tasks will be made global. Additionally, you cannot associate a local resource with a global resource (e.g., you cannot add a local task to a global URD).

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.

5.3.2 Advanced Operations

The Resource Manager provides several tools to make it easier to deal with large numbers of resources.

- To find unused resources of the selected type, choose **Administration > Selection > Show Only Unused**.



You can clear this selection by choosing **Administration > Selection > Show All**.



- To find duplicate resources, choose **Administration > Selection > Show Only Duplicates**.



In the Select Comparison Criteria 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 clear this selection by choosing **Administration > Selection > Show All**.

- You can merge multiple resources of the same type into one resource. This can be helpful when you have, for example, several models that all represent the identical distribution or constant value, or several tasks that are identical. This might happen if, for example, you have imported diagrams that all reference the same model, but the imports happened at separate times; the model would be imported each time a diagram that referenced it was imported. There are two ways to merge resources:

  IMPORTANT: For both methods, you should be certain that the resources that you are merging are identical before proceeding. 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.

  - To merge resources manually, select the resources and choose **Administration > Actions > Merge and Delete**.

    The number of resources that can be selected at once is limited by the value in the **Administration > Selection > Max Selection** field.

    In the window that appears, select the resource to merge the other resources into (i.e., the resource name that will be retained). This resource will be used in all places where the other selected resources were previously used.

  - To find duplicate resources and merge them into a single resource, choose **Administration > Actions > Merge All Duplicates**.

    You will be asked to confirm that you want to merge all duplicate instances of each resource of the selected type into the first occurrence of the resource. In the Select Columns window that then appears, select the properties that must match in order for the resources to be considered duplicates. Once you have selected the columns to compare, click **OK** to merge the duplicate resources.

  Note: If you are merging models, they must be of the same category in order to be merged. You cannot merge models that were created via publication.

5.3.3 Exporting Information from the Resource Manager

To print the contents of the Resource Manager, choose **Home > Edit > Print**.



You can send the contents of the Resource Manager to Excel or, for Synthesis applications that offer analysis workbooks, to an analysis workbook by choosing **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) to an analysis workbook or to Excel.

5.3.4 Resource Manager Display Characteristics

To rearrange columns in a Resource Manager table, 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.

You can resize the columns by dragging the edge of the column header to the desired position. To automatically resize the 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. Choosing **Best Fit (all columns)** resizes all columns in the table to fit the data that they contain.

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 change the columns that appear in the table for a resource type via the Column Chooser, which is accessed by choosing **Home > Columns > Column Chooser**. (*For details about the Column Chooser, see Section 5.3.5 on page 177*).



You also can click the **Columns** icon at the bottom of the navigation panel to display a list of available columns in the navigation panel.

There are several ways to filter the resources that appear in the table.

- The Auto Filter row appears by default immediately below the column headers. Enter text in any cell in this row to display only those resources that contain 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**.



- The Find panel, which is hidden by default, allows you to search for text within the resources that are currently displayed in the table. Right-click any column header and choose **Show Find Panel** to display the panel. Enter text in the field and click **Find** to filter the resources displayed in the table. 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**.
- Choose **Home > Filter > Enable Advanced Filters** to gain access to more detailed filtering functionality. *See Section 5.3.6 on page 177*.



- 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.

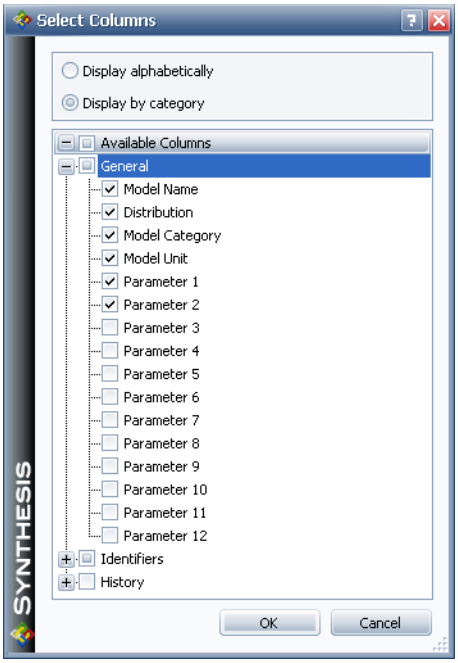
The number of resources displayed in the table is determined by the value in the **Home > Filter > Maximum Display** field.

5.3.5 Column Chooser

The Column Chooser allows to you specify which columns will appear in the Resource Manager or in BlockSim's Item Properties Table for the type of item you are currently working with. To access the Column Chooser, click **Column Chooser** or right-click any column header and choose **Column Chooser**.



The columns that are available in this window will vary depending on the type of item you are working with. Note that when you are working with an RBD in the Item Properties Table, the properties in question are those that can appear in the Block Properties window for any type of block that can appear in an RBD. The Column Chooser for models is shown next.



You can sort the available columns by category, as shown here, or you can select the **Display alphabetically** option to sort the columns alphabetically; this may be helpful if you are looking for a particular column. The order that you sort by in this window does not affect the column positions in the Resource Manager or Item Properties Table.

You can select or clear an individual column's check box to display or hide that column. You can also select or clear the check box in the **Available Columns** header to select or clear all check boxes. If you have sorted the available columns by category, you can select or clear the check box for the category to select or clear all of the check boxes for the columns that make up that category.

The selections you make in this window will apply to all items of the current type. For example, if you are working with a fault tree in the Item Properties Table, the selections you make in the Column Chooser will apply when you are working with all fault trees.

5.3.6 Advanced Filters

In the Resource Manager and in BlockSim's Item Properties Table, you can click **Enable Advanced Filters** to gain access to more detailed filtering functionality.



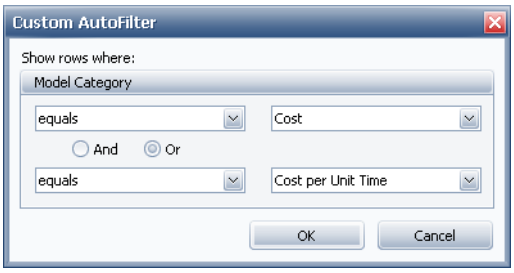
This functionality is also always enabled in the XFRACAS to RDW interface and in the RDW to Folio window.

When advanced filters are enabled, a **Filter** icon will appear when you point to any column header in the table.

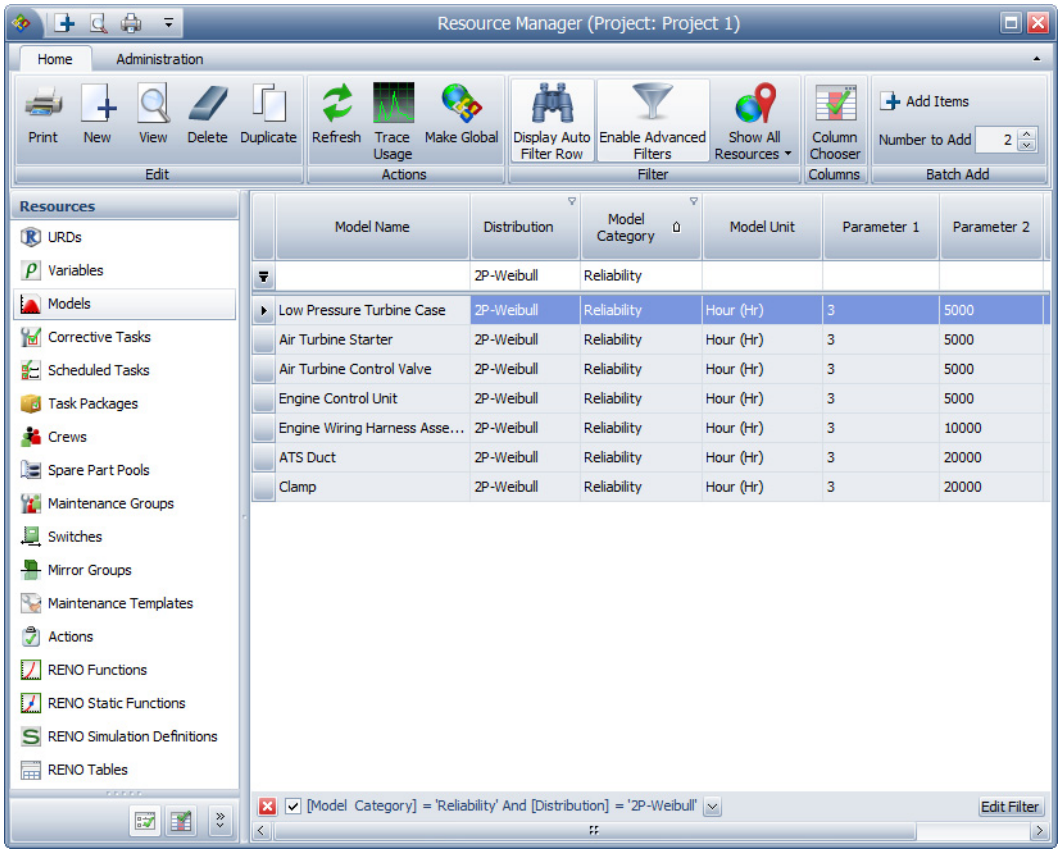


Click this icon to see a drop-down list of criteria that you can apply for the currently selected column. Only those resources that use the selected criterion will be displayed in the table.

In the Resource Manager and the Item Properties Table, if no predefined criteria 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.

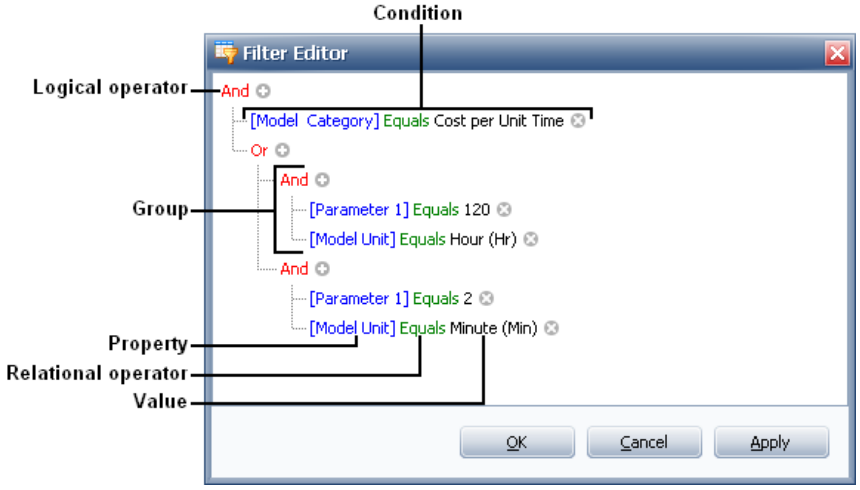


You can apply multiple filters in this way to narrow the resources in the table as needed. Each column that has a filter applied will continue to display the Filter icon in the column header. The filters that you apply will appear at the bottom of the table, as shown next.



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**.

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**. The Filter Editor window allows you to modify your filter(s) as needed.



Click any colored element in the window to view a drop-down list of available values. Each group is headed by a logical operator (And, Or, Not And, Not Or), which is shown in red, and expresses the connection between the conditions below it. Each condition is made up of a property that you can filter by (e.g., Model Name, Model Category, Created By, etc.), which is shown in blue, and a relational operator (e.g., equals, is greater than, is between, is any of), which is shown in green. The filter shown here will show only cost per unit time models equal to either $120 per hour or $2 per minute.

To add a condition to a group, click the plus sign beside the group's logical operator, click the logical operator and choose **Add Condition** or, with some element of the group in focus, press the **PLUS** or **INSERT** key. To remove a condition from a group, click the **X** beside the condition or, with some element of the condition in focus, press the **MINUS** or **DELETE** key.

To add a subgroup to a group, click the main group's logical operator and choose **Add Group**. To remove a subgroup from a group, click the **X** beside the subgroup's logical operator, click the subgroup's logical operator and choose **Remove Group** or, with the subgroup's logical operator in focus, press the **MINUS** or **DELETE** key.

5.3.7 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.



The Select Resource window is similar to the Resource Manager, with the following differences:

- Only the resource type that you are currently working with appears in the **Resources** area.
- 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.

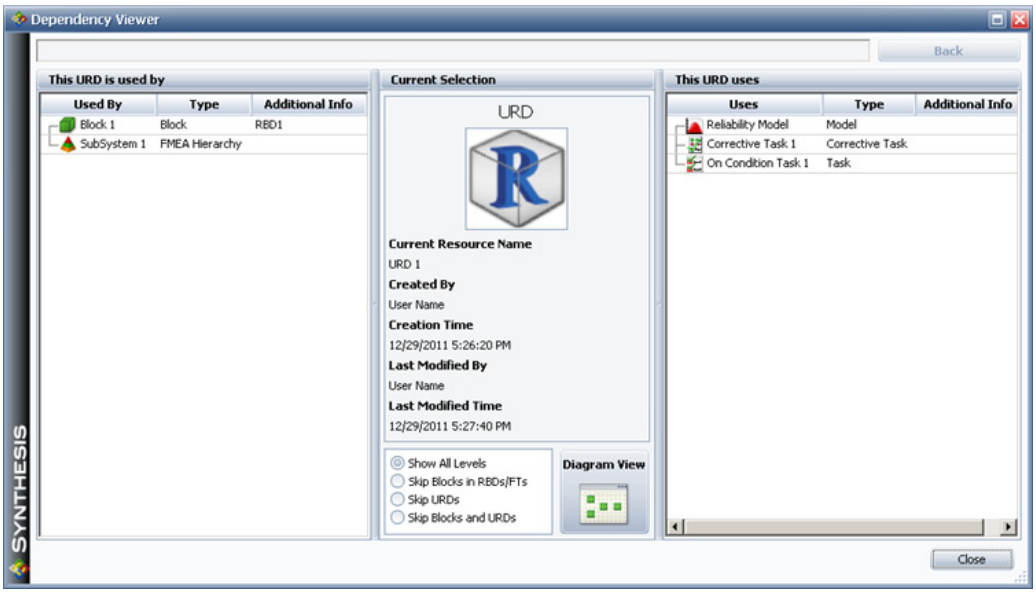
5.4 Dependency Viewer

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, accessible 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 any additional resources that the currently selected resource itself uses.



The currently selected resource is shown in the **Current Selection** area in the middle of the Dependency Viewer.

If the selected resource uses any resources, those resources (i.e., its dependents) will be displayed in the **This [resource] uses** area on the right. For example, if the selected resource is a URD, the model and any tasks it uses will be displayed, as shown next.



You can drill down by double-clicking a resource in the **This [resource] uses** area or by right-clicking and choosing **Trace Item** on the shortcut menu. This resource will then become the new selected resource, and any resources it uses will then be displayed. 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. The items selected will be shown in order in the field at the top of the window; click **Back** to go back through the path to the previously selected items.

If the selected resource is used anywhere, the items that use it (i.e., its precedents) will be displayed in the **This [resource] is used** by area on the left. For example, you can see in the image above that URD 1 is used by Block 1 in a BlockSim diagram and SubSystem 1 in an FMEA analysis. The **This [resource] is used by** area works similarly to the **This [resource] uses** area; you can double-click an item to make it the current selection in the window.

You can use the options in the **Current Selection** area to control what is shown in the area on the left. For example, imagine that the currently selected item is the "Reliability Model" from the image above. This model is used by URD 1, which is used by Block 1 in RBD 1 and by SubSystem 1.

- If **Show All Levels** is selected, URD 1 (the immediate precedent) will be shown in the **This [resource] is used by** area. Double-clicking this will select the URD, and Block 1 and SubSystem 1 will be shown as its precedents. You can then double-click the block to select it, showing RBD 1 as its precedent. You could double-click SubSystem 1 instead of the block, showing that it has no precedents.
- If **Skip Blocks in RBDs/FTs** is selected, URD 1 (the immediate precedent) will be shown in the **This [resource] is used by** area. Double-clicking this will select the URD, and RBD 1 and SubSystem 1 will be shown as its precedents, leaving out the block between the URD and the RBD. Note that blocks are always skipped in RENO flowcharts (i.e., only the flowchart name will be shown as the precedent for a resource used by any block within the flowchart).
- If **Skip URDs** is selected, Block 1 and SubSystem 1 will be shown as the precedents, skipping the URD between the model and the block. Double-click the block to select it, and to show RBD 1 as its precedent.
- If **Skip Blocks and URDs** is selected, RBD 1 and SubSystem 1 will be shown as the precedents for the model.

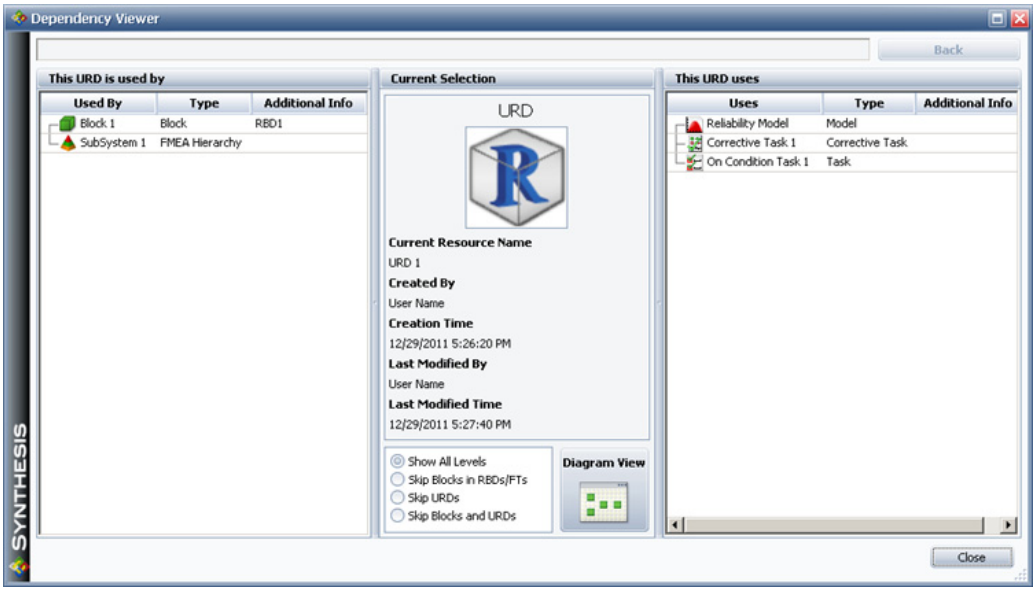
You can click the **Diagram View** button to open the Graphical Dependency Viewer (The next section describes how to use the Graphical Dependency Viewer). The currently selected item will be the main block in the diagram that is created.

Note that 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.

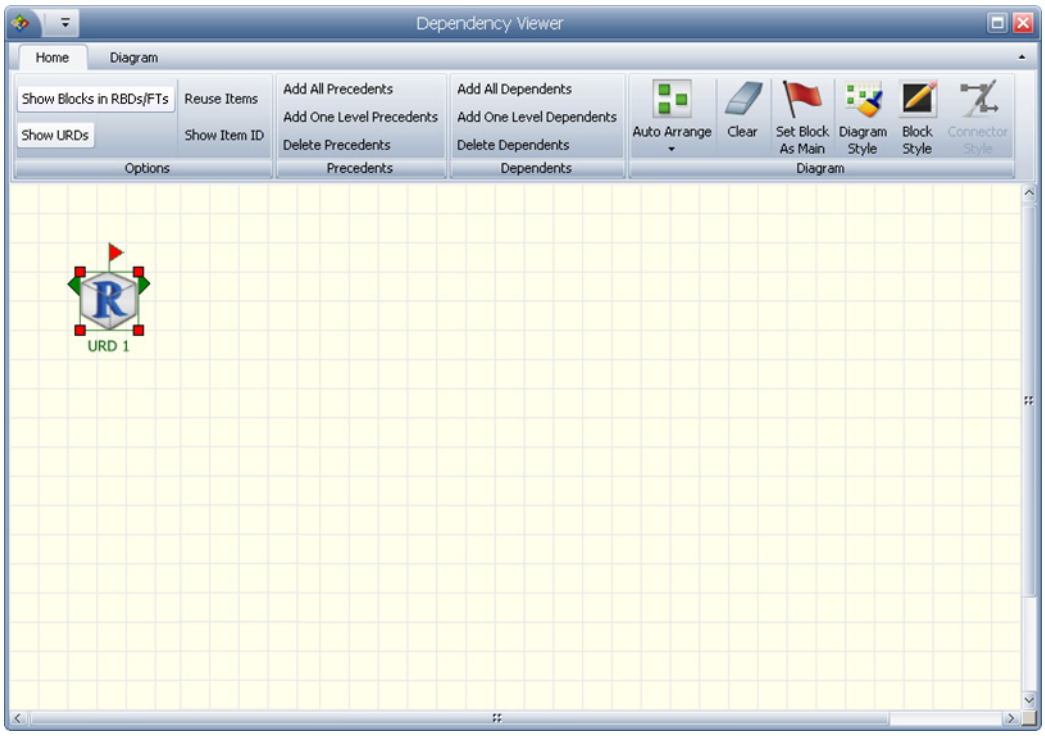
5.4.1 Graphical Dependency Viewer

The Graphical Dependency Viewer, accessed by clicking the **Diagram View** button in the Dependency Viewer, offers a diagram-based alternative to help you understand the connections among resources in your project.

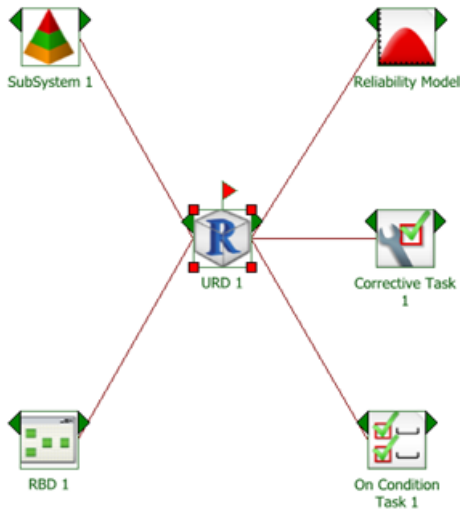
To understand how to use the Graphical Dependency Viewer, consider the example shown in the previous section (*Section 5.4*):



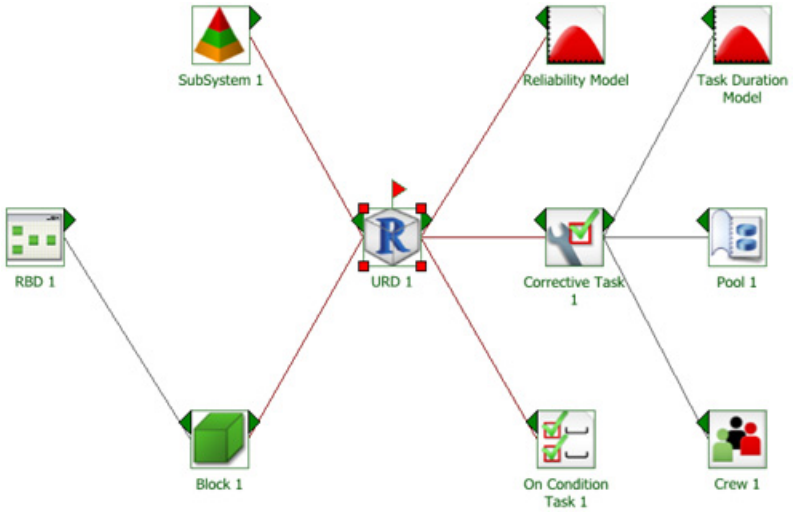
Clicking the **Diagram View** button in the window will yield a diagram view that shows only the main item (in this case, the URD), which is marked with a flag:



To view a graphical representation of the information shown in the Dependency Viewer (i.e., one level of precedents and one level of dependents), choose **Home > Precedents > Add One Level Precedents** and **Home > Dependents > Add One Level Dependents**:



Further, assume that you want to see all levels of precedents and dependents for the URD. Choose **Home > Precedents > Add All Precedents** and **Home > Dependents > Add All Dependents**:

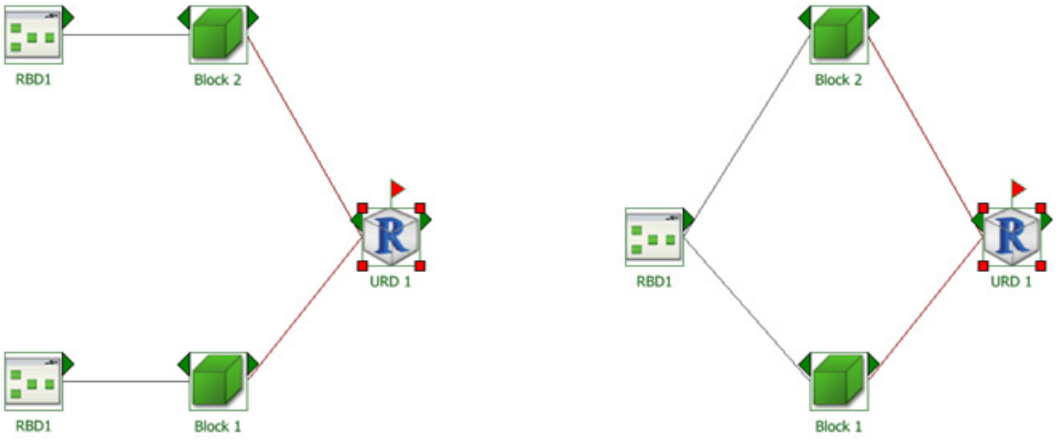


We can now see that Block 1 is used in RBD 1, and that Corrective Task 1 uses a model, a spare part pool and a crew.

You can select any item shown in the diagram and use the Add and Delete commands for precedents and dependents to expand or refine which dependencies are shown. To set a different item as the main item in the diagram, select the item and choose **Home > Diagram > Set Block as Main**.

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 **Home > Precedents > Add All Precedent**s, 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** and **Show URDs** commands to determine whether blocks and/or URDs are shown as intermediate steps between diagrams and models.
- Select **Show Item ID** to display the internal ID assigned to each resource, block and/or diagram.

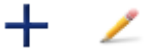
The Diagram tab of the Ribbon offers zoom options and printing options, as well as the ability to copy the diagram or to export it as a graphics file.

5.5 Universal Reliability Definitions (URDs)

Universal reliability definitions (URDs) are used to represent a set of properties that can include failure behavior and maintenance tasks. URDs are resources that can be shared among analyses and can be managed via the Resource Manager.

Note: Depending on where you are using a URD, only those properties that are relevant are applied. For example, in BlockSim, if you apply a URD to a block in an analytical diagram, only the failure model associated with the URD will be applied to the block. If you apply a URD to a block in a simulation diagram, the failure model and all maintenance tasks associated with the URD will be applied. Similarly, if you apply a URD to an item or FMEA record in Xfmea, only the failure model associated with the URD will be applied to the item or FMEA record. If you apply a URD to an item or FMEA record in RCM++/RBI, the failure model and all maintenance tasks associated with the URD will be applied.

The Universal Reliability Definition window allows you to create, view and edit URDs. It can be accessed from the URDs page of the Resource Manager by choosing **Home > Edit > New**, by selecting a URD and choosing **Home > Edit > View** or by double-clicking a URD. It can also be accessed by clicking the **Create New** or **View/Edit** icon in the wizard available from the **URD** field wherever URDs are used.



At the top of the window, you can specify the URD name. By default, URDs are named "URD [increment]" (i.e., the first URD created is URD 1, the next is URD 2 and so on). You can replace this with your own name of up to 150 characters, if desired. 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 Universal Reliability Definition window offers two views of the properties associated with the URD. In both views, information about the creation and last modification of the URD is displayed at the bottom of the window.

The **Trace Usage** icon opens the Dependency Viewer, allowing you to see where the resource is used and what other resources are associated with it.



5.5.1 Hierarchical View

The Hierarchy tab displays information about the URD in a standard hierarchical format. Each resource directly assigned to the URD is shown, along with a subset of its properties.

The table contains the following settings to configure the URD:

- **Model** allows you to 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; once you have specified the model, its category will be displayed as part of the field name.
- **Corrective Task** allows you to choose or create a corrective task for the URD. Corrective maintenance consists of the action(s) taken to restore a failed component to operational status.
- **Scheduled Tasks** allows you to choose or create multiple scheduled tasks for the URD. These tasks may be preventive maintenance, inspections and/or on condition tasks. The 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.

- **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.*



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.



5.5.2 Filtered View

The Filtered tab displays all resources of a specified type that are associated with the URD, either by being assigned to it directly or by being assigned to a resource that is assigned to the URD. Select a resource type in the **Filter By** field to view the relevant resources. You can double-click a resource on the Filtered tab to access its properties window; you cannot, however, change which resources are associated with the URD on this tab.

You can specify the properties columns that are displayed in the table by clicking the **Select Columns** icon.



You can select or clear an individual column's check box to display or hide that column. You can also select or clear the check box for the category to set all of the check boxes for the columns that make up that category, or use the check box in the Available Columns header to set all check boxes in the window.

The columns that are available in this window will vary depending on the type of resource you have selected to view in the Filtered tab; they correspond to the properties that are available for configuring that type of item.

5.6 Models

Models are resources that can be shared among analyses and can be managed via the Resource Manager.

5.6.1 Defining and Viewing/Editing Models

Models can be defined manually or they can be created by publishing results from an analysis in a Synthesis application.

For manually defined models, there are two interfaces that allow you to define or view the model:

- The Model window can be accessed from the Models page of the Resource Manager by choosing **Home > Edit > New**, by selecting a model and choosing **Home > Edit > View** or by double-clicking a model. It can also be accessed by clicking the **View/Edit** icon in the Model Wizard. Finally, it can be accessed by double-clicking the model's hyperlink where it is used or by clicking the **View/Edit** icon in the Preview area that is displayed when the model is selected in an equation in a RENO resource or block. *See Section 5.6.4 on page 189*.
- When you need to define a model from within the BlockSim/RENO Block Properties window or a resource properties window, the Model Wizard can be accessed by clicking the arrow in any field that requires a model,

such as the **Model** field in the Universal Reliability Definition window, the **Task Duration** field in the Maintenance Task window, etc. *See Section 5.6.5 on page 190*.

For models published from an analysis, the properties window provides information about which analysis the model is based on, whether the model is synchronized with the latest analysis and the model parameters and other properties.

- In most Synthesis applications, the model can be published from the data folio or diagram. See the "Control Panel Publishing Page" section in the "Interface" chapter.
- For DOE++, see "Publishing Models from R-DOE Results" in the DOE++ documentation.
- For Lambda Predict, see "Publishing Models from Failure Rate Predictions" in the Lambda Predict documentation.

When you are publishing the model, you can also specify its name and category. To edit a published model, you will return to the original data source, make the necessary changes, recalculate and republish the model so it is once again synchronized with the original analysis. This will cause the model to be changed everywhere that it is used.

5.6.2 Model Categories

There are six categories of models, each serving a specific purpose:

- Probability models represent a likelihood of occurrence. They can take three forms:
  - Reliability models
  - Probability of failure models
  - Event occurrence models

  These models are used by URDs and by switches.
- Duration models represent a length of time. These models are used for task durations and for logistic delays associated with crews and spare part pools.
- Cost models can take two forms:
  - Cost per unit time models are used by tasks, crews and spare part pools for costs that accrue over time.
  - Cost models are used by tasks, crews and spare part pools for costs that arise on a per incident basis.

Note: All categories of models can be used in equations in RENO resources and/or blocks.

5.6.3 Model Types

In addition, each model can be one of three types:

- A **constant** model represents a fixed probability, duration or cost. For duration models and cost per unit time models, you will also need to enter the time units that the model uses.

  In BlockSim, applying a model that uses a fixed probability to a block causes the block to be considered "static." A static block can be interpreted either as a block with a reliability value that is known only at a given time (but the block's entire failure distribution is unknown) or as a block with a fixed reliability (i.e, a reliability that is constant over time). Systems can contain static blocks, time-dependent blocks or a mixture of the two.

- A **distribution** model represents behavior that varies based on factors such as time and/or applied stress. To define a model that uses a distribution:
  - You can select a distribution from the drop-down list and then enter the required parameter(s), which will vary depending on the selected distribution.
  - You can use the Quick Parameter Estimator (QPE), which 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.

  The available distributions and their required inputs are as follows:

  - **Distributions**:
    - **Weibull**: You can select a 2-parameter or 3-parameter Weibull distribution.
      - **Beta** is the shape parameter.
      - **Eta** is the scale parameter.
      - **Gamma** is the location parameter. This field is available only if you have selected a 3-parameter Weibull distribution.
    - **Mixed**: You can select to use a mixed Weibull distribution with 2, 3 or 4 subpopulations. Mixed Weibull distributions are used in cases where multiple subpopulations or failure modes exist within a data set; each subpopulation is represented using a distinct Weibull distribution and the different Weibull distributions are then pieced together to form a continuous function called the *mixed Weibull* distribution (also known as *multimodal Weibull*). For these distributions, you will need to specify parameters for each subpopulation. To do this, select the subpopulation in the **Subpopulation** field, then enter the following values:
      - **Beta** is the shape parameter for the selected subpopulation.
      - **Eta** is the scale parameter for the selected subpopulation.
      - **Portion** is the portion (or percentage) of the total population represented by the selected subpopulation. The sum of the portion values for all subpopulations must be equal to 1.

      Notice that the number of the selected subpopulation appears next to each parameter name to indicate the subpopulation the parameter applies to.
    - **Normal**
      - **Mean** is the location parameter.
      - **Std** (standard deviation) is the scale parameter.
    - **Lognormal**
      - **Mean** is the location parameter.
      - **Std** (standard deviation) is the scale parameter.
    - **Exponential**: You can select a 1-parameter or 2-parameter exponential distribution. You have the option of defining the exponential parameter as lambda (i.e., failure rate) or mean time (i.e., MTBF). This option can be set on the Calculations page of the Application Setup.
      - **Mean Time** is a location parameter.
      - **Lambda** is the scale parameter.
      - **Gamma** is a location parameter. This field is available only if you have selected a 2-parameter exponential distribution.

- **G-Gamma**
  - **Mu** (exp(mu)) is the scale parameter.
  - **Sigma** is a shape parameter.
  - **Lambda** is a shape parameter. Please note that for any fixed value of lambda, the generalized-gamma distribution is a log-location-scale distribution.
- **Gamma**
  - **Mu** is the scale parameter.
  - **K** is the shape parameter.
- **Logistic**
  - **Mean** is the location parameter.
  - **Std** (standard deviation) is the scale parameter.
- **Loglogistic**
  - **Mean** is the location parameter.
  - **Std** (standard deviation) is the scale parameter.
- **Gumbel**
  - **Mean** is the location parameter.
  - **Std** (standard deviation) is the scale parameter.

- **Life-Stress Relationships with Distributions**: These models describe behavior based on three variables: reliability, stress and time. They can be used for contained load sharing blocks to describe how the life distribution changes as the load changes. Such models can also be used for standard blocks. For simulation purposes, the use stress serves as an additional block-specific parameter on the model; the block is considered to use only that stress level all the way through the simulation.
  - **Arrhenius-Weibull** sets the life-stress relationship to Arrhenius and the distribution to Weibull.
    - **Beta** is the Weibull distribution shape parameter.
    - **B** is the first Arrhenius parameter.
    - **C** is the second Arrhenius parameter.
    - **Use Stress** is the stress level at which the model will be used and evaluated.
  - **Arrhenius-Lognormal** sets the life-stress relationship to Arrhenius and the distribution to lognormal.
    - **Log-Std** is the lognormal distribution scale parameter.
    - **B** is the first Arrhenius parameter.
    - **C** is the second Arrhenius parameter.
    - **Use Stress** is the stress level at which the model will be used and evaluated.
  - **Arrhenius-Exponential** sets the life-stress relationship to Arrhenius and the distribution to exponential.
    - **B** is the first Arrhenius parameter.
    - **C** is the second Arrhenius parameter.
    - **Use Stress** is the stress level at which the model will be used and evaluated.
  - **Eyring-Weibull** sets the life-stress relationship to Eyring and the distribution to Weibull.
    - **Beta** is the Weibull distribution shape parameter.
    - **A** is the first Eyring parameter.
    - **B** is the second Eyring parameter.
    - **Use Stress** is the stress level at which the model will be used and evaluated.

- **Eyring-Lognormal** sets the life-stress relationship to Eyring and the distribution to lognormal.
  - **Log-Std** is the lognormal distribution scale parameter.
  - **A** is the first Eyring parameter.
  - **B** is the second Eyring parameter.
  - **Use Stress** is the stress level at which the model will be used and evaluated.
- **Eyring-Exponential** sets the life-stress relationship to Eyring and the distribution to exponential.
  - **A** is the first Eyring parameter.
  - **B** is the second Eyring parameter.
  - **Use Stress** is the stress level at which the model will be used and evaluated.
- **IPL-Weibull** sets the life-stress relationship to the inverse power law and the distribution to Weibull.
  - **Beta** is the Weibull distribution shape parameter.
  - **K** is the first IPL parameter.
  - **n** is the second IPL parameter.
  - **Use Stress** is the stress level at which the model will be used and evaluated.
- **IPL-Lognormal** sets the life-stress relationship to the inverse power law and the distribution to lognormal.
  - **Log-Std** is the lognormal distribution scale parameter.
  - **K** is the first IPL parameter.
  - **n** is the second IPL parameter.
  - **Use Stress** is the stress level at which the model will be used and evaluated.
- **IPL-Exponential** sets the life-stress relationship to the inverse power law and the distribution to exponential.
  - **K** is the first IPL parameter.
  - **n** is the second IPL parameter.
  - **Use Stress** is the stress level at which the model will be used and evaluated.

- 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 information on variables, see Section 5.7 on page 201.*
  - 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.

5.6.4 Model Window

The Model window allows you to define and/or view the properties for a manually defined model. The following settings are available on the Main tab to configure the model:

- The **Model Name** field allows you to specify a name for the model. You can replace the default name with your own name of up to 150 characters, if desired.
- In the **Model Category** field, you must specify what category of behavior the model represents (i.e., reliability, probability of failure, event occurrence, duration, cost or cost per unit time), thereby restricting where and how it can be used. Once a model has been created, you cannot change its category.

- In the **Model Type** field, specify whether the model is constant, varies with time (i.e., a distribution) or is based on a variable (i.e., dynamic).

  For constant models, enter the fixed probability, duration or cost value. The type of value that you enter is dependent upon the model category. For duration models and cost per unit time models, you will also need to enter the time units that the model uses.

  For distribution models, select the distribution and specify its parameters. You can:

  - Select a distribution from the drop-down list and then enter the required parameters, which will vary depending on the selected distribution. *For a list of the available distributions and their required inputs, see Section 5.6 on page 185.*
  - Click the **QPE** icon to use the Quick Parameter Estimator (QPE), which helps 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.

  For dynamic models, specify the variable that the model is based on. For duration and cost per unit time models, the time unit is the default unit defined 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.

Information about the creation and last modification of the model is displayed at the bottom of the tab.

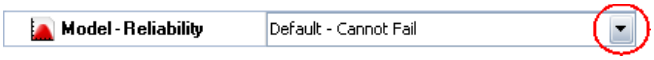
The **Trace Usage** icon opens the Dependency Viewer, allowing you to see where the resource is used and what other resources are associated with it. *See Section 5.4 on page 180.*



The Identifiers tab of the Model window allows you to enter additional identifying information that can be used to search for this resource.

5.6.5 Model Wizard

When you need to assign a model, you can click the field that requires the model as input and click the arrow that appears in the field, as shown next.



This opens the Model Wizard, which allows you to select an existing model or create a model on the fly; it retains your most recent settings so that you do not have to re-enter commonly used settings.

You do not have to specify the model category when you create a model using the Model Wizard; it is determined by where the model is used, as follows:

- Models assigned to URDs are reliability models.
- Models assigned to switches are reliability models.
- Models used for task durations and for logistic delays associated with crews and spare part pools are duration models.
- Models used by tasks, crews and spare part pools for costs that accrue over time are cost per unit time models.
- Models used by tasks, crews and spare part pools for costs that arise on a per incident basis are cost models.

- Models assigned directly to blocks in BlockSim RBDs and to system hierarchy items in Xfmea/RCM++/RBI are reliability models.
- Models assigned directly to blocks in BlockSim fault trees and to FMEA records in Xfmea/RCM++/RBI (i.e., functions, failures or causes) are probability of failure models.

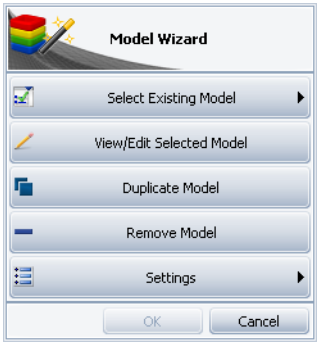
If you need to create a model of a different category, you must create the model via the Model window. *See Section 5.6.4 on page 189*.

The Main page provides access to all available functions within the wizard. If a model has not been assigned in the field, you can select an existing model or create a new one.



If a resource has already been assigned, you can:

- Select another existing model to replace the current model. *See Section 5.6.5.1 on page 192*.
- Create a duplicate of the current model and assign the new copy in the field (e.g., if you want to use the same/similar properties but remove the association with the original model that may be used in a different way elsewhere in the project). *See Section 5.6.5.2 on page 192*.
- View/edit the current model. *See Section 5.6.5.3 on page 194*.
- Remove the model from the current location (it will still be available for other records within the current project). *See Section 5.6.5.4 on page 194*.
- Use the Settings page to specify what is shown in the wizard. *See Section 5.6.5.5 on page 194*.

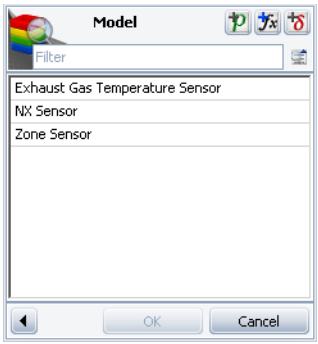


Most functions are also available as icons on the Select page of the wizard, which is the default view except when no model has yet been assigned. You can, however, always return to the Main page of the wizard by clicking the **Back** arrow in the bottom left corner of the Model Wizard.

5.6.5.1 Selecting an Existing Model

The Select page provides a list of the models of the specified type that currently exist in the project. This is the default view once a model has been assigned, and can also be accessed by clicking the **Select Existing** button or icon.





You can filter the models that are displayed in the list by entering text in the **Filter** field (marked with a magnifying glass); clear the filter by clicking the **Clear Filter** icon in the field (red x). Select a model and click **OK** (or double-click it) to assign the model to the current location. If a model was already assigned, the newly selected model will replace it.

You can click the **Select Resource** icon to open the Select Resource window, allowing you to select an existing model. The Select Resource window displays more information about each resource and offers a wider range of filtering tools.

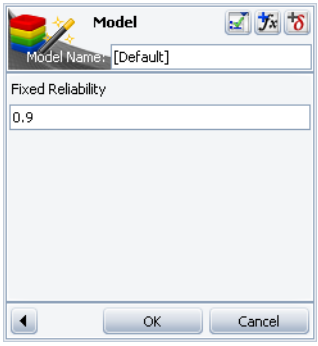


5.6.5.2 Creating a Model

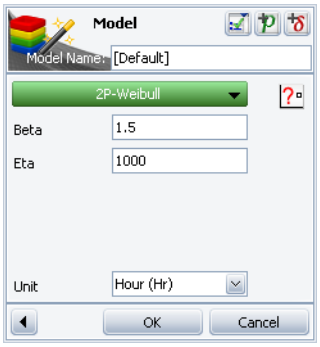
To create a new model from the wizard, click the **New Constant**, **New Distribution** or **New Dynamic** button or icon.



The New Constant page allows you to enter a fixed probability, duration or cost value for a constant model. The type of value that you enter is dependent upon the model category. For duration models and cost per unit time models, you will also need to enter the time units that the model uses.



The New Distribution page allows you to select the distribution and specify its parameters.

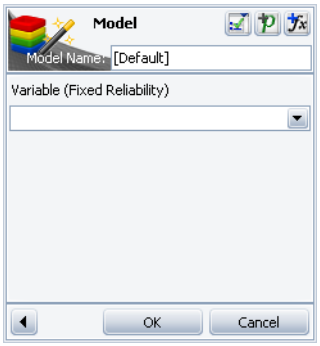


You can:

- Select a distribution from the drop-down list and then enter the required parameters, which will vary depending on the selected distribution. *For a list of the available distributions and their required inputs, see Section 5.6 on page 185.*
- Click the **QPE** icon to use the Quick Parameter Estimator (QPE), which helps 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.



The New Dynamic page allows you to choose the variable that the model is based on. For duration and cost per unit time models, the time unit is the default unit defined 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.



Once you have specified the properties for the new model, click **OK** to create the model and assign it to the current location.

Once you have created a given category of model via the Model Wizard, your settings for that kind of model will be retained and shown the next time you create that same kind of model. For example, the first time you create a reliability model, you will choose whether to create a constant model or a distribution model. If you create a distribution model, then the next time you create a reliability model, you will automatically see the options for a distribution model.

If a model has already been assigned and you wish to remove the association with the original model but use the same (or similar) properties for the current location, you can click the **Duplicate** button on the Main page of the wizard. The new copy will then be assigned in place of the original.

5.6.5.3 Editing a Model

If a model has already been assigned, you can view and/or edit its properties in the Model window by clicking the **View/Edit** button or icon.



IMPORTANT: Any changes you make to the model will apply everywhere that the model is used. This includes analyses in other Synthesis applications.

5.6.5.4 Removing a Model

If a model has already been assigned, you can remove the model from the current location by clicking the **Remove** button or icon.

The model will still be available to be assigned to other records in the current project. If a model is not assigned to any other locations within the project and you want to completely remove it from the database, choose **Project > Synthesis > Resource Manager**.

5.6.5.5 Using the Settings Page

On the Settings page of the wizard, you can specify the following:

- Whether you want to see all models in the wizard, only local models (i.e., those models that are available for use only in the current project, which is the default setting for resources) or only global models (i.e., those models that are available for use in any project within the database).

  Note: When you are associating a model with a global URD, only global models will be shown in the wizard.

- The maximum number of models that will be displayed in the wizard. Larger numbers of models may take a longer time to load. If not all models are shown in the wizard, you can click the **Select Resource** icon to open the Select Resource window, which displays all existing models.



- Whether model parameters should be shown. Hiding parameters may reduce time to load large numbers of models.
- The precision used in the wizard, which is the number of decimal places shown when parameters are displayed.
- The tolerance used in the wizard, which is the point at which the software will switch to scientific notation when parameters are displayed. 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.

5.6.6 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).

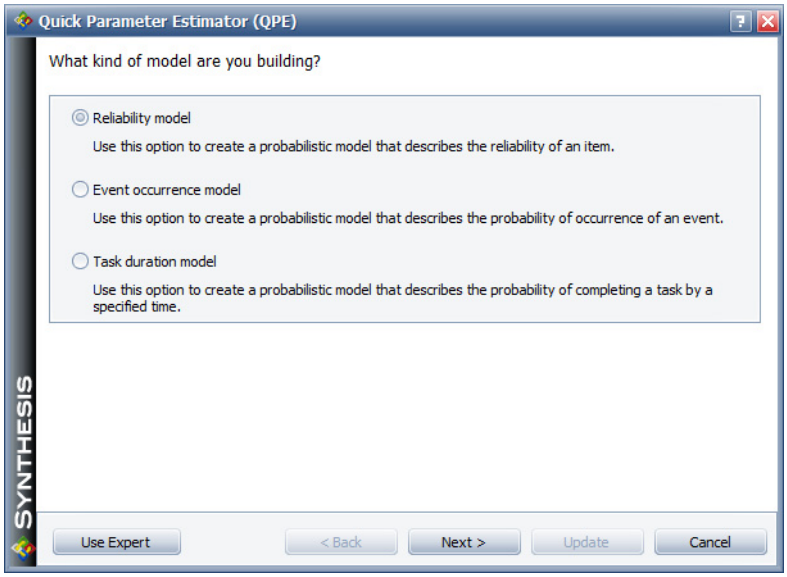
The QPE includes a Wizard view and an Expert view, described below. You can toggle between the different views by clicking the **Use Expert** or **Use Wizard** button at the bottom-left corner of the window.

- 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.

5.6.6.1 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:

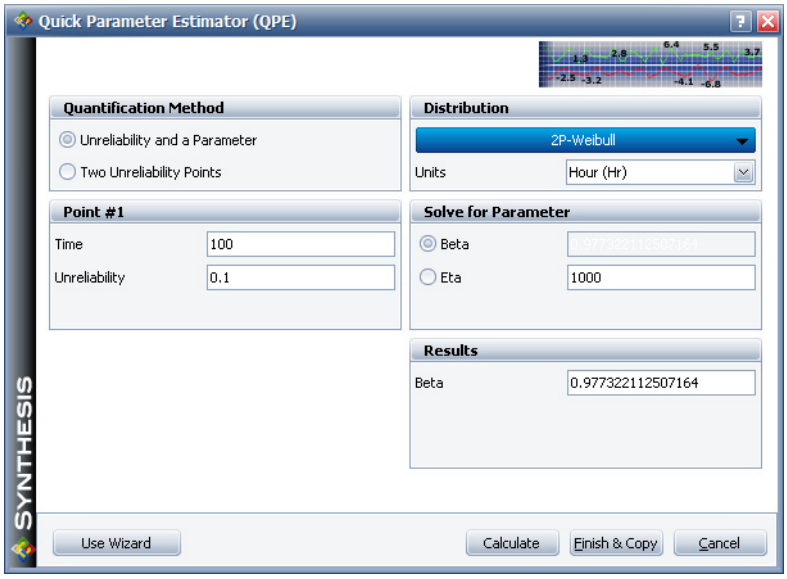
- Best-case, worst-case and most likely estimates for how long it will take to complete the task

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.

5.6.6.2 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.

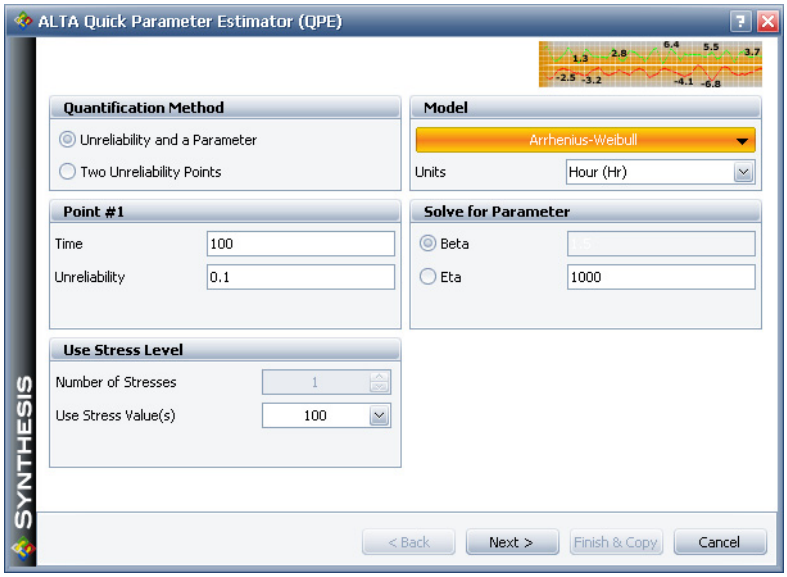
- Click **Calculate** to solve for the unknown parameter(s).
  - If you opened the QPE from an analysis folio or utility, you may click the **Update** button if you wish to update that window using the selected distribution, calculated parameters and selected time units.
  - In Weibull++/ALTA, if you opened the QPE from the Ribbon, you may click the **Finish & Copy** button if you wish to copy the results to the Windows Clipboard.

5.6.6.3 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)**.



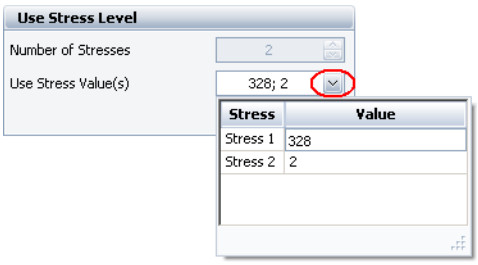
The window will appear as shown next.



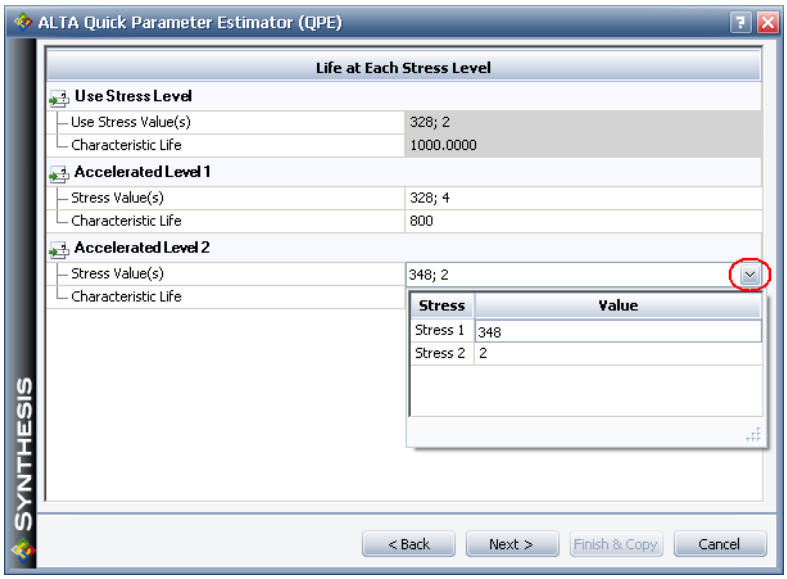
You can also open the ALTA QPE from the ALTA standard folio (by leaving the folio's data sheet empty and clicking **Calculate**), as well as the ALTA Monte Carlo and SimuMatic utilities.

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.

5.6.7 Published Model Properties

For models that are published from a Synthesis analysis (as opposed to being defined manually), the properties window is intended to help users determine which analysis the model is based on, whether the model is synchronized with the latest analysis and whether the model is appropriate for a particular situation in another Synthesis analysis.

When you are creating a new published model, the window is called **Publish Model**. When you are viewing the properties for an existing model, it is called **View Model**. In either case, the following properties and tools are available.

- **Source** is the Synthesis analysis that the model was published from.
- **Application** is the Synthesis application (e.g., Weibull++, BlockSim, etc.) 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 originating 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."
- **Model Definition**: For analytical models that are published from BlockSim, the "model" is actually a direct link to the system reliability equation for the diagram and therefore this area will simply display "Diagram Analysis." (See Publishing Fitted and Analytical Models in the BlockSim documentation for more information.) For all other published models, this area displays the specific model parameters and other relevant details. Note that:
  - **PNZ** stands for percent non-zero and represents the proportion of the population with non-zero failure times. A value of 1 indicates that there are no zero failure times in the data set (which is the most common scenario in life data analysis); a decimal value indicates that the data set includes zero failure times (such as out-of-the-box failures, for example). The model parameters are calculated based on non-zero failure times, and the PNZ value is used as a multiplier when calculating certain metrics (e.g., reliability, unreliability) based on the model.
  - Certain model parameters may have an associated **Unit**. Units are defined for the database via the Define Units window (**File > Manage Repository > Manage Units**).
  - For an explanation of the different **Model Categories**, see Section 5.6 on page 185.
- **Identifiers** are the specific identifying details that can be used to search for the model when it is needed in another analysis. For most applications, this information is defined on the Identifiers page of the control panel for the analysis that the model is published from. For Lambda Predict, these fields are managed from the Model tab in the Properties panel.

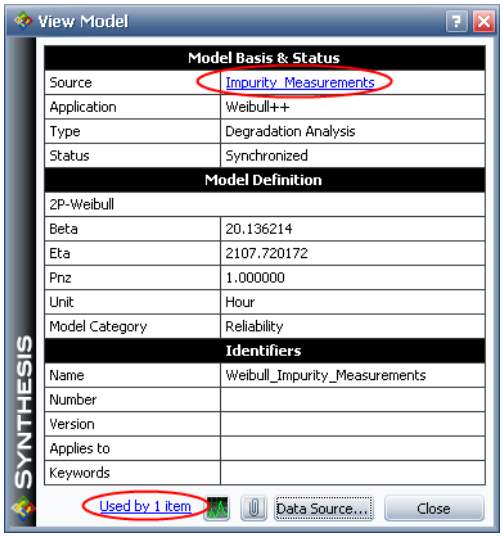
Once the model has been published, a link at the bottom of the window indicates how many times the model is being used in another Synthesis analysis or resource. To explore the specific details in the Dependency Viewer, you can click either the link or the **Trace Usage** icon. *See Section 5.4 on page 180*.



If desired, you can also use the **Attachments** icon to view/create any links or attachments that will accompany the model. For example, this could include any additional documentation that might help users determine when and how the model might be appropriate to use.



Finally, if you have the ability to view the original analysis that was used to publish the model (i.e., if the required Synthesis application is activated on your computer and your user account has permissions to access the analysis), a **Data Source** button will also be displayed at the bottom of the window and the **Source** field at the top of the window will also be configured as a link, as shown next. This gives you the option to explore the original analysis in more detail before you select to use the model, or to make changes to the analysis if necessary.



5.7 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. When all simulations are complete, the variable always returns to its initial starting value. 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 Section 5.6 on page 185*), 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.

5.7.1 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:

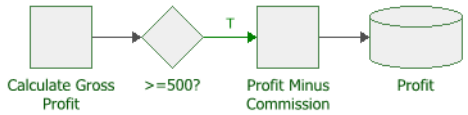
- **Reset After Each Simulation** (i.e., reset after a single pass through the flowchart)
- **Reset After Each Run** (i.e., reset after each set of simulations)
- **Reset After Analysis** (i.e., reset after each set of runs)
- **Never Reset**

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.

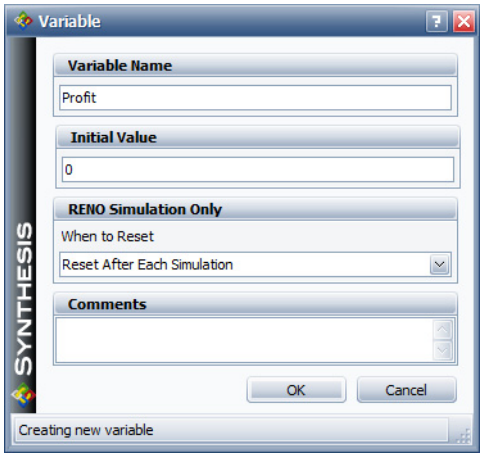
5.7.1.1 Example

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.

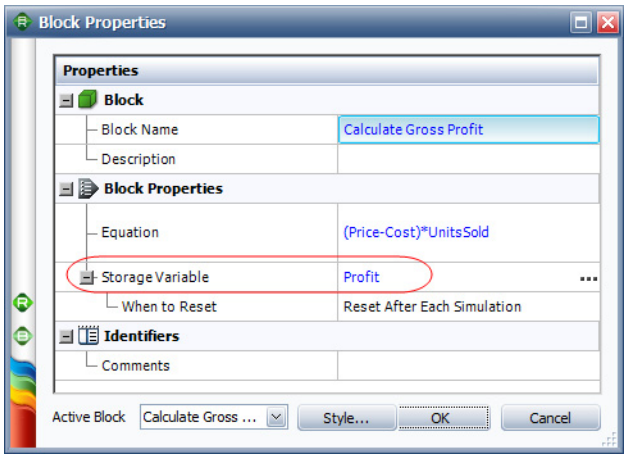
In the following configuration, the first block calculates the gross profit from a sale, and then stores its output into a variable called “Profit.” The conditional block determines whether the gross profit exceeds 500. If the output is true, the third block deducts the sale's commission from the profit, and overwrites the stored value with the new output.



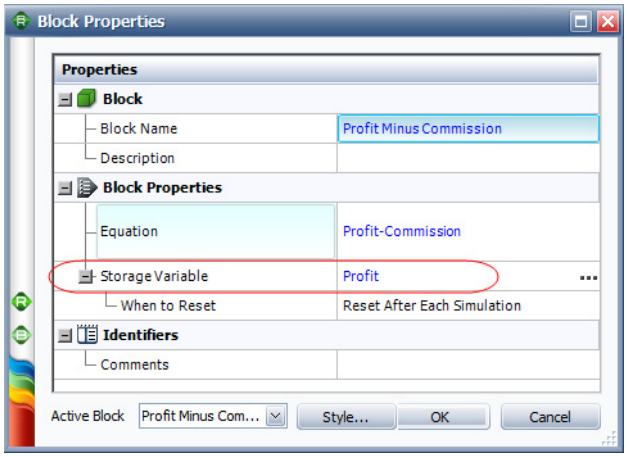
The following picture shows the properties window of the variable called “Profit.” The initial value of this variable is set to 0.



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.



Weibull++ Standard Folios 6

The objective of life data analysis is to fit a statistical distribution to failure time data in order to understand the reliability performance of a product over time or to make projections about future behavior. From the analysis, you can estimate several life characteristics, such as probability of failure, reliability, mean life or failure rate.

WEIBULL++ Only

The Weibull++ standard folio for life data analysis is the most basic folio in the application. This chapter describes how to use the folio for performing an analysis, including:

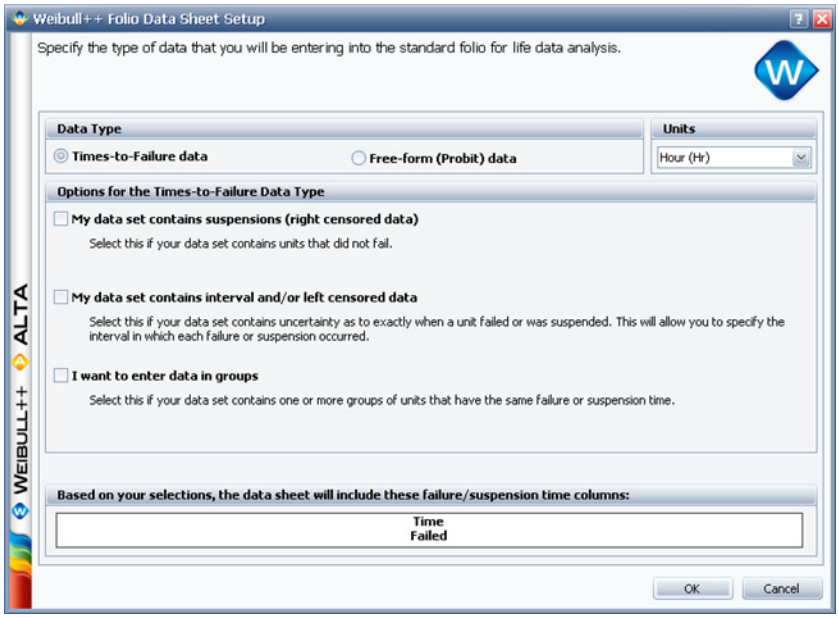
- Setting up a Weibull++ standard folio - Section 6.1 (p. 206).
- Entering different types of life data - Section 6.2 (p. 207).
- Analyzing life data, including:
  - Using the control panel - Section 6.3 (p. 210).
  - Generating life data analysis plots - Section 6.4 (p. 217).
  - Performing reliability calculations via the Quick Calculation Pad (QCP) - Section 6.5 (p. 219).
- Example of a simple life data analysis - Section 6.6 (p. 223).
- Additional analysis tools, including:
  - Choosing a life distribution via the Distribution Wizard - Section 6.7.1 (p. 226).
  - Extracting subsets of data via the batch auto run process - Section 6.7.2 (p. 229).
  - Evaluating the fit of a distribution with the Goodness of Fit Results tool - Section 6.7.3 (p. 231).
  - Grouping together data points via the Auto Group Data tool - Section 6.7.4 (p. 232).
- Special analysis methods, including:
  - Mixed Weibull analysis - Section 6.8.1 (p. 233).
  - Bayesian-Weibull analysis - Section 6.8.2 (p. 236).
  - Competing Failure Modes (CFM) analysis - Section 6.8.3 (p. 240).
  - What-if analysis, including:
    - Altering the parameters of a model - Section 6.8.4 (p. 244).
    - Using the Specify Points feature to adjust a rank regression analysis - Section 6.8.5 (p. 245).
    - Performing calculations with no data - Section 6.8.6 (p. 246).
    - Using the Fill Median Ranks feature to obtain an unreliability estimate of a failure - Section 6.8.7 (p. 248)

6.1 Weibull++ Standard Folio Setup

To add a Weibull++ standard folio to a project, choose **Insert > Folios > Weibull++ Standard Folio** or right-click the **Weibull++ Folios** folder in the project explorer and choose **Add Weibull++ Standard Folio** on the shortcut menu.



A setup window for configuring your folio will be displayed, as shown next:



The first step is to specify the type of data you will be working with. There are two types: times-to-failure data or free-form data. In addition, times-to-failure data may contain three forms of censoring: suspensions (right censored), interval censored or left censored data. Any of the times-to-failure data types can also be entered as grouped data.

The data type you select determines the data entry columns that appear in the data sheet. Notice that when you select a data option, the preview area at the bottom of the setup window shows you the names of the data entry columns that will appear in the data sheet. *To learn more about each data type, see Section 6.2 on page 207.*

6.1.1 Time Units

The next step is to specify the time units appropriate for your data. Time units may be measured in mileage, distance, weight, etc. The appropriate columns in the data sheet will be automatically configured for the time units you select. If you later wish to change the time units of an existing data sheet, click the **Change Units** icon on the Main page of the control panel of the data sheet.



This opens the Change Units window, which gives you several options for converting the time units of the existing data. *See page 113 in Chapter 3*.

What's Changed? In the Synthesis version, you now 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. See *See page 39 in Chapter 2*.

6.2 Data Types

This section describes the different types of data you can use in a life data analysis. In Weibull++, there are two types of life data: times-to-failure data and free-form data. In addition, times-to-failure data may contain three forms of censoring: suspensions, interval censored or left censored data. The following sections describe each data type and show you how to enter them in a Weibull++ standard folio data sheet.

Tip: In a Weibull++ standard folio, the data type you select determines the data entry columns that appear in the data sheet. You can change the data entry columns for an existing data sheet anytime by choosing **Life Data > Format and View > Alter Data Type**.

A Note About Subset IDs

By default, Weibull++ standard folio data sheets include a **Subset ID** column for logging any pertinent information or comments about the data. You can choose to add a second subset ID column (labeled "Subset ID 2" by default) by choosing **Life Data > Format and View > Add/Remove Columns**.



The information in the subset ID columns do not affect the calculations in the folio unless you select to use analysis methods that categorize data based on their subset IDs. The following is a list of the affected analysis methods in Weibull++. Note that in these cases, only the information in the first subset ID column (labeled "Subset ID 1" by default) affects the analysis.

- Competing failure modes (CFM) analysis. *See Section 6.8.3 on page 240*.
- Extracting subsets of data via the batch auto run process. *See Section 6.7.2 on page 229*.
- Grouping together data points via the Auto Group Data tool. *Section 6.7.4 on page 232*.
- Using the Specify Points feature to adjust a rank regression analysis. *See Section 6.8.5 on page 245*.

In the following sections, the subset ID column in the examples is used for explanatory purposes, and is not intended to demonstrate how the column would normally be used.

6.2.1 Times-to-Failure Data

Times-to-failure data sets, also known as *complete data*, are obtained by recording the exact times when the units failed. For example, if we tested five units and they all failed, and we recorded the time when each failure occurred, we would then have complete information as to the time of each failure in the sample.

To use this data type, select the **Times-to-Failure data** check box on the setup window. Your data sheet will have a **Time Failed** column for recording the times-to-failure. The example shown next displays a standard folio data sheet where all the units in the sample failed. The exact time of each failure is recorded.

	Time Failed (Hr)	Subset ID
1	12	failed unit
2	24	failed unit
3	32	failed unit
4	56	failed unit
5	80	failed unit
6	104	failed unit

If multiple units failed at the same time, you can choose to enter the data in groups. To do this, select the **I want to enter data in groups** check box on the setup window. This adds a third column to the data sheet, as shown next. In this example, the units that failed at the same time are grouped together. The **Number in State** column indicates the

number of failed units in that group (the software will automatically enter a 1 if you leave this cell blank) and the **State End Time** column indicates the exact time the units in each group failed.

	Number in State	State End Time (Hr)	Subset ID
1	1	12	1 failed unit
2	3	24	3 failed units
3	1	32	1 failed unit
4	5	56	5 failed units
5	1	80	1 failed unit
6	10	104	10 failed units

6.2.2 Times-to-Failure with Suspensions (Right Censored Data)

The term *suspension* describes the units did not fail during the observation period. Such units are also known as right censored data. For example, if we tested five units and only three had failed by the end of the test, the observed operating time of the two units that did not fail would be referred to as right censored data. The term *right censored* means that the event of interest (i.e., the time-to-failure) is to the right of our data point on the time scale.

To use this data type, select the **Times-to-Failure** data check box and the **My data contains suspensions (right censored data)** check box on the setup window. Your data sheet will have a **State F or S** column for recording whether the data point is a failure (F) or suspension (S). In the example shown next, a test was conducted and the first five units in the sample failed but the sixth unit was still operational when the test terminated at 190 hours.

	State F or S	Time to F or S (Hr)	Subset ID
1	F	75	failed unit
2	F	93	failed unit
3	F	100	failed unit
4	F	120	failed unit
5	F	185	failed unit
6	S	190	suspended unit

Tip: If you enter the time as a negative number (e.g., -190), Weibull++ will automatically assign it as a suspension. This time-saving feature applies only to data sheets that have been configured to support right censored data.

When entering this type of data in groups, units that have the same state (i.e., failed or suspended) and have the same value in the State End Time column can be grouped together, as shown next.

	Number in State	State F or S	State End Time (Hr)	Subset ID
1	1	F	75	1 failed unit
2	3	F	93	3 failed units
3	1	F	100	1 failed unit
4	5	F	120	5 failed units
5	1	F	185	1 failed unit
6	10	S	190	10 suspended units

6.2.3 Times-to-Failure with Interval and Left Censored Data

The term *interval censored data* describes the uncertainty as to the exact time the unit failed within an interval. This type of data frequently comes from tests or situations where the units are not constantly monitored. For example, if we are running a test on five units and inspecting them every 100 hours, we only know that a unit failed or did not fail between inspections. Specifically, if we inspect a certain unit at 100 hours and find it operating, and then perform another inspection at 200 hours to find that the unit is no longer operating, then the only information we have is that the unit failed at some point in the interval between 100 and 200 hours.

Left censored data is a special case of interval censored data. In left censored data, the interval is between time = 0 and some inspection time.

To use this data type, select the **Times-to-Failure data** check box and the **My data set contains interval and/or left censored data** check box on the setup window. In the example shown next, the first unit was observed at 150 hours and found to be failed. The second unit was still operating at 150 hours but failed at some point prior to the next observation at 300 hours, and so on.

	Last Inspected (Hr)	State End Time (Hr)	Subset ID
1	0	150	left censored unit
2	150	300	interval censored unit
3	300	450	interval censored unit
4	450	600	interval censored unit
5	600	750	interval censored unit
6	750	900	interval censored unit

Note: Although this example shows a scenario in which the inspections occur at regularly scheduled intervals, Weibull++ does not require the start time of the interval to be equal to the end time of the previous interval.

When entering this type of data in groups, all units that failed within the same interval are considered to be part of the same group.

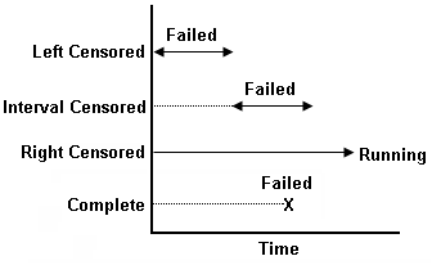
	Number in State	Last Inspected (Hr)	State End Time (Hr)	Subset ID
1	3	0	150	3 left censored units
2	1	150	300	1 interval censored unit
3	7	300	450	7 interval censored units
4	3	450	600	3 interval censored units
5	1	600	750	1 interval censored unit
6	1	750	900	1 interval censored unit

6.2.4 Times-to-Failure with All Types of Data

Times-to-failure data with different types of censoring can be added to a single data sheet. The data may be entered in groups or ungrouped. The following is an example of a grouped data set with complete, suspended, interval and left censored data.

	Number in State	Last Inspected	State F or S	State End Time	Subset ID
1	2	0	F	24	2 left censored units
2	10	24	F	48	10 interval censored units
3	1	48	F	96	1 interval censored unit
4	1	96	F	96	1 failed unit
5	2	96	F	120	2 interval censored units
6	5	120	S	150	5 suspended units

The next illustration summarizes the differences between complete data and the different types of censored data.



6.2.5 Free-Form (Probit) Data

The free-form data type is used to analyze the relationship between an independent variable (x-axis data) and the percentage of failures (y-axis data) in response to the variable. The y-axis data are treated as probability units, or probits, and they show how the percentage of failures increase in response to a variable (e.g., time, distance, stress level, etc.). For example, you could use the free-form data type to record the Y percentage of automobiles that will need repair after X number of miles, or record the Y percentage of product failures at X degrees of temperature.

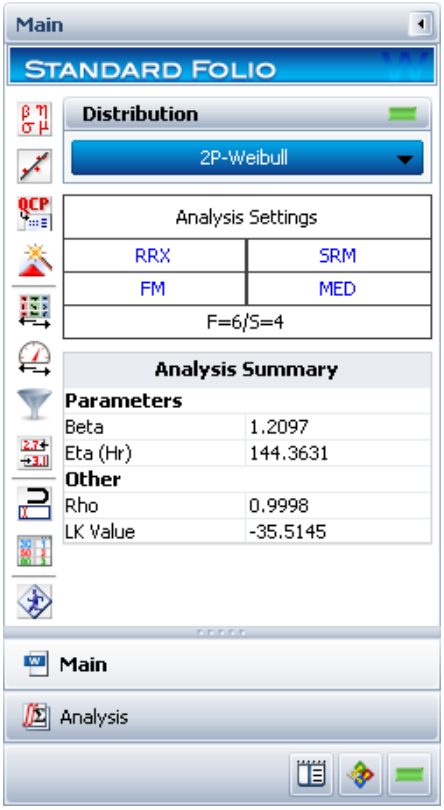
To use this data type, select the **Free-form (Probit) data** check box on the setup window. In the data sheet, enter the values for the independent variable in the **X-Axis value** column and enter the cumulative percentage of failures in the **Y-Axis value** column. The following is an example of a data set that shows the effect of time (x-axis data) on the probability of failure (y-axis data) of a product.

	X-Axis value (Hr)	Y-Axis value,(%)	Subset ID
1	100	28.9	at 100 hr, the prob. of failure is 28.90%
2	200	31.38	at 200 hr, the prob. of failure is 31.38%
3	300	49.99	at 300 hr, the prob. of failure is 49.99%
4	400	68.61	at 400 hr, the prob. of failure is 68.61%
5	500	87.05	at 500 hr, the prob. of failure is 87.05%
6	600	89.2	at 600 hr, the prob. of failure is 89.20%

6.3 Weibull++ Standard Folio Control Panel

The Weibull++ standard folio control panel allows you to configure the analysis settings for the data sheet and view/access the results. It consists of multiple pages, each containing options for performing particular tasks. This section focuses on the Main page of the Weibull++ standard folio control panel, which contains most of the tools you will need to perform life data analysis. *For more information about the control panel in general, see page 114 in Chapter 3*.

The Main page of the Weibull++ standard folio control panel is shown next.



The Main page includes the following settings:

- The **Distribution** drop-down list allows you to select a life distribution to model the failure behavior of the data. There are nine distributions and three special analysis methods to choose from. Any distribution can be used to analyze a data set; however, if the distribution does not match the expected failure rate behavior of the data, the accuracy of the predictions will be compromised.
  - The nine distributions are Weibull, exponential, lognormal, normal, generalized gamma, gamma, logistic, loglogistic and Gumbel.[1]

> **Tip:** If you are not certain which distribution would provide the best fit for your data, you can use the Distribution Wizard to guide you through the selection process. *See Section 6.7.1 on page 226*.

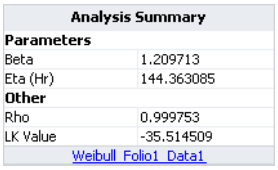
  - The special analysis methods are:
    - **Mixed Weibull** analysis is used for analyzing data sets that reflect different trends in the failure behavior. This method may be useful when dealing with failure modes that cannot be assumed to be independent (i.e., the occurrence of one failure mode affects the probability of occurrence of the other mode) and/or when it is not possible to identify the failure mode responsible for each individual data point. To use mixed Weibull analysis, you will need to specify how many subpopulations exist in the data set (2, 3 or 4). The software will determine the proportion of units that fall under each subpopulation, and calculate the Weibull parameters of those subpopulations. The overall reliability of the full data set is then calculated by taking the sum of the proportional reliability contributions of each subpopulation. *See Section 6.8.1 on page 233*.
    - **Bayesian-Weibull** analysis is based on the concepts of Bayesian statistics. It is typically used in situations when you have a small sample size and a strong prior knowledge of what the failure behavior is likely to be. *See Section 6.8.2 on page 236*.
    - **Competing Failure Modes (CFM)** analysis is used for analyzing data sets that contain failures due to different types of failure modes. The occurrence of one failure mode results in a failure for the product; hence, the failure modes are said to "compete" to cause the failure. This analysis can be used when the failure modes are assumed to be statistically independent (i.e., the occurrence of one failure mode does not affect the probability of occurrence of the other mode). To use CFM analysis, you will need to categorize the times in the data set according to failure modes and then choose one type of life distribution to model the failure behavior of each mode separately. The overall reliability is computed by obtaining the reliability due to each failure mode and then multiplying together the reliability values. *See Section 6.8.3 on page 240*.
- The **Analysis Settings** area provides a quick summary of the settings that will be used to fit the distribution. Click the options displayed in blue text on the control panel to toggle between the available options.

[1] The ReliaWiki resource portal has information on the statistical characteristics of each distribution at: http://www.ReliaWiki.org/index.php/Life_Distributions.

The following table serves as a guide to the abbreviations. These settings are also available on the Analysis page of the control panel (*for a description of these settings, see Section 6.3.2 on page 215*). Note that the available options will vary depending on the data type, distribution and analysis method you have selected.

Analysis Method	Rank Method	Confidence Bound Method	Rank Method
Rank Regression on X (RRX)	Standard Ranking Method (SRM)	Fisher Matrix Confidence Bounds (FM)	Median Ranks (MED)
Rank Regression on Y (RRY)	ReliaSoft Ranking Method (RRM)	Likelihood Ratio Confidence Bounds (LRB)	Kaplan-Meier Estimator (K-M)
Non-Linear Rank Regression (NLRR)		Bayesian Confidence Bounds (BSN)	
Maximum Likelihood Estimation (MLE)		Beta-Binomial Confidence Bounds (BB)	

- The **Analysis Summary** area displays the calculated results. The following example shows the calculated parameters for a particular data set.



Analysis Summary	
Parameters	
Beta	1.209713
Eta (Hr)	144.363085
Other	
Rho	0.999753
LK Value	-35.514509
Weibull_Folio1_Data1	

You can click anywhere within the **Analysis Summary** area to open the Results window, which displays the calculated results in a worksheet. From the Results window, you can edit, copy or print the results.

Note: If the distribution and parameters are associated with a published model, then the model's name will appear as a link at the bottom of the **Analysis Summary** area. Click the link to view the model's properties. *For details on how to publish the results as a model, see page 116 in Chapter 3*.

Because the calculated parameters are estimates based on observed data points from a limited sample size, there is uncertainty in the results. Therefore, in addition to displaying the calculated parameters, the control panel also returns other results that may be used to evaluate how well the chosen distribution or analysis method fits the data set. Depending on your application settings, the following information may be displayed:

- The **Rho** value (correlation coefficient) is automatically calculated and displayed in the results if the parameters were estimated using rank regression. Rho is a measure of how well the regression line falls onto a straight line on a probability plot. The closer the value of rho is to +/-1, the better the linear fit. A value of zero would indicate that the data points are randomly scattered and have no pattern or correlation to the regression line model.
- The **LK Value** (likelihood function value) is automatically calculated and displayed in the results regardless of the chosen analysis method. It allows you to compare how well different models fit the same data set. The model with the highest LK value is considered to have the best fit statistically for that data. However, because this value is influenced by the sample size and the variability of the data set, it varies across different data sets and cannot be used by itself to evaluate a particular model.

- If the data set contains failures at time = 0, the calculation for the **Pnz** (percent non-zero) will be automatically displayed in the results. Pnz represents the proportion of the population with non-zero failure times. A value of 1 indicates that there are no zero failure times in the data set (which is the most common scenario in life data analysis); a decimal value indicates that the data set includes zero failure times (such as out-of-the-box failures, for example). The model parameters are calculated based on non-zero failure times, and the PNZ value is used as a multiplier when calculating certain metrics (e.g., reliability, unreliability) based on the model.
- If you use the 3-parameter Weibull distribution, you have the option to display the **theta** parameter, which is the sum of the values of the eta and gamma parameters. To include the theta parameter in the results, select the **Show Theta** check box on the Weibull++ Folios page of the Application Setup (**File > Application Setup**). Note that for the 1-parameter and 2-parameter Weibull distributions, the value of gamma is zero; therefore, if you select to display theta for these distributions, the value of theta will be identical to the value of eta.

  As an alternative, you can select to always display the theta parameter instead of eta. To do this, click the Calculations page of the Application Setup and select the **Use Theta parameter scheme on Weibull** check box.

Additional Tools

The following tools may also be used in conjunction with your own engineering knowledge about the model in order to evaluate the fit of the distribution to the data set.

- For a rank regression analysis, the **Probability Plot** shows you how well the solution line tracks the plotted data points. This allows you to visually assess the fit of a distribution. This method can be used only when the parameters have been calculated using rank regression.
- **Confidence Bounds** allow you to quantify the amount of uncertainty in the parameter estimates and in the reliability predictions due to sampling error. The narrower the confidence bounds, the closer the estimates are to the true value of the parameters and the better the precision on the reliability predictions. You can visualize the confidence bounds by using plots or obtain point calculations on the confidence bounds by using the Quick Calculation Pad. *See Section 6.5 on page 219*.
- The **Goodness of Fit Results** is a utility that uses three different tests to determine how well the parameters of the distribution fit the data. Note that this utility can only be used to evaluate data sets that do not contain censored data. *See Section 6.7.3 on page 231*.

6.3.1 Folio Tools

The folio tools are arranged on the left side of the Main page of the control panel. Use these tools to manage data and experiment with the results of your analysis.



Calculate estimates the parameters of the chosen distribution, based on the current data set and the specified analysis settings. This tool is also available by choosing **Life Data > Analysis > Calculate**.



Plot creates a new sheet in the folio that provides a choice of applicable plot types. For standard folios, this includes plots such as reliability vs. time, failure rate vs. time, probability plot, etc. This tool is also available by choosing **Life Data > Analysis > Plot**. *See Section 6.4 on page 217*.



QCP opens the Quick Calculation Pad, which allows you to calculate results based on the analyzed data sheet, such as the probability of failure, mean life, etc. This tool is also available by choosing **Life Data > Analysis > Quick Calculation Pad**. *See Section 6.5 on page 219*.



Distribution Wizard opens the Distribution Wizard, which helps you select the distribution that best fits your data based on the selected parameter estimation method (i.e., RRX, RRY or MLE). The tool is also available by choosing **Life Data > Analysis > Distribution Wizard**. *See Section 6.7.1 on page 226*.



Alter Data Type opens the Weibull++ Folio Data Sheet Setup window, which allows you to change the data entry columns in the current data sheet. Note that when you change the setup, you may lose information already entered in the data sheet. For example, switching the data sheet from one that supports interval data to one that supports only right censored data will remove all the inspection times. This tool is also available by choosing **Life Data > Format and View > Alter Data Type**. *See Section 6.1 on page 206*.



Change Units opens the Change Units window, which allows you to change the units for the time values in the current data sheet. *See page 113 in Chapter 3*.



Auto Group Data opens the Auto Group Data window, which allows you to group together data points with the same failure/suspension times. Grouping data significantly speeds up calculations when you have a very large data set. This tool is also available by choosing **Life Data > Options > Auto Group Data**. *See Section 6.7.4 on page 232*.



Alter Parameters (also called Alter Parameters w/o Recalculation) allows you to manually alter the value of one (or more) parameters, while keeping the original values of the rest of the parameters and the variance/covariance matrix the same. The LK value, the solution line on plots and all subsequent analyses will be based on the modified set of parameters.



Specify Points opens the Specify Points window, which allows you to adjust the way that the line is fitted to the points for a rank regression analysis. The ranks will be calculated based on all of the data points but the regression line will be fitted only to the points you have specified. This tool is also available by choosing **Life Data > Options > Specify Points**. *See Section 6.8.5 on page 245*.



Color Data Values allows you to apply different font colors to the entries in the data sheet for easy visualization. The following commands are available:

- **Default Color** applies a black font color to all the entries in the data sheet
- **Based on Data Type** applies different font colors based on the type of censoring. The colors are: red for failure data (no censoring), green for suspension data, magenta for left-censored data and blue for interval data.
- **Based on Subset ID 1** applies different font colors based on unique subset IDs in the Subset ID 1 column. If a data point has no entry in the Subset ID 1 column, then the font color is black.
- **Based on Subset ID 2** applies different font colors based on unique subset IDs in the Subset ID 2 column. If a data point has no entry in the Subset ID 2 column, then the font color is black.
- **Based on Subset ID Combination** applies font colors based on unique combinations of subset IDs in the Subset ID 1 and Subset ID 2 columns. If a data point has no subset IDs, the font color is black, but if a data point has at least one subset ID, then it is treated as having a unique combination of subset IDs.

Note that whenever you edit or add new data to the data sheet, you must choose one of the Color Data Values commands again in order to apply the correct font colors.



Batch Auto Run opens the Batch Auto Run window, which allows you to extract data from an existing data set based on the subset ID. This tool is also available by choosing **Life Data > Options > Batch Auto Run**. *See Section 6.7.2 on page 229*.

6.3.2 Weibull++ Standard Folio Analysis Settings

The Analysis page of the Weibull++ standard folio control panel includes the settings that are also displayed on the Main page, along with any additional settings that may be applicable for the current data sheet. This section provides a brief description of all the settings.[2]



The Analysis page of the Weibull++ standard folio control panel includes the following settings:

- The **Analysis Method** setting allows you to choose the method for estimating the parameters of your chosen distribution. There are four options to choose from: rank regression on X (RRX), rank regression on Y (RRY), non-linear rank regression (NLRR) and maximum likelihood estimation (MLE). Note the following:
  - For free-form data, the only analysis methods available are RRX and RRY.
  - The NLRR method is available only when you choose the 3-parameter Weibull distribution.

  As a rule of thumb, data sets with small sample sizes and mostly complete data may be best analyzed with rank regression, while MLE may be more appropriate for data sets with a high proportion of suspensions, interval data or many observed failures. This is because the MLE method is based on the likelihood function, which considers each time-to-suspension in the estimate of the parameters, unlike in rank regression where the solution is based on the plotting positions of the times-to-failure data. However, the MLE solution tends to be badly biased (statistically distorted) when performed on small sample sizes. As the sample size gets larger, the difference between the two methods become less important. To determine whether your sample size is large enough for

2. The ReliaWiki resource portal provides more information on the background theory of each analysis method at: http://www.ReliaWiki.org/index.php/Parameter Estimation.

MLE, you will need to take into account factors such as the amount of variability in your data set and the acceptable level of risk or margin of error in your calculations.[3]

When choosing between rank regression on X (RRX) and rank regression on Y (RRY), note that data are best analyzed when regressed in the direction of uncertainty. For example, in reliability tests, the times-to-failure (x-axis) vary from test to test but the probabilities of failure (y-axis) remain consistent. Therefore, if the uncertainty is on the times-to-failure data, RRX may be the preferred analysis method.

On the other hand, in situations where the time value (x-axis) is known but the probability of failure (y-axis) varies for each time value, the RRY method may be the appropriate choice because the uncertainty is on the unreliability estimates.

- The **Rank Method** setting allows you to choose the method for calculating the unreliability estimates of the times-to-failure data. There are two options available: median ranks and the Kaplan-Meier estimator.

  The median ranks method assigns unreliability estimates based on the failure order number and the cumulative binomial distribution. Alternatively, the Kaplan-Meier estimator uses the product of the surviving fractions, producing a modified empirical distribution. In general, the median ranks method is preferable and more widely used for unreliability estimation. Thus, it is a good idea to use the median ranks method unless one has a specific reason to use the Kaplan-Meier methodology.

  - The **Use RRM for interval data** check box is for the ReliaSoft ranking method (RRM). This option is automatically selected when you have interval censored data in the data sheet; otherwise, the software will use the default standard ranking method (SRM). The RRM method provides a more accurate estimate of the time the failure occurred in the interval by iteratively recomputing the failure times and the ranks until the parameter values converge. For non-interval data (exact times-to-failure data) there is no difference in the results between the SRM and RRM methods.[4]
  - The **Confidence Bounds Method** setting allows you to choose the method for calculating the confidence bounds. The methods available depend on your chosen distribution method or analysis method. Note that confidence bounds calculations are not available for free-form data.[5]
    - The Fisher Matrix method is the standard way of calculating confidence bounds, and it is the default method in the software. For data sets with a large number of data points, there is no significant difference in the results between the Fisher Matrix method and the likelihood ratio method; however, when working with very small sample sizes, the Likelihood Ratio method may be more accurate.
    - The Bayesian method calculates the confidence bounds by analyzing prior information about the assumed distribution's parameters along with sample data in order to evaluate the amount of uncertainty in the estimated values.
    - The Beta-Binomial method is a non-parametric approach to confidence bounds calculations, meaning that no underlying distribution is assumed. This method can be used only with the mixed Weibull distribution.
- The **Sort before calculation** option allows you to sort the failure/suspension times in the data sheet in ascending order. This will not affect the results of the calculation because, in the standard folio, all data points are assumed to be independent and identically distributed (i.i.d.).

---

3. For more information on rank regression vs. the MLE parameter estimation method, please read "*Comparison of MLE and Rank Regression Analysis When the Data Set Contains Suspensions*" at: http://www.Weibull.com/hotwire/issue16/relbasics16.htm.

4. The ReliaWiki resource portal provides more information on the RRM method at: http://www.ReliaWiki.org/index.php/ReliaSoft's Alternate Ranking Method.

5. The ReliaWiki resource portal provides more information on the background theory of confidence bounds at: http://www.ReliaWiki.org/index.php/Confidence Bounds.

- The **Grouped Data Settings** affect how the unreliability estimates (median ranks or Kaplan-Meier estimates) are plotted on the probability plot. These settings are available only when working with grouped data.

  When using rank regression with grouped data, the software will plot only the data points that correspond to the highest median rank position in each group. For example, for two groups of data with 10 units each, the software will plot only two data points: the 10th rank position out of 20 and the 20th rank position out of 20. The regression line is then fitted to these two points. When you select the **Ungroup on regression** check box, the software will plot each individual data point in the group. The regression line is then fitted to all the data points. In general, grouped data analysis provides parameter estimates that have wider confidence bounds, while ungrouped data analysis provides parameter estimates with much narrower confidence bounds.[6]

  When using MLE analysis, the **Ungroup on MLE** check box ungroups the data during calculations and then plots all the data points in each group on the probability plot; however, the solution of the line is obtained from the likelihood function and not by the plotting positions of the data points. Therefore, the line is not expected to track the points on the plot. This option is useful for adjusting the way the grouped data points are displayed on the MLE probability plot.

  Note: When working with grouped interval censored data, you have to be cautious about ungrouping the data. In interval censored data, it is assumed that the failures occur at some time in the interval between the previous and the current time-to-failure. For example, a group of 10 units may be interval censored between zero and 100 hours. If you select the **Ungroup on Regression** check box, the software will treat the 10 units as failures that occur at *exactly* 100 hours.

- The **Biasing** option is available when you select the 2-parameter Weibull distribution with MLE analysis, and only if the Bayesian confidence bounds method is not selected. The MLE method is known to obtain biased (statistically distorted) estimates when performed on small sample sizes. The **Unbias parameters** check box corrects the biased estimate of the Weibull beta parameter due to the MLE sampling error. Weibull++ uses the correction factor proposed by Ross and Hirose to unbias both censored and non-censored data.[7] [8]

6.4 Life Data Analysis Plots

Weibull++ includes multiple plot types you can use to visualize the results of your life data analysis. You can create plots by choosing **Life Data > Analysis > Plot** or by clicking the icon on the Main page of the control panel.



The scaling, setup, exporting and confidence bounds settings are similar to the options available for all other Weibull++ plot sheets. *To learn how to customize a plot, see Chapter 15*.

Tip: Weibull++ includes two additional plots you could use across all types of data folios: the overlay plot, which allows you to compare different data sets or distributions; and the side-by-side plot, which allows you to display different plots of a single data set all in a single window for easy comparison. *See Chapter 14*.

6. The ReliaWiki resource portal provides more information about using rank regression with grouped data at: http://www.ReliaWiki.org/index.php/Grouped_Data_Parameter_Estimation.

7. The ReliaWiki resource portal provides more information about the correction factors at: http://www.ReliaWiki.org/index.php/The_Weibull_Distribution#Unbiased_MLE.

8. For more information on how you might correct biased estimates, please read "*Unbiasing Parameters in Weibull++*" at: http://www.Weibull.com/hotwire/issue109/relbasics109.htm. (Note that this article uses Weibull++ 7, but the theory remains the same.)

The following is a description of the different types of plots that can be created in a Weibull++ standard folio:

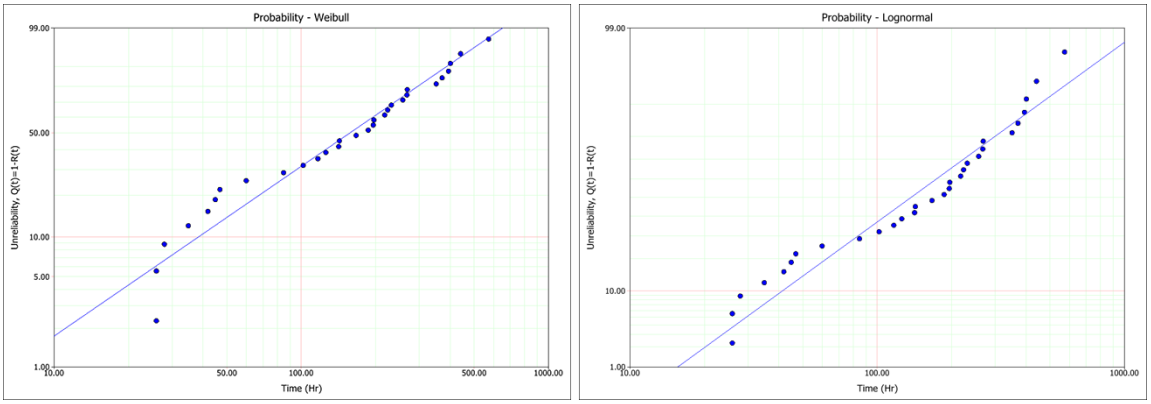
- The **Probability** plot shows the trend in the probability of failure over time. The plotting positions of the data points are determined by the failure/suspension times in the data set (x-axis) and their corresponding unreliability estimates (y-axis).

  Note: Unlike the probability plots for other distributions, the y-axis in an exponential probability plot always indicates the reliability instead of the unreliability. This tradition arose from the time when probability plotting was performed "by hand." The exponential reliability model starts with R = 1 at T = 0 (or gamma). Thus, if the unreliability were plotted, the axis would start at Q = 1 - R = 0, which is not possible, given that the y-axis scale is logarithmic.

  When the parameters have been calculated using rank regression analysis, the regression line is fitted to the data points on the plot in order to obtain the distribution parameters. Therefore, the plot can also be used to compare how different distributions fit a particular data set. The closer the regression line tracks the points on the plots, the better the fit.

  In contrast, the maximum likelihood estimation (MLE) method obtains the solution of the line from the likelihood function, not by the plotting positions of the data points. Therefore, the line is not expected to track the points on the plot, and the plot should not be used to evaluate the fit of a distribution when using MLE.

  The next figures show the rank regression analysis of single data set using a Weibull distribution and a lognormal distribution. As you can see, the probability plot shows that the Weibull distribution presents the better fit to this particular data set.



- The **Reliability vs. Time** plot shows the reliability values over time, capturing trends in the product's failure behavior.
- The **Unreliability vs. Time** plot shows the probability of failure of the product over time.
- The ***pdf*** plot shows the probability density function of the data over time, allowing you to visualize the distribution of the data set.
- The **Failure Rate vs. Time** plot shows the failure rate function of the product over time.
- The **Contour Plot** shows a 3D surface on a 2D plot. The plot represents the 3D surface by plotting constant z slices, called *contours*, on a 2D format. The shape of a contour is determined by plotting the parameters of the data set (x- and y-axis) at the position where a given confidence level occurs (z-axis).

  You can use the overlay plot feature to superimpose two contour plots from two different data sets at the same confidence level in order to determine the level at which the two data sets are statistically different. If the contour lines overlap, then the data sets do not show a statistically significant difference at that confidence level. *See Section 14.4 on page 394*.

- The **Failures/Suspensions Histogram** is a bar chart that shows the number of failures/suspensions that fall into different time intervals. This allows you to evaluate the frequency distribution of the failures and suspensions in the data set. The chart has two display options:
  - **Probability Density Values** displays a chart where the height of each bar is proportional to the frequency of occurrence of failures in each interval. Select the **Superimpose pdf** check box to display the pdf line on the same chart.
  - **Failures** displays a chart where the height of each bar is proportional to the number of failures in each interval. Select the **Show suspensions** check box to display a bar chart of the number of suspensions in each interval.

  You have the option to adjust the time intervals to get a good indication of how the data points are distributed. Note that if the histogram has intervals that are, in theory, infinitesimally small, the chart will result in a plot of the probability density function (*pdf*).
- The **Failures/Suspensions Pie** chart displays the ratio and proportion of failures and suspensions to the whole data set.
- The **Failures/Suspensions Timeline** plots are similar to horizontal bar charts. The values of the failures and suspensions are plotted on the y-axis and the time periods are on the x-axis. All the lines are anchored to the y-axis, giving a common point of measurement.

6.5 Quick Calculation Pad (QCP)

The Quick Calculation Pad (QCP) provides a convenient way of calculating a variety of useful metrics. To access the tool, click the **QCP** icon on the Main page of the control panel.



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 a particular analysis, you can go directly to the topic of interest:

- Life Data Analysis. *See Section 6.5.2 on page 221*.
- Accelerated Life Testing Data Analysis. *See page 260 in Chapter 7*.
- Recurrent Event Data Analysis. *See page 383 in Chapter 13*.

6.5.1 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 data sheet. For example, in a standard folio, you could enter the failure/suspension times 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 details about managing time units for a Synthesis repository, see page 39 in Chapter 2*.

- If applicable, use the **Bounds** drop-down list to specify what type of confidence bounds to calculate. The following graphics illustrate the types.[9]



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 calculation options are disabled when the underlying data type is free-form data.

- 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.[10]

  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 time with 2-sided confidence bounds at the 90% confidence level, the first picture shows the display with captions and the second picture shows without.



QCP window with captions on.

9. The ReliaWiki resource portal provides more information on confidence bounds at: http://www.ReliaWiki.org/index.php/Confidence Bounds.

10. There is one exception. If you choose **Parameter Bounds**, the information for all parameters will not fit in the results area at the top of the window. For this calculation, you must select a type from the **Bounds** drop-down list and enter the confidence level under **Inputs**. When you click **Calculate**, the information will be displayed directly in the Results window.



QCP window 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 folios and data sheets. The calculations performed in the QCP will be based on the currently active data sheet. If this option is not selected, you will need to close the QCP to access any data sheet or folio. 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**).

6.5.2 QCP Calculations for Life Data Analysis

Weibull++ includes a Quick Calculation Pad (QCP) for computing useful life data analysis metrics. You can access the QCP by clicking its icon on the Main page of the control panel.



To perform a calculation, select the appropriate option and enter any required inputs in the **Input** area, then click **Calculate**. *For more detailed information on how to use the QCP in general, see Section 6.5 on page 219*.

The following calculations are available for life data analyses:

Probability

- **Reliability**

  Calculates the probability that a new product 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 units are fielded, then 90 of them will still be operating at the end of 3 years.

- **Probability of Failure**

  Calculates the probability that a new product 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 product 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 length of the interval in the **Mission Additional Time** field.

  For example, a product may have a reliability of 90% for 3 years. If the product 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%.

- **Conditional Probability of Failure**

  Calculates the probability that a product 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 length 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 units are fielded, then 90 of them will be still be operating at the end of 4 years.

  What's Changed? In previous versions of Weibull++, 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% Life At** field. For example, a B10 life of 4 years means that 10% of the fielded units are expected to be failed at the end of 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 average time at which a product is expected to operate before failure. In the Weibull++ standard folio, the mean life is the mean time to failure (MTTF) based on the fitted model.

  Note: The term *mean time to failure* (MTTF) is used as a metric for the analysis of *non-repairable* components. In the Weibull++ standard folio, all data are assumed to come from non-repairable components that are independent and identically distributed (i.i.d.). On the other hand, the term *mean time between failures* (MTBF) is used as a metric in repairable systems analysis, where the same system may fail and be repaired multiple times. To analyze simple repairable system data in Weibull++, use the recurrent event data analysis (RDA) folio. *See Chapter 13*.

  For more complex repairable system analyses, see ReliaSoft's BlockSim and RGA++ software.

Rate

- **Failure Rate**

  Calculates the instantaneous number of failures per unit time that can be expected at a certain time given that a unit survives to that age. Enter the time at which you wish to calculate the failure rate in the **Mission End Time** field.

  For example, a failure rate of 0.01 at 100 hours means that each unit that survives to 100 hours has approximately a 1% probability of failure in the next hour.

Bounds

- **Parameter Bounds**

  Calculates the specified bounds on the parameter estimates, allowing you to quantify the amount of uncertainty in those estimates. This option is available only when you have specified the type of confidence bounds to use from the **Bounds** drop-down list. When you click **Calculate**, the Results Window will open to display the estimated parameters and their bounds.

6.6 Life Data Analysis Example

Ten identical units of a prototype device were reliability tested at the same application and operation stress levels. Six of the units failed during testing after operating for the following times: 16, 34, 53, 75, 93, 120 hours. Four other units were still operating after 120 hours.

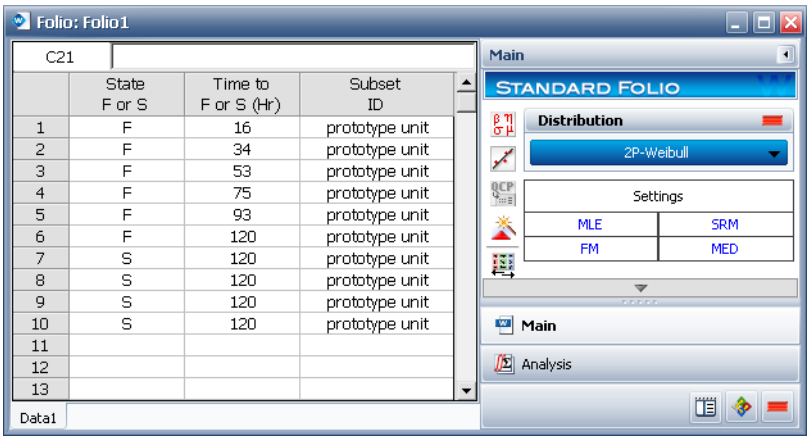
The objective is to obtain the reliability of the prototype device at 60 hours, at the lower 1-sided 90% confidence bound.

1. The first step is to add a new Weibull++ standard folio by choosing **Insert > Folios > Weibull++ Standard Folio**.



2. In the Weibull++ Folio Data Sheet Setup window, select **Times-to-Failure data** for the data type and then select **My data sheet contains suspensions (right censored data)**. For the time units, select **Hour**. Click **OK** to create the folio.

3. Enter the given data in the data sheet, and then on the Main page of the control panel, choose the **2P-Weibull** distribution and the **MLE** parameter estimation method, as shown next. (You may also choose the parameter estimation method on the Analysis page of the control panel.)



4. Choose **Life Data > Analysis > Calculate** or click the icon on the Main page of the control panel.



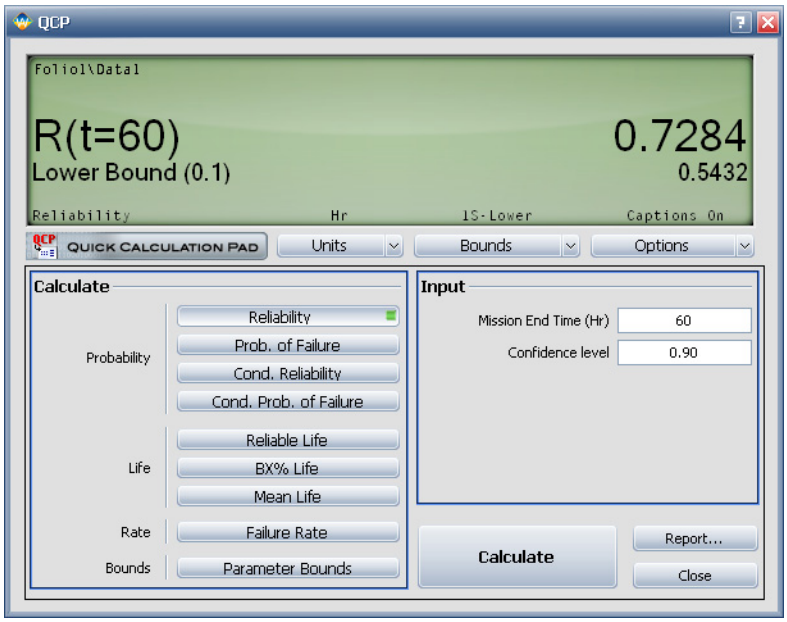
The parameters of the distribution will be automatically computed and displayed in the **Analysis Summary** area of the control panel. The parameters are beta = 1.5070 and eta = 128.6414.

5. To estimate the reliability of the device, open the QCP by choosing **Life Data > Analysis > Quick Calculation Pad** or by clicking the icon on the Main page of the control panel.



6. In the QCP, choose to calculate the **Reliability** with **Lower One-Sided** confidence bounds. Select **Hour** for the time units and then make the following inputs: Mission End Time = **60** and Confidence Level = **0.9**.

Click **Calculate** to obtain the results, as shown next.

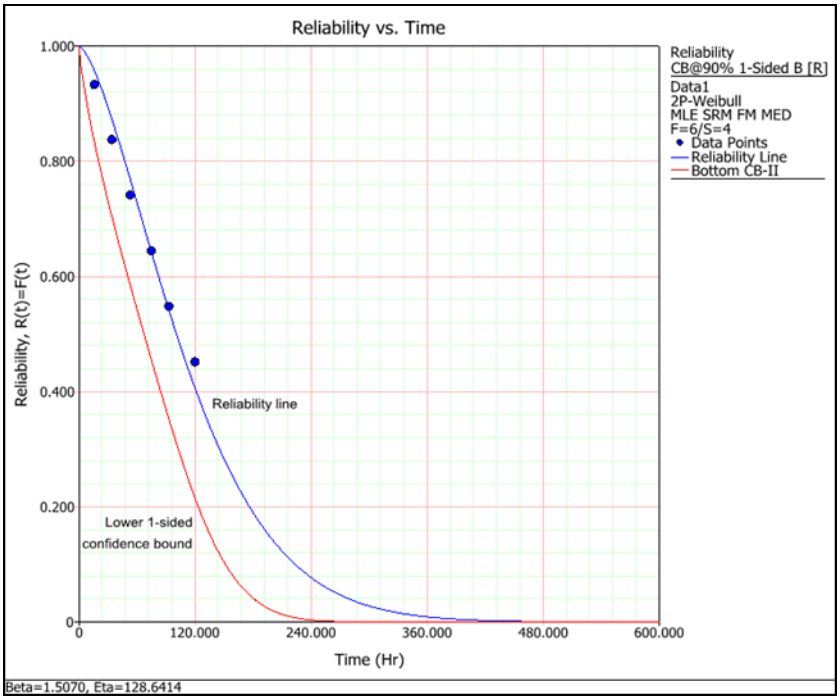


The result shows that the reliability of the prototype device at 60 hours is 72.84%. The lower one-sided 90% confidence bound is estimated to be 54.32%. This means that there is 90% probability that the reliability of the device will be at least 54.32% after 60 hours of operation.

7. To visualize the reliability of the device over time, close the QCP and then create a plot by choosing **Life Data > Analysis > Plot**, or by clicking the icon on the Main page of the control panel.



8. Next, display the confidence bounds in the plot by choosing **Plots > Confidence Bounds > Confidence Bounds** or by right-clicking the plot sheet and then choosing Confidence Bounds on the shortcut menu. In the Confidence Bounds Setup window, make the following selections:
   - Sides: **One-Sided; Show Bottom**
   - Type: **Reliability (Type II)**
   - Confidence Level: **90%**
9. Click **OK**. On the control panel of the plot sheet, choose the **Reliability vs. Time** plot type. The following picture shows the resulting plot. As you can see, the line representing the confidence bound is fairly far from the reliability line, indicating a significant amount of uncertainty in the results. Based on this result, you may choose to perform further analysis on the data, such as testing those 4 suspended units to failure to see whether the uncertainty is due to the small number of failures in the data set or due to the variability inherent in the device itself.



6.7 Additional Tools

6.7.1 Weibull++ Distribution Wizard

The Distribution Wizard performs multiple goodness of fit tests to determine the best distribution for a data set based on the chosen parameter estimation method. Note that the Distribution Wizard only serves as a guide. You should compare its suggestion with your own engineering knowledge about the product being modeled before making the final decision on which distribution to use for your data set.

Comparing Distributions

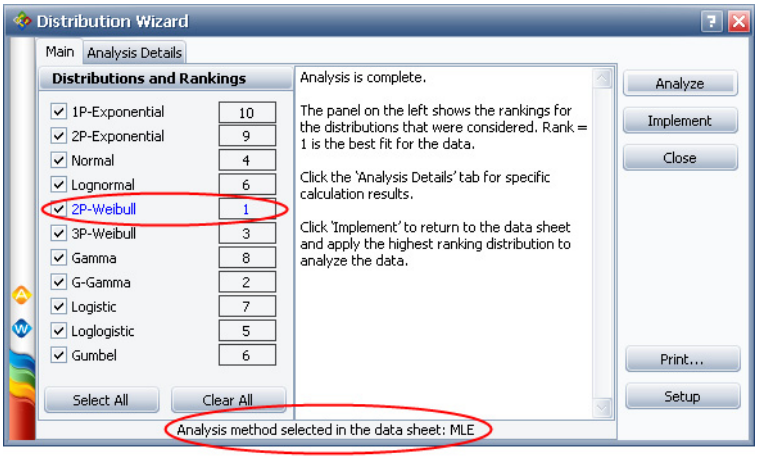
To use the Distribution Wizard, you must have at least two failure data points in the current data sheet.

The first step is to choose a parameter estimation method from the control panel. Next, open the Distribution Wizard by choosing **Life Data > Analysis > Distribution Wizard** or by clicking the icon on the Main page of the control panel.



On the Main tab of the Distribution Wizard, select the distributions you would like to consider and then click **Analyze** to start the evaluation. The distributions will be ranked according to how well they fit the data, with rank 1 being the best fit.

In the example shown next, the 2P-Weibull distribution is the suggested distribution for the data set. The parameter estimation method selected in the data sheet (in this case, MLE) is displayed at the bottom of the window.



To calculate the parameters of the distribution in the rank 1 position, click the **Implement** button. This closes the Distribution Wizard, and the results will appear in the Analysis Summary area of the control panel.

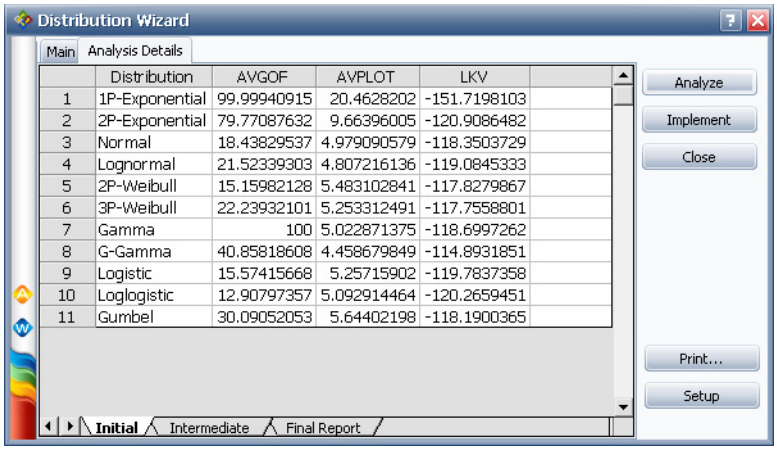
Note: You may find that the Distribution Wizard often recommends either the 3-parameter Weibull distribution or the generalized gamma (G-Gamma) distribution. This is because these distributions have the ability to mimic the attributes of other distributions based on the values of the distribution's parameters. In such cases, if you have reason to believe that the recommended distribution is not applicable, you may prefer to select the distribution with the next highest ranking.

Goodness of Fit Tests

To see the calculations behind the ranking, click the **Analysis Details** tab. The Distribution Wizard performs three goodness of fit tests to determine the rank of the distributions:

- **The Kolmogorov-Smirnov (GOF) test**, which tests for statistical difference (the difference between the expected and obtained results).
- **The correlation coefficient (PLOT) test**, which measures how well the plotted points fit a straight line.
- **The Likelihood Value (LKV) test**, which computes the value of the log-likelihood function given the parameters of the distribution.

On the Analysis Details tab, the **Initial** sheet contains the values computed from the three tests. The **AVGOF** column contains the average values from the GOF test, the **AVPLOT** column contains the average values from the PLOT test and the **LKV** column contains the average values from the LKV test, as shown next.



	Distribution	AVGOF	AVPLOT	LKV
1	1P-Exponential	99.99940915	20.4628202	-151.7198103
2	2P-Exponential	79.77087632	9.66396005	-120.9086482
3	Normal	18.43829537	4.979090579	-118.3503729
4	Lognormal	21.52339303	4.807216136	-119.0845333
5	2P-Weibull	15.15982128	5.483102841	-117.8279867
6	3P-Weibull	22.23932101	5.253312491	-117.7558801
7	Gamma	100	5.022871375	-118.6997262
8	G-Gamma	40.85818608	4.458679849	-114.8931851
9	Logistic	15.57415668	5.25715902	-119.7837358
10	Loglogistic	12.90797357	5.092914464	-120.2659451
11	Gumbel	30.09052053	5.64402198	-118.1900365

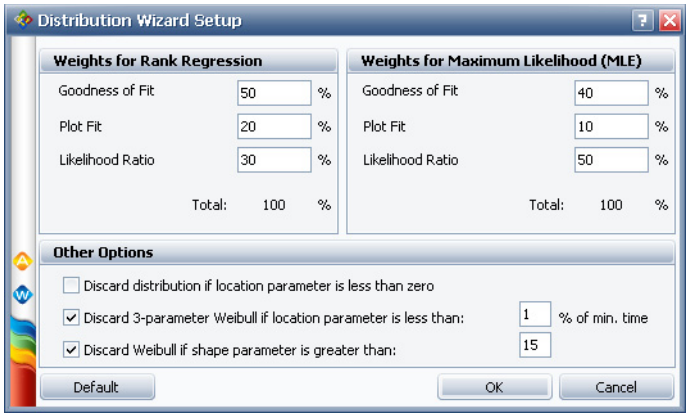
The **Intermediate** sheet displays the ranking of the distributions based on the values from the Initial sheet. The **RAVGOF** column contains the ranking of the GOF test, the **RAVPLOT** column contains the ranking of the PLOT test and the **RLKV** column contains the ranking of the LKV test.

The values from the Initial sheet are first weighted (e.g., AVGOF * weight) and then summed into one overall DESV value (i.e., weighted decision variable). The distribution with the lowest DESV value is considered to be the best fit for the data. The weights assigned to each test are based on the parameter estimation method. For example, by default, a higher weight is given to the LKV test if the parameter estimation is MLE; whereas in rank regression, higher weights are given to the K-S and Rho tests. The weights can be adjusted in the Setup window of the Distribution Wizard.

The **Final Report** sheet displays all the distributions in their final ranking order and the calculated parameters for each distribution.

Distribution Wizard Setup Window

You have the option to change the weighs that are used to determine the ranking of the distributions. Click the **Setup** button at the lower right corner of the Distribution Wizard to view the settings, as shown next.



You can use different weights for the rank regression method and the MLE method. The sum of the three weights for each parameter estimation method must equal 100%. The weights are:

- **Goodness of Fit** represents the weight percentage for the GOF test.
- **Plot Fit** represents the weight percentage for the PLOT test.
- **Likelihood Ratio** represents the weight percentage for the LKV test.

The settings in the **Other Options** area specifically affect the Weibull and the exponential distributions:

- **Discard if location parameter is less than zero** will exclude the Weibull and exponential distributions from the ranking if their location parameters are negative. A negative location parameter indicates that failures occurred before time = 0, or before a product operated for the first time.
- **Discard 3-parameter Weibull if location parameter is less than (_)% of min. time** will exclude the Weibull distribution from the ranking if the value of the location parameter is smaller than the first time-to-failure. The default setting is for a value that is less than 1% of the first time-to-failure, which is a value close to zero.
- **Discard Weibull if shape parameter is greater than (_)** will exclude the Weibull distribution from the ranking if the shape (beta) parameter is greater than a specified value. The default setting is to discard the distribution if the beta value is greater than 15.

6.7.2 Batch Auto Run

The Batch Auto Run utility allows you to extract data subsets from an existing data set. For example, consider a data sheet that contains the failure times for a product manufactured in two different plants. By using Batch Auto Run, you can filter the data by manufacturing plant and then analyze the failures from each plant separately.

Batch Auto Run Setup

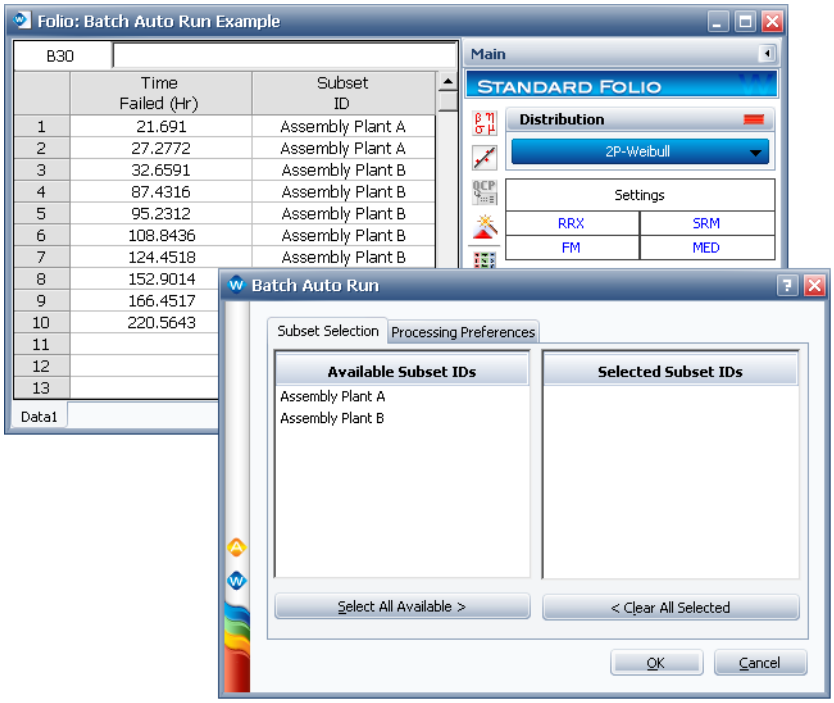
To use the tool, you must first define a subset ID for each data point you want to extract, making sure that the data points you wish to group together all have the same subset ID.

Note: The subset ID can be any text up to 30 characters, including spaces. For example, "A _ _ X" is not the same as "A _ X" where " _ " is used to designate a space.

Open the utility by choosing **Life Data > Options > Batch Auto Run** or by clicking the icon on the Main page of the control panel.



You will be presented with a list of all the subset IDs in your data sheet, similar to the inset shown next.



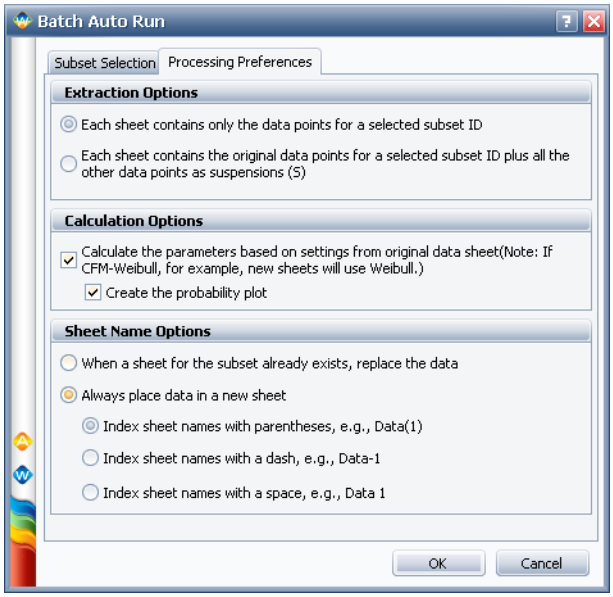
Double-click an available subset ID to include it in the batch auto run process or drag the subset ID to the **Selected Subset IDs** column. You can also click the **Select All Available** button to automatically include all subset IDs in the batch auto run process. Each selected subset ID will be extracted to its own separate data sheet at the end of the batch auto run process.

Processing Preferences

You can set your preferences for how the data will be processed on the Processing Preferences tab. When you change the settings and click **OK**, your preferences will be saved as the default settings for the next time you apply the batch auto run process for any Weibull++ standard folio.

The following options are available:

- **Extraction Options** allow you to specify how you want the data of the selected subset IDs to be extracted. The first option simply extracts the data points for each subset ID into separate data sheets so that you can analyze the data separately.

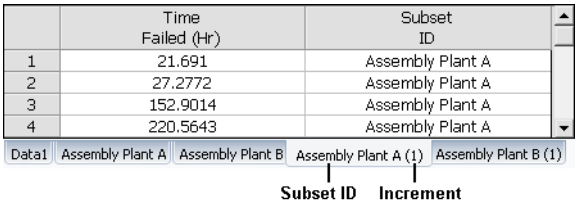


  The second option allows you to extract the data sets that are required to perform a competing failure modes (CFM) analysis. For example, consider a device that has three competing failure modes A, B and C. When you extract the data points of mode A, the Batch Auto Run utility will retain all the data points due to mode A but mark all other data points due to modes B and C as suspensions. In CFM analysis, the data points of modes B and C are considered to be suspensions because if they did not exist, all the units will have failed at some point in the future due to mode A. *For an example on how to use this setting, see Section 6.8.3.1 on page 242*.

- **Calculation Options** allow you select whether to automatically analyze the extracted data sheets and create a probability plot. The parameters of the extracted data sheet will be calculated using the same analysis settings specified for the original data sheet.

- **Sheet Names Options** allow you specify where you wish to put the extracted data. The first time you run the batch auto run process, the utility will extract the data points into new data sheets with the same names as the subset IDs. If you run the process again for the same subset IDs, you will have the option to either replace the data in the existing sheets or to always extract the data to new sheets.

  If you select the **Always place data in a new sheet** option, the data will be extracted to another new data sheet with the same name as the subset ID plus an increment to reflect the number of additional data sheets that have been created for the same subset ID. The increments can be separated by a parenthesis, dash or space. The following example shows the increment number separated by parentheses.



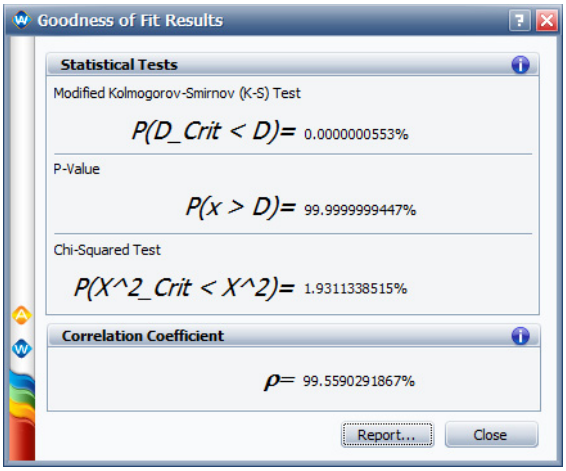
6.7.3 Goodness of Fit Results

The Goodness of Fit Results is a utility that uses statistical tests and the correlation coefficient to determine how well the parameters of the distribution fit the data. Note that this utility can only be used to evaluate data sets that do not contain censored data.

To use the utility, calculate the parameters of the distribution of your data set first and then choose **Life Data > Options > Goodness of Fit Results**.



The following picture shows sample results for a particular data set.



The results return the computed values of the following:

- Both the **Modified Kolmogorov-Smirnov (K-S)** test and the **Chi-Squared** test return the probability that the respective critical value (*D_Crit* for K-S and *X^2_Crit* for chi-squared) is less than the calculated statistic value (*D* statistic for K-S and *X^2* statistic for chi-squared). The calculated statistic value is the difference between the observed probability and the predicted probability. High probability values, close to 1, indicate that there is a significant difference between the theoretical distribution and the data set. In other words, it is an indication that the model may not be a good fit for the data. Note that the Chi-Squared test may not produce valid results for smaller sample sizes (e.g., minimum of 35).
- The **P-Value** shows the probability that the calculated statistic value is greater than a random variable. If the p-value is smaller than your specified level of significance, then the model may not be a good fit for the data.
- The **Correlation Coefficient** test determines how well the plotted points fit a straight line. This test is only available if the parameters were estimated using rank regression. The closer the value of rho (ρ) is to 1, the better the linear fit. A value closer to zero would indicate that the data points are randomly scattered and have no pattern or correlation to the regression line model.

6.7.4 Auto Group Data

The Auto Group Data tool is available for both Weibull++ and ALTA standard folios. It automatically groups together data points with the same failure/suspension times. Grouping data significantly speeds up calculations when you have a very large data set.

To group your data, click the **Auto Group Data** icon on the Main page of the control panel.

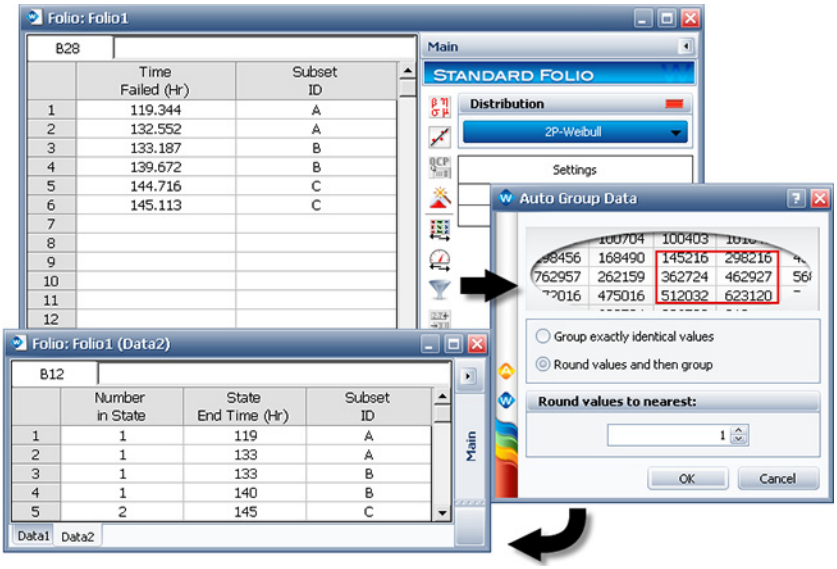


In addition to grouping exactly identical values, you have the option to round the values in the data sheet to the nearest whole number or to the nearest multiple of 10, and then group them. This can be useful when you need to set the accuracy of your computations. For example, a recorded observation of 509.71 cycles may be known to be accurate only when rounded to the nearest whole number.

In Weibull++, the utility considers the values (or rounded values) to be identical only if they have identical subset IDs, as shown in the example below.

In ALTA, the values are considered to be identical only if they have identical Use Stress levels *AND* identical subset IDs.

Note that if you have a second subset ID column in the data sheet, the information in that column is not taken into account and you may lose any information entered in that column when the values are grouped.



6.8 Special Analysis Methods

6.8.1 Mixed Weibull Analysis

Mixed Weibull analysis (also call *multimodal* Weibull) is a method that can be used in situations when dealing with failure modes that cannot be assumed to be independent (i.e., the occurrence of one failure mode affects the probability of occurrence of the other mode) and/or when it is not possible to identify the failure mode responsible for each individual data point.

To use mixed Weibull analysis, you will need to specify how many subpopulations exist in the data set (2, 3 or 4). The software will then determine the proportion of units that fall under each subpopulation and calculate the Weibull parameters of those subpopulations. The overall reliability of the full data set is then calculated by taking the sum of the proportional reliability contributions of each subpopulation.[11]

Tip: If the failure modes are assumed to be statistically independent and the data set can be directly categorized into distinct subpopulations, competing failure modes (CFM) analysis may be more appropriate. *See Section 6.8.3 on page 240*.

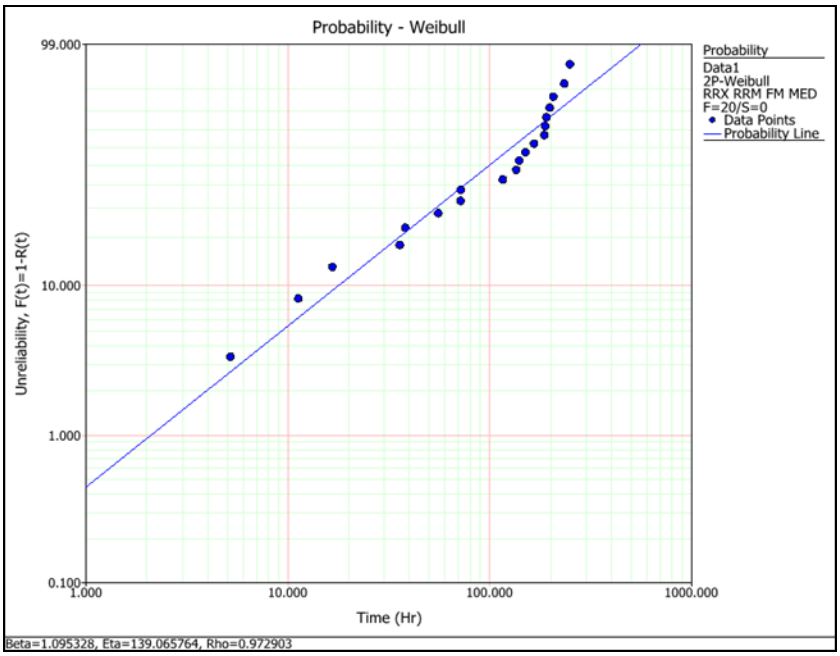
Example

Data sets with mixed populations are often recognized by the "dogleg" or S-shaped curve pattern of the data points on a probability plot. For example, consider the following data set.

Failure Times (in Hours) for 20 Test Units			
5.2	55.7	140.1	191.3
11.3	71.8	150.5	198.8
16.6	72	166	206.7
35.9	116.4	186.7	234.3
38.2	135.8	188.8	249.8

[11] The ReliaWiki resource portal provides more information about the mixed Weibull distribution at: http://ReliaWiki.org/index.php/The_Mixed_Weibull_Distribution.

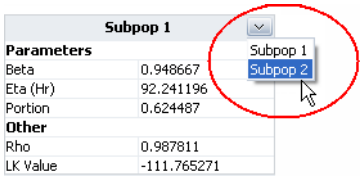
When the data set is analyzed with a 2-parameter Weibull distribution and plotted on a probability plot, as shown next, the pattern of the data points looks as if it could be modeled by two straight lines of different slopes. This indicates the possible existence of subpopulations in the data set.



Based on the pattern observed on the probability plot, let us assume that there are two subpopulations in the data set. To use the mixed Weibull distribution, follow the steps below:

1. Enter the data from the table into a Weibull++ standard folio.
2. On the Main page of the control panel, click the **Distribution** drop-down list. Choose **Mixed Weibull** and then choose the number of subpopulations to consider (for this example, choose **2 Subpop-Mixed Weibull**). Click **Calculate**.

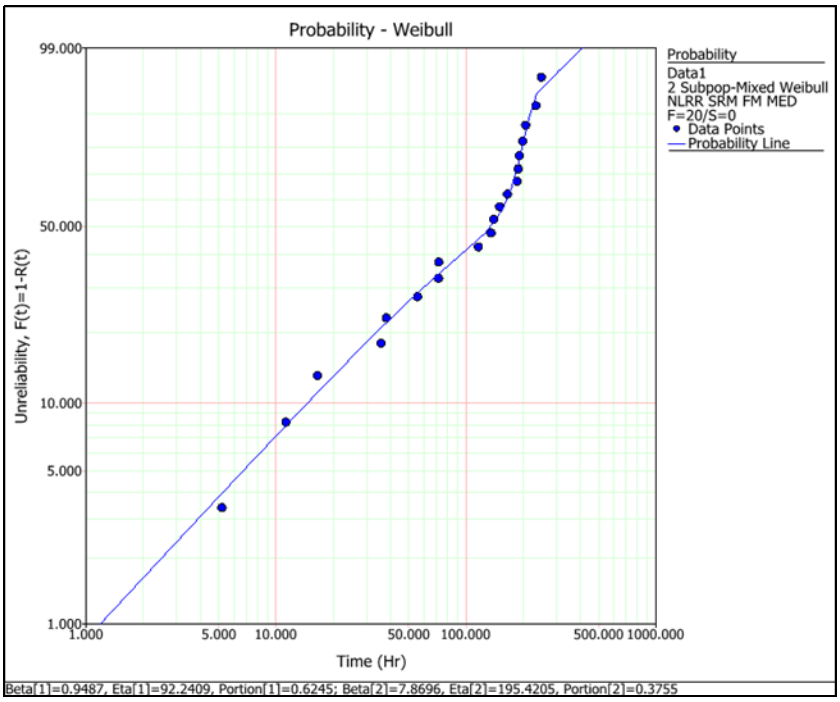
   Weibull++ will automatically fit a 2 subpopulation mixture model with Weibull parameters and portion values. In the results shown next, the **Subpop 1** area displays the parameters that were calculated for the first subpopulation. The result showing Portion = 0.6245 means that Subpop 1 comprises 62.45% of the data points. To see the parameters calculated for the other 37.55% of the data points, click the arrow on the control panel and choose **Subpop 2**, as shown next.



3. To view a worksheet containing a summary of all the subpopulations and their corresponding parameters, click anywhere within the parameter results area of the control panel. This opens the Results window, which displays the calculated results in a worksheet. From the Results window, you can edit, copy or print the results.

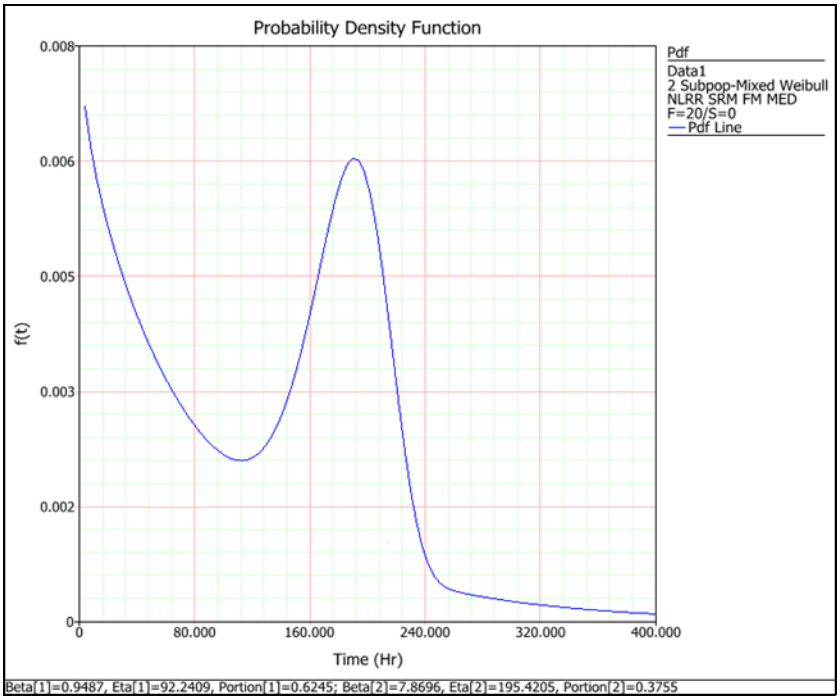
> **Note:** The subpopulation numbers are ordered based on the value of the scale parameter (eta). The order is used only to identify the subpopulation and does not affect the analysis.

4. To view the probability plot of the data set, click the **Plot** icon on the control panel, as shown next. As you can see, the 2-subpopulation mixed Weibull distribution provides a better fit to the pattern of the data points compared to the 2P-Weibull distribution.



You can use the Quick Calculation Pad (QCP) to make reliability predictions within the range of the observations; however, care must be taken when making projections outside the range of the observed failure times (i.e., extrapolation)

5. To visualize the mixed Weibull distribution, choose **Pdf Plot** in the Plot Type drop-down list, as shown next.



From time = 0 to approximately time = 100, the pdf exhibits the "decreasing with time" trend characteristic of a Weibull distribution with a beta value less than or equal to 1. This region characterizes the contribution of subpopulation 1. From time = 100 onward, the pdf exhibits the bell-shaped curve typical of a Weibull distribution with a beta value in the vicinity of 5. This region characterizes the contribution of subpopulation 2.

Note that the mixed Weibull analysis method does not specify which data point belongs to which subpopulation. If reliability predictions are needed at the subpopulation level, you will need to gather additional information in order to identify which data point belongs to which subpopulation.

6.8.2 Bayesian-Weibull Analysis

In Weibull++, the Bayesian-Weibull analysis method combines the concept of Bayesian statistics with the properties of the Weibull distribution to make predictions about the life characteristics of a product. This type of analysis is particularly useful when there is limited life data for a given product or failure mode but there is a strong prior understanding of the failure rate behavior.

The prior information in the analysis is in the form of a distribution fitted to a range of likely beta values. In the context of life data analysis, the beta parameter is used to define the shape of the Weibull distribution, which reflects the failure behavior of a product (i.e., whether the failure rate increases, decreases or remains constant over time). The range of beta values could be obtained from data for a similar product design or from engineering knowledge. When a distribution is fitted to the range of beta values, the distribution is called the *prior distribution* of beta and it serves to model the most likely failure behavior of the product.[12]

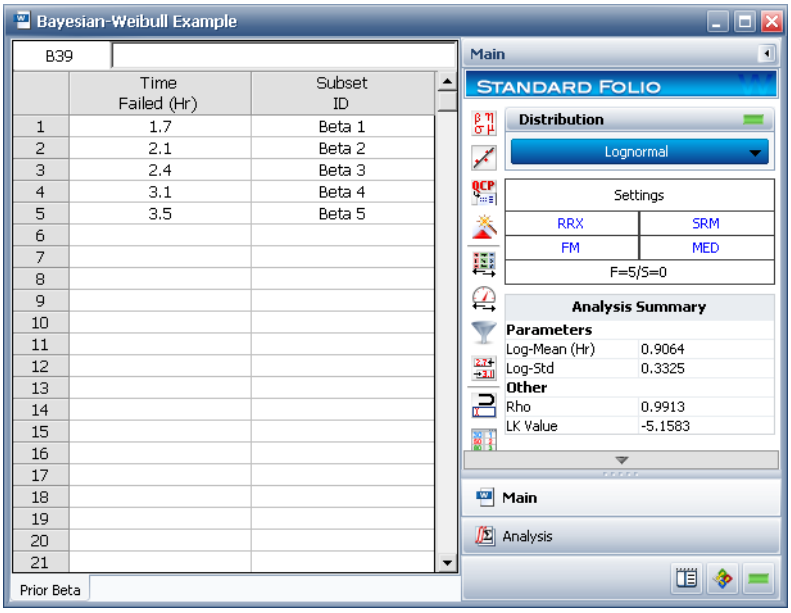
Example

Take for example a device that was redesigned for improved reliability. A test performed on the new design produced only a few failures; however, the same failure modes as the original design were observed. To use the Bayesian-Weibull approach, you will first need to obtain the prior distribution of beta in order to describe the failure behavior

[12] The ReliaWiki resource portal provides more information about Bayesian-Weibull analysis at: http://www.ReliaWiki.org/index.php/Bayesian-Weibull_Analysis.

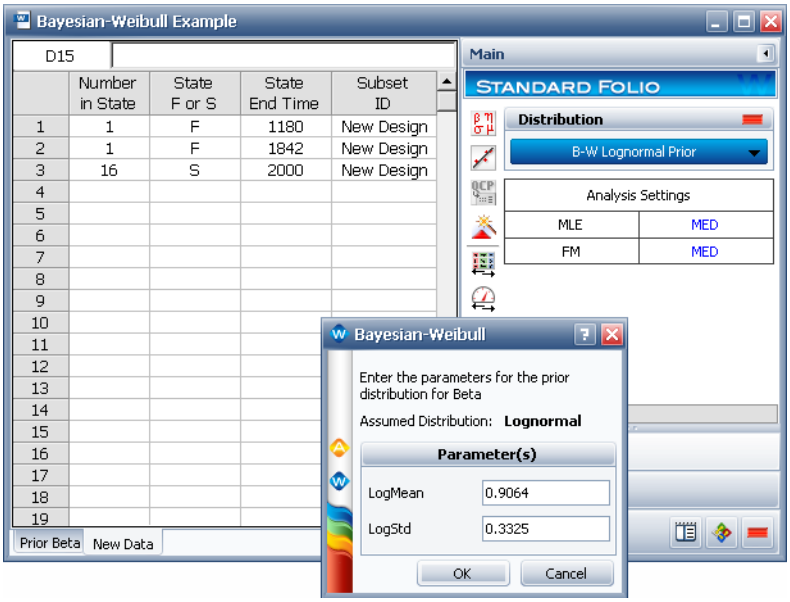
observed for the original design. In Weibull++, there are four distributions available: normal, lognormal, exponential and uniform.

The following example shows a range of known beta values obtained from prior analyses of the original design. The prior distribution of beta is determined to be a lognormal distribution with a log-mean of 0.9064 and a log-standard deviation of 0.3325, as shown next.



Next, collect the current times-to-failure data of the new design and enter the information in a new data sheet, as shown next. On the Main page of the control panel, click the **Distribution** drop-down list and choose **Bayesian-Weibull > B-W Lognormal Prior**.

Click **Calculate**. A window similar to the one shown in the inset will be displayed.



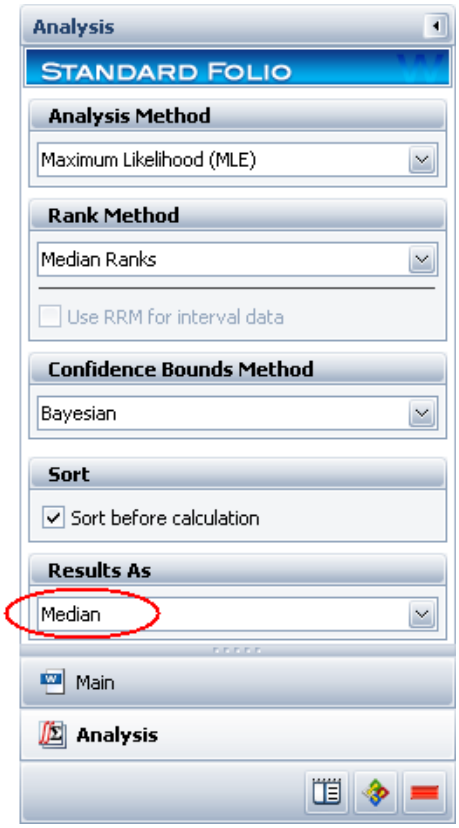
Enter the calculated parameter values of the prior distribution of beta: LogMean = 0.9064 and LogStd = 0.3325. Click **OK**.

The following picture shows the results of the analysis.

Analysis Summary	
Prior Distribution - Lognormal	
LogMean	0.9064
LogStd	0.3325
Posterior Distribution	
Beta (Median)	2.3612
Eta (Median)	5321.6319

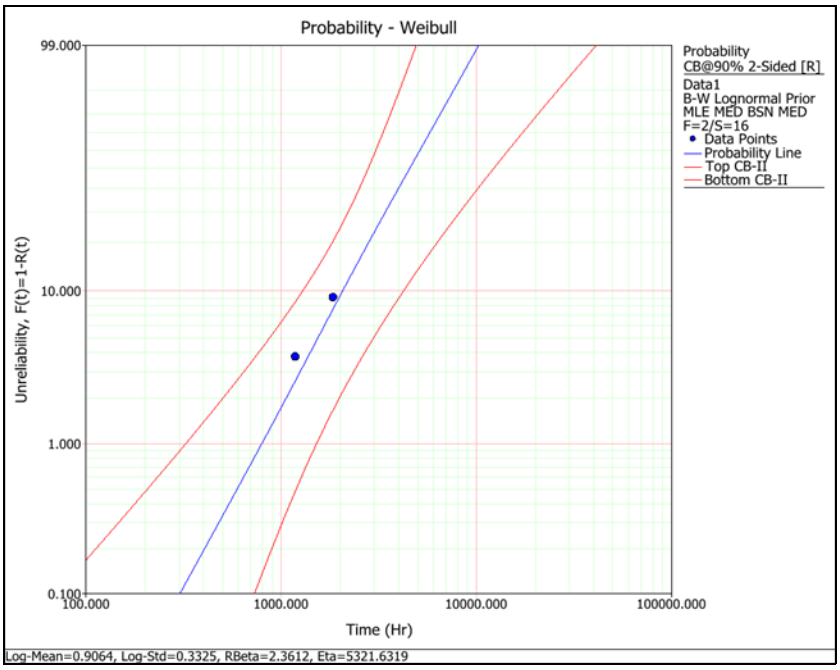
Note that instead of a single calculated value for each of the parameters, the parameters of the Bayesian-Weibull distribution are distributed. This result is known as a *posterior distribution*. In the example, the result showing that Beta (Median) = 2.3612 means that the median value of the posterior distribution of the beta parameter is 2.3612. For convenience, this is used as the single point estimate for the beta parameter.

By default, Weibull++ calculates the point estimates from the *median value* of the posterior distribution. This is because the median value always corresponds to the 50th percentile of a distribution. You have the option to change this setting and calculate the *mean value* of the posterior distribution instead; however, note that the mean is not a fixed percentile and this could cause issues when comparing results across different data sets. The option to change this setting is available on the Analysis page of the control panel when you choose the Bayesian-Weibull analysis, as shown next.

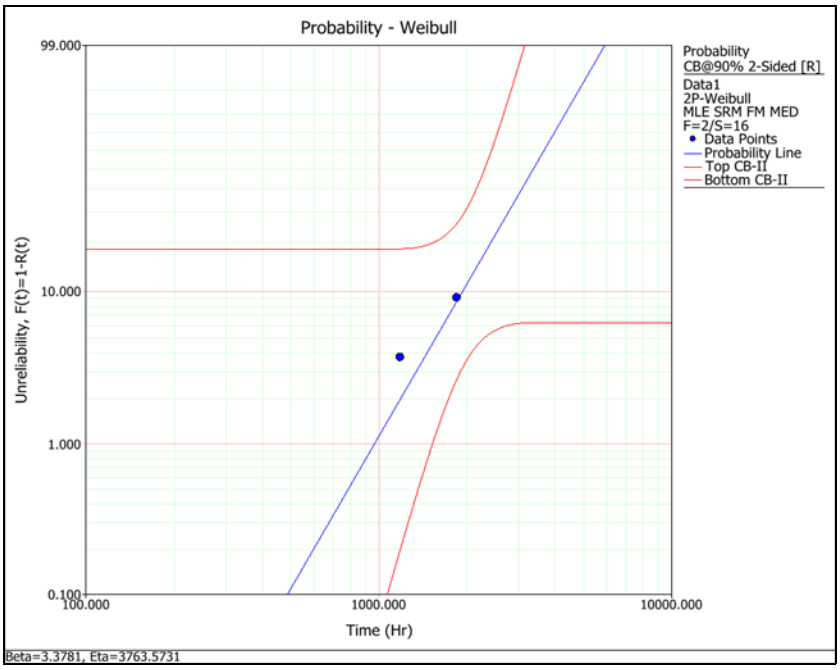


To obtain point estimates of a metric such as the probability of failure or the reliability at a given point in time, use the Quick Calculation Pad (QCP). The QCP will automatically compute the metric of interest using the median value of the posterior distribution (or the mean, if you switch the setting on the Analysis page of the control panel).

The following plots show the confidence bounds for the Bayesian-Weibull analysis that incorporates the prior information compared to the confidence bounds of a 2-parameter Weibull analysis of the new design data only. As you can see, the confidence bounds of the Bayesian-Weibull analysis are tighter, which may lead to a more precise analysis of whether the redesign improved the life of the device.



Confidence bounds of the Bayesian-Weibull analysis



Confidence bounds of the 2-parameter Weibull analysis

6.8.3 Competing Failure Modes (CFM) Analysis

Often, a product could fail due to any of several possible causes of failure. For a non-repairable component, each unit can only fail once, hence, if the occurrence of any one failure mode results in a failure of the component, then the failure modes are said to "compete" to cause the failure. This can be viewed as a series reliability model, where the overall reliability is equal to the product of the reliability equations for all of the failure modes, or:

$$R(t) = R_1(t) \cdot R_2(t) \cdot \ldots \cdot R_n(t) \quad (1)$$

where n is the number of failure modes considered. Note that competing failure modes (CFM) analysis can be performed only when the product is non-repairable and when it can be assumed that the failure modes are statistically independent (i.e., the occurrence of one failure mode does not affect the probability of occurrence for the other failure modes). Furthermore, the Competing Failure Modes option that is built-in to the Weibull++ standard folio supports the analysis of up to four competing failure modes and applies the same distribution to analyze all of the modes. If your analysis does not fit these conditions, it may be more appropriate to use a reliability block diagram (RBD) analysis approach. *See Section 6.8.3.1 on page 242*.[13]

Tip: When it is not possible to categorize your data set into distinct modes, the mixed Weibull analysis may be more appropriate. *See Section 6.8.1 on page 233*.

Example

Consider a water pump that is considered failed when either its gasket or its valve fails. To use CFM analysis in a Weibull++ standard folio, you must categorize each failure time into distinct subset IDs. The following data sheet shows the categorized failure times.

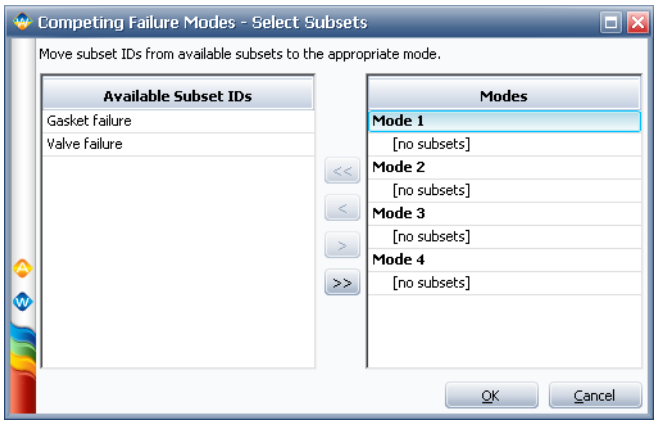
	State F or S	Time to F or S (Hr)	Subset ID
1	F	9	Gasket failure
2	F	95	Gasket failure
3	F	132	Gasket failure
4	F	156	Valve failure
5	F	196	Valve failure
6	F	220	Gasket failure
7	F	250	Valve failure

Note: The subset ID can be any text up to 30 characters, including spaces. For example, "A _ _ X" is not the same as "A _ X" where " _ " is used to designate a space.

On the Main page of the control panel, choose **Competing Failure Modes** and then choose a distribution that is appropriate to analyze the data for each failure mode (for this example, choose **CFM-Weibull**). Next, select **MLE** as

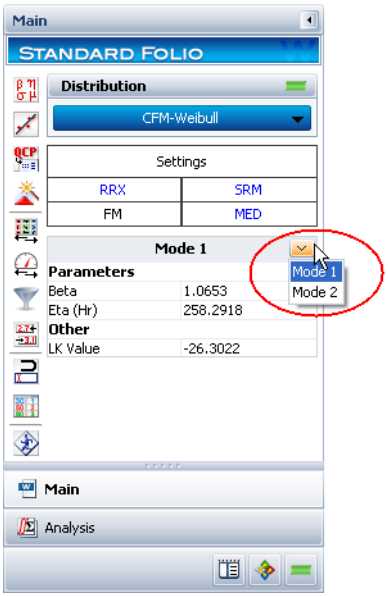
[13] The ReliaWiki resource portal provides more information about competing failure modes analysis at: http://ReliaWiki.org/index.php/Competing_Failure_Modes_Analysis.

the parameter estimation method and click **Calculate**. You will be presented with a list of all the subset IDs in your data sheet, as in the figure shown next.



You can define up to four modes to include in the CFM analysis, and each mode can be associated with one or more subset IDs.[14] First, select the subset ID(s) in the **Available Subsets** column. Next, select a mode from the **Modes** column and then click the arrow buttons to assign the selected subset ID(s) to the selected mode. You can also double-click or drag a subset ID to assign it to a mode.

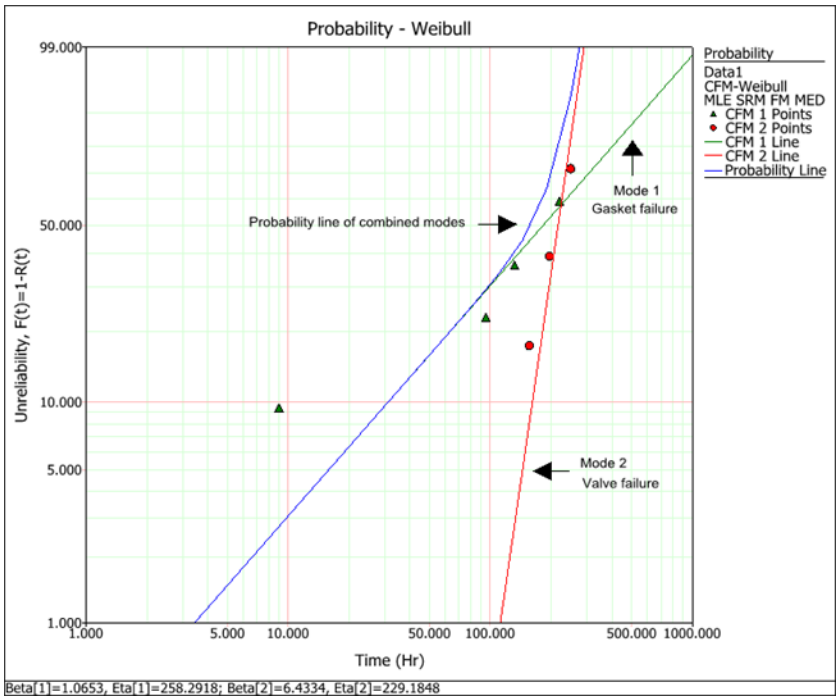
Assign "Gasket failure" to "Mode 1" and "Valve failure" to "Mode 2," and then click **OK**. Weibull++ will automatically calculate the parameters for each mode separately. You can view the parameters of a mode by clicking the arrow in the control panel and then choosing the mode from the drop-down list, as shown next.



To view a list of all the modes and their corresponding parameters, click anywhere within the parameter results area of the control panel. This opens the Results window, which displays the calculated results in a worksheet. From the Results window, you can edit, copy or print the results.

[14] For example, consider a product with components A, B and C, where components B and C are identical, used in two places in the product and subjected to identical stresses. In this case, components B and C are expected to exhibit identical failure modes; therefore, they can be analyzed together and grouped under a single mode for CFM analysis.

To visualize how each failure mode contributes to the water pump's overall probability of failure, click the **Plot** icon on the control panel. The solution of the probability line is based on Eqn. (1). All other plots and the calculations you perform via the Quick Calculation Pad (QCP) are also based on the same equation.



6.8.3.1 Using Reliability Block Diagrams (RBDs) for Failure Modes Analysis

As mentioned above, the competing failure modes (CFM) analysis options that are built into the Weibull++ standard folio (e.g., CFM-Weibull, CFM-lognormal, etc.) assume a simple series reliability model and use only one type of distribution to analyze all the modes. When your analysis is more complicated, Weibull++ offers a different method for failure modes analysis. The method described next applies in any or all of the following circumstances:

- The analysis involves more than four failure modes.
- The failure modes are described by different life distributions.
- The relationship between the failure modes does not follow a series configuration. For example, if more than one failure mode must occur together in order for the component to fail, a parallel configuration must be used for the analysis and this is referred to as "complex failure modes analysis" rather than "competing failure modes analysis."

Example

Using the same water pump system in the previous example, create an RBD that illustrates the relationship between the failure modes.

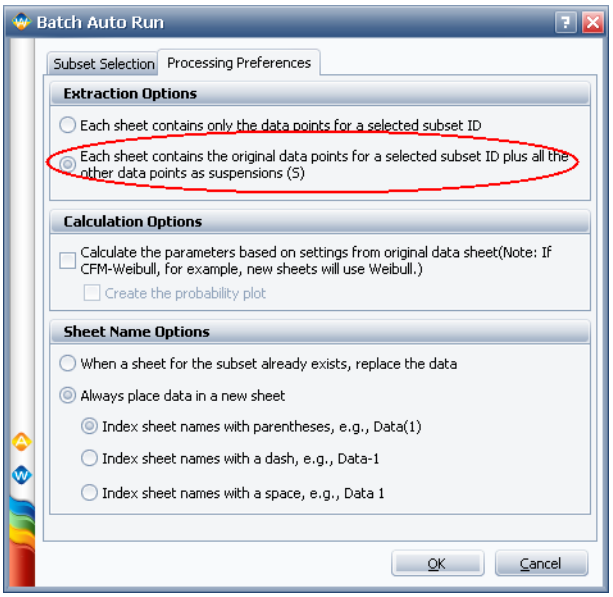
To analyze a data set that contains competing failure modes, the first step is to separately analyze the data due to each mode. When you analyze the data set of a particular mode, the failure times for all other competing modes are considered to be suspensions. This captures the fact that those units operated for a period of time without experiencing the failure mode of interest before they were removed from observation when they failed due to another mode. You can easily perform this step via the Batch Auto Run utility.

To do this, follow the steps below:

1. In the data sheet that contains the failure times for the gasket and valve, open the Batch Auto Run utility by choosing **Life Data > Options > Batch Auto Run** or by clicking the icon on the Main page of the control panel.



2. In the Batch Auto Run window, click the **Select All Available** button at the bottom of the window to automatically include all the modes in the batch auto run process.
3. Click the **Processing Preferences** tab and select the option shown next.

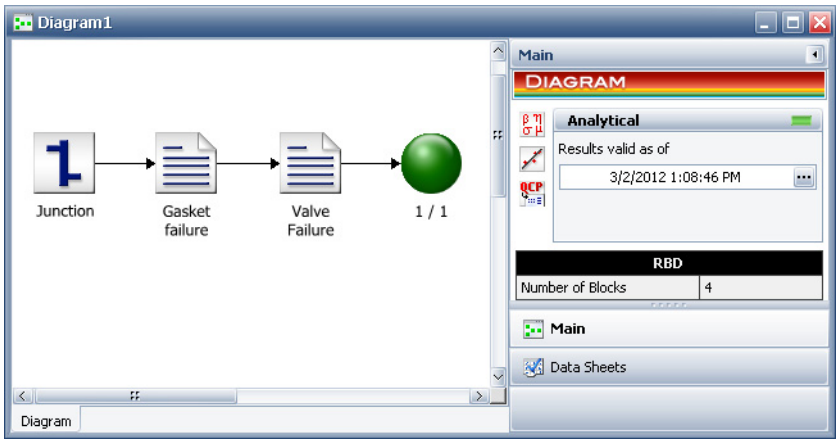


4. Click **OK**. The data set for each mode will be extracted to separate data sheets in the folio. Select an appropriate distribution for each failure mode and estimate the parameters for each separate data sheet. (For this example, use the **2P-Weibull** distribution and the **MLE** parameter estimation method for all the data sheets).
5. The next step is to describe the relationships between the failure modes. To do this, add an RBD to the project by choosing **Insert > Tools > Diagram**.



This creates a reliability block diagram (RBD) folio, which gives you the ability to create blocks that represent the failure modes. You can use arrows to connect the blocks in an appropriate configuration to describe the

relationship between the failure modes and show how each mode can result in a failure for the product. The following RBD shows that the water pump will fail if either the gasket or valve fails.



Following are brief descriptions on how to build the diagram shown here. *For complete instructions, see page 456 in Chapter 18.*

- Add a junction block by choosing **Diagram > Diagram Tools > Add Junction**. The junction block serves as a unique starting point for the diagram and has no properties associated with it.
- Add blocks by choosing **Diagram > Diagram Tools > Add Block**. You will be asked to select the data sheet of the failure mode that the block will represent.
- Add nodes by choosing **Diagram > Diagram Tools > Add Node**. The nodes act as switches that the diagram paths move through.
- Add arrows by choosing **Diagram > Diagram Tools > Connect Blocks**.

6. To analyze the diagram, choose **Diagram > Analysis > Analyze** or click the icon on the control panel of the RBD folio.



All reliability calculations performed via the QCP tool of the RBD folio will be based on the resulting reliability equation of the diagram.

Tip: ReliaSoft's advanced reliability block diagram software, BlockSim, can handle advanced configurations and analyses, such as dependent failure modes (i.e., the occurrence of one failure mode affects the probability of occurrence of the other modes).

6.8.4 Alter Parameters

The Alter Parameters feature is available for both Weibull++ and ALTA standard folios. It enables you to experiment with possible alternative scenarios by altering the estimated parameters of the fitted model.

After you calculate the parameters for your data set, you can access the tool by choosing **[Life Data/ALTA] > Options > Alter Parameters** and then selecting one of the following options:

- **Alter Parameters (w/o Recalculation)** allows you to alter the value of one (or more) parameters, while keeping the original values of the rest of the parameters and the variance/covariance matrix the same.
- **Alter One Parameter (and Recalculate)** allows you to alter one of the parameter values, and then have the software estimate the other parameter values that would result in a good fit for the model. In Weibull++ standard folios, this option is available only for data sets that were analyzed with the MLE analysis method and either the

2P- or 3P-Weibull distribution. If you wish to modify the value of the gamma parameter of the 3P-Weibull distribution, you must first select the **Use true 3-P MLE on Weibull** check box in the Calculations page of the Application Setup (**File > Application Setup**), and then estimate all three parameters using MLE. Note that the pure MLE solution is not always stable, and may fail for some data sets.

For both options, the LK values, solution line on plots and all subsequent analyses will be based on the modified set of parameters. To go back to the true parameters, click the **Calculate** icon on the folio's control panel.

6.8.5 Specify Points

The Specify Points feature enables you to experiment with possible alternative scenarios by allowing you to adjust the way that the line is fitted to the points for a rank regression analysis. The ranks will be calculated based on all of the data points but the regression line (which determines the parameter estimates) will be fitted only to the points you have specified. Any plots or QCP results will be based on the adjusted line, and therefore may be appropriate only within the context of your specific "what-if" analysis.

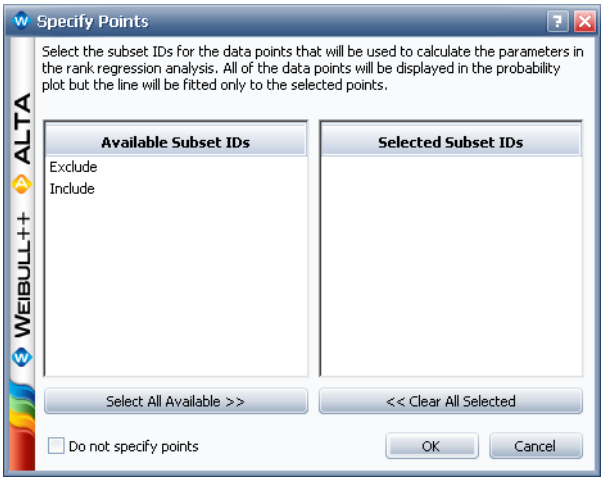
To use the Specify Points feature, you will first need to identify the data points that you wish to include in the regression line/parameter estimation. In a Weibull++ standard folio data sheet, assign the selected data points to a specific subset ID, as shown in the following example.

	Time Failed (Hr)	Subset ID
1	21.691	Exclude
2	27.2772	Exclude
3	32.6591	Exclude
4	87.4316	Include
5	95.2312	Include
6	108.8436	Include
7	124.4518	Include
8	152.9014	Include
9	166.4517	Include
10	220.5643	Include

Choose **Life Data > Options > Specify Points** or click the icon on the Main page of the control panel.



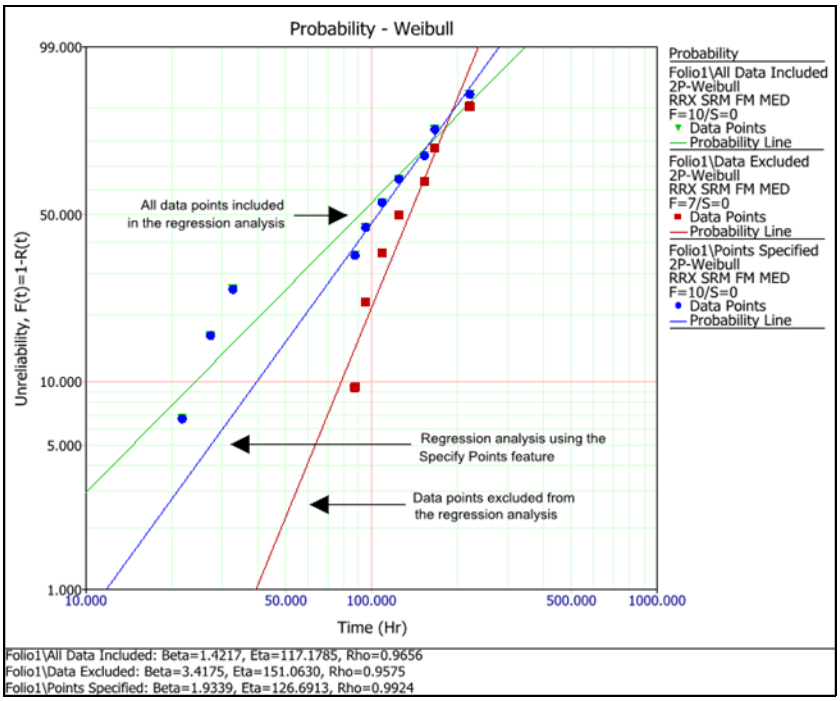
You will be presented with a list of all the subset IDs in your data sheet as shown next.



Double-click the subset ID of the data points you wish to include in the analysis (in this case, it is subset ID "Include"). You can also drag the subset ID to the **Selected Subset ID** column. Click **OK**.

Next, choose an appropriate distribution for the data set and calculate the parameters. The control panel will display the parameters of the regression model and also display a status to indicate that the standard analysis method has been altered and the parameters were calculated based on a regression line fitted only through the specified data points.

The overlay plot shown next illustrates the changes to the estimates of the parameters between the cases where the three data points are either included or excluded from the regression analysis.



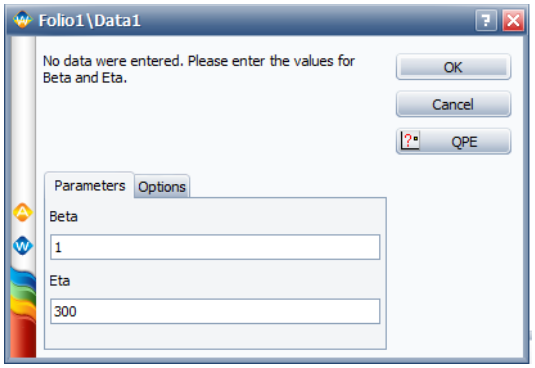
6.8.6 Calculations with No Data Entered

Weibull++ gives you the flexibility to generate plots and calculate metrics for reliability models without entering the data that the reliability model is based upon. To do this, create a Weibull++ standard folio data sheet, and without entering any data, choose a distribution and click the **Calculate** icon on the Main page of the control panel.



With no data specified, this opens an input window that allows you to enter the parameters for the selected model. After you enter the parameters and click **OK**, the parameters of the specified model will be displayed in the **Analysis**

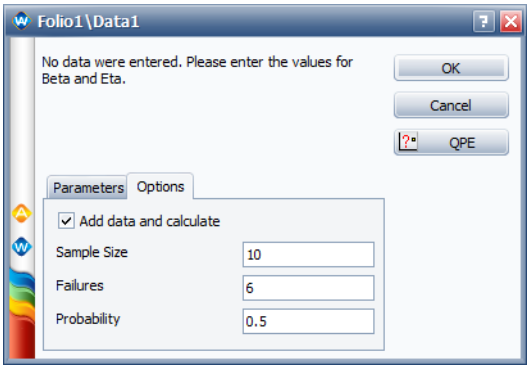
Summary area of the control panel. Any plots or QCP results you produce from this data sheet will be based on this model.



Tip: If you don't know the model parameters, you have the option to use the Quick Parameter Estimator (QPE) to estimate them based on what you know about the behavior over time. *For more information about the QPE, see page 194 in Chapter 5.*

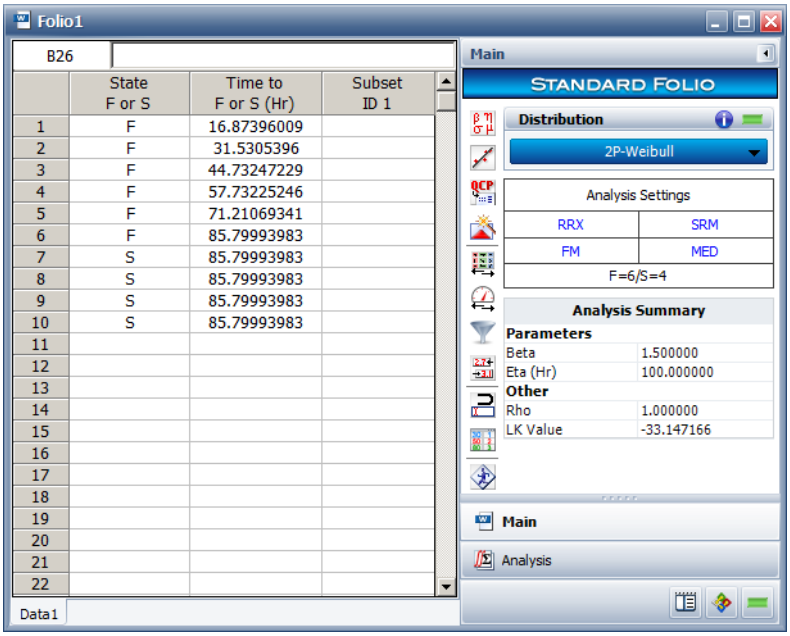
Generating a Data Set Based on the Specified Reliability Model

Weibull++ gives you the option to generate a data set based on the specified model. To do this, click the **Options** tab in the input window and then select the **Add data and calculate** check box, as shown next.



With this setting enabled, Weibull++ will generate a data set that is based on the specified model and your desired sample size, number of failures and probability. The probability value is the chance that the i^{th} failure will occur after the generated failure time T_i. For example, the following picture shows the generated failure times for a 2-parameter Weibull distribution using the following settings:

- Beta = 1.5
- Eta = 100
- Sample size = 10
- Number of failures = 6
- Probability = 50%



In this case, the 1st failure time (T_1) is estimated to be about 16.874 hours. This means that when you conduct a test on a product using the same sample size and reliability model, there is a 50% chance that the 1st failure will occur after 16.874 hours. Therefore, 16.874 can also be thought of as the one-sided lower bounds for the 1st failure time at a confidence level of 50%. The rest of the failure times in the data set can be read in a similar manner.

6.8.7 Fill Median Ranks

The Fill Median Ranks feature is available only for data sheets that have been configured to support free-form data.[15] It enables you to obtain the median ranks for a given set of failure times, allowing you to experiment with the unreliability estimates for each failure and see how they affect the solution of an analysis.

To generate median ranks, enter the times-to-failure data in the "X-Axis value" column of the data sheet, then choose **Life Data > Options > Fill Median Ranks**.



For each time-to-failure value in the "X-Axis value" column, the "Y-Axis value" column is filled with the corresponding median ranks value. On the control panel, choose a distribution and then choose **Life Data > Analysis > Calculate** or click the icon on the Main page of the control panel.



The software will compute the parameters of the distribution based on the data set. You can use the plots and the QCP to obtain results based on the calculated parameters; however, calculations involving confidence bounds are not available.

[15] The ReliaWiki resource portal has more information on median ranks at: http://www.ReliaWiki.org/index.php/Parameter_Estimation.

ALTA Standard Folios 7

ALTA Only

In basic life data analysis, units are tested under normal use conditions in order to obtain information about a product's failure behavior. In some cases, however, it is desirable to obtain this information more quickly than is possible with use level testing. For example, products with long lifetimes or short periods between design and release, as well as products that are operated continuously, may produce results too slowly with basic life data analysis.

The ALTA standard folio is used to perform quantitative accelerated life testing data analysis. In this analysis, the engineer extrapolates a product's failure behavior at normal conditions from life data obtained at accelerated stress levels. Since products fail more quickly at accelerated stress levels, this sort of analysis allows the engineer to obtain reliability information about a product (e.g., mean life, probability of failure at a specific time, etc.) in a shorter time.[1]

This chapter describes how to use the ALTA standard folio, including:

- Setting up an ALTA standard folio - Section 7.1 (p. 250).
- How to analyze life data, including:
  - Using the control panel - Section 7.2 (p. 253).
  - Generating ALTA plots - Section 7.3 (p. 257).
  - Performing reliability calculations via the Quick Calculation Pad (QCP) - Section 7.7 (p. 260).
- Example of a single-stress accelerated life testing data analysis - Section 7.8 (p. 263).
- Example of a two-stress accelerated life testing data analysis - Section 7.9 (p. 265).
- Working with time-varying stresses, including:
  - Using stress profiles to represent time-varying stresses - Section 7.10 (p. 267).
  - Example of an analysis with time-varying stresses - Section 7.10.3 (p. 271).
- Additional analysis tools, including:
  - Choosing a life distribution using the Distribution Wizard - Section 7.11.1 (p. 274).
  - Adding and removing stress and subset ID columns - Section 7.11.2 (p. 274).
  - Performing a likelihood ratio test - Section 7.11.3 (p. 275).
  - Transferring data to a Weibull++ standard folio - Section 7.11.4 (p. 277).
  - Converting stress values from one unit of measurement to another - Section 7.11.5 (p. 278).
  - Performing calculations with insufficient test data - Section 7.11.6 (p. 279).

1. The ReliaWiki resource portal has more information on the basics of accelerated life testing at: http://www.ReliaWiki.org/index.php/Introduction_to_Accelerated_Life_Testing.

7.1 ALTA Standard Folio Setup

The ALTA standard folio contains the tools for performing an accelerated life testing data analysis. A setup window for configuring your folio will be displayed when you:

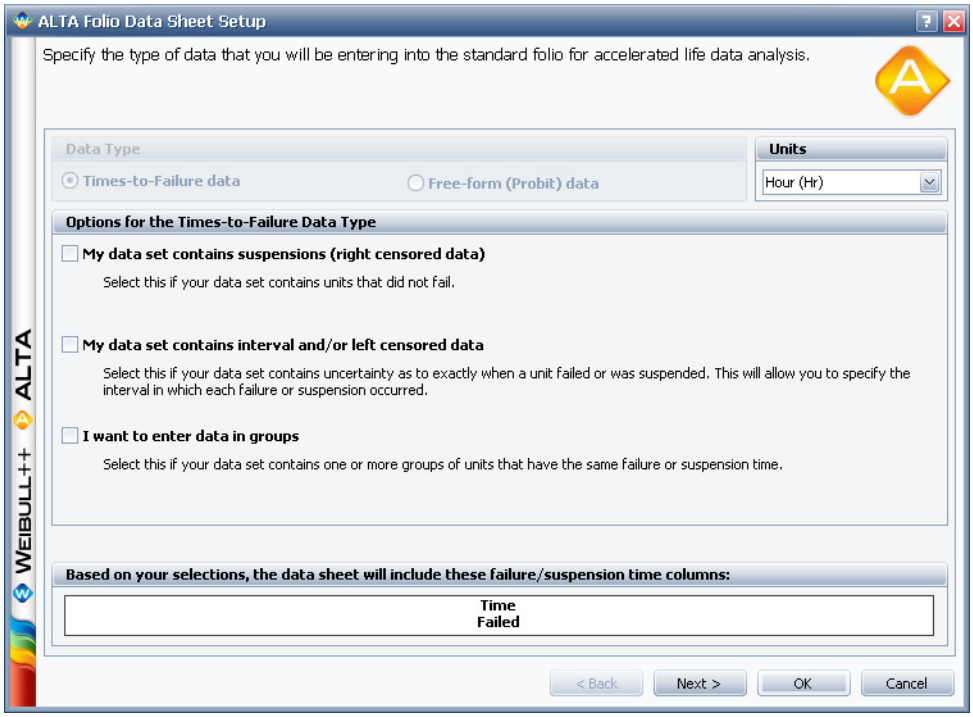
- Add an ALTA standard folio to an existing project by choosing **Insert > Folios > ALTA Standard Folio**. You can also right-click the **ALTA Folios** folder in the current project explorer and choose Add Standard Folio on the shortcut menu.



- Change the types of data that are included in your data sheet by clicking the **Alter Data Type** icon on the control panel, or by choosing **ALTA > Format and View > Alter Data Type**.



The first page of the Data Sheet Setup window is shown next.



This page of the window is used to select what types of data can be included in the data sheet. The available options are described next. *For a more detailed description of data types, see page 207 in Chapter 6.*

- The **My data set contains suspensions** option allows you to record data for units that did not fail during the observation period. These data points are known as *suspensions*. The suspension time is the last time the unit was observed as not failed. In the example shown next, a test was conducted and five units in the sample failed, but one unit (marked with an "S" in the first column) was still operational when the test terminated at 7,500 hours.

	State F or S	Time to F or S (Hr)	Temperature K
1	F	6580	393
2	F	7140	393
3	S	7500	393
4	F	4560	408
5	F	4920	408
6	F	5280	408

- The **My data set contains interval and/or left censored data** option allows you to record data when there is uncertainty as to the exact time when a unit fails. This type of data frequently comes from situations where the units are not constantly monitored. For example, if you are running a test on five units and inspecting them every 100 hours, you only know that a unit failed or did not fail between inspections. (If you select to enter both suspensions and interval data, you can enter suspensions as intervals.) *Left censored data* is a special case of interval censored data. With left censored data, the interval is between time = 0 and some inspection time.

  In the example shown next, the first unit was operational at the start of the test but found failed at 150 hours (thus, it is a left censored data point). The second unit was still operating at 150 hours, but it failed at some point prior to the next observation at 300 hours, and so on.

	Last Inspected (Hr)	State End Time (Hr)	Temperature K
1	0	150	408
2	150	300	408
3	300	450	408
4	450	600	393
5	600	750	393
6	750	900	393

- The **I want to enter data in groups** option allows you to group multiple data points together. The following is an example of a grouped data set with complete, suspended, interval and left censored data. (Grouping in this way significantly speeds up calculations when you have a very large data set.)

	Number in State	Last Inspected (Hr)	State F or S	State End Time (Hr)	Temperature K
1	2	0	F	24	408
2	10	24	F	48	408
3	1	48	F	96	408
4	1	100	F	100	393
5	2	96	F	120	393
6	5	120	S	150	393

  The first row represents 2 left censored data points (2 units were observed at 24 hours and found to be failed). The fourth row contains an exact failure time (failure at 100 hours) for 1 unit. The sixth row represents an interval suspension for 5 units that were found operational at 120 hours and then also at 150 hours. The remaining rows represent interval data for units that failed during the test.

Note that when you select a data type, the preview table at the bottom of the setup window will show you the name of the data entry columns that will appear in the data sheet.

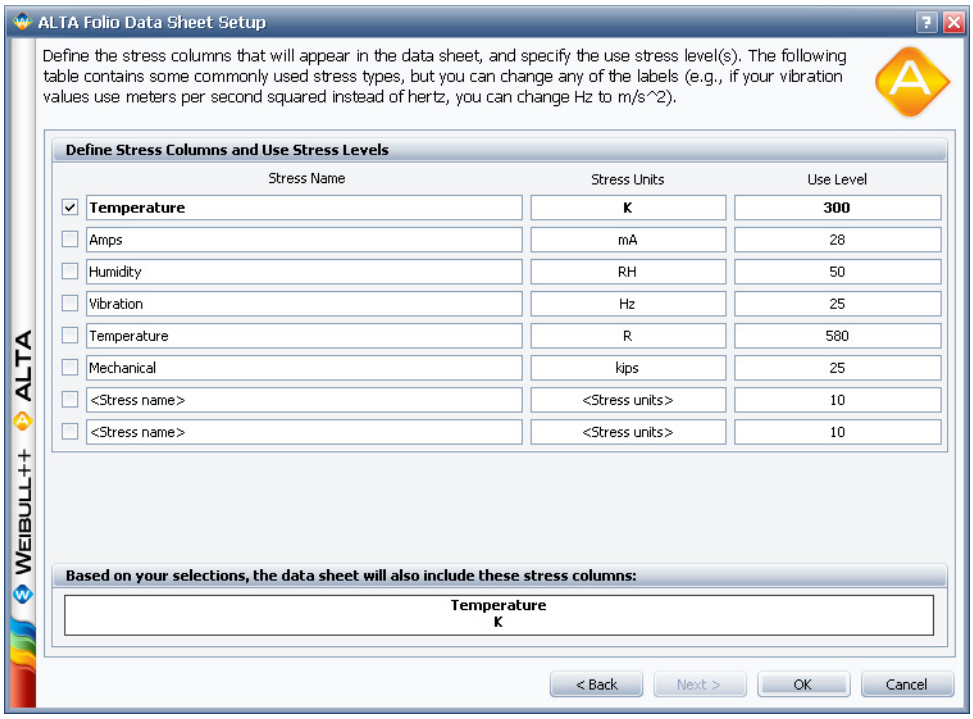
If you are creating a new folio, you will also need to specify the time units appropriate for your data. Units may be measured in time, distance, weight, etc. The appropriate columns in the data sheet will be automatically configured for the units you select. If you later wish to change the time units of an existing data sheet, click the **Change Units** icon on the Main page of the control panel of the data sheet. *See page 113 in Chapter 3*.



Note: In the Synthesis version, you now 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. *See page 39 in Chapter 2*.

If you wish to use the same stress column and stress level settings that were used when you last set up an ALTA standard folio—or if you are only changing the data type of an existing folio—click **OK** to close the window and view your folio. Otherwise, click **Next>** to display the second page of the window.



You can use this page to do the following:

- Adjust the number of stress columns to be used in your folio by selecting the appropriate check boxes.
  - Every stress column represents a stress type. So, for example, a data sheet with two stress columns would be used for data obtained from a two-stress test. In this case, each row would contain a failure/suspension time and the two stress values that define the stress level at which the data point was obtained (e.g., a unit might have failed at 100 hours while tested at 343 K and 80% relative humidity).
  - If you would like to change the number of stress columns in a folio you have already created, click the **Add or Remove Columns** icon on the Main page of the folio's control panel.



- Change the labels of each stress column by changing the text under the **Stress Name** or **Stress Units** fields. These fields are merely for adding text to the headers of the stress columns, so you can keep track of which stress type is associated with each stress column. The information is not relevant to the software's calculations.
- Define the product's normal stress level by entering values under the **Use Level** heading for each stress type (e.g., the product that was tested under accelerated temperature and humidity might normally operate at 323 K and 50% humidity). After the folio is created, you can change this value in the Use Stress Level window. *See Section 7.6 on page 260*.

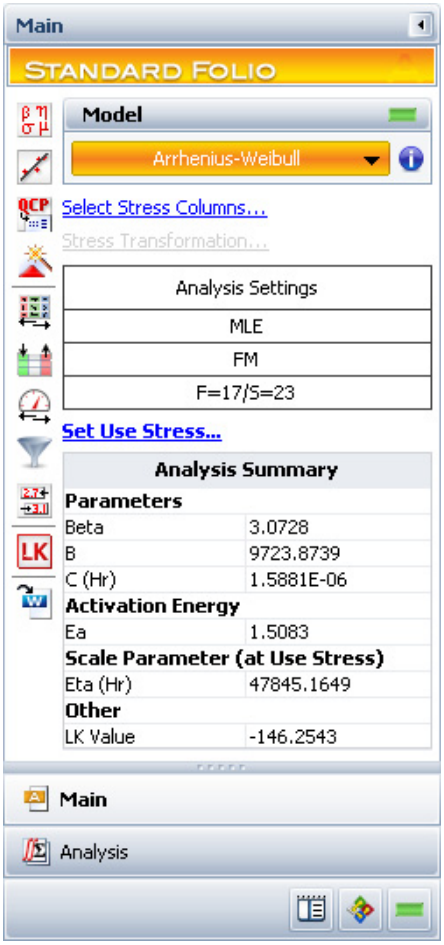
Like the first page of the setup window, there is a preview at the bottom of the page that shows you how the names of the associated columns will appear in the data sheet.

IMPORTANT: Temperature values must always be entered in absolute units (e.g., Kelvin or Rankine). To convert temperature values in the data sheet that were entered in degrees Celsius or Fahrenheit to absolute units, choose **ALTA > Options > Convert Stress Values**.

7.2 ALTA Standard Folio Control Panel

The ALTA standard folio control panel allows you to configure the analysis settings for the data sheet and view/access the results. It consists of multiple pages, each containing options for performing particular tasks. This section focuses on the Main page of the control panel, which contains most of the tools you will need to perform accelerated life testing analysis. *For more information about the control panel in general, see page 114 in Chapter 3*.

The Main page of the ALTA standard folio control panel is shown next.



The Main page includes the following settings:

- The **Model** drop-down list allows you to select the life-stress relationship and life distribution that will be used to extrapolate failure behavior at normal operating conditions. The selected life distribution is fitted to the data obtained at each accelerated stress level used in the test, and then the selected life-stress relationship is used to extrapolate the probability density function (*pdf*) at the use stress level from the *pdfs* at the accelerated levels.

  What's Changed? In previous versions of ALTA, the life-stress relationship and life distribution were selected separately, in two different drop-down lists. Now, a single drop-down list is used to select a single model that combines both. For example, if you choose **Arrhenius-Weibull** from the drop-down list, your analysis will use a combination of the Arrhenius life-stress relationship and the Weibull life distribution.

  While any distribution can be used to analyze a data set, if the distribution does not fit the data well, your results may contain too much uncertainty. Moreover, each of the available life-stress relationships is designed to work with specific stress types (e.g., the Arrhenius relationship is designed for data obtained from accelerated

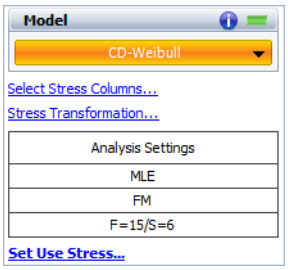
temperatures). Therefore, to ensure accurate predictions for the product's use level reliability, the appropriate life distribution and life-stress relationship must be selected.[2] There are five life-stress relationships to select from in ALTA Standard, and there are four additional relationships in ALTA PRO:

- The five life-stress relationships in ALTA Standard are:
  - **Arrhenius**: a single-stress model typically used when temperature is the accelerated stress.
  - **Eyring**: a single-stress model typically used when temperature or humidity is the accelerated stress.
  - **Inverse power law (IPL)**: a single-stress model typically used with a non-thermal stress, such as vibration, voltage or temperature cycling.
  - **Temperature-humidity (TH)**: a double-Arrhenius model that is typically used when temperature and humidity are the acceleration variables.
  - **Temperature-nonthermal (TNT)**: a combination of the Arrhenius and IPL relationships that is typically used when one stress is temperature and the other is non-thermal (e.g., voltage).
- In addition, ALTA PRO offers four other models for more advanced applications:
  - **Generalized Eyring (GER)**: a variation of the Eyring relationship that is typically used when both temperature and humidity are accelerated.
  - **Proportional hazards (PPH)**: analyzes data with up to 8 simultaneous stress types using the exponential relationship for each stress. It also allows the use of zero as a stress value, which enables the analysis of data with indicator variables (e.g., 0 = on/off and 1 = continuous operation).
  - **General log-linear (GLL)**: supports the analysis of data with up to 8 simultaneous stress types and provides the flexibility of specifying the life-stress relationship (Arrhenius, IPL or exponential) for each stress. Like the PPH model, this model also allows for the analysis of data with indicator variables.
  - **Cumulative damage (CD)**: analyzes data with up to 8 stress types and/or situations where the stress varies with time.

The three available life distributions are 2-parameter Weibull, lognormal (not available with the proportional hazards relationship) and 1-parameter exponential.

> **Tip:** If you are not certain which life distribution would provide the best fit for your data, you can use the Distribution Wizard to guide you through the selection process. The choice of the life-stress relationship depends on various factors, including the types and number of stresses used in the test. *See Section 7.11.1 on page 274*.

- Directly underneath the Model drop-down list are options for selecting the stress columns (*see Section 7.4 on page 259*) and, when applicable, the stress transformations that will be used in the analysis (*see Section 7.5 on page 260*), as shown next.



- The **Analysis Settings** area (shown in the picture above) displays “MLE” and “FM” to indicate that the maximum likelihood estimation (MLE) method will be used to estimate your model’s parameters and the Fisher matrix (FM) method will be used to solve for the confidence bounds. In ALTA, these settings cannot be changed. This area

2. The ReliaWiki resource portal has more information on the distributions and life-stress relationships at: http://www.ReliaWiki.org/index.php/Accelerated_Life_Testing_Data_Analysis_Reference.

also displays the number of failures and suspensions in your data set (e.g., "F=17/S=23" means there are 17 failures and 23 suspensions).

- The **Set Use Stress** link directly underneath the Analysis Settings area allows you to define the normal stress level for every stress that is selected to be used in the analysis. *See Section 7.6 on page 260*.
- The **Analysis Summary** area displays the calculated parameter values and other values for your selected model. The following example shows the calculated parameters of a CD-Weibull model (i.e., the cumulative damage life-stress relationship combined with the Weibull life distribution) for a particular data set.

Analysis Summary	
Parameters	
Beta	0.731186
Alpha	
Alpha(0) (Hr)	147.345975
Alpha(1)	-19.945776
Scale Parameter (at Use Stress)	
Eta (Hr)	1.111139E+44
Other	
LK Value	-104.253233
ALTA_Folio1_Data1	

You can click anywhere within the **Analysis Summary** area to open the Results window, which displays the calculated results in a worksheet. From the Results window, you can edit, copy or print the results.

Note: If the model and parameters are associated with a published model, then the published model's name will appear as a link at the bottom of the **Analysis Summary** area. Click the link to view the model's properties. *For details on how to publish the results as a model, see page 116 in Chapter 3.*

Depending on your settings in the Application Setup (**File > Application Setup**), the following information may also be displayed:

- The **Ea** (activation energy) value represents the effect of temperature. It appears when temperature-related models are used, such as Arrhenius, Eyring, generalized Eyring, temperature-humidity, and temperature-nonthermal. A large activation energy means temperature has a large effect on the life of the product.
- The **Scale Parameter** value represents the product's characteristic life.
  - 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.
- The **LK Value** (likelihood function value) is the logarithm of the likelihood value for the model parameters. It can help you compare how well different models fit the same data set, as described below.

Additional Tools

Accelerated life testing results are estimates based on observed data points from a limited sample size and obtained at increased stress levels. For this reason, there is always some uncertainty in the results. It is therefore important to evaluate how well the chosen model fits the data set. There are several ways to evaluate the model's fit and compare it to other models. To make a sound assessment, the following tools may be used in conjunction with your own engineering knowledge about the model:

- The **Use Level Probability** and **Standardized Residuals** plots show you how well the solution line tracks the plotted data points. This allows you to visually assess the fit of the selected model. *See Section 7.3 on page 257*.
- The **Likelihood Ratio Test** is used to confirm the assumption of a common shape parameter among the data obtained at the various accelerated stress levels. This assumption underlies the use of any model that includes the Weibull or lognormal life distribution. *See Section 7.11.3 on page 275*.

- **Confidence Bounds** allow you to quantify the amount of uncertainty in the parameter estimates and other functions, such as reliability predictions, due to sampling error. The narrower the confidence bounds at a given confidence level, the closer the estimates are to the true value of the parameters and the better the precision on the reliability predictions. *See page 392 in Chapter 14*.

  You can visualize the confidence bounds by using plots, or you can view the calculated values of the confidence bounds by using the Quick Calculation Pad.

- The **LK Value** (likelihood function value) allows you to compare how well different models fit the same data set. The model with the highest LK value is considered to have the best fit statistically for that data. However, because this value is influenced by the sample size and the variability of the data set, it varies across different data sets and cannot be used by itself to evaluate a particular model.

  - The LK value for the selected model is automatically displayed on the Main page of the control panel.
  - To view a 3D plot of the likelihood surface, choose **ALTA > Analysis > 3D Plot**, and then choose **Likelihood Function** from the **Plot Type** drop-down list. *See page 399 in Chapter 14*.
  - ALTA's Distribution Wizard uses LK values to automatically compare life distributions. The Wizard assumes the life-stress relationship selected on the control panel. *See Section 7.11.1 on page 274*.

7.2.1 Folio Tools

The folio tools are arranged on the left side of the Main page of the control panel. Use these tools to manage data and experiment with the results of your analysis.



Calculate estimates the parameters of the chosen model, based on the current data set and the specified analysis settings. This tool is also available by choosing **ALTA > Analysis > Calculate**.



Plot creates a new sheet in the folio that provides a choice of applicable plot types. For standard folios, this includes plots such as Use Level Probability, Reliability vs. Time, Life vs. Stress, etc. You can also create a Plot sheet by choosing **ALTA > Analysis > Plot**. *See Section 7.3 on page 257*.



QCP opens the ALTA Quick Calculation Pad, which allows you to calculate results, such as the mean life and the probability of failure, based on the currently active data sheet. This tool is also available by choosing **ALTA > Analysis > Quick Calculation Pad**. *See Section 7.7 on page 260*.



Distribution Wizard opens the Distribution Wizard, which helps you select the life distribution that will best fit your data. This tool is also available by choosing **ALTA > Analysis > Distribution Wizard**. *See Section 7.11.1 on page 274*.



Alter Data Type opens the Data Sheet Setup window, which allows you to change the data entry columns in the current data sheet. Note that when you change the setup, you may lose information already entered in the data sheet. For example, switching the data sheet from one that supports interval data to one that supports only right censored data will remove all the inspection times. This tool is also available by choosing **ALTA > Format and View > Alter Data Type**. *See Section 7.1 on page 250*.



Add or Remove Columns allows you to add/remove columns for stress levels and unit IDs. *See Section 7.11.2 on page 274*.



Change Units opens the Change Units window, which allows you to change the units for the time values in the current data sheet. *See page 113 in Chapter 3*.



Auto Group Data opens the Auto Group Data window, which allows you to group together data points with the same failure/suspension times. Grouping data significantly speeds up calculations when you have a very large data set. *See page 232 in Chapter 6*.

Alter Parameters (also called Alter Parameters w/o Recalculation) allows you to manually alter the value of one (or more) parameters, while keeping the original values of the rest of the parameters and the variance/covariance matrix the same. The LK value, the solution line on plots and all subsequent analyses will be based on the modified set of parameters.

Likelihood Ratio Test opens the Likelihood Ratio Test tool, which allows you to assess the assumption of a common shape parameter across all the stress levels used in the test. This assumption underlies the use of any model that includes the Weibull or lognormal life distribution. You can also open this tool by choosing **ALTA > Options > Likelihood Ratio Test**. *See Section 7.11.3 on page 275*.

Transfer Data to Weibull++ allows you to transfer data from the ALTA standard folio to a Weibull++ standard folio. *See Section 7.11.4 on page 277*.

7.3 ALTA Plots

ALTA includes multiple plot types you can use to visualize the results of your accelerated life testing data analysis. You can create plots by choosing **ALTA > Analysis > Plot** or by clicking the icon on the Main page of the control panel.



Note that clicking the **Plot** icon before the parameters have been calculated will automatically calculate the parameters and then plot the data. The scaling, setup, exporting and confidence bounds settings are similar to the options available for all other plot sheets. *To learn how to customize a plot, see Chapter 15*.

Tip: ALTA includes two additional plots you could use across all types of data folios: the overlay plot, which allows you to compare different data sets; and the side-by-side plot, which allows you to simultaneously view multiple plots of one data set in a single window. *See Chapter 14*.

The following is a description of the different types of plots that can be created in an ALTA standard folio. Note that the use stress level for plots is by default the level you entered on the Main page of the control panel. However, you may adjust the level for plots by clicking **Set Use Stress** directly underneath the **Analysis Summary** area of the Plot page. This will not change the use stress level specified on the Main page.

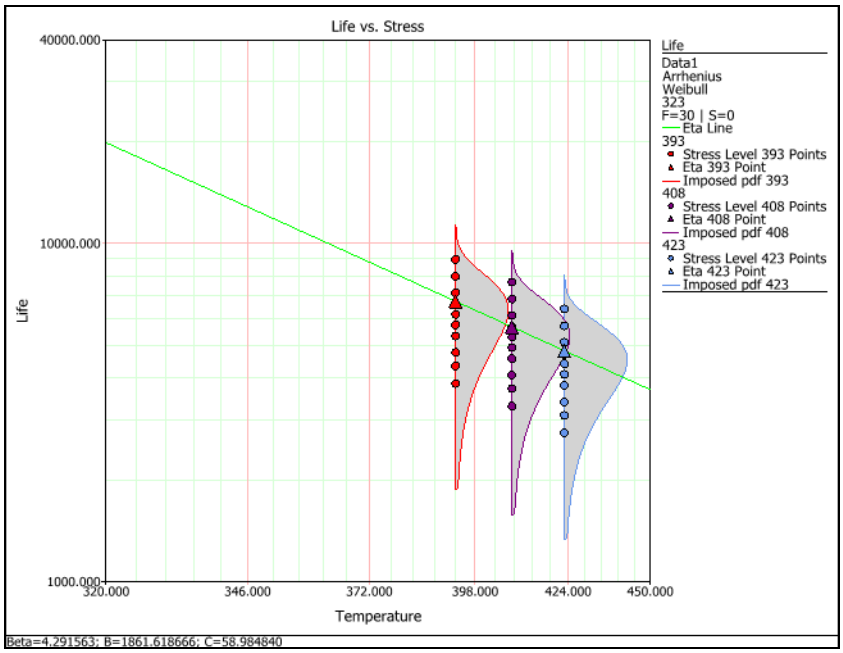
- The **Use Level Probability** plot shows the trend in the probability of failure over time at the specified use stress level. The plot also shows data points from the test that are transformed from the accelerated stress levels to the use stress level. The relationship between unreliability and time is linearized wherever possible, which results in non-linear axis scales. Like the standardized residuals plot, this plot is useful for comparing models that use the same life-stress relationship.
- The **Probability** plot shows the probability of failure (i.e., unreliability) as a function of time at every stress level used in the test. The plot also displays the extrapolated line for unreliability at the specified use stress level.

Note: Unlike the probability plots for other distributions, the y-axis in an exponential probability plot always indicates the reliability instead of the unreliability. This tradition arose from the time when probability plotting was performed "by hand." The exponential reliability model starts with R = 1 at T = 0 (or gamma). Thus, if the unreliability were plotted, the axis would start at Q = 1 - R = 0, which is not possible, given that the y-axis scale is logarithmic.

- The **Reliability vs. Time** plot shows the reliability values over time at the specified use stress level, capturing trends in the product's failure behavior. The plot also shows data points from the test that are transformed from the accelerated stress level to the use stress level.
- The **Unreliability vs. Time** plot shows the probability of failure of the product over time at the specified use stress level. Unlike the Use Level Probability plot, the axis scales are linear.

- The ***pdf* Plot** shows the probability density function of data over time at the specified use stress level. This allows you to visualize the distribution of the data set.
- The **Failure Rate vs. Time** plot shows the failure rate of the product over time at the specified use stress level.
- The following plots show the relationship between stress level and another reliability metric. In multi-stress situations, one stress is varied and the remaining stresses are fixed. Click **Set Use Stress** on the Plot page of the control panel to select the varied stress.
  - The **Life vs. Stress** plot shows the effect of a stress on the life of the product. Multiple *pdfs*, each at a different stress level used in the test, are displayed on the plot. The failure times obtained at each stress level are shown at the base of the associated *pdf*. Note that the lines and *pdfs* are mapped to the use stress level, but the failure times are plotted at the tested conditions.

    If a model that uses the Weibull distribution is selected, an eta *line* will be displayed as well. The eta line estimates the time by which 63.2% of units in the population are expected to fail. The plot may also include other life lines that show the relationship between stress level and the time by which other specified percentages of a population are expected to fail (*see Section 14.5 on page 395*). The life-stress relationship is linearized whenever possible.

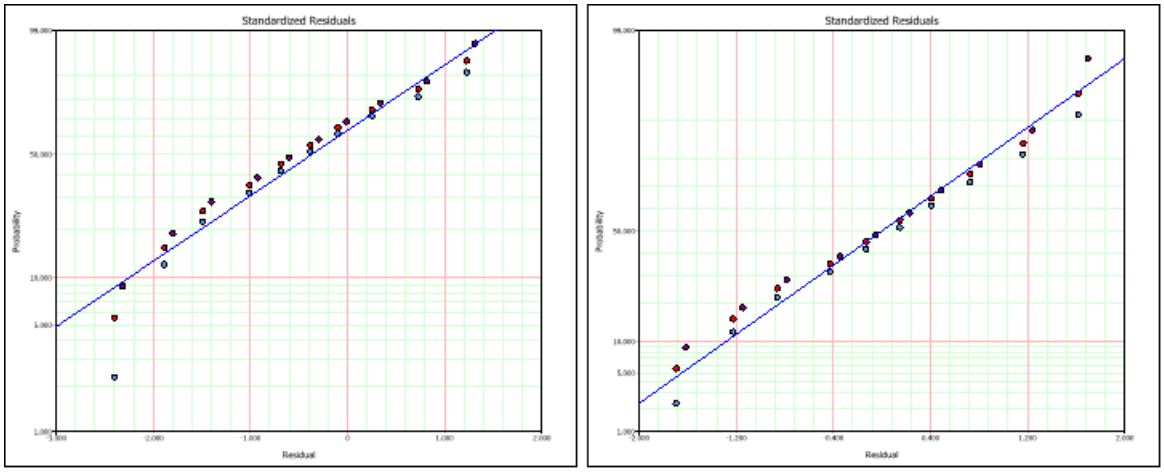


  - The **Std vs. Stress** plot shows the standard deviation of failure time as a function of stress level, thus providing information about the spread of data at every given stress level on the x-axis.
  - The **AF vs. Stress** plot shows the acceleration factor as a function of stress level. The acceleration factor is determined by dividing the product's life under normal operating conditions by its life at an accelerated stress level. For example, if the acceleration factor is 2 at an accelerated stress level of 350 K, then the product's life at 350 K is expected to be half of its life at the specified use stress level.
- The following plots are residual plots. In these plots, a residual value for each data point is displayed. As a result, these plots are useful for assessing model assumptions, revealing inadequacies in the model and revealing any extreme observations.[3]

3. The ReliaWiki resource portal has more information on residual plots at: http://www.ReliaWiki.org/index.php/Accelerated_Life_Testing_and_ALTA#Residual_Plots.

- The **Standardized Residuals** plot is useful for determining the adequacy of the selected model for the data. The appropriate probability transformation is given on the y-axis and the values of the residuals are given on the x-axis. The residual values for each data point are color-coded to indicate which accelerated stress level the associated data point was obtained from. If the model adequately fits the data, the points should track the plot line.

  For example, the next figures show the results of an Arrhenius-Weibull model and an Arrhenius-lognormal model using the same data set. As you can see, the plot shows that the lognormal distribution presents the better fit to this particular data set.



Arrhenius-Weibull Model Arrhenius-Lognormal Model

- The **Cox-Snell Residuals** plot is similar to the standardized residuals plot, except the line is plotted on an exponential probability plotting paper and is on the positive domain.
- The **Standard vs. Fitted Value** plot helps to detect behavior that isn't modeled in the underlying relationship. It plots the standardized residuals versus the scale parameter of the underlying life distribution (which is a function of stress) on log-linear paper (linear on the y-axis). Note that when heavy censoring is present, the plot is more difficult to interpret.

7.4 Select Stress Columns

The Select Stress Columns window is used to select which of the stress columns in your data sheet will be used in the analysis. When applicable, this window also includes drop-down lists that allow you to categorize the selected stresses. To access the window, click **Select Stress Columns** directly underneath the **Model** area of the control panel.

All of the stress columns in your data sheet are listed in the **Available Stress Columns** area. To use a column in the analysis, select the corresponding check box. Note that the text directly above the **Available Stress Columns** area provides information on how many stresses are required and how many can be used with the selected model.

Tip: You can also choose which stress columns to include in the analysis by selecting the check boxes in the column headers of the data sheet.

If you chose a temperature-nonthermal (TNT) or generalized Eyring (GER) model on the control panel, you will be required to categorize each stress as thermal or nonthermal. To use either of these models, you must select exactly two stress columns: one for a thermal stress and one for a nonthermal stress.

7.5 Stress Transformation

When you select a general log-linear (GLL) or cumulative damage (CD) model, you must use the Stress Transformation window to choose one transformation for each stress that will be used in the analysis. To access this window, click **Stress Transformation** immediately above the **Analysis Settings** area of the control panel.

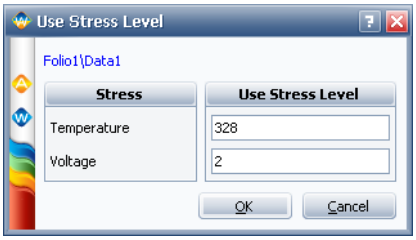
The **Transformation** drop-down list provides the following three options for how the stress will be transformed within the model.

- Use the **None X=V** transformation for stress types associated with the exponential life-stress relationship (LSR). This transformation is commonly used for indicator variables (e.g., 0 = on/off and 1 = continuous operation).
- Use the **Reciprocal X=1/V** transformation for stress types associated with the Arrhenius LSR. This transformation is commonly used for thermal stresses.
- Use the **Logarithmic X=ln[V]** transformation for stress types associated with the inverse power law LSR. This transformation is commonly used for non-thermal stresses.

7.6 Use Stress Level

The Use Stress Level window is used to define the normal stress levels for every stress that is selected to be used in the analysis. This value can also be defined during the setup of the standard folio. To access the window, click **Set Use Stress** directly underneath the **Analysis Settings** area of the control panel.

The **Stress** area of the window lists every stress that will be used in the calculation. In the **Use Stress Level** area, enter the stress level that the product will experience under normal conditions. For example, to specify that the normal operating temperature for a product is 328 K and the normal operating voltage is 2 V, you would enter the following into the window:



If the analysis is for a cumulative damage (CD) model, you will also be able to use stress profiles to describe time-varying stress conditions that the product will experience during normal use. *See Section 7.10.2 on page 270*.

7.7 QCP Calculations for ALTA

ALTA includes a Quick Calculation Pad (QCP) for computing useful metrics. You can access the QCP by clicking its icon on the Main page of the control panel.



To perform a calculation, select the appropriate option and enter any required inputs in the **Input** area, then click **Calculate**. *For more detailed information on how to use the QCP in general, see page 219 in Chapter 6*.

The following calculations are available for accelerated life testing data analysis:

Probability

- **Reliability**

  Calculates the probability that a new product will operate without failure for a given period of time at the stress level specified in the **Stress** field. 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 units are fielded, then 90 of them will still be operating at the end of 3 years.

- **Probability of Failure**

  Calculates the probability that a new product will be failed in a given period of time at the stress level specified in the **Stress** field. 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 product will successfully operate at a specific time interval given that it has operated successfully up to a specified time and at the stress level specified in the **Stress** field. Enter the start time of the interval in the **Mission Start Time** field and enter the length of the interval in the **Mission Additional Time** field.

  For example, a product may have a reliability of 90% for 3 years if it operates at a stress level of 10 volts. If the product 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%.

- **Conditional Probability of Failure**

  Calculates the probability that a product will be failed at a specific time interval given that it has not failed up to a specified time and at the stress level specified in the **Stress** field. Enter the start time of the interval in the **Mission Start Time field** and enter the length of the interval in the **Mission Additional Time** field.

  For example, a product may have a 10% probability of failure for 3 years if it operates at a stress level of 10 volts. 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 at the stress level specified in the **Stress** field. 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 units are fielded, then 90 of them will be still be operating at the end of 4 years.

  What's Changed? In previous versions of ALTA, this calculation was called "Warranty Time."

- **BX% Life**

  Calculates the estimated time at which a specified probability of failure will be achieved at the stress level specified in the **Stress** field. Enter the probability of failure in the **BX% Life At** field. For example, a B10 life of 4 years means 10% of the fielded units are expected to be failed at the end of 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 life, B5 life, etc.).

- **Mean Life**

  Calculates the average time at which a product is expected to operate before failure at the stress level specified in the **Stress** field. In the standard folio, the mean life is the mean time to failure (MTTF) based on the fitted model.

  Note: The term *mean time to failure* (MTTF) is used as a metric for the analysis of *non-repairable* components. In the ALTA standard folio, all data are assumed to come from non-repairable components that are independent and identically distributed (i.i.d.). On the other hand, the term *mean time between failures* (MTBF) is used as a metric in repairable systems analysis, where the same system may fail and be repaired multiple times. Simple repairable system data can be analyzed in Weibull++ using the Recurrent Event Data Analysis (RDA) folio. *See Chapter 13*.

  For more complex repairable system analyses, see ReliaSoft's BlockSim and RGA++ software.

Rate

- **Failure Rate**

  Calculates the instantaneous number of failures per unit time that can be expected at a certain time and at the stress level specified in the **Stress** field, given that a unit survives to that age. Enter the time at which you wish to calculate the failure rate in the **Mission End Time** field.

  For example, a failure rate of 0.01 at 100 hours and at a stress level of 10 volts means that each unit that survives to 100 hours has approximately a 1% probability of failure in the next hour.

Acceleration

- **Acceleration Factor**

  Calculates the ratio of the product's use level life to its life at an accelerated stress level. For example, if the product has a life of 100 hours at the use stress level, and it has a life of 50 hours at an accelerated level, then the acceleration factor at the specified stress levels would be 2. Click the arrow in the **Stress** field to enter the use level stress values for every stress that was used to calculate the data sheet. Click the arrow in the **Acceleration Stress** field to enter the accelerated stress values.

Bounds

- **Parameter Bounds**

  Calculates the specified bounds on the parameter estimates, allowing you to quantify the amount of uncertainty in those estimates. This option is available only when you have specified the type of confidence bounds to use from the **Bounds** drop-down list. When you click **Calculate**, the Results Window will open to display the estimated parameters and their bounds.

7.8 ALTA Single-Stress Example

A manufacturer asks a team of engineers to determine, with a confidence level of 90%, whether a prototype device has a reliability of at least 90% at 6,000 hours. Since the life of the product under normal operating conditions is expected to be more than 15,000 hours, it is not feasible to test under these conditions and obtain results in an acceptable timeframe. So the engineers decide to run an accelerated test with a duration of 7,500 hours, by which time they expect that the majority of units will have failed. The normal operating temperature for this product is 323 K (50°C), and temperature is the only acceleration variable.

The data collected from the test for three different accelerated stress levels are shown next.

Stress Level (K)	393	408	423
Failure Time (Hr)	3850	3300	2750
	4340	3720	3100
	4760	4080	3400
	5320	4560	3800
	5740	4920	4100
	6160	5280	4400
	6580	5640	4700
	7140	6120	5100
	Did not fail	6840	5700
	Did not fail	Did not fail	6400

1. Add a new ALTA standard folio by choosing **Insert > Folios > ALTA Standard Folio**. On the first page of the setup window, make sure only the **My data set contains suspensions** check box is selected. Then, select **Hour (Hr)** from the **Units** drop-down list and click **Next**.
2. On the second page of the setup window, select **Temperature** as the stress type. Set the use level temperature to **323** with **K** as the unit.
3. Click **OK** to close the setup window and view your new ALTA standard folio. Enter the failure/suspension data into the folio's data sheet. The first fifteen rows of the data sheet are shown next.

	State F or S	Time to F or S (Hr)	Temperature K
1	F	3850	393
2	F	4340	393
3	F	4760	393
4	F	5320	393
5	F	5740	393
6	F	6160	393
7	F	6580	393
8	F	7140	393
9	S	7500	393
10	S	7500	393
11	F	3300	408
12	F	3720	408
13	F	4080	408
14	F		
15	F		

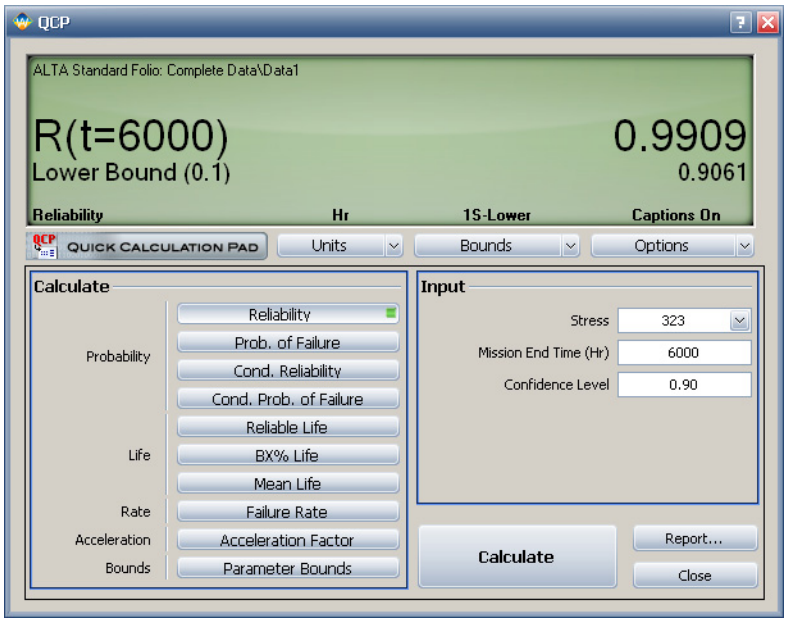
Note: Complete data set is not shown.

4. On the Main page of the folio's control panel, select the **Arrhenius-Weibull** model and click **Calculate**. The parameters of the model will be automatically computed and displayed in the **Analysis Summary** area.
5. To estimate the reliability of the device at 6,000 hours, choose **ALTA > Analysis > Quick Calculation Pad** or click the icon on the control panel.



6. In the QCP, choose to calculate the **Reliability** with **Lower One-Sided** confidence bounds. Select **Hour** for the time units and then make the following inputs:
   - Stress = **323**
   - Mission End Time = **6000**
   - Confidence Level = **0.90**

   Click **Calculate** to display the results, as shown next.



The results show that the device's estimated reliability at 6,000 hours is 99.09%. The lower one-sided 90% confidence bound is shown to be 90.61%. In other words, the test shows with 90% confidence that the device's reliability a 6,000 hours is at least 90.61%, thus meeting the demands of the manufacturer.

7.9 ALTA Two-Stress Example

A team of reliability engineers is instructed to perform an accelerated life test and use the accelerated test data to extrapolate a product's use level failure behavior. They are asked to estimate, with 90% confidence, the time at which the product will have an unreliability of 10% (i.e., the B10 life).

The normal operating temperature for this product is 328 K, and the normal operating voltage is 2 V. It was decided to accelerate both stresses in the test. The following table shows the data from the test.

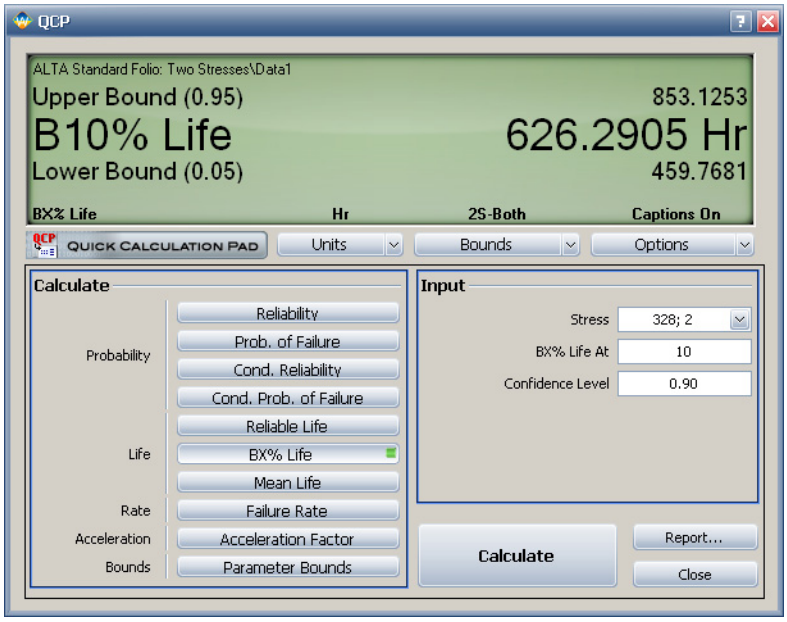
Time Failed (Hr)	Temperature (K)	Voltage (V)
620	348	3
632	348	3
658	348	3
822	348	3
216	378	3
246	378	3
332	378	3
400	378	3
380	378	5
416	378	5
460	378	5
596	378	5

1. Add a new ALTA standard folio by choosing **Insert > Folios > ALTA Standard Folio**. Since the data set contains only exact failure times, do not select any of the check boxes in the **Options for the Times-to-Failure Data Type** area. Then, select **Hour (Hr)** from the **Units** drop-down list and click **Next**.
2. On the second page of the setup window, select **Temperature** and **Voltage** as the stress types. Set the use level temperature to **328** with **K** as the unit, and set the use level voltage to **2** with **V** as the unit.
3. Click **OK** to close the setup window and view your new standard folio. In the data sheet of your new folio, select the check boxes inside the **Temperature** and **Voltage** column headers. Both stresses will now be used in the calculation for the selected model.
4. On the Main page of the folio's control panel, select the **TNT-Weibull** model (i.e., the temperature-nonthermal life-stress relationship combined with the Weibull life distribution).
5. Enter the above data into the data sheet, then click **Calculate**. The parameters of the model will be automatically computed and displayed in the **Analysis Summary** area, as shown next.
6. To obtain the two-sided 90% confidence bounds on the product's B10 life, choose **ALTA > Analysis > Quick Calculation Pad** or click the icon on the control panel.



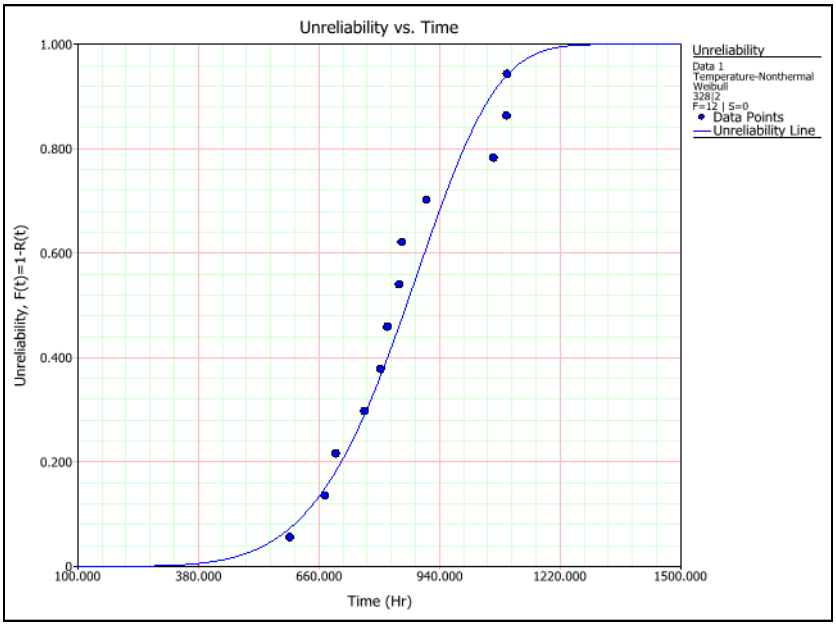
7. In the QCP, choose to calculate the **BX% Life** with **Two-Sided Bounds**. Select **Hour** for the time units and then make the following inputs:
   - Stress = **328; 2**
   - BX% Life At = **10**
   - Confidence Level = **0.9**

   Click **Calculate** to display the results, as shown next.



The results show, with 90% confidence, that the time at which the product will have an unreliability of 10% is between 459.7681 and 853.1253 hours.

8. To view a plot of the unreliability vs. time at the use stress level, go to the Plot page of the control panel and select **Unreliability vs. Time** from the **Plot Type** drop-down list. The plot is shown next.



7.10 Time-Dependent Stress Profiles

Stress levels under test conditions and/or during normal use may vary with time. For example, an engineer may decide that each unit in a sample will be tested at 310 K for the first 10 hours of a test, 320 K for the next 10 hours, and 330 K for the remainder of the test. ALTA PRO allows you to define and save *stress profiles* that describe how stress levels will vary with time. When you select a cumulative damage model in the ALTA standard folio, all of the stress profiles validated in your project become available to describe use level stress conditions and stress conditions during testing.

Tip: The stress profile functionality is available only in ALTA PRO. If you are an ALTA Standard user who needs to use time-dependent stress profiles in your analysis, please contact ReliaSoft for information about upgrading to ALTA PRO.

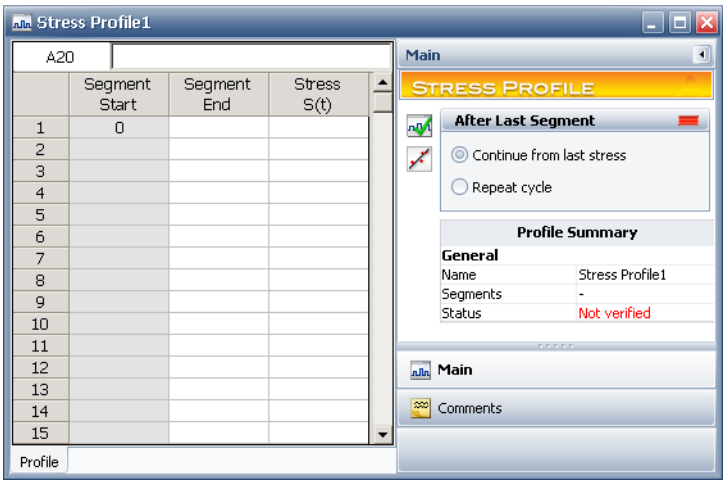
This section describes how to use time-dependent stress profiles, including:

- Creating stress profiles - Section 7.10.1 (p. 267).
- Using stress profiles in an analysis - Section 7.10.2 (p. 270).
- An example involving a time-varying stress - Section 7.10.3 (p. 271).

7.10.1 Creating Stress Profiles

A stress profile is a representation of how stress levels will vary with time. It consists of a basic pattern that either repeats as a cycle or occurs once and then continues from its last defined stress setting. To create and save a stress profile, follow the steps below.

- To add a new stress profile to your project, choose **Insert > Tools > ALTA Stress Profile**. A stress profile icon will appear under the Tools heading of the current project explorer, and the Stress Profile folio will open, as shown next.



- To rename the stress profile, right-click the profile's icon in the current project explorer and choose **Rename**.

- Next, you must define a pattern for how stress levels will change over a specified period of time. The pattern you will define consists of a series of segments, where each segment has a specified duration and either:
  - a constant stress value, or
  - a function that takes a time value and returns a stress value.

  These segments are defined in the data sheet, as shown next.

Segment Start	Segment End	Stress S(t)
0	1100	328
1100	2200	338
2200	3300	348

- **Segment Start** is calculated automatically by the software and cannot be entered manually. The first segment's start time is always 0. For every subsequent segment, the start time is identical to the prior segment's end time.
- **Segment End** allows you to enter the end time for each segment, which must be smaller than the end time of the next segment.
- **Stress S(t)** is the stress value or function for the segment. If the stress level will stay constant during a segment, enter a constant stress value (e.g., 30). If the stress level will change during a segment, enter the stress as a function of time (e.g., at time = 20, the function $t + 20$ will return a stress value of 40). When entering a function, you must use *t* or *T* as the time variable.

  Tip: Because stress units (e.g., volts) and time units (e.g., hours) are not defined in the stress profile, it is important to apply stress profiles only to folios that are intended to use the same stress and time units. In the Stress Profile window, you may want to rename column headers (by double-clicking inside the header) or use the Comments page as a reminder of which units are applicable to that profile. You can also create sub-folders under the Tools heading in the current project explorer to organize your profiles in a way that will make them easier to manage for future use.

  For example, suppose the stress unit is psi and the time unit is seconds. Now suppose you wanted to define the following 120-second pattern: a stress value of 30 psi for 60 seconds, followed by a stress value of 50 psi for 30 seconds, followed by a stress function that begins at 50 psi and gradually decreases at a rate of 1 psi per second for the remainder of the pattern. For this pattern, you could fill out the data sheet as follows:

Segment Start	Segment End	Stress S(t)
0	60	30
60	90	50
90	120	50 - (t - 90)

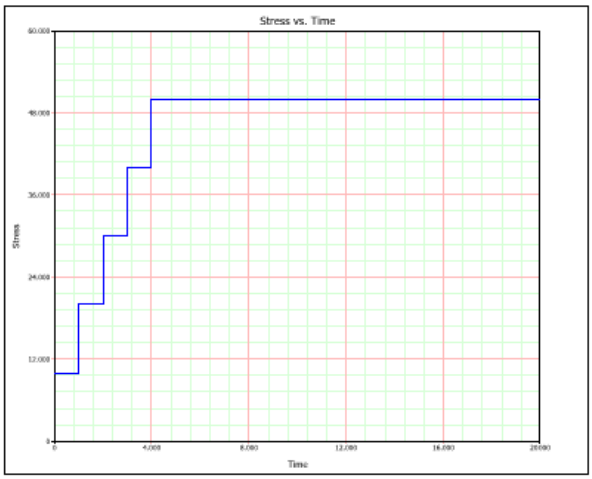
  Notice that the last segment of this profile uses a stress function. Since *t* is the test time (or, if the profile is cyclical, the time since the pattern was last restarted), t = 90 when the last segment begins. So the last segment's stress level starts at 50 - (90 - 90) = 50. Then, at 91 seconds, the stress level will have dropped to 50 - (91 - 90) = 49. After another second it will have dropped to 50 - (92 - 90) = 48, and so on.

- Next, choose the appropriate option in the **After Last Segment** area of the control panel. Your selection here will determine what happens after the end time of the profile's last segment (in the above example, after time = 120).

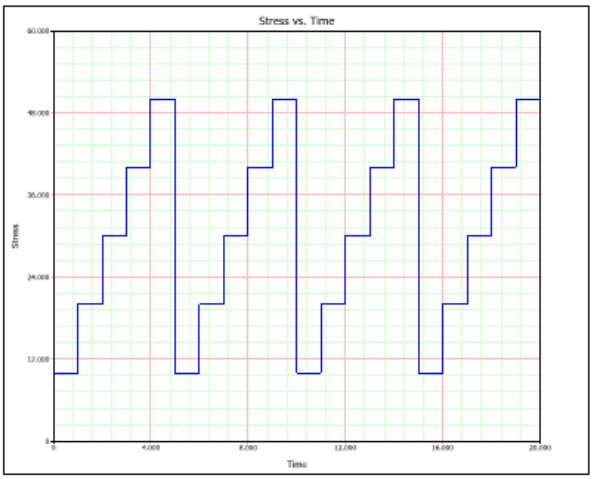
  If you select **Continue from last stress**, all times after the last segment will use the stress value/function defined in the last segment.

  If you select **Repeat cycle**, the entire pattern of segments will be treated as a repeating cycle.

For example, suppose you defined a pattern that is made up of five segments, each an hour long and increasing stepwise from the segment before. The graphs below illustrate the difference between continuing from the last stress (left) and selecting to repeat (right). In this example, the test has a duration of 20 hours, and the graphs explain what would happen through the entire duration of the test.



Continue from last stress



Repeat cycle

- Before you can use the new stress profile in your analysis, you must save any changes you have made and validate the current profile settings. To do this, click the **Validate Stress Profile** icon.



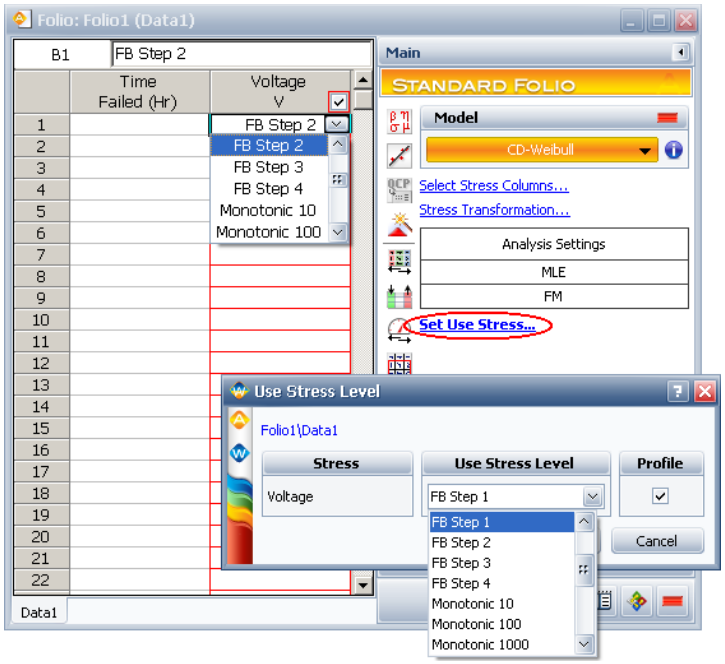
- After you save the changes in your profile, the **Profile Summary** area will appear. Click the area to view a report of the current stress profile in the tool's Results window.
- You may also wish to view a graphical representation of your profile. To do this, click the **Plot** icon to save any profile changes and view a stress vs. time plot. *For general information on plots, see Chapter 14.*

> **Note:** Clicking the **Validate Stress Profile** icon will not automatically update your stress vs. time plot. To make sure your plot reflects the most recent profile information, click the **Plot** icon.

- If desired, you can use the Comments page of the control panel to enter notes or other text that will be saved with the stress profile.

7.10.2 Using Stress Profiles in the ALTA Standard Folio

There are two ways stress profiles can be used in the ALTA standard folio. First, they can be used to describe the time-varying stress conditions that sample units will experience during testing (as shown in the data sheet below). Second, they can be used to describe the time-varying stress conditions that the product will experience during normal use (as shown in the Use Stress Level window below).



- To describe the test's stress conditions:
  - In the ALTA standard folio data sheet, each row represents a tested unit (or group of units). To choose the profile that describes the test conditions for a unit or group, first make sure a cumulative damage life-stress model is selected on the control panel. Then select the cell in the appropriate stress column for the unit or group. A drop-down list will appear in the cell. The list displays all the validated stress profiles in the currently active project.
- To describe the product's use level stress conditions:

> **Note:** Some plots (e.g., the use level probability plot) are not available when you use a stress profile to describe the product's use level conditions.

  - To choose a stress profile that represents the product's normal operating conditions, first make sure a cumulative damage life-stress model is selected on the control panel. Then click the **Set Use Stress** link to open the Use Stress Level window. Select the **Profile** check box for all applicable stresses. The **Use Stress Level** drop-down list displays all the validated stress profiles in the currently active project.

> **Note:** If you update a stress profile that is associated with an ALTA standard folio, the calculations will not be updated automatically. To update the ALTA standard folio calculations, click the **Calculate** icon on the control panel.

7.10.3 Example

A reliability engineer is asked to determine, with a confidence level of 90%, whether an electronic component has a B10 life of at least 350 hours under ordinary conditions.

To simplify the analysis, the engineer will assume that the ordinary stress conditions for the component consist of a constant stress level of 2 V. However, because the ordinary stress conditions are known to involve escalating stress values, the component will be subjected to a time-varying voltage stress during testing. The following table describes the stress values that will be applied during the test.

Starting Time (Hr)	Ending Time (Hr)	Stress Level (V)
0	250	2
250	350	3
350	370	4
370	380	5
380	390	6
390	400	7

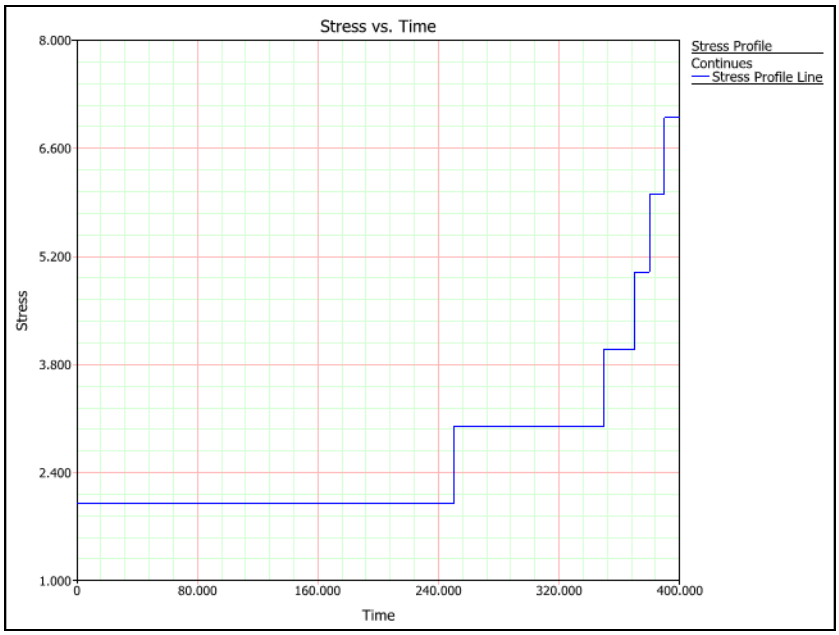
All eleven units were tested to failure using the same stress profile. The following failure times were observed in the test.

Failure Time (Hr)	Stress Level
280	Time-varying
310	Time-varying
330	Time-varying
352	Time-varying
360	Time-varying
366	Time-varying
371	Time-varying
374	Time-varying
378	Time-varying
381	Time-varying
385	Time-varying

1. Add a new ALTA standard folio by choosing **Insert > Folios > ALTA Standard Folio**. Since all the data are exact failure times, clear all of the check boxes in the **Options for the Times-to-Failure Data Type** area. Then, select **Hour (Hr)** from the **Units** drop-down list and click **Next**.
2. On the second page of the setup window, select **Voltage** as the stress types. Set the use stress level to **2** with **V** as the unit.
3. Click **OK** to close the setup window and view your new standard folio. Next, create a stress profile for voltage by choosing **Insert > Tools > ALTA Stress Profile**.
4. A new stress profile icon will appear in the current project explorer. Rename the new stress profile to "Voltage Step" by right-clicking the new stress profile icon and choosing **Rename Item** from the shortcut menu. Enter segment end times and stress values into the data sheet, as shown next.

	Segment Start	Segment End	Stress S(t)
1	0	250	2
2	250	350	3
3	350	370	4
4	370	380	5
5	380	390	6
6	390	400	7

5. Select **Continue from last stress** on the control panel. Click the **Validate Stress Profile** icon to save and validate the information in your new profile. Finally, click the **Plot** icon to view a stress vs. time plot for your profile. It should appear as shown next.



6. After creating the stress profile, return to your ALTA standard folio. Select **CD-Weibull** (i.e., the cumulative damage life-stress relationship combined with the Weibull life distribution) from the **Model** drop-down list. Click **Stress Transformation** and apply the **Logarithmic** transformation to the stress. Then, in the data sheet of the

folio, enter the failure times. To assign the stress profile to a failure time, click the corresponding cell in the Voltage column of the data sheet. Click the arrow in the cell and choose **Voltage Step** from the drop-down list. After you have filled out the data sheet, it should appear as shown next.

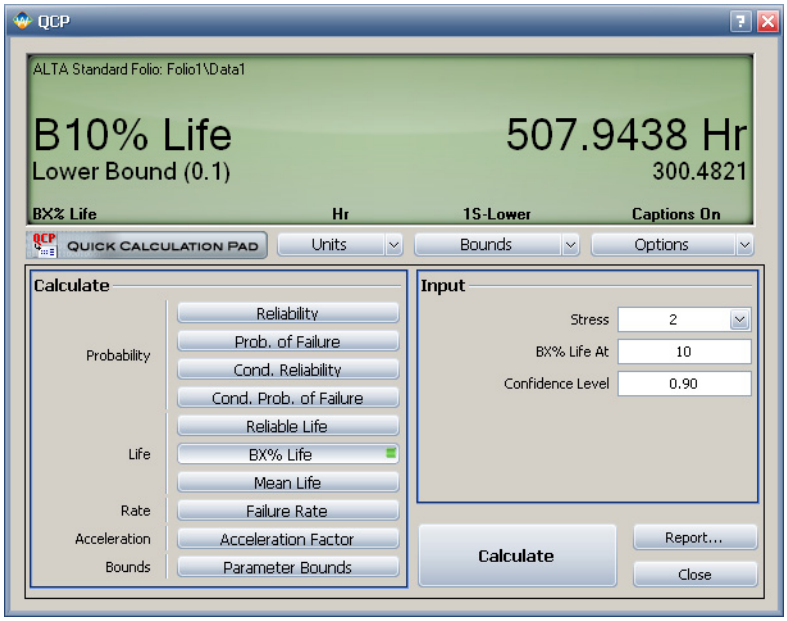
	Time Failed (Hr)	Voltage V
1	280	Voltage Step
2	310	Voltage Step
3	330	Voltage Step
4	352	Voltage Step
5	360	Voltage Step
6	366	Voltage Step
7	371	Voltage Step
8	374	Voltage Step
9	378	Voltage Step
10	381	Voltage Step
11	385	Voltage Step

7. Click the **Calculate** icon. The parameters of the model will be automatically computed and displayed in the **Analysis Summary** area.
8. To estimate the B10 life of the device, choose **ALTA > Analysis > Quick Calculation Pad** or click the icon on the control panel.



9. In the QCP, choose to calculate the **BX% Life** with **Lower One-Sided** confidence bounds. Select **Hour** for the time units and then make the following inputs:
   - Stress = **2**
   - BX% Life At = **10**
   - Confidence Level = **0.9**

   Click **Calculate** to display the results, as shown next.



The results show that the component's estimated B10 life is 507.9438 hours. The lower one-sided 90% confidence bound is shown to be 300.4821 hours. Thus, the test's results do not show with 90% confidence that the component's B10 life is at least 350 hours.

7.11 Additional Tools

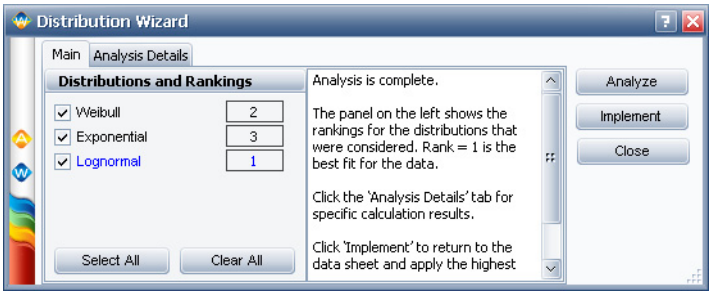
7.11.1 ALTA Distribution Wizard

The Distribution Wizard for ALTA standard folios performs a Likelihood Value (LKV) test. The test computes the value of the log-likelihood function given the calculated parameters of the distribution in order to determine the best distribution for a data set. Note that the Distribution Wizard only serves as a guide for you to decide which distribution to use in your selected model, and it assumes the life-stress relationship you selected on the control panel. You should compare its suggestion with your own engineering knowledge about the product being modeled before making the final decision.

Open the Distribution Wizard by choosing **ALTA > Analysis > Distribution Wizard** or by clicking its icon in the Main page of the control panel.



On the Main tab of the Distribution Wizard, select the distributions you would like to consider and then click **Analyze** to compare them. The distributions will be ranked according to how well they fit the data, with rank 1 being the best fit. The highest ranked distribution will also be highlighted with blue text, as shown in the following example.



Click the **Implement** button to close the Distribution Wizard and automatically calculate the parameters for the highest ranked distribution.

The data sheet on the Analysis Details tab summarizes the ranking results and displays the value of the log-likelihood function for each distribution given the calculated parameters. The calculated parameters for each considered distribution are also shown.

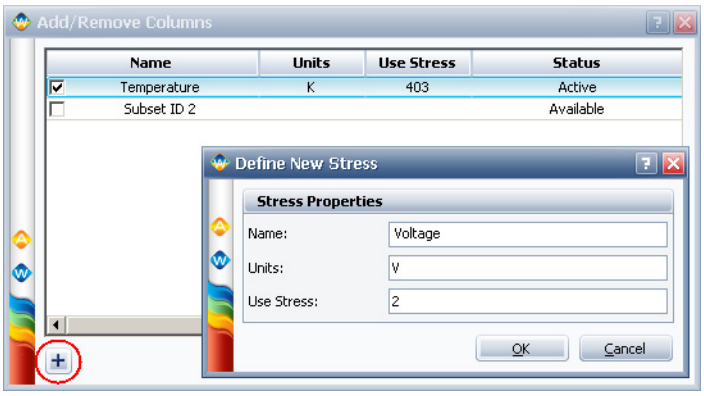
7.11.2 Add or Remove Columns

The Add or Remove Columns window allows you to change the number of stress columns in your data sheet. You can also add/remove a second subset ID column by selecting/clearing the **Subset ID 2** check box. To add a stress column:

- Open the window by choosing **ALTA > Format and View > Add or Remove Columns** or clicking its icon on the control panel.



- In the window that appears, click the **Add Column** icon in the bottom-left corner. The Define New Stress window will appear.



- Enter the name, units of measurement and normal stress level for the new stress. Click **OK** to return to the Add or Remove Columns window.
- The new stress will appear in the window with a selected check box and "Add" displayed in the Status column.

Name	Units	Use Stress	Status
Temperature	K	323	Active
Voltage	V	2	Add
Subset ID 2			Available

- Click **OK** to add the stress to your data sheet and close the window.

To remove a column, clear the check box for that column. "Remove" will be displayed in the Status column. Click **OK**.

Tip: If you wish to later change the heading of any stress or subset ID column, you can do so by double-clicking the column heading in the data sheet.

7.11.3 Likelihood Ratio Test

The Likelihood Ratio Test tool is used to assess the assumption of a common shape parameter among the data obtained at the various accelerated stress levels. This assumption underlies the use of any model that includes a life distribution with a shape parameter. By confirming this assumption, you confirm that units will fail in the same manner across different stress levels. The tool is available for all models in ALTA that include the Weibull or lognormal distribution, except the cumulative damage models in ALTA PRO.

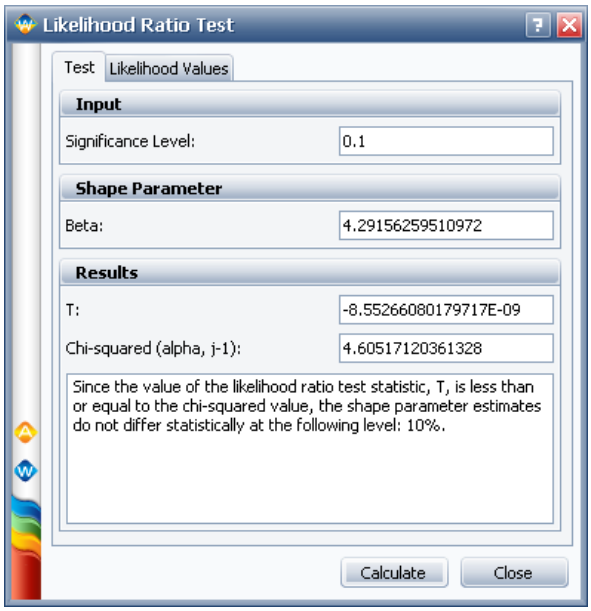
To use the Likelihood Ratio Test tool, first calculate the parameters in your ALTA standard folio. After the shape parameter is calculated in the folio, open the tool by choosing **ALTA > Options > Likelihood Ratio Test** or clicking its icon on the main page of the ALTA standard folio's control panel.



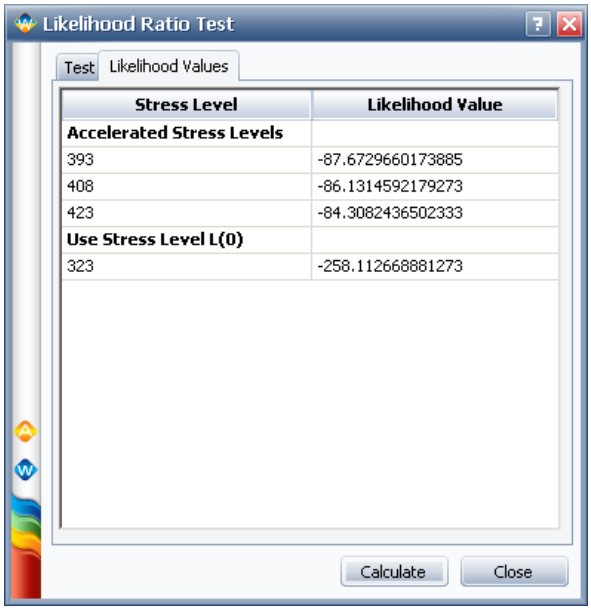
Next, enter a significance level in the **Input** area. The tool uses your specified significance level to determine whether there is a statistically significant difference in the shape parameter estimates at each stress level. For example, if you enter **0.1**, then the tool will determine whether the estimated shape parameters are statistically the same at a confidence level of 0.9.

Finally, click **Calculate** to view the results of the test. The **Results** area of the window displays the calculated likelihood ratio test statistic, *T*, and chi-squared value. If *T* is not greater than the chi-squared value, then the

estimated shape parameters do not differ significantly at the specified level. A written report of the results also appears in this area, as shown next.



The Likelihood Values tab (shown next) of the window displays the likelihood values obtained by two methods.



The values in the **Accelerated Stress Levels** area are calculated by fitting a separate distribution to the data from each of the test stress levels (with two or more exact failure times). The likelihood value in the **Use Stress Level** area is calculated by fitting a single model with a common shape parameter and a separate scale parameter for each of the stress levels, where the stress levels are represented by indicator variables.

7.11.4 Transfer Data to Weibull++

The Transfer Data to Weibull++ window is used to transfer the data in an ALTA standard folio data sheet to a Weibull++ standard folio. This allows you to perform life data analysis on your accelerated test data. For example, you could use this window to separately analyze the data obtained from each stress level from the accelerated testing. To access the window, click the **Transfer Data to Weibull++** icon on the control panel.



After you click the icon, you will have the option to either transfer the data to a single Weibull++ data sheet or transfer it to multiple sheets.

- If you select the **Transfer all data to a single sheet** check box, the subset ID column of that sheet will display the stress level at which each data point was obtained. (If desired, you can then use the Batch Auto Run tool to extract data subsets from the data sheet. *See page 229 in Chapter 6*.)
- If you select the **Transfer data for each stress to a separate data sheet** check box, the data will be grouped according to stress level (where, in multiple-stress situations, a stress level is a combination of stress values), and then each group will be placed into its own data sheet.
  - If you select to transfer the data to multiple sheets, the **Calculate parameters after transfer** check box will appear. Select this check box if you wish to have the software automatically calculate each data sheet using the life distribution from your selected model. If you select this check box, you will also have the option to automatically add a Contour plot that will compare the calculated parameters for each data sheet. For example, you could use this plot to help you determine whether the shape parameter estimates for the data obtained at different stress levels differ significantly at a specified confidence level.

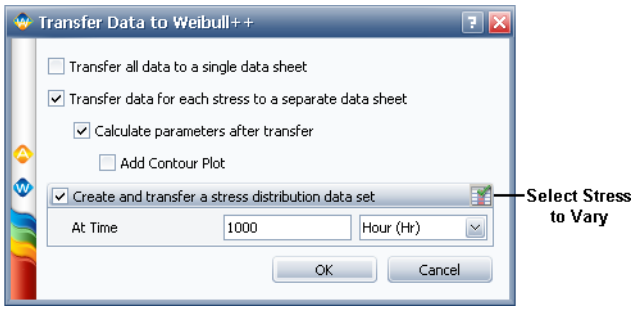
This window also includes a check box labeled **Create and transfer a stress distribution data set**. If you select this option, the tool will calculate the product's unreliability at the specified time and at eleven different stress levels. The different stress levels will range from the use stress level to the maximum stress level used in the test, with equal increments in between. This information will then be exported to a free-form data sheet in a new Weibull++ standard folio, like the data sheet shown next.

	X-Axis value (K)	Y-Axis value (%)	Subset ID
1	356	31.5703213539804	
2	363	37.7433304078836	
3	370	44.4030918456798	
4	377	51.3944426342843	
5	384	58.5196014754012	
6	391	65.5509968237365	
7	398	72.2510488907534	
8	405	78.3968226952862	
9	412	83.8057016695541	
10	419	88.3570754849994	
11	426	92.0051178700479	

In this example, the product's use stress level is 356 K, and the maximum stress level used in the test is 426 K. Thus, the X-axis values range from 356 to 426. The Y-axis values represent the unreliability (as a percentage) for each stress level. So, if the time specified in the Transfer Data to Weibull++ window was 1,000 hours, then, according to

the first row of this data sheet, the product has a probability of failure of 31.57% at 1,000 hours at a stress level of 356 K. If you wish, you can calculate this data to further analyze the relationship between stress level and unreliability.

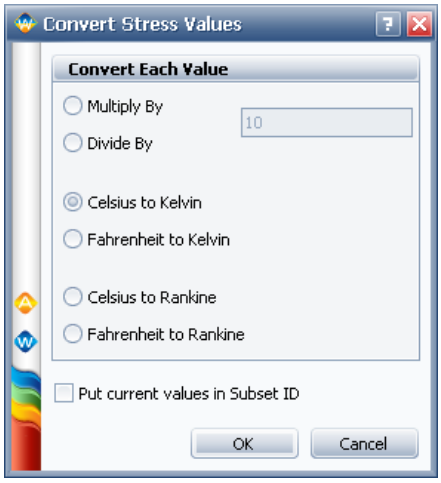
In multi-stress situations, the **Select Stress to Vary** icon will be enabled in the window, as shown next.



If you click this icon, a window will appear that allows you to select which stress type will be varied in the software's unreliability calculations. The remaining stress types will be fixed at their specified use stress values.

7.11.5 Convert Stress Values

The Convert Stress Values window (shown next) is used to convert all the values entered in one of the stress columns of an ALTA standard folio data sheet.



You can have the stress values multiplied or divided by a specified value, or you can convert temperature values that were entered in Celsius or Fahrenheit to one of the absolute scales supported by the software (i.e., Kelvin or Rankine).

Follow the steps outlined below:

- Select the stress column that contains the values you want to convert by clicking the header of that column or selecting any cell in that column.

> **Note:** The software will always convert all of the values in a column. For example, if you select one cell and convert the contained value, all the other values in that column will be converted as well.

- Choose **ALTA > Options > Convert Stress Values** to open the window.



- In the **Convert Each Value** area, select the conversion option you wish to use.
  - If you select the **Multiply By** or **Divide By** option, you will enable an input field. In this field, enter the value that you wish to have each stress value multiplied or divided by.
- Select the **Put current values in Subset ID** check box if you want the software to move the original, unconverted values into the first subset ID column in the data sheet for future reference. *If there is already information in the first subset ID column, the software will overwrite that information.*

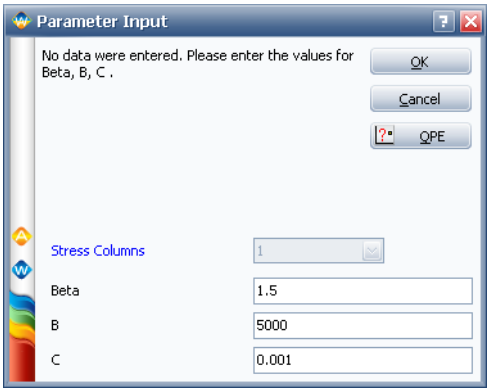
> **Tip:** If you only have one subset ID column, and you want to move its information to a column that will not be overwritten when you select the **Put current values in Subset ID** check box, first use the Add or Remove Columns window to add a second subset ID column (*see Section 7.11.2 on page 274*). Then paste the information from the first subset ID column to the new column. The software will only overwrite values in the first ID column.

- Click **OK** to convert all the values in the selected stress column.

7.11.6 Calculations with Insufficient Data Entered

The Parameter Input window will appear when you attempt to calculate an ALTA data sheet that does not contain enough data to fit a model.

If you attempt to calculate a data sheet in an ALTA standard folio with no data entered, the Parameter Input window will appear, as shown next, and allow you to either enter the parameters for the selected ALTA model directly or use the Quick Parameter Estimator (QPE) to estimate the model's parameters based on information you have about the reliability of the product at normal and accelerated stress levels.

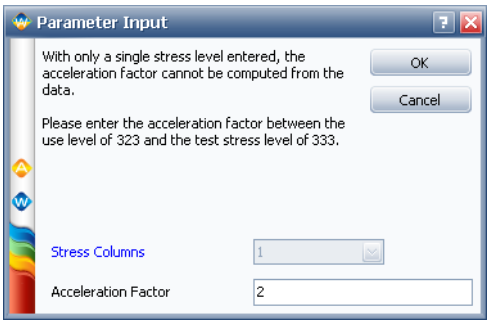


After you enter the parameters and click **OK**, the data sheet will be updated automatically with the specified model. Any plots or QCP results you produce from this data sheet will be based on this model.

> **Note:** If you have more than one stress column selected to be used in the analysis and you attempt to calculate an empty data sheet using a proportional hazards or general log-linear model, the **Columns** drop-down list will be enabled in the Parameter Input window. With this drop-down list, you can select how many stresses to include in the model. You cannot choose to use more stresses than are selected to be used in the analysis.

To perform an accelerated life testing data analysis, you need information about how the product behaves at more than one stress level. If you select to use an Arrhenius, Eyring or inverse power law (IPL) model and attempt to calculate the data sheet with data for only one stress level, the Parameter Input will appear, as shown next, and ask

you to specify an acceleration factor that allows the software to calculate how the product behaves at other stress levels.



The acceleration factor is obtained by dividing the product's life at the use stress level by its life at the accelerated stress level used in the test. For example, if the product has a life of 100 hours at the use stress level, and it was tested at an accelerated stress level which reduces its life to 50 hours, then you would enter **2** in the **Acceleration Factor** field.

Note: This method of calculating the data sheet does not support the calculation of confidence bounds.

Weibull++ Degradation Analysis

8

Degradation analysis is a technique that uses the performance measurements (degradation measurements) of a product over time to predict the point at which each unit in the sample is expected to fail. This analysis is useful for tests performed on products with very high reliability, where it is not possible to test the products to failure under normal operating conditions.

The Weibull++ degradation analysis folio extrapolates the times at which the amount or level of degradation is serious enough that the unit is considered to be failed. In addition, the folio automatically performs life data analysis on the extrapolated failure times. This allows you to obtain the life characteristics of the product.[1]

This chapter describes how to use the Weibull++ degradation analysis folio, including:

- Setting up the data sheet and using the control panel - Section 8.1 (p. 282).
- Viewing a Degradation vs. Time plot - Section 8.2 (p. 285).
- A degradation analysis example - Section 8.3 (p. 285).
- Using the Weibull++ Degradation Model Wizard - Section 8.4 (p. 288).

1. The ReliaWiki resources portal has more information on degradation analysis at: http://www.ReliaWiki.org/index.php/Degradation_Data_Analysis.

8.1 Weibull++ Degradation Analysis Folio

To add a Weibull++ degradation analysis folio to a project, choose **Insert > Folios > Weibull++ Degradation** or right-click the **Specialized Folios** folder in the current project explorer and choose **Add Weibull++ Degradation** on the shortcut menu.



The following example shows a degradation analysis of two automobile tires, A and B.

	Inspection Time (mi)	Degradation	Unit ID
1	5000	6.1	A
2	5000	5.9	B
3	10000	5	A
4	10000	5.1	B
5	15000	4	A
6	15000	4.3	B
7	20000	3.2	A
8	20000	3.3	B
9	25000	2.8	A
10	25000	2.9	B

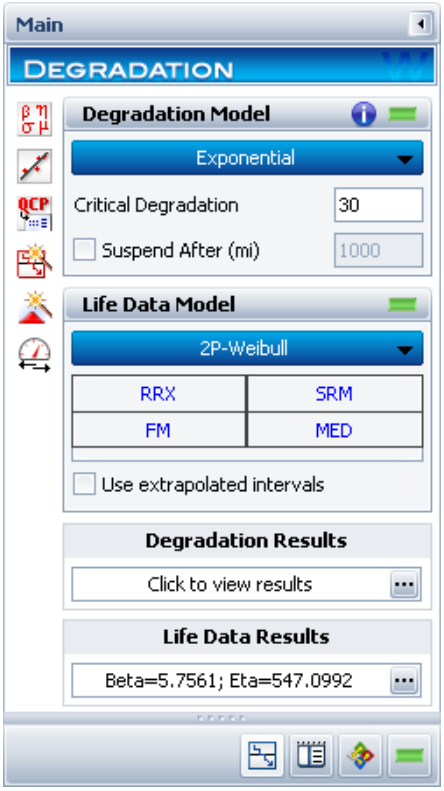
The data sheet contains the following information:

- The **Inspection Time** column indicates the time at which the degradation of the unit was measured. Time may also be measured as the amount of usage the unit accumulates or as the factor that degrades the unit. For example, in the data sheet, the inspection time is in mileage because of its direct relation to the wear on the tires. The level of degradation may be measured either continuously or at predetermined intervals of time.
- The **Degradation** column indicates the degradation measurement of a unit. In this example, the degradation is obtained by directly measuring the depth of the tire tread. In some cases, measuring the level of degradation is not possible without invasive techniques or destroying the specimen. In those situations, the degradation may be measured through a performance indicator, such as the luminosity of a bulb or resistance of a dielectric.
- The **Unit ID** column indicates the unit being measured. The software extrapolates the point of failure for each unique unit based on its degradation measurements.

Control Panel Settings

The Weibull++ degradation folio control panel allows you to configure the analysis settings for the data sheet and view/access the results. It consists of multiple pages, each containing options for performing particular tasks. This section focuses on the Main page of the Weibull++ degradation folio control panel, which contains the tools you will need to analyze the data set. *For more information about control panels in general, see page 114 in Chapter 3*.

The Main page of the Weibull++ degradation folio control panel is shown next.



The Main page includes the following settings:

- The **Degradation Model** area allows you select the mathematical model to use for extrapolating the times at which the critical degradation level would be reached for each unique unit ID in the data sheet. The following is a list of all available models and their corresponding equations. The variable x represents the failure time, y represents the critical degradation, and variables a, b and c are the model parameters to be solved for in the extrapolation.
  - Linear: $y = a*x+b$
  - Exponential: $y = b*\exp(a*x)$
  - Power: $y = b*(x^a)$
  - Logarithmic: $y = a*\ln(x)+b$
  - Gompertz: $y = a*b^(c^x)$
  - Lloyd-Lipow: $y = a-b/x$

> **Tip:** If you are not certain which degradation model would provide the best fit for your data, you can use the Degradation Model Wizard to guide you through the selection process. *See Section 8.4 on page 288.*

- The **Critical Degradation** field is for defining the level of degradation that is considered a failure. For example, you might wish to define failure as the time when the depth of the tire tread is less than 2 millimeters.
- The **Suspend After** option allows you to define a censoring time past which no failure times are extrapolated. In practice, there is usually a rather narrow band in which the censoring time has any practical meaning. If the censoring time is relatively low, no failure times will be extrapolated, which defeats the purpose of the degradation analysis. On the other hand, a relatively high censoring time would occur after all of the theoretical

failure times, thus rendering the censored data meaningless. Nevertheless, certain situations may arise in which it is beneficial to censor the degradation data. For example, if your product has a maximum warranty life of 10,000 hours, then you may want to treat all failures beyond that time as suspensions.

- The **Life Data Model** area contains the settings for performing life data analysis on the failure times that were extrapolated via the degradation analysis. These settings have no effect on the degradation analysis calculations; they affect only the life data analysis calculations. The settings in this area are identical to the settings on the control panel of a Weibull++ standard folio, except for the following option:
  - The **Use extrapolated intervals** option allows you to extrapolate the failure/suspension times as interval censored data. The intervals are calculated as bounds around the extrapolated data points.

What's Changed? In previous versions of Weibull++, you used the Associate Specialized Folio tool to transfer the extrapolated data points to a standard folio data sheet in order to perform life data analysis. In the Synthesis version, you can conveniently perform life data analysis calculations within the degradation analysis folio.

When you click the **Calculate** icon on the control panel, the software extrapolates the points at which the degradation measurements will reach the point of critical degradation. It then performs a separate life data analysis on the extrapolated points based on the settings you have specified in the **Life Data Model** area.

- The **Degradation Results** area displays the parameters of the chosen degradation model and the extrapolated failure/suspension times. Click the **Degradation Results (...)** button to open the Results window, which allows you to view, edit and print the calculations.
- The **Life Data Results** area displays the parameters of the life data model. Click the **Life Data Results (...)** button to open the Results window.

Note: If the distribution and parameters are associated with a published model, then the model's name will appear as a link at the bottom of the **Life Data Results** area. Click the link to view the model's properties. *For details on how to publish the results as a model, see page 116 in Chapter 3*.

Folio Tools

The folio tools are arranged on the left side of the Main page of the control panel. Use these tools to manage data and experiment with the results of your analysis.



Calculate extrapolates the points at which the degradation measurements will reach the point of critical degradation (i.e., the times at which the degradation is serious enough that the unit is considered to be failed). The software then performs life data analysis calculations on the extrapolated data points. This tool is also available by choosing **Degradation > Analysis > Calculate**.



Plot creates a new sheet in the folio that provides a choice of applicable plot types. This includes the degradation vs. time plot in addition to the life data analysis plots. This tool is also available by choosing **Degradation > Analysis > Plot**. *See Section 8.2 on page 285*.



QCP opens the Weibull++ Quick Calculation Pad, which allows you to calculate life data analysis metrics, identical to the calculations you can perform in a Weibull++ standard folio. This tool is also available by choosing **Degradation > Analysis > QCP**.



Degradation Model Wizard opens the Degradation Model Wizard, which helps you select the degradation model that best fits your data. This tool is also available by choosing **Degradation > Analysis > Model Wizard**. *See Section 8.4 on page 288*.



Distribution Wizard opens the Distribution Wizard, which helps you select the distribution that best fits the data set based on the chosen parameter estimation method (i.e., RRX or RRY). In the degradation folio, the ranking in the Distribution Wizard is based on the life data model. This tool is also available by choosing **Degradation > Analysis > Distribution Wizard**. *See page 226 in Chapter 6*.



Change Units opens the Change Units window, which allows you to change the units for the time values in the current data sheet. *See page 113 in Chapter 3*.

8.2 Weibull++ Degradation Plot Sheet

After you analyze the degradation data in the Weibull++ degradation analysis folio, you can create plots of your results by choosing **Degradation > Analysis > Plot** or by clicking the **Plot** icon on the Main page of the control panel.



The **Plot Type** area on the plot sheet of the control panel allows you to select the type of plot you wish to view and the units of measurement you want the x-axis of the plot to be displayed in. The plot types available include all of the life data analysis plots (*see page 217 in Chapter 6*). These are generated by analyzing the extrapolated failure times for each unit under test.

In addition to the life data analysis plots, you can view the following plots, which pertain specifically to the extrapolation of failure times from the degradation measurements.

- **Degradation vs. Time (Linear)** plots the degradation of each test unit over time, using a linear scale for time and degradation. A thick horizontal line marks the critical degradation level (i.e., the level at which a unit is considered failed) that was specified on the Main page of the control panel. You can use this plot to visualize how failure times were extrapolated for each unit.
- **Degradation vs. Time (Log)** plots the same information as the Degradation vs. Time (Linear) plot, but using logarithmic axes for time and degradation.

The **Unit IDs** area of the control panel will display the unit IDs you entered in the data sheet of the folio. To remove a test unit from the plot, clear the associated check box.

8.3 Weibull++ Degradation Analysis Example

The data set used in this example is 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 "Degradation - Crack Propagation."

An analyst needs to determine the time when 10% of the turbine blades in a sample are expected to fail (i.e., B10 life). To determine performance, the blades are tested for crack propagation. The units are cyclically stressed and inspected every 100,000 cycles for crack length. Failure is defined as a crack length of 30mm or greater.

The data sheet in the "Crack Propagation" folio displays the results of the inspections for the five units, A through E.

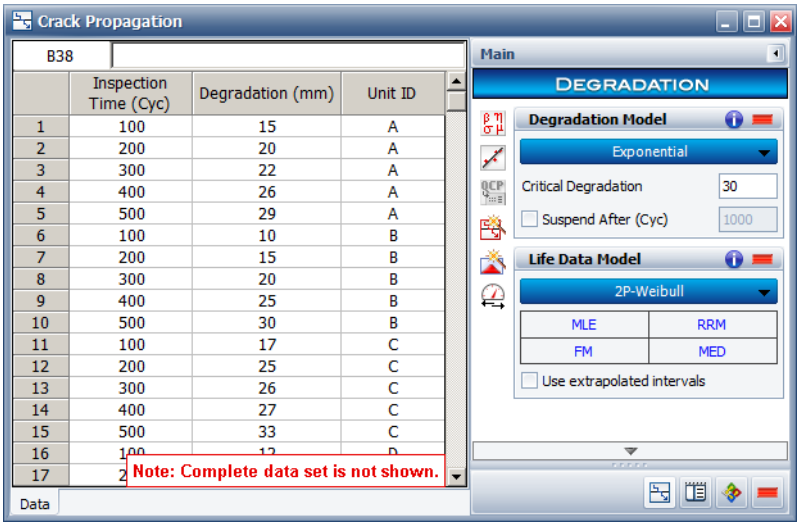
Using degradation analysis, project the failure times for the blades using an exponential model for the extrapolation, and then calculate the reliability of the blades at a 90% confidence level.

1. The first step is set up the data sheet for the analysis. Click the **Change Units** icon on the Main page of the control panel.



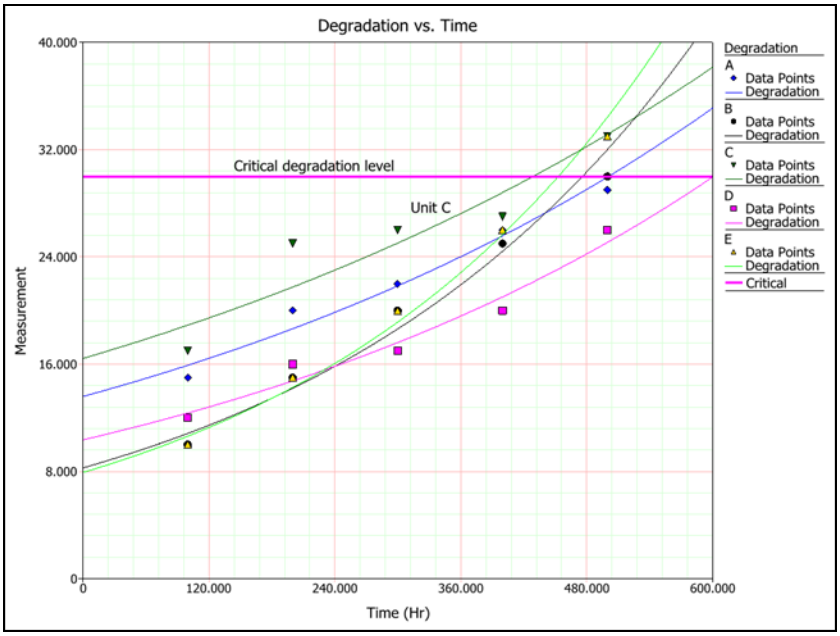
   In the Change Units window, select to change the time units to **Cycles**.

2. On the control panel, select the **Exponential** degradation model and enter a value of **30** for the **Critical Degradation**. For the life data model, select the **2-P Weibull** distribution and the **MLE** analysis method. Use the default settings for all other options, as shown next.

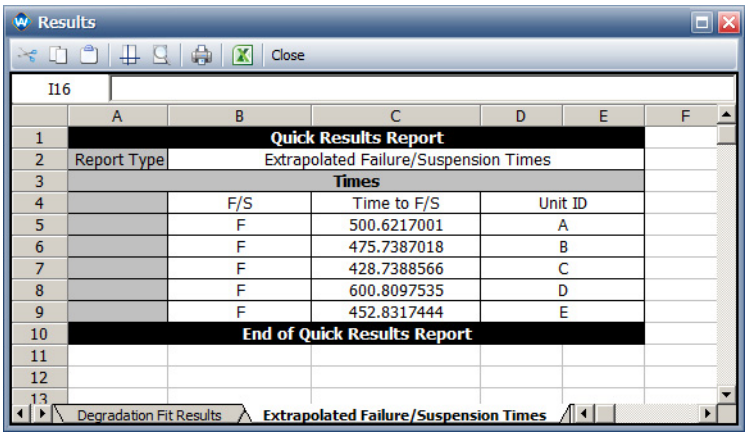


3. Click **Calculate** and then click the **Plot** icon on the Main page of the control panel.

   The Degradation vs. Time (Linear) plot shows that unit C is expected to reach the critical degradation level first, compared to the other units. (Plot shown with annotations to make the information easier to read.)

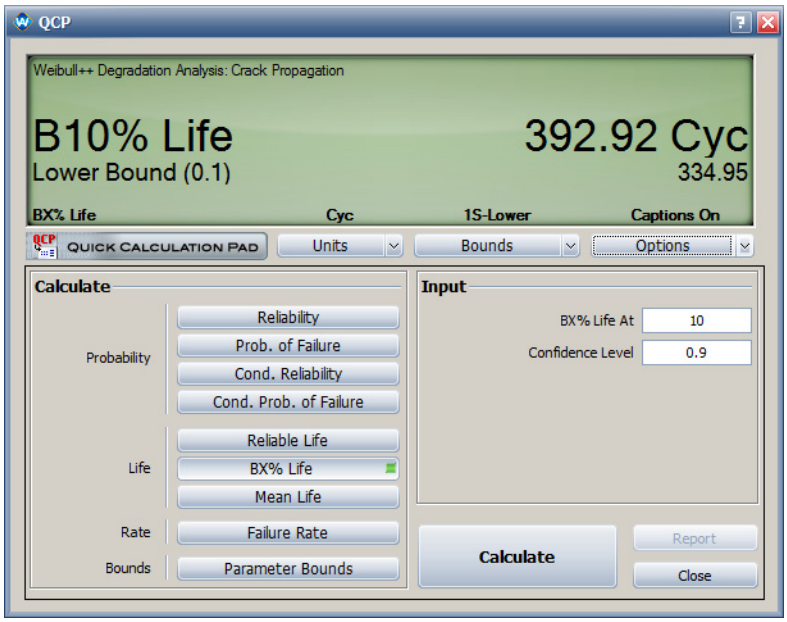


4. To view the times when the units will reach the critical degradation level, return to the data sheet of the folio. On the control panel, click the **Degradation Results (...)** button. In the Results window, select the second data sheet. The report shows that Unit C will reach critical degradation at about 428.74 cycles.



5. To view the results of the life data analysis, close the Results window and click the **Life Data Results (...)** button on the control panel. The parameters of the life data model are estimated to be beta = 8.055074 and eta = 519.555393.
6. The next step is to estimate the time when 10% of the blades in the sample are expected to fail. Close the Results window and click the **QCP** icon on the control panel.

   In the QCP, choose to calculate the **BX% Life** with **lower 1-sided** confidence bounds. Select **Cycles** for the units and enter a value of **10** in the **BX% Life At** field and **0.9** in the **Confidence Level** field. Click **Calculate** to obtain the results, as shown next:



The results indicate that 10% of the turbine blades will have failed (i.e., B10 life) at approximately 392.92 cycles of operation, and the 90% lower one-sided confidence bound on the B10 life is 334.95 cycles.

8.4 Degradation Model Wizard

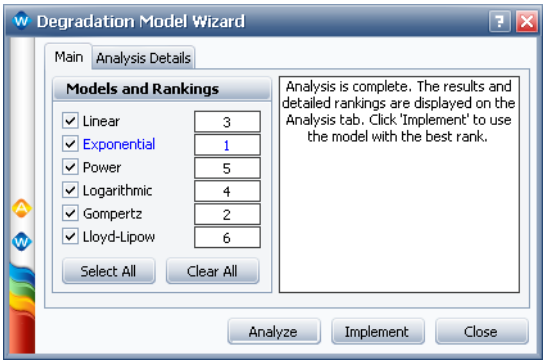
The Degradation Model Wizard performs a goodness of fit test to determine the best degradation model for your data. Note that the degradation model wizard only serves as a guide. You should compare its suggestion with information about the product being modeled before making the final decision.

Evaluating and Using Degradation Models

To use the degradation model wizard, you must first have at least two data points for each unique unit ID in the current data sheet. Access the wizard by choosing **Degradation > Analysis > Model Wizard** or by clicking its icon on the Main page of the control panel.



On the Main tab of the Degradation Model wizard, select the models you would like to consider. Click **Analyze** to start the evaluation. The results of the evaluation will be presented, as in the example shown next.

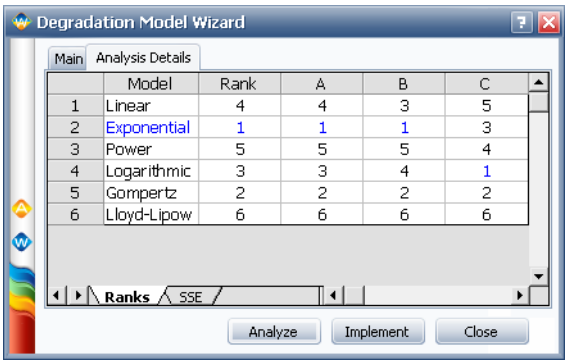


The models will be ranked according to how well they fit the data, with rank 1 being the best fit. In the example shown above, the exponential model is the suggested model for the data set. Click **Implement** to extrapolate failure times using the suggested model.

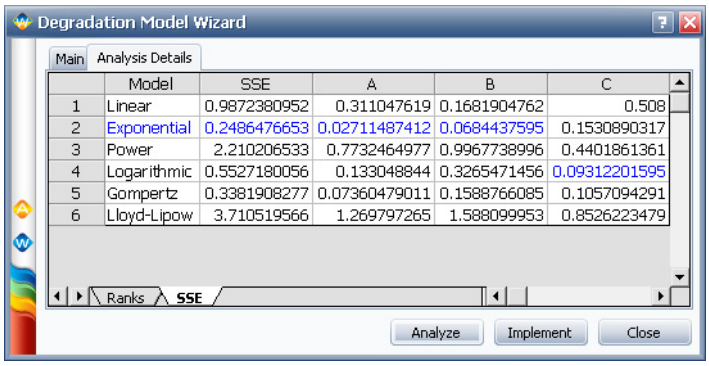
Viewing the Analysis Details

The wizard uses the Sum of Square Error (SSE) to evaluate the fit of the data. The calculations behind the ranking can be viewed on the Analysis Details tab, which contains two data sheets:

- The **Ranks** sheet displays the overall ranking as well as the ranking for each model and unit ID based on the SSE evaluation. Due to the randomness of materials, a model that is good for one unit may not be the best for the other units. In the example shown next, the exponential model is the best model for analyzing the data collected for Devices A and B, while the logarithmic model is a better fit for Device C.



- The **SSE** sheet shows the sum of square error for each model and unit ID. The values are obtained by first calculating the distance (the error) vertically from each data point to its corresponding value on the fitted model. The error value is squared, and then all the squared values are added up. The SSE column shows the calculated sum of squared errors for each model. The highest rank is given to the model with the lowest SSE value.



	Model	SSE	A	B	C
1	Linear	0.9872380952	0.311047619	0.1681904762	0.508
2	Exponential	0.2486476653	0.02711487412	0.0684437595	0.1530890317
3	Power	2.210206533	0.7732464977	0.9967738996	0.4401861361
4	Logarithmic	0.5527180056	0.133048844	0.3265471456	0.09312201595
5	Gompertz	0.3381908277	0.07360479011	0.1588766085	0.1057094291
6	Lloyd-Lipow	3.710519566	1.269797265	1.588099953	0.8526223479

ALTA Degradation Analysis 9

Degradation analysis is a technique that uses the performance (degradation) measurements of a product over time to predict the point at which each unit in the sample is expected to fail. This analysis is useful for tests performed on products with very high reliability, where it is not possible to test the units to failure under normal conditions. ALTA degradation analysis uses accelerated stress levels during testing to further reduce the test time required to obtain the predicted failure times. The predicted failure times are failures that would occur under these accelerated stress levels.

After the failure times under accelerated stresses are extrapolated, the ALTA degradation analysis folio will automatically perform an accelerated life testing data analysis on those failure times. This allows you to obtain the life characteristics of the product at normal operating conditions.

This chapter describes how to use the ALTA degradation analysis folio, including:

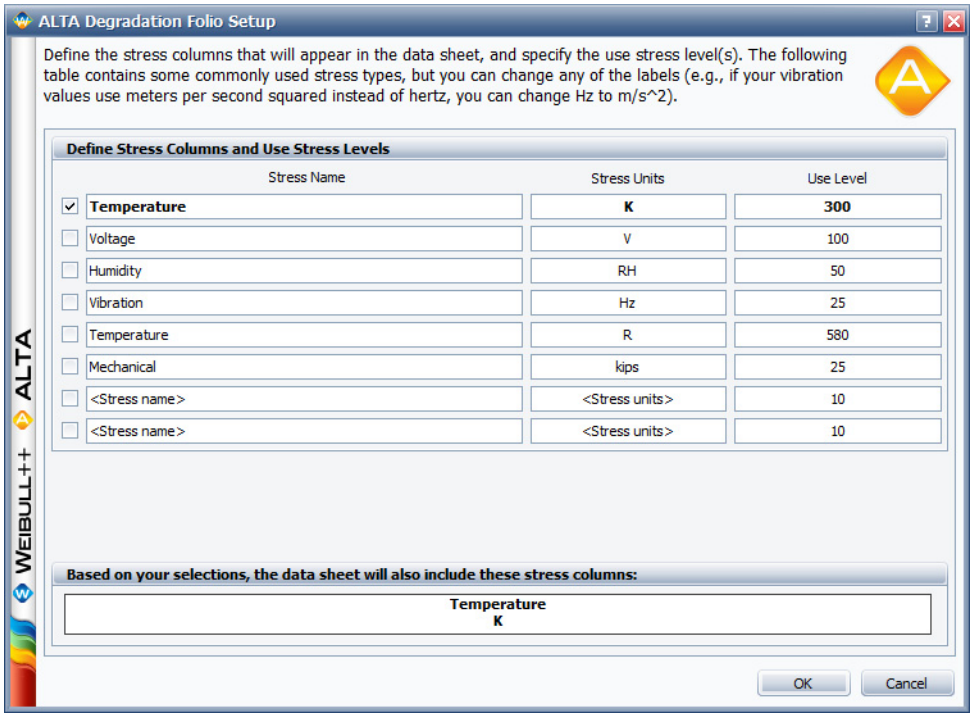
- Setting up the data sheet and using the control panel - Section 9.1 (p. 292)
- Viewing a Degradation vs. Time plot - Section 9.2 (p. 296)
- A degradation analysis example - Section 9.3 (p. 297)

9.1 ALTA Degradation Analysis Folio

To add an ALTA degradation analysis folio to an existing project choose **Insert > Folios > ALTA Degradation Folio** or right-click the **Specialized Folios** folder in the current project explorer and choose **Add ALTA Degradation** on the shortcut menu.



The Setup window will appear, as shown next.



The setup window is used to do the following:

- Determine the number of stress columns to be used in your degradation folio's data sheet by selecting or clearing the various check boxes. Every stress column represents a stress type. So, for example, a data sheet with two stress columns would be used for data obtained from a two-stress test.
- Change the labels for each stress column by changing the text under the **Stress Name** and **Stress Units** headings. There is a preview table at the bottom of the window that shows you how the names of the associated columns will appear in the data sheet.

Note: The **Stress Name** and **Stress Units** fields are merely for adding text to the headers of the stress columns, so you can keep track of which stress type is associated with each stress column. This information is not relevant to the software's calculations.

- Define the stress level of each stress that is expected for normal use conditions by entering values under the **Use Level** heading.

After you configure your folio, click **OK** in the setup window to create and view it.

Each row in the ALTA degradation analysis data sheet is a degradation measurement for a specified test unit at a given stress level. For example, the following shows data for two LED units, A1 and A2, whose light intensity was inspected every 50 hours.

	Inspection Time (Hr)	Degradation	Amps mA	Unit ID
1	50	95.1	35	A1
2	100	86	35	A1
3	150	77.6	35	A1
4	200	70	35	A1
5	250	66.7	35	A1
6	50	93.3	35	A2
7	100	87.1	35	A2
8	150	79.7	35	A2
9	200			
10	250	Note: Complete data set is not shown.		

The data sheet contains the following information:

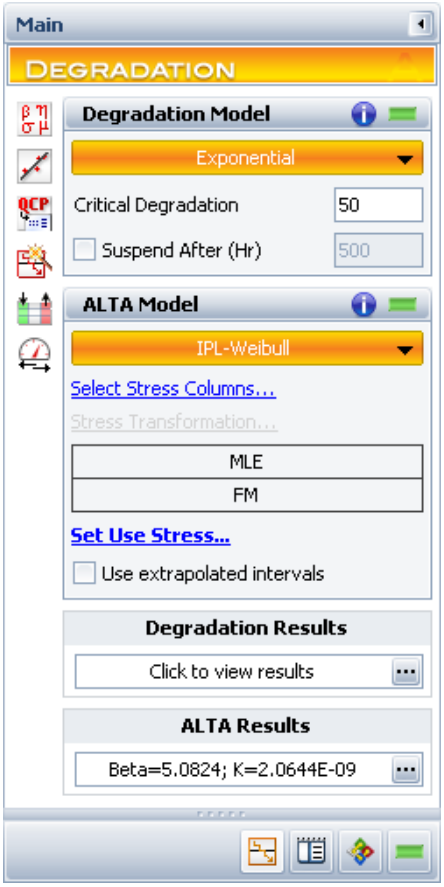
- The **Inspection Time** column indicates the time at which the degradation of the unit was measured. Time may also be measured as the amount of usage the sample accumulates or as the factor that degrades the unit. The level of degradation may be measured either continuously or at predetermined intervals of time.
- The **Degradation** column indicates the degradation measurement of a unit (in this case, measured in mcd).

  Note that in some cases, measuring the level of degradation is not possible without invasive techniques or destroying the specimen. In those situations, the degradation may be measured through a performance indicator, such as the luminosity of a bulb (as in this example) or resistance of a dielectric.
- The **Amps** column indicates the stress type and level (in this case, measured in mA) of the unit being tested.
- The **Unit ID** column indicates the unit being measured. The software extrapolates the failure time for each unique unit based on its degradation measurements.

Control Panel Settings

The ALTA degradation folio control panel allows you to configure the analysis settings for the data sheet and view/access the results. It consists of multiple pages, each containing options for performing particular tasks. This section focuses on the Main page of the ALTA degradation folio control panel, which contains the tools you will need to analyze the data set. *For more information about control panels in general, see page 114 in Chapter 3*.

The Main page of the ALTA degradation folio control panel is shown next.



The Main page includes the following settings:

- The **Degradation Model** area includes a drop-down list you must use to select a mathematical model. This model will be used to extrapolate the times at which the critical degradation level would be reached for each unique unit ID in the data sheet. The following is a list of all available models and their corresponding equations. The variable x represents the failure time, y represents the critical degradation, and variables a, b and c are the model parameters to be solved for in the extrapolation.
  - Linear: $y = a*x+b$
  - Exponential: $y = b*\exp(a*x)$
  - Power: $y = b*(x^a)$
  - Logarithmic: $y = a*\ln(x)+b$
  - Gompertz: $y = a*b^(c^x)$
  - Lloyd-Lipow: $y = a-b/x$

> **Tip:** If you are not certain which degradation model would provide the best fit for your data, you can use the Degradation Model Wizard to guide you through the selection process. *See page 288 in Chapter 8*.

- The **Critical Degradation** field allows you to define the level of degradation that is considered a failure. For example, you might wish to define failure as the time when the depth of a tire tread is less than 2 millimeters.
- **Suspend After** allows you to define a censoring time past which no failure times are extrapolated. In practice, there is usually a rather narrow band in which the censoring time has any practical meaning. If the censoring time is relatively low, no failure times will be extrapolated, which defeats the purpose of the degradation analysis. On the other hand, a relatively high censoring time would occur after all of the theoretical failure times, thus rendering the censored data meaningless. Nevertheless, certain situations may arise in which it is beneficial to censor the degradation data. For example, if your product has a maximum warranty life of 10,000 hours, then you may want to treat all failures beyond that time as suspensions.
- The **ALTA Model** area contains the settings for performing accelerated life testing data analysis on the failure times that were extrapolated via the degradation analysis. These settings have no effect on the degradation analysis calculations; they only affect the accelerated life testing analysis calculations. The settings in this area are identical to the settings on the control panel of an ALTA standard folio, except for the following option.
  - The **Use extrapolated intervals** option extrapolates the failure/suspensions times as interval censored data. The intervals are calculated as bounds around the extrapolated data points.

> **What's Changed?** In previous versions of ALTA, you used the Associate Specialized Folio tool to transfer the extrapolated data points to an ALTA standard folio data sheet in order to perform accelerated life testing data analysis. In the Synthesis version, you can conveniently perform this analysis within the ALTA degradation analysis folio.

When you click the **Calculate** icon on the control panel, the software extrapolates the points at which the degradation measurements will reach the point of critical degradation. It then performs a separate accelerated life testing data analysis on the extrapolated points based on the settings you have specified in the **ALTA Model** area.

- The **Degradation Results** area displays the parameters of the chosen degradation model and the extrapolated failure/suspension times. Click the **Degradation Results (...)** button to open the Results window, which allows you to view, edit and print the calculations.
- The **ALTA Results** area displays the parameters of the model that combines your selected life distribution and life-stress relationship. Click the **ALTA Results (...)** button to open the Results window.

> **Note:** If the model and parameters are associated with a published model, then the published model's name will appear as a link at the bottom of the **ALTA Results** area. Click the link to view the model's properties. *For details on how to publish the results as a model, see page 116 in Chapter 3.*

Folio Tools

The folio tools are arranged on the left side of the Main page of the control panel. Use these tools to manage data and experiment with the results of your analysis.



Calculate extrapolates the times at which the degradation will reach the critical degradation level (i.e., the times at which the degradation is serious enough that the unit is considered to be failed). The software then performs accelerated life testing analysis calculations on the extrapolated data points. This tool is also available by choosing **Degradation > Analysis > Calculate**.



Plot creates a new sheet in the folio that provides a choice of applicable plot types. This includes the degradation vs. time plot in addition to the ALTA plots. This tool is also available by choosing **Degradation > Analysis > Plot**. *See Section 9.2 on page 296*.



QCP opens the Quick Calculation Pad, which allows you to calculate accelerated life testing analysis metrics, identical to the calculations you can perform in an ALTA standard folio. This tool is also available by choosing **Degradation > Analysis > QCP**. *See page 260 in Chapter 7*.



Degradation Model Wizard opens the Degradation Model Wizard, which helps you select the degradation model that best fits your data. This tool is also available by choosing **Degradation > Analysis > Model Wizard**. *See page 288 in Chapter 8*.



Distribution Wizard opens the Distribution Wizard, which helps you select the life distribution that best fits the data set. In the degradation folio, the ranking in the Distribution Wizard is based on the life distribution included in the selected ALTA model. This tool is also available by choosing **Degradation > Analysis > Distribution Wizard**. *See page 274 in Chapter 7*.



Add or Remove Columns allows you to add/remove columns for stress levels and unit IDs. *See page 274 in Chapter 7*.



Change Units opens the Change Units window, which allows you to change the units for the time values in the current data sheet. *See page 113 in Chapter 3*.

9.2 ALTA Degradation Plot Sheet

After you analyze the degradation data in the ALTA degradation analysis folio, you can create plots of your results by choosing **Degradation > Analysis > Plot** or by clicking the **Plot** icon on the Main page of the control panel.



The **Plot Type** area on the plot sheet of the control panel allows you to select the type of plot you wish to view and the units of measurement you want the x-axis of the plot to be displayed in. The plot types available include all of the ALTA plots (*see page 257 in Chapter 7*). These are generated by analyzing the extrapolated failure times at the accelerated stress levels.

In addition to the ALTA plots, you can view the following plots, which pertain specifically to the extrapolation of failure times from the degradation measurements.

- **Degradation vs. Time (Linear)** plots the degradation of each test unit over time, using a linear scale for time and degradation. A thick horizontal line marks the critical degradation level (i.e., the level at which a unit is considered failed) that was specified on the Main page of the control panel. You can use this plot to visualize how failure times were extrapolated for each unit.
- **Degradation vs. Time (Log)** plots the same information as the Degradation vs. Time (Linear) plot, but using logarithmic axes for time and degradation.

The **Unit IDs** area of the control panel will display the unit IDs you entered in the data sheet of the folio. To remove a test unit from the plot, clear the associated check box.

9.3 ALTA Degradation Analysis Example

A manufacturer of LEDs wants to determine, with 90% confidence, whether the B10 life of the LED is at least 700 hours. During operation, the light intensity of the LED diminishes over time, and for present purposes the level at which an LED can be considered failed (i.e., the critical degradation level) is 50 mcd. The LEDs normally operate at a stress level of 28 mA, but it would take too long for units tested at this stress level to decrease to a light intensity of 50 mcd. So it is decided to perform an ALTA degradation analysis.

5 units were tested at 35 mA, and 5 were tested at 40 mA. The light intensity of each unit was recorded five times, at 50 hour intervals, as shown next.

Inspection Time (Hr)	Unit A1	Unit A2	Unit A3	Unit A4	Unit A5
50	95.1	93.3	98.3	96.6	95.8
100	86.0	87.1	92.4	88.2	89.0
150	77.6	79.7	89.0	85.1	84.0
200	70.0	74.3	84.3	81.4	81.0
250	66.7	73.0	83.0	78.6	80.0

Light Intensity (mcd) of Units Tested at 35 mA

Inspection Time (Hr)	Unit B1	Unit B2	Unit B3	Unit B4	Unit B5
50	86.6	82.1	82.7	79.8	75.1
100	78.7	71.4	70.3	68.3	66.7
150	76.0	65.4	64.0	62.3	62.8
200	71.6	61.7	61.3	60.0	59.0
250	68.0	58.0	59.3	59.0	54.0

Light Intensity (mcd) of Units Tested at 40 mA

1. Open an ALTA degradation analysis folio by choosing **Insert > Folios > ALTA Degradation Folio**. In the **Define Stress Columns and Use Stress Levels** area of the setup window, select the first check box in the Stress Name column and rename the stress to "Amps," then enter **mA** in the Stress Units column and **28** in the Use Level column.

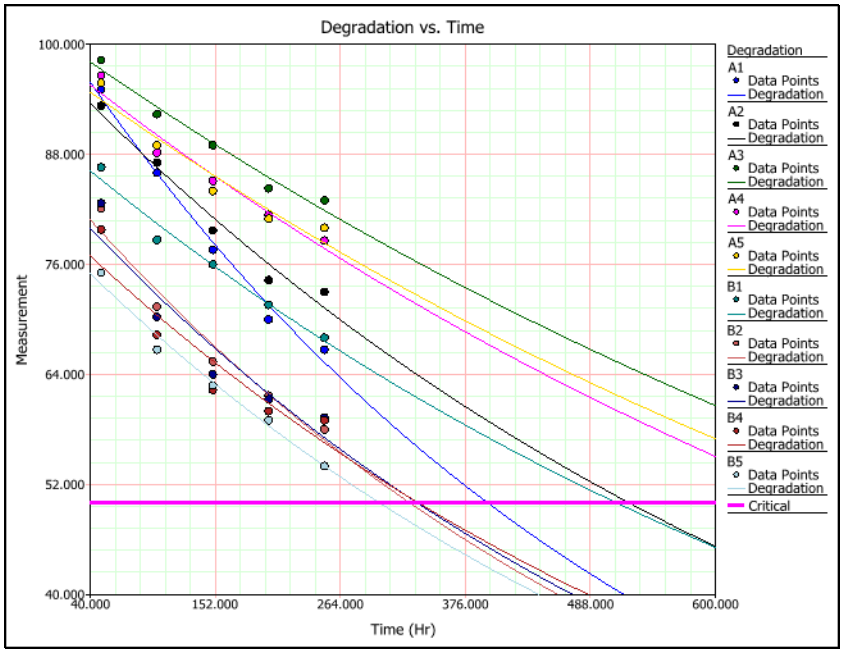


2. Click **OK** to view your new ALTA degradation folio. Then enter the data for all units into the data sheet. As an example, the data50 for units A1 and A2 are entered as shown next.

	Inspection Time (Hr)	Degradation	Amps mA	Unit ID
1	50	95.1	35	A1
2	100	86	35	A1
3	150	77.6	35	A1
4	200	70	35	A1
5	250	66.7	35	A1
6	50	93.3	35	A2
7	100	87.1	35	A2
8	150	79.7	35	A2
9	200			
10	250	Note: Complete data set is not shown.		

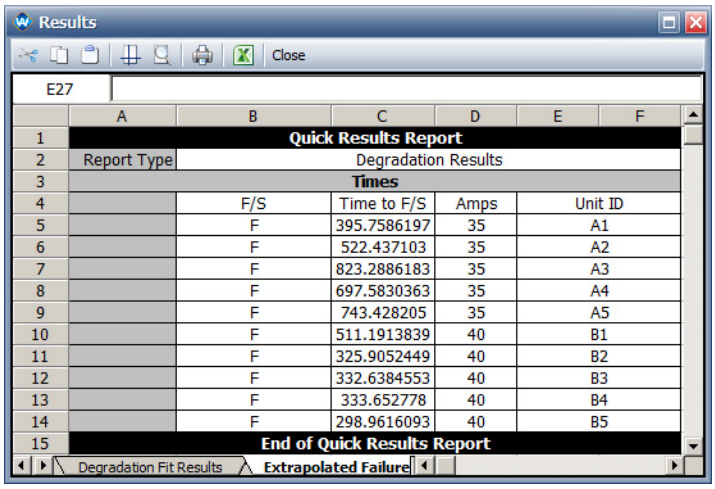
3. In the **Degradation Model** area, select **Exponential** from the drop-down list and enter **50** in the **Critical Degradation** area. In the **Model** area, select the **IPL-Weibull** model (i.e., the inverse power law life-stress relationship combined with the Weibull life distribution) from the drop-down list.
4. Click the **Calculate** icon. The software will predict the failure time for each unit under the specified accelerated stress conditions, and it will perform an accelerated life testing data analysis on those failure times in order to extrapolate to normal use conditions. To view a plot of the results click the **Plot** icon.

   The Degradation vs. Time plot shows how the performance of each unit ID degrades over time. You can select to plot the data points on a linear or logarithmic scale. The following example shows the linear scale plot. The pink line at the bottom of the plot marks the critical degradation level that was specified on the control panel.



5. To view the parameters of the chosen degradation model and the extrapolated failure/suspension times, return to the data sheet. On the control panel, click **Degradation Results (...)**. The parameters of the degradation model for

each unit ID are displayed in the first data sheet of the Results window while the extrapolated failure times for all units are displayed in the second sheet, as shown next.



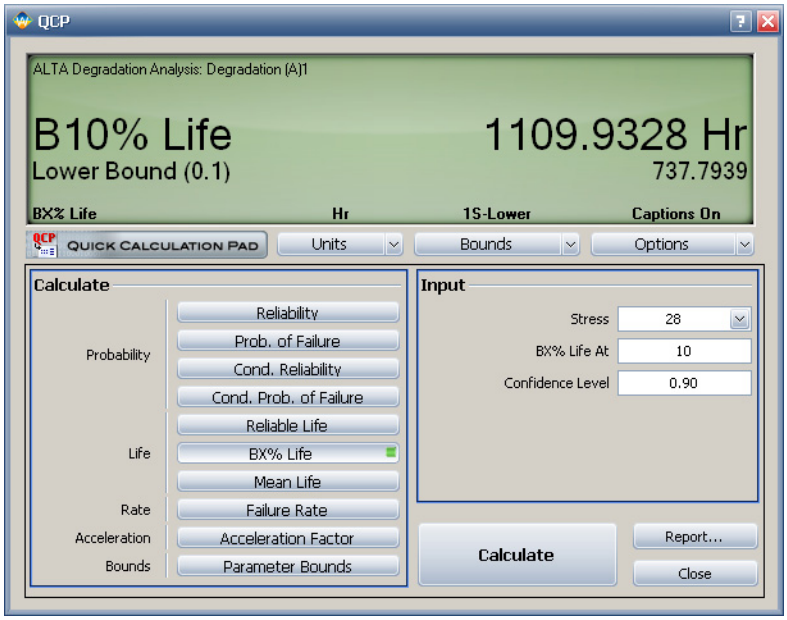
	A	B	C	D	E	F
1	Quick Results Report					
2	Report Type	Degradation Results				
3	Times					
4		F/S	Time to F/S	Amps	Unit ID	
5		F	395.7586197	35	A1	
6		F	522.437103	35	A2	
7		F	823.2886183	35	A3	
8		F	697.5830363	35	A4	
9		F	743.428205	35	A5	
10		F	511.1913839	40	B1	
11		F	325.9052449	40	B2	
12		F	332.6384553	40	B3	
13		F	333.652778	40	B4	
14		F	298.9616093	40	B5	
15	End of Quick Results Report					

6. To view the results of the accelerated life testing data analysis, close the Results window and click **ALTA Results (...)** on the control panel.
7. To estimate the time at which 10% of units are expected to fail under normal conditions, open the QCP by choosing **ALTA > Analysis > Quick Calculation Pad** or by clicking the icon on the Main page of the control panel.



In the QCP, choose to calculate the **BX% Life** with **Lower One-Sided** confidence bounds. Select **Hour** for the time units and then make the following inputs: BX% Life At = **10** and Confidence Level = **0.9**

Click **Calculate** to obtain the results, as shown next.



The results show that the B10 life of the LED is estimated to be 1109.9328 hours. The lower one-sided 90% confidence bound is estimated to be 737.7939 hours. Thus, the LED meets the manufacturer's requirement.

Non-Parametric LDA Folio 10

WEIBULL++ Only

Non-parametric life data analysis (LDA) allows you to analyze data sets without assuming an underlying life distribution. This technique is useful when dealing with unknown failure modes, when there is not enough data to assume a life distribution or when the data set does not fit any life distribution in a satisfactory way. Non-parametric methods can give the "big picture" of how products are behaving without the level of technical detail that standard (parametric) life data analysis provides, and without the risk of errors brought about by making incorrect assumptions about the distribution; however, the non-parametric analysis does not provide reliability predictions outside of the points of observation (e.g., if failures occurred at 100 and 200 hours, the analysis will calculate the reliability at those two times, but it cannot be used to calculate the reliability at 150 hours).

In cases where you might want to estimate (i.e., interpolate) the reliability of a product between the points of observation, the non-parametric LDA folio gives you the ability to create a parametric model that is based on the analysis. This means that you convert the reliability values obtained by the non-parametric analysis into unreliability values, and then fit a distribution to the free-form data set (x = time and y = unreliability). The parametric analysis allows you to interpolate (and to some extent, extrapolate) the life characteristics of the product using the Quick Calculation Pad and a wider variety of plots.[1]

This chapter describes how to use the non-parametric LDA folio, including:

- Setting up a non-parametric folio data sheet - Section 10.1 (p. 302).
- Using the control panel settings - Section 10.2 (p. 303).
- Transferring the data to a Weibull++ standard folio - Section 10.3 (p. 306).
- Example of a non-parametric life data analysis - Section 10.4 (p. 306).

1. For more information about non-parametric vs. parametric analysis, please read "*Data Analysis and Reporting*" at: http://www.Weibull.com/hotwire/issue44/hottopics44.htm.

10.1 Non-Parametric LDA Folio Data Sheet

To create a non-parametric LDA folio, choose **Insert > Folios > Non-Parametric LDA** or right-click the **Specialized Folios** folder in the current project explorer and choose **Add Non-Parametric LDA** on the shortcut menu.



Once the folio is created, select a non-parametric analysis method from the **Data Type** area of the control panel. The method you select will determine the columns that appear in the data sheet.

There are three non-parametric analysis methods available: Kaplan-Meier, Actuarial-Simple and Actuarial-Standard. In general, the Kaplan-Meier method is recommended for data sets with few suspensions, while the Actuarial-Standard and Actuarial-Simple methods are best for cases where the data set is mostly suspensions or when working with interval data, provided that none of the intervals are overlapping. There are no significant differences in the results between the two actuarial methods; however, in some cases, the approximation method of the Actuarial-Simple analysis may provide more conservative estimates.[2]

Kaplan-Meier Analysis

The following example shows the data sheet for the Kaplan-Meier analysis.

	Number in State	State F or S	State End Time (Hr)
1	3	F	9
2	1	S	9
3	1	F	11
4	1	F	13
5	1	F	17
6	1	S	19

The **Number in State** column indicates the number of units in a group, the **State F or S** column indicates whether the units in the group are failures (F) or suspensions (S) and the **State End Time** column records the end time of the interval.

Actuarial-Simple Analysis

The following example shows the data sheet for the Actuarial-Simple analysis.

	Start time (Hr)	End time (Hr)	Available Units	Number Failed	Number Suspended
1	0	50	55	2	4
2	50	100	49	0	5
3	100	150	44	2	2
4	150	200	40	3	5
5	200	250	32	2	1
6	250	300	29	1	2
7	300	350	26	2	1
8	350	400	23	3	3
9	400	450	17	3	4
10	450	500	10	1	2
11	500	550	7	2	1
12	550	600	4	1	0
13	600	650	3	2	1

2. The ReliaWiki resource portal has more information on these analysis methods at: http://www.ReliaWiki.org/index.php/Non-Parametric_Life_Data_Analysis.

The data sheet contains information about the start time and end time of each interval, as well as the number of units in each interval that are either failures or suspensions. The **Available Units** column shows the number of units at the start of each interval. The information in this column is automatically calculated by the software based on all of the entries in the **Number Failed** and **Number Suspended** columns.

Actuarial-Standard Analysis

The following example shows the data sheet for the Actuarial-Standard analysis.

	Start time (Hr)	End time (Hr)	Available Units	Adjusted Units	Number Failed	Number Suspended
1	0	50	55	53	2	4
2	50	100	49	46.5	0	5
3	100	150	44	43	2	2
4	150	200	40	37.5	3	5
5	200	250	32	31.5	2	1
6	250	300	29	28	1	2
7	300	350	26	25.5	2	1
8	350	400	23	21.5	3	3
9	400	450	17	15	3	4
10	450	500	10	9	1	2
11	500	550	7	6.5	2	1
12	550	600	4	4	1	0
13	600	650	3	2.5	2	1

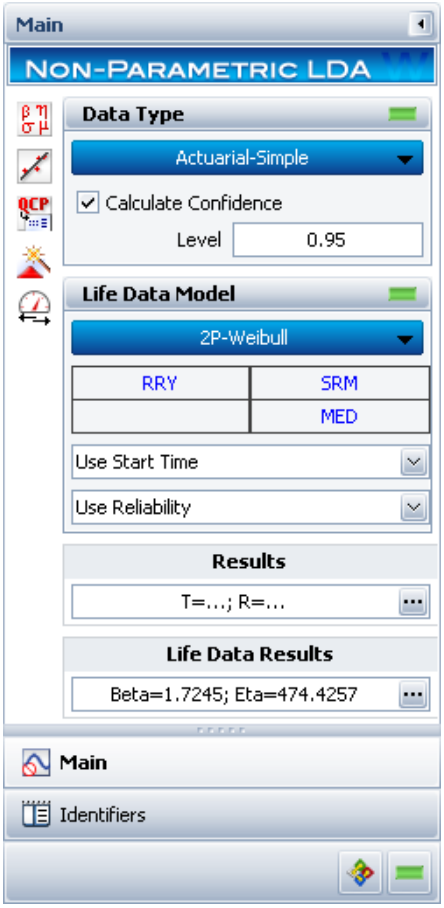
For the Actuarial-Standard analysis, the calculations are similar to the Actuarial-Simple analysis except that the number of units at the start of an interval is reduced by half of the number of suspensions in that interval. This results in reliability estimates that may be lower (i.e., more conservative) when compared to the results obtained by the Actuarial-Simple method.

The data sheet contains information about the start time and end time of each interval, as well as the number of units in each interval that are either failures or suspensions. The **Available Units** column shows the number of units at the start of the interval and the **Adjusted Units** column shows the value reduced by half of the number of suspensions. The data in these two columns are automatically calculated by the software based on all of the entries in the **Number Failed** and **Number Suspended** columns.

10.2 Control Panel Settings

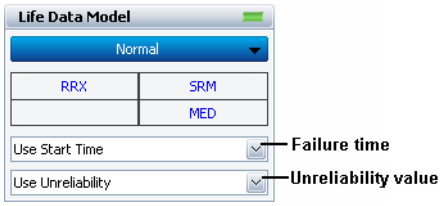
The non-parametric LDA folio control panel allows you to configure the analysis settings for the data sheet and view/access the results. It consists of multiple pages, each containing options for performing particular tasks. This section focuses on the Main page of the non-parametric LDA control panel, which contains the tools you will need to analyze the data set. *For more information about control panels in general, see page 114 in Chapter 3*.

The Main page of the non-parametric LDA folio control panel is shown next.



The Main page includes the following settings:

- The **Data Type** area allows you to select the non-parametric method for approximating the reliability. The method you select will determine the columns that appear in the data sheet.
- The **Calculate Confidence** check box allows you to calculate the lower and the upper one-sided confidence bounds of the reliability estimates at a specified confidence level. Specify the desired confidence level in the **Level** field. Selecting this option also allows you to automatically include the confidence bounds in the plot of the analysis.
- The **Life Data Model** area contains the settings for creating a parametric model that is based on the data set obtained from the non-parametric analysis. These settings have no effect on the non-parametric analysis. The options in this area are identical to the ones on the control panel of a Weibull++ standard folio, except for the two drop-down lists shown next:



The two options allow you to choose which values the chosen life distribution will be fitted to. The options are:

- **Failure time**: If you use the Kaplan-Meier analysis, the failure times are the end times of the intervals. If you use either the Actuarial-Simple or Actuarial-Standard analysis, each failure time may be the start time of the interval, end time of the interval, or the average between the start time and end time.
- **Unreliability value**: If you select to include confidence bounds for the non-parametric analysis, you will have the option to select either the unreliability estimates calculated by the non-parametric analysis, or the upper confidence bounds or lower confidence bounds of the unreliability estimate.

When you click the **Calculate** icon on the control panel, the software analyzes the data set based on the chosen non-parametric method. It then automatically performs a separate standard life data analysis based on the settings you have specified in the **Life Data Model** area on the control panel.

- The **Non-Parametric Results** area allows you to view the reliability estimates calculated by the non-parametric analysis and the free-form data set that will be used for parametric analysis. Click the **Non-Parametric Results (...)** button to open the Results window, which allows you to view, edit and print the report.
- The **Life Data Results** area allows you to view the parameters of the life data model. Click the **Life Data Results (...)** button to open the Results window. You can also use the Quick Calculation Pad (QCP) to obtain results based on the life data model, such as the probability of failure or mean life.

Folio Tools

The folio tools are arranged on the left side of Main page of the control panel. Use these tools to manage data and experiment with the results of your analysis.



Calculate calculates the reliability estimates based on the chosen non-parametric method (i.e., Kaplan-Meier, Actuarial-Simple or Actuarial-Standard). It then performs a separate standard (parametric) life data analysis based on the settings in the Life Data Model area of the control panel. This tool is also available by choosing **Non-Parametric LDA > Analysis > Calculate**.



Plot creates a Reliability vs. Time plot of the non-parametric analysis. The scaling, setup, exporting and other features are similar to the options available for other Weibull++ plot sheets. Features that are not applicable to this plot will be hidden or disabled. Clicking the **Plot** icon before the folio has been calculated will automatically calculate the folio and then plot the data. This tool is also available by choosing **Non-Parametric LDA > Analysis > Plot**.



QCP opens the Weibull++ Quick Calculation Pad, which allows you to calculate results, such as mean life and probability of failure, based on the results of the standard life data analysis. It uses the same calculations that you can perform in a Weibull++ standard folio QCP. The tool is also available by choosing **Non-Parametric LDA > Analysis > Quick Calculation Pad**. *See page 221 in Chapter 6.*



Distribution Wizard opens the Distribution Wizard, which helps you select the distribution that best fits the data set based on the chosen parameter estimation method (i.e., RRX or RRY). In the non-parametric LDA folio, the ranking in the Distribution Wizard is based on the life data model. This tool is also available by choosing **Non-Parametric LDA > Analysis > Distribution Wizard**. *See page 226 in Chapter 6.*



Change Units opens the Change Units window, which allow you to change the units for the time values in the current data sheet. *See page 113 in Chapter 3.*

10.3 Transferring Data to a Weibull++ Standard Folio

The Life Data Model area of the non-parametric LDA control panel gives you the ability to create a parametric model that is based on the analysis. In order to assess whether the parametric model provides a good fit to the results of the analysis, you can transfer the data to a Weibull++ standard folio. To do this, choose **Non-Parametric LDA > Transfer Life Data > Transfer Life Data to a New Folio**.



The data will be transferred to a free-form data sheet, where the failure time values will be copied to the "X-Axis value" column and the unreliability values will be copied to the "Y-Axis value" column. In the standard folio, you could use the plots and other results to evaluate the fit of the distribution to the values from the non-parametric analysis. If the distribution does not provide a good fit, you could experiment with other distributions to see which one would provide a better fit. Once you have selected an appropriate parametric model, you can use it to obtain calculations from the standard folio's QCP, or to generate other graphical plots in the standard folio's plot sheet. If you use the same analysis settings in both the standard folio and non-parametric LDA folio, then you will obtain the same results from the QCPs in both folios.

10.4 Non-Parametric Life Data Analysis Example

The data set used in this example is 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 "Non-Parametric LDA - Three Methods" and the folio that contains the data is called "Kaplan-Meier."

An analyst is performing a field test for a prototype product. A group of 21 participants agreed to test the product and all participants are to return after appointed periods of time to report their experience with the product.

Units that are reported to have malfunctioned during the test period are marked as failed (F), while units that did not exhibit any problems are marked as suspensions (S). At the end of the 41 week test, the analyst's log shows the following information.

Number of Units	State (F or S)	Test Period (Weeks)
3	F	9
1	S	9
1	F	11
1	S	12
1	F	13
1	S	13
1	S	15
1	F	17
1	F	21
1	S	22
1	S	24

Number of Units	State (F or S)	Test Period (Weeks)
1	S	26
1	F	28
1	F	30
1	S	32
2	S	35
1	S	39
1	S	41

The objective is to create a Reliability vs. Time plot in order to predict the failure rate behavior of the product.

1. The first step is to create a non-parametric LDA folio. Choose **Insert > Folios > Non-Parametric LDA**.



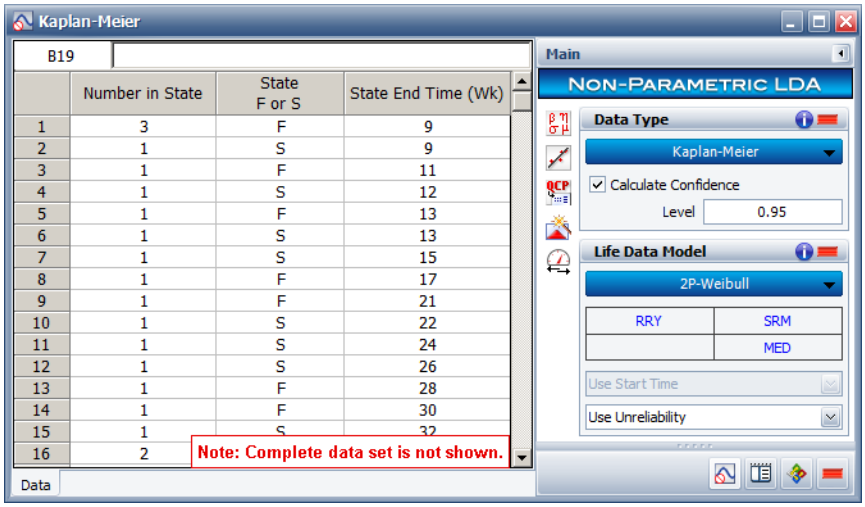
2. On the Main page of the control panel, choose the **Kaplan-Meier** analysis. This will format the data sheet correctly for the non-parametric analysis. Select the **Calculate Confidence** check box and enter **0.95** for the confidence level.
3. Click the **Change Units** icon.



In the Change Units window, change the time unit to **Week**. Click **OK**.

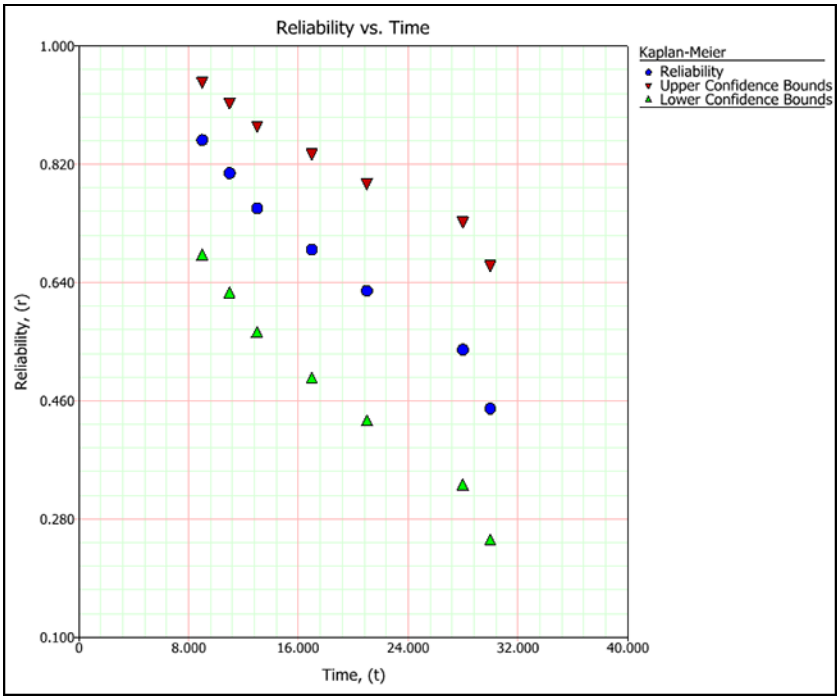
4. Enter the information from the table into the data sheet.
5. The non-parametric LDA folio will obtain the reliability estimates only for the failure times that were entered in the data sheet (in this example, it is for weeks 9, 11, 13, 17, 21, 28 and 30). In order to estimate (i.e., interpolate) the reliability of the product at other times, you will need to create a parametric model. To do this, make the following selections in the **Life Data Model** area of the control panel:
   - **2P-Weibull**.
   - **RRY** (Rank Regression on Y).
   - **Use Unreliability**. This means that the free-form data set will be based on the unreliability estimates calculated from the non-parametric analysis, rather than the upper or lower confidence bound.

The following picture shows the completed setup.

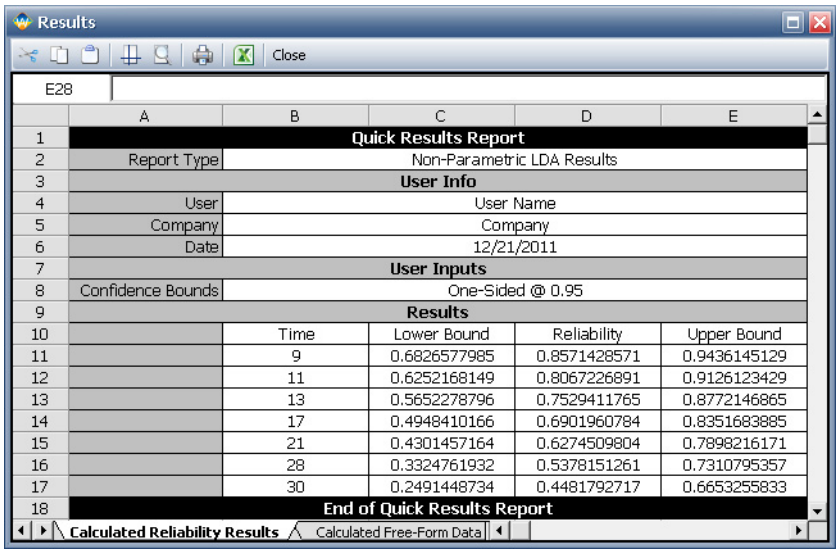


6. Click the **Calculate** icon on the control panel. The software will perform a non-parametric analysis on the data set, and then automatically perform a separate standard life data analysis based on the settings specified in the **Life Data Model** area.
7. To create a Reliability vs. Time plot, click the **Plot** the icon on the control panel.

   In the following example, the dots on the plot show the reliability estimates at each point of failure, the triangles show the lower 1-sided confidence bounds of the estimates, and the inverted triangles show the upper 1-sided confidence bounds. The pattern of the data points shows that the product exhibits a sharp decline in reliability over the course of a few weeks.



8. To view the calculated reliability values, click the **Non-Parametric Results (...)** button on the control panel to open the Results window, as shown next.



Results

Close

E28

	A	B	C	D	E
1	Quick Results Report				
2	Report Type	Non-Parametric LDA Results			
3	User Info				
4	User	User Name			
5	Company	Company			
6	Date	12/21/2011			
7	User Inputs				
8	Confidence Bounds	One-Sided @ 0.95			
9	Results				
10		Time	Lower Bound	Reliability	Upper Bound
11		9	0.6826577985	0.8571428571	0.9436145129
12		11	0.6252168149	0.8067226891	0.9126123429
13		13	0.5652278796	0.7529411765	0.8772146865
14		17	0.4948410166	0.6901960784	0.8351683885
15		21	0.4301457164	0.6274509804	0.7898216171
16		28	0.3324761932	0.5378151261	0.7310795357
17		30	0.2491448734	0.4481792717	0.6653255833
18	End of Quick Results Report				

Calculated Reliability Results | Calculated Free-Form Data

The results show that the reliability of the product at 9 weeks of operation is estimated to be 85.71%; however, by 30 weeks, the reliability estimate is 44.81% and may be as low as 24.91%.

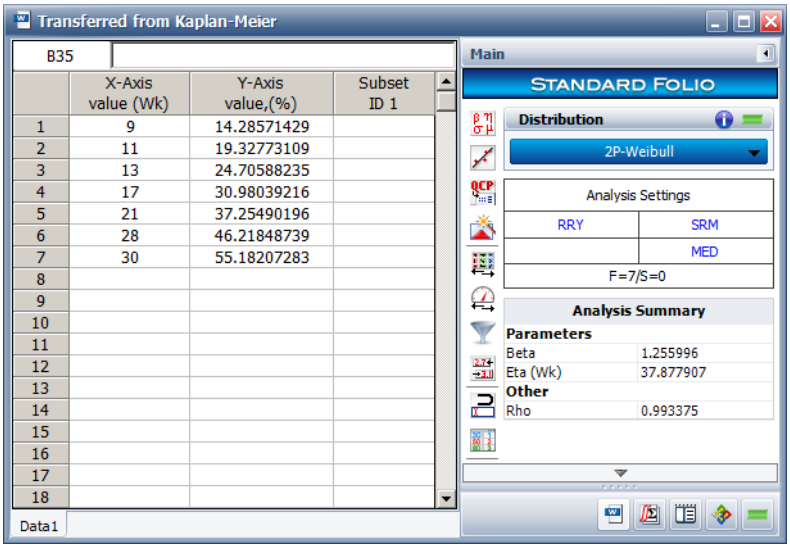
Evaluating the Life Data Model

9. To assess whether the assumed distribution (that was chosen in the Life Data Model area of the control panel) provides a good fit, you can transfer the results of the non-parametric analysis to a Weibull++ standard folio. To do this, choose **Non-Parametric LDA > Transfer Life Data > Transfer Life Data to a New Folio**.



The data will be transferred to a free-form data sheet, where the failure time values will be copied to the "X-Axis value" column and the unreliability values will be copied to the "Y-Axis value" column.

10. In the standard folio, select the same analysis settings that you chose in the non-parametric LDA folio (i.e., 2P-Weibull and RRY), and then click the **Calculate** icon on the control panel. The following picture shows the completed analysis.



Transferred from Kaplan-Meier

B35

	X-Axis value (Wk)	Y-Axis value,(%)	Subset ID 1
1	9	14.28571429	
2	11	19.32773109	
3	13	24.70588235	
4	17	30.98039216	
5	21	37.25490196	
6	28	46.21848739	
7	30	55.18207283	
8			
9			
10			
11			
12			
13			
14			
15			
16			
17			
18			

Data1

Main

STANDARD FOLIO

Distribution

2P-Weibull

Analysis Settings

RRY | SRM

MED

F=7/S=0

Analysis Summary

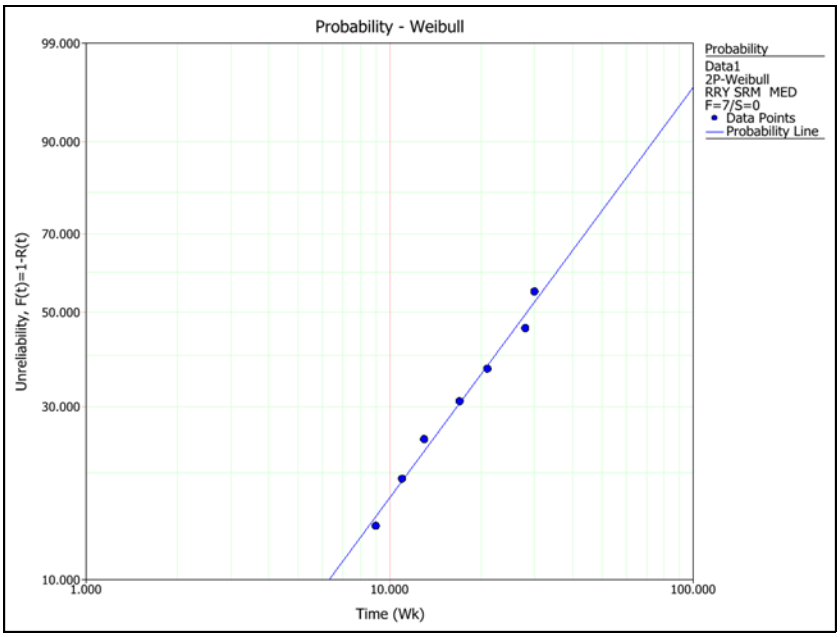
Parameters

Beta	1.255996
Eta (Wk)	37.877907

Other

Rho	0.993375

11. Click the **Plot** icon on the control panel. The following probability plot shows the probability line with respect to the values obtained from the non-parametric analysis. As you can see, the 2P-Weibull provides a good fit to the values.



If the distribution did not provide a good fit, you could experiment with other distributions to see which one would provide a better fit. Once you have selected an appropriate parametric model, you can use it to obtain calculations from the Quick Calculation Pad (QCP).

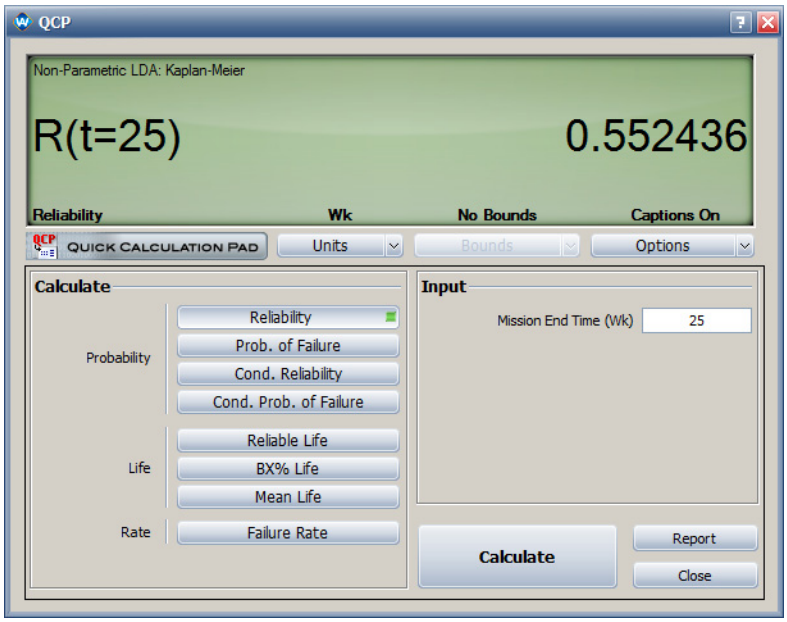
Obtaining Results for Other Points in Time

12. To obtain a reliability estimate at a time that was not calculated in the non-parametric analysis, say, 25 weeks, use the Quick Calculation Pad (QCP). You can use the QCP either from the standard folio with the transferred data or from the non-parametric LDA folio. If you use the same analysis settings (i.e., distribution, parameter estimation method) in both the standard folio and non-parametric LDA folio, then you will obtain the same results from the QCPs in both folios.

    Click the **QCP** icon on the control panel.



In the QCP, choose to calculate the **Reliability**. Select **Week** for the time units and then enter **25** for the mission end time. Click **Calculate** to display the results. The reliability is estimated to be 55.2436%, as shown next.



Note: Confidence bounds calculations are not available for free-form data.

Warranty Analysis 11

Despite best efforts to design and produce products that will last the expected lifetime, field failures are a reality. In order to anticipate customer needs and ensure customer satisfaction, there must be a process for studying field failures and correcting existing, as well as potential, reliability problems of fielded products. The analysis of warranty claims data is one such method.

WEIBULL++ Only

Warranty claims data typically consist of quantities of units sold and quantities returned within a specified warranty period. Performing reliability calculations on this type of raw data can be tedious and time-consuming. To speed up your calculations, the Weibull++ warranty analysis folio is designed to convert your existing warranty claims data sets into failure/suspension data sets so that they can be easily analyzed with traditional life data analysis methods. In addition, the folio gives you the ability to predict future warranty returns so you can detect and correct potential product quality problems in the field and plan for warranty fulfillment needs such as repair costs and the number of spares to stock.[1]

This chapter describes how:

- Create a warranty analysis folio and set up the control panel - Section 11.1 (p. 314).
- Use the different warranty analysis folio formats, including:
  - Nevada chart format - Section 11.2 (p. 317).
  - Times-to-failure format - Section 11.3 (p. 328).
  - Dates of failure format - Section 11.4 (p. 333).
  - Usage format - Section 11.5 (p. 338).
- Forecast warranty returns - Section 11.6 (p. 350).
- Use the warranty analysis plot settings - Section 11.7 (p. 352).
- Manage warranty units in Weibull++ - Section 11.8 (p. 352)

1. The ReliaWiki resources portal has more information on warranty analysis at: http://www.ReliaWiki.org/index.php/Warranty_Data_Analysis.

11.1 Warranty Analysis Folio Setup

To add a warranty analysis folio, choose **Insert > Folios > Warranty** or right-click the **Specialized Folios** folder in the current project explorer and choose **Add Warranty** on the shortcut menu.



A setup window will prompt you to select a format to use with the folio. Select the format that is appropriate for the type of warranty claims data you have available. If you select the Nevada chart format, click **Next** to go to the next step to configure your analysis; otherwise, click **OK** to immediately create the folio.

Converting Formats

You have the option to convert the format of an existing warranty analysis folio to a new format. To do so, open the existing folio and choose **Warranty > Tools > Convert Data**, and then choose the format to convert to.

Converting formats creates a new folio and does not overwrite the contents of the existing folio. It is an automatic process, except when converting from the dates of failure format to the Nevada chart format, and vice versa. In that case, you will be prompted to define the time unit of each period (e.g., dates, month or years) in order to proceed with the conversion. The following table shows a summary of the data conversion options:

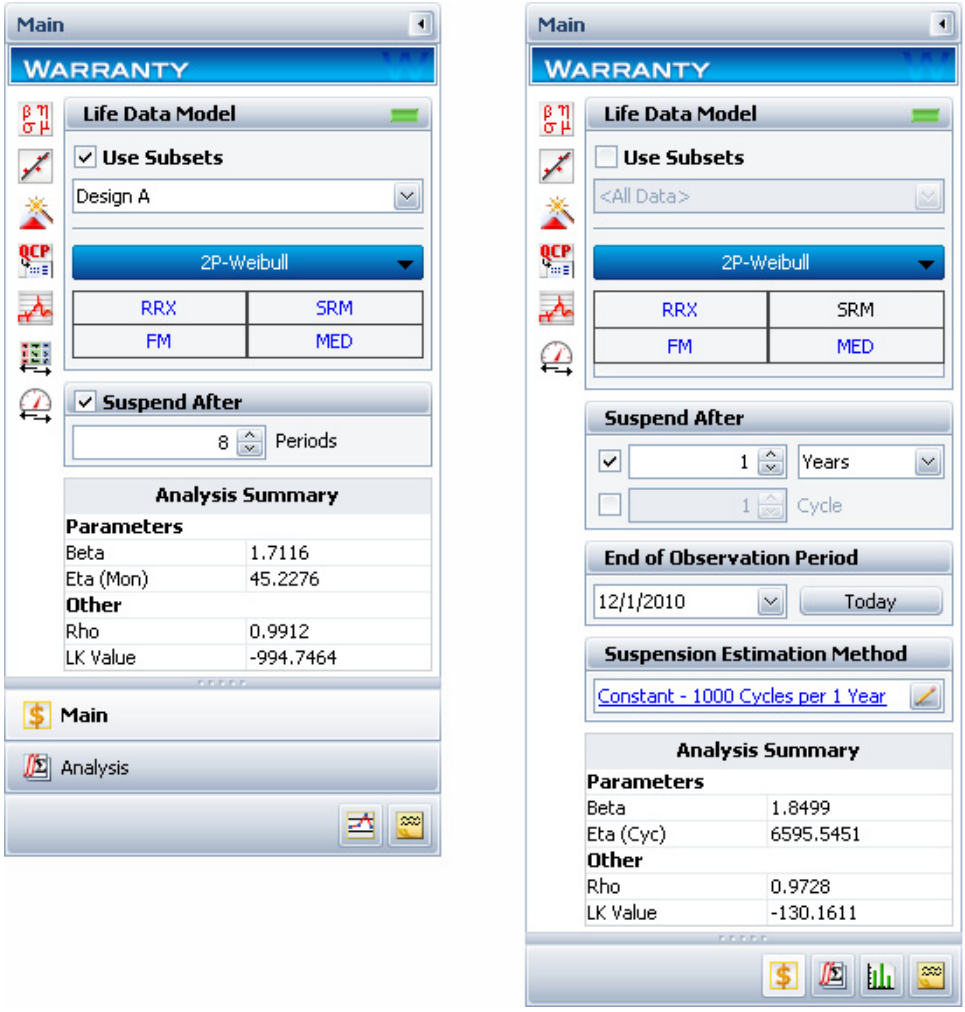
From	Convert To
Nevada Chart Format	Dates of Failure Format
	Times-to-Failure Format
Dates of Failure Format	Nevada Chart Format
	Times-to-Failure Format
Times-to-Failure Format	N/A
Usage Format	Times-to-Failure Format

11.1.1 Warranty Analysis Folio Control Panel

The warranty analysis folio control panel allows you to configure the analysis settings for the data sheet and view/access the results. It consists of the following pages:

- The **Main** page contains all the tools you will need to perform a warranty analysis. These tools are described in the next section below.
- The **Analysis** page contains the settings that are also displayed on the Main page of the control panel.
- The **Comments** page allows you to enter notes or other text that will be saved with the folio.
- The **Statistical Process Control** page is available only for the Nevada chart format. It contains optional tools for monitoring warranty returns data. *See Section 11.2.3 on page 322*.
- The **Suspensions** page is available only for the Usage format. It contains the settings for estimating the amount of usage for fielded units. *See Section 11.5.2 on page 341*.

As an example, the following picture shows the Main page of the control panels of the Nevada chart format and usage format, respectively.



Control Panel Main Page

The Main page of the warranty analysis folio contains most of the tools you will need to perform a warranty analysis. The following settings are available for all formats:

- The **Life Data Model** area contains the options for performing life data analysis. When you click **Calculate**, Weibull++ converts the warranty claims data into failure/suspension data and then performs a life data analysis based on the settings you specified in this area.
  - **Use Subsets** gives you the option to analyze multi-population data sets. To use this setting, you must first assign each subpopulation group in your data sheet to a specific subset ID. Select the **Use Subsets** check box, and then use the drop-down list to switch between the subset IDs and alter the analysis settings (i.e., distribution and parameter estimation method) for each one. When you click **Calculate**, the software will separately analyze each subset of data. You can view the parameters of each subset ID by choosing the subset ID from the drop-down list or by clicking anywhere within the **Analysis Summary** area.

- The **Suspend After** check box allows you to specify a period beyond which, all additional data will be treated as suspensions. Most product warranties are of a certain duration, and in many cases, data after the warranty period are incomplete or unreliable. This setting allows you ignore any failure times after the specified period and treat them as suspensions. This option is not available for the times-to-failure format.

  Note that:

  - In the Nevada chart format, the warranty period uses the same time units used in the Nevada chart data sheets.
  - In the Dates of Failure format, you can specify the warranty period to be in days, months or years.
  - In the Usage format, you have two options: you can specify the warranty period to be in days, month or years, or you can consider a warranty period based on usage (e.g., 100K miles or 10 years, whichever comes first).
- The **Analysis Summary** area displays the results of the life data analysis. Click anywhere within the area to open the Results window, which shows the converted failure/suspension data and the parameters of the life distribution model. From the Results window, you can edit, copy or print the results.
- The **End of Observation Period** setting is s available only in the dates of failure format and usage format. It is used to determine the time in service for each unit that had not yet been returned by the end of the observation period (i.e., the suspensions). The date is typically set to the last day the warranty data was collected.
- The **Suspensions Estimation Method** area is available only in the usage format. It displays information about how the product usage is defined. Clicking the link will take you to the Suspensions page, which contains the options for estimating the amount of usage accumulated by suspensions. *See Section 11.5.2 on page 341*.

Folio Tools

The folio tools are arranged on the left side of the Main page of the control panel. Use these tools to manage data and experiment with the results of your analysis.



Calculate converts the warranty claims data to failure/suspension times and then estimates the parameters of the chosen life distribution based on the converted data. If you select the **Use Subsets** option on the Main page of the control panel, the parameters of the distribution of each subset ID will be separately calculated. This tool is also available by choosing **Warranty > Analysis > Calculate**.



Plot creates a new sheet in the folio that provides a choice of applicable plot types. In warranty analysis folios, this includes plots such as expected failures vs. period, reliability vs. time, probability plot, etc. You can plot the data in the warranty analysis folio either with or without generating a forecast. This tool is also available by choosing **Warranty > Analysis > Plot**.



Distribution Wizard opens the Distribution Wizard, which helps you select the distribution that best fits your data based on the selected parameter estimation method (i.e., RRX, RRY or MLE). In warranty analysis folios, the ranking in the Distribution Wizard is based on the converted failure/suspension times. This tool is also available by choosing **Warranty > Analysis > Distribution Wizard**. *See page 226 in Chapter 6*.



QCP opens the Quick Calculation Pad, which allows you to calculate life data analysis metrics. It uses the same calculations that you can perform in a Weibull++ standard folio QCP. The tool is also available by choosing **Warranty > Analysis > Quick Calculation Pad**. *See page 221 in Chapter 6*.



Forecast opens the Forecast Setup window, which allows you to specify the time periods at which you wish to estimate the number of expected warranty returns. Clicking **Forecast** before the parameters have been calculated will automatically calculate the parameters and open the Forecast Setup window. This tool is also available by choosing **Warranty > Tools > Forecast**. *See Section 11.6 on page 350*.



Data Sheet Setup is available only for the Nevada chart format. It opens the Warranty Setup window, which allows you to change the time periods of the warranty claims data in the current data sheet. *See Section 11.2 on page 317*.



Change Units opens the Change Units window, which allows you to select the units to use in the data sheet. This tool is not available in the dates of failure format. The units that are available in the drop-down list are defined at the database level (*see page 39 in Chapter 2*). Note that:

- In the Nevada chart format, this tool is available only when you choose to label the sales and returns periods in terms of numbers during the setup process. *See Section 11.2 on page 317*.
- In the usage format, only the units that are related to usage (e.g., cycles, miles, etc.) are available in the drop-down list.
- In the times-to-failure format, all the time units that were defined at the database level are available in the drop-down list.

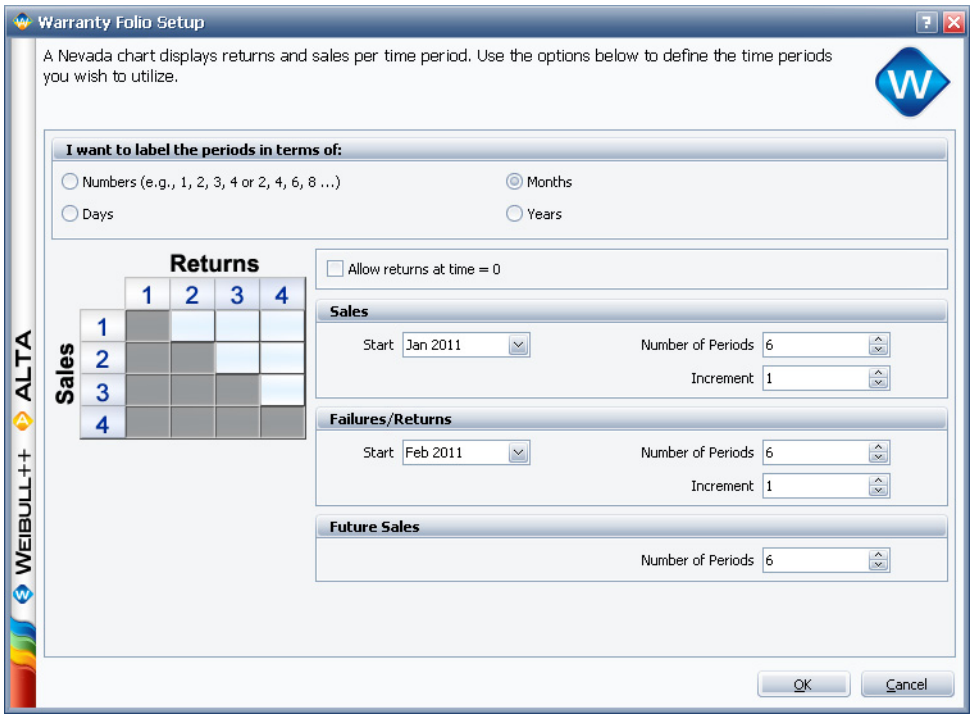
11.2 Nevada Chart Format

If you keep track of the period in which each returned unit was sold and the period in which it was returned, you can use the Nevada chart format to convert your warranty claims data into failure/suspension data. The following sections describe how to work with this format, including:

- How to set up a Nevada chart and enter sales and return data - Section 11.2.1 (p. 317)
- Using zero-failure times in the analysis - Section 11.2.2 (p. 319)
- Using the statistical process control (SPC) feature - Section 11.2.3 (p. 322)
- An example of warranty analysis involving SPC - Section 11.2.4 (p. 324)

11.2.1 Setting Up the Chart and Entering Data

The setup window of the Nevada chart format allows you to define the time periods of interest. The value in the **Increment** field indicates the length of the time period. In the following example, the time unit is in terms of months; therefore, an increment of 1 would mean that 1 period is equivalent to 1 month, while an increment of 2 would mean that 1 period is equivalent to 2 months.



The setup window also includes the following optional settings:

- The **Future Sales** area allows you to define the number of future sales periods. This setting will include additional rows in the Sales data sheet for entering the projected sales figures. Information regarding future sales is used only in the forecast analysis.
- **Allow Returns at Time = 0** allows you to include zero-time failures in the analysis, which are failures that occurred before the units made it to the field. These failures may be due to manufacturing defects, insufficient quality control or shipping damages.

Time Units

In the Nevada chart format, selecting a new unit only changes the name of the unit. The data set is not converted to the new unit. The appropriate columns in the data sheet will be automatically configured for the time units you selected, and the new unit will be displayed next to the results in the control panel.

There are two ways you could edit the time periods for an existing Nevada chart data sheet:

- If the periods are labeled in terms of days, months or years, you can change the time units and periods anytime by choosing **Warranty > Tools > Data Sheet Setup** or by clicking the icon on the control panel.



  This opens the Warranty Folio Setup window, which allows you to edit the time periods. Note that when you change the existing setup, you may lose information already entered in the data sheet.

- If the periods are labeled in terms of numbers, you can use any time units appropriate for your data. To select a time unit, click the **Change Units** icon on the Main page of the control panel of the data sheet.



  This opens the Change Units window, which gives you a selection of time units to use. The units that are available in the drop-down list are defined at the database level. *See page 39 in Chapter 2*.

Sales and Return Data

The Nevada chart folio consists of two data sheets: Sales and Returns. The picture shown next is an example of the Sales data sheet.

Period	Quantity In-Service	Subset ID
Jan 10	100	Batch 1
Feb 10	150	Batch 2
Mar 10	200	Batch 3
Apr 10	250	Future Sales
May 10	250	Future Sales
Jun 10	250	Future Sales

Note: For the column headings in this sheet, the general term *in-service* represents the closest estimate of the time at which each unit started operating in the field. For example, if you have access to both the manufacturing date and the shipping date for a group of parts, you may choose to perform the analysis based on the shipping date because it is a closer indicator of the time the product started operating in the field.

The Sales data sheet contains the following information:

- The **Period** column shows the time periods that were defined during setup.
- The **Quantity In-Service** column indicates the number of units that are assumed to have started operating during each period.
- The **Subset ID** column is for logging any pertinent information or comments about the data. You can also use the Subset ID column to categorize subpopulations in your data set. You will be able to choose a distribution appropriate for each subset of data by selecting the **Use Subsets** option on the Main page of the control panel.

> **Note:** The subset ID can be any text up to 30 characters, including spaces. For example, "A _ _ X" is not the same as "A _ X" where " _ " is used to designate a space.

- If you entered future sales periods during setup, yellow rows will be added in the data sheet to identify the future sales data. The projected sales figures will be used in the forecast analysis. *See Section 11.6 on page 350*.

The picture shown next is an example of the Returns data sheet. The column headers represent the warranty periods and the row headers represent the sales periods.

	Feb 10	Mar 10	Apr 10
Jan 10	3	5	7
Feb 10		2	4
Mar 10			5

In the example shown here, there were 3 returns in February from the batch that was in-service in January. In March, 5 more units from the batch that was in-service in January were returned and 2 units from the batch that was in-service in February were also returned. The rest of the chart can be read in a similar manner.

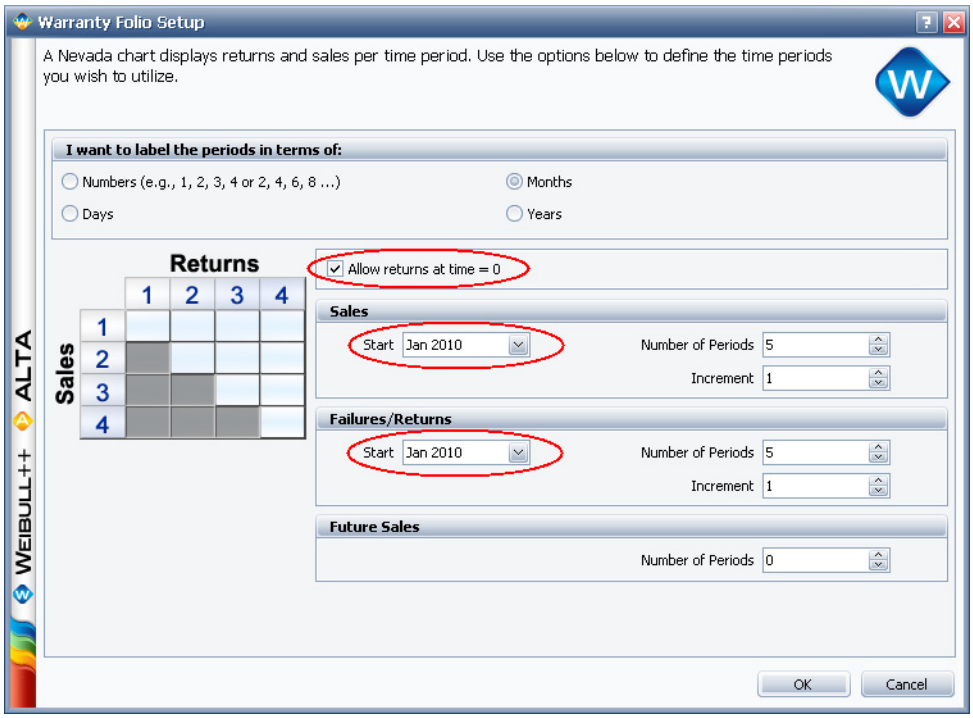
To perform the warranty analysis, go to the Main page of the warranty folio control panel and choose a failure distribution for the data set and a parameter estimation method (you can also choose the parameter estimation settings from the Analysis page of the control panel). Click **Calculate**. Weibull++ automatically converts the warranty data into failure/suspension times and also estimates the parameters of the chosen distribution.

To view the failures/suspension data, click anywhere within the **Analysis Summary** area of the control panel to open the Results window, which shows the converted failure/suspension data and the parameters of the life distribution model. From the Results window, you can edit, copy or print the results.

11.2.2 Zero-Time Failures

In many cases, it is possible for failures to occur immediately at the beginning of the life of the unit. These types of failures are known as out-of-the-box failures or zero-time failures. Examples are products that failed quality inspections prior to shipping or were damaged during shipping and then returned within the same period they were sold.

To include zero-time failures in the Nevada chart warranty analysis, select the check box during setup and set the return start date to be the same as the sales start date, as shown in the following example.

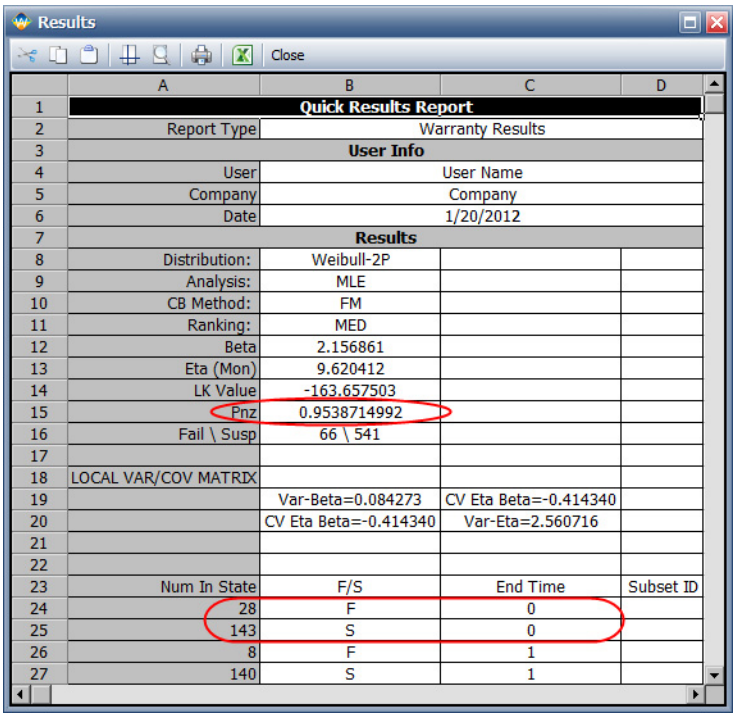


When you enable this setting and click **OK**, the Returns sheet will allow you to enter returns for the same month in which the units were in-service, as shown next.

	Jan 10	Feb 10	Mar 10	Apr 10	May 10
Jan 10	5	2	6	5	3
Feb 10		3	1	5	4
Mar 10			5	3	7
Apr 10				8	2
May 10					7

In this example, 5 units from the January sales were damaged during shipping and so were returned within the January sales period. These 5 failures will be treated as zero-time failures. In February, 3 units were also returned within the February sales period. These 3 failures are also zero-time failures. The rest of the chart can be read in a similar manner.

When you calculate the parameters and view the results in the Results window, the converted data set will contain a certain number of data points with failures at time = 0, as shown in the next example.



	A	B	C	D
1	**Quick Results Report**			
2	Report Type	Warranty Results		
3	**User Info**			
4	User	User Name		
5	Company	Company		
6	Date	1/20/2012		
7	**Results**			
8	Distribution:	Weibull-2P		
9	Analysis:	MLE		
10	CB Method:	FM		
11	Ranking:	MED		
12	Beta	2.156861		
13	Eta (Mon)	9.620412		
14	LK Value	-163.657503		
15	Pnz	0.9538714992		
16	Fail \ Susp	66 \ 541		
17				
18	LOCAL VAR/COV MATRIX			
19		Var-Beta=0.084273	CV Eta Beta=-0.414340	
20		CV Eta Beta=-0.414340	Var-Eta=2.560716	
21				
22				
23	Num In State	F/S	End Time	Subset ID
24	28	F	0	
25	143	S	0	
26	8	F	1	
27	140	S	1	

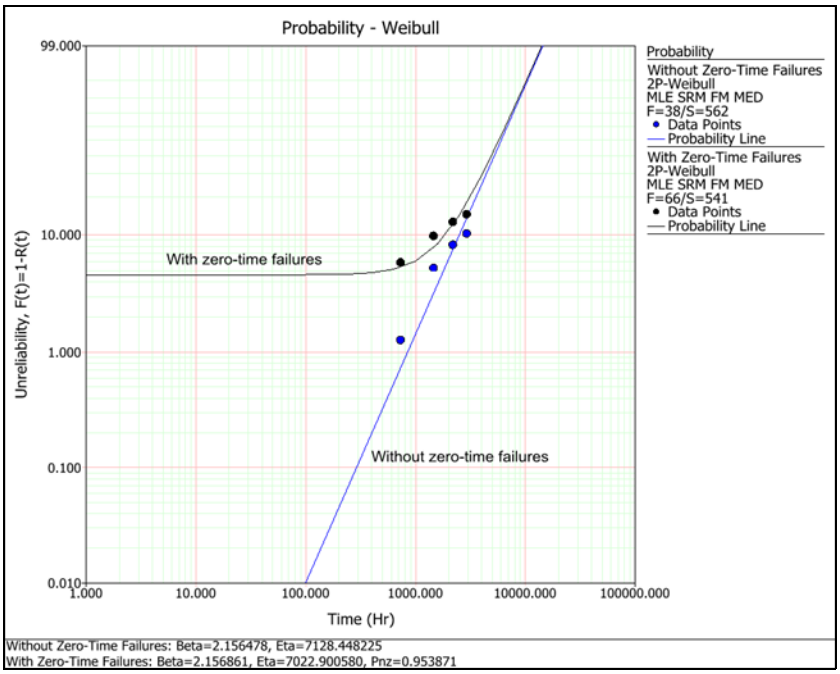
Pnz represents the proportion of the population with non-zero failure times. In this sample report, the percentage of the population with non-zero failure times is approximately 95.3871%.

The distribution parameters are calculated without the zero-time failures but any subsequent reliability calculations for the full data set will be multiplied by the Pnz value, such that:

$$R'(t) = Pnz \times R(t)$$

where $R'(t)$ is the reliability function for the entire data set and $R(t)$ is the reliability function for the data set without the zero-time failures. Therefore, when Pnz is equal to 1, all the failures are assumed to occur after time = 0.

The following overlay plot illustrates the effect of the Pnz value on the reliability function. The straight line is the unreliability function for the set of non-zero failure and suspension times. The inclusion of the zero-time failures causes the unreliability function to curve towards the value of unreliability at time = 0, which is 1-Pnz.



When you use the Quick Calculation Pad (QCP), the reliability, unreliability and probability density calculations of the entire data set are influenced by the value of Pnz, while the conditional reliability and failure rate functions are not affected.

11.2.3 Statistical Process Control

The Nevada chart format includes a statistical process control (SPC) method for analyzing warranty returns data. The method uses a chi-squared test to help you detect unusually high or low return rates for any given period, alerting you to any possible deviations in manufacturing, quality control or any other factors that may adversely affect the reliability of the product in the field. This enables you to intervene immediately and avoid increased warranty costs or more serious repercussions. The SPC application is available only for the Nevada chart format.

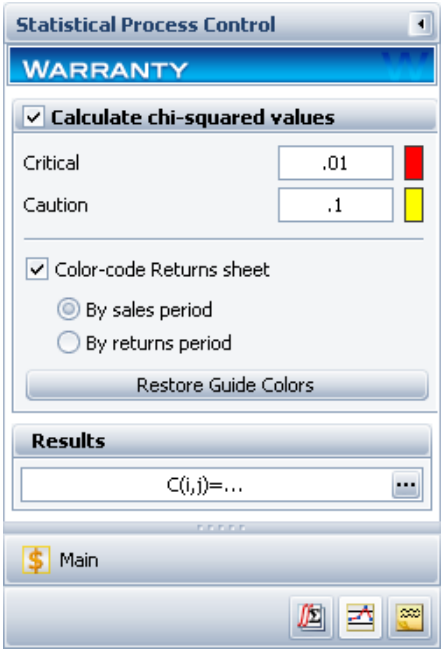
Note: The SPC feature applies only to homogeneous data sets and is therefore unavailable when you select the **Use Subsets** option on the Main page of the control panel.

The SPC method evaluates the discrepancy between the actual number of failures and the number of failures predicted by the distribution parameters. To use SPC, click the **SPC page** icon on the control panel.



Select the **Calculate chi-squared values** check box. The fields for the **Critical** and **Caution** control values will become available. The control values are the upper probability levels of the chi-squared distribution below which the sales or return periods will be flagged as outliers. Enter the desired critical and caution levels in decimal form. The

values must be greater than zero and less than one. By default, the critical value is 1% and the caution value is 10%, as shown next.



Calculate the chi-squared values by choosing **Warranty > Analysis > Calculate** or by clicking the icon on the Main page of the control panel.



If you wish to view the table of chi-squared values, click the **Show Results (...)** button.

Tip: If you wish to perform a manual evaluation of the table of chi-squared values, you can use Weibull++'s Quick Statistical Reference (QSR) tool to look up chi-squared values for different control values and degrees of freedom. You can access the QSR by choosing **Home > Tools > Quick Statistical Reference**. *See Chapter 26*.

Analysis Results

To provide an easy visualization of the results, select the **Color-code Returns sheet** check box. You can select to evaluate the results in two ways: **By sales period** (results are color-coded by rows) or **By returns period** (results are color-coded by columns).

The default color indicator are:

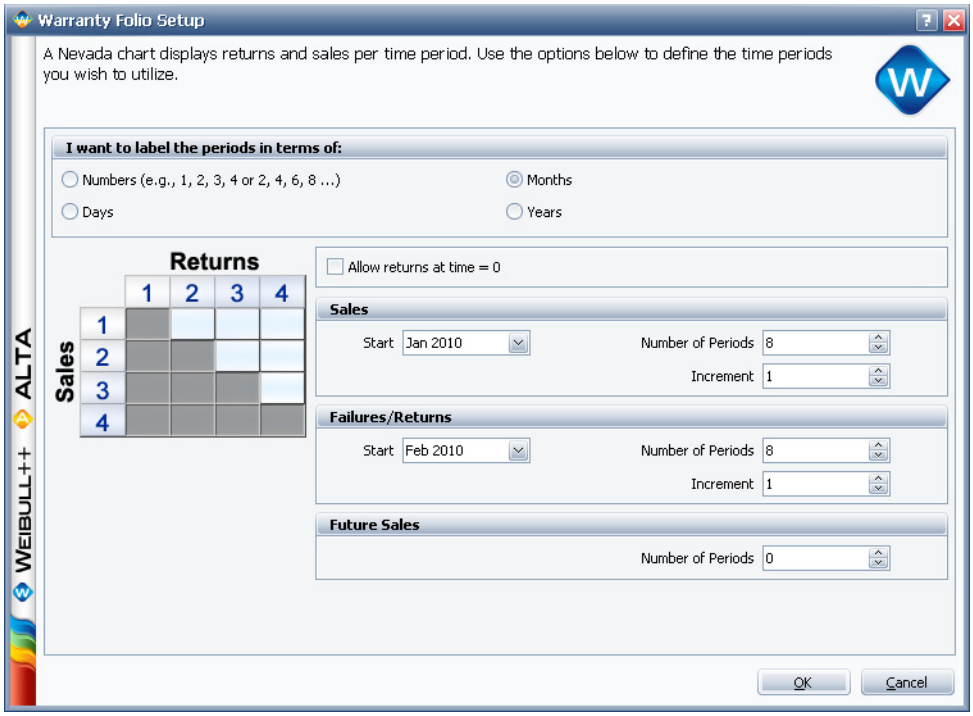
- Green indicates that the number of product returns is within the expected range.
- Yellow indicates that the number of product returns is between the caution and critical levels.
- Red indicates that the number of product returns is outside the expected range.

You can opt to change the colors in the Returns sheet by selecting the desired cells or row/column header and then choosing **Sheet Options > Format and View > Fill Color**.

To reset the colors back to their default settings, choose **Warranty > Tools > Restore Guide Colors** or click the **Restore Guide Colors** button on the SPC page of the control panel.

11.2.4 Example

An electronics manufacturer wants to study the expected number of returns for a product they created using materials obtained from two suppliers. The sales and returns data for the product are collected and entered into a Nevada chart warranty analysis folio, as shown next.

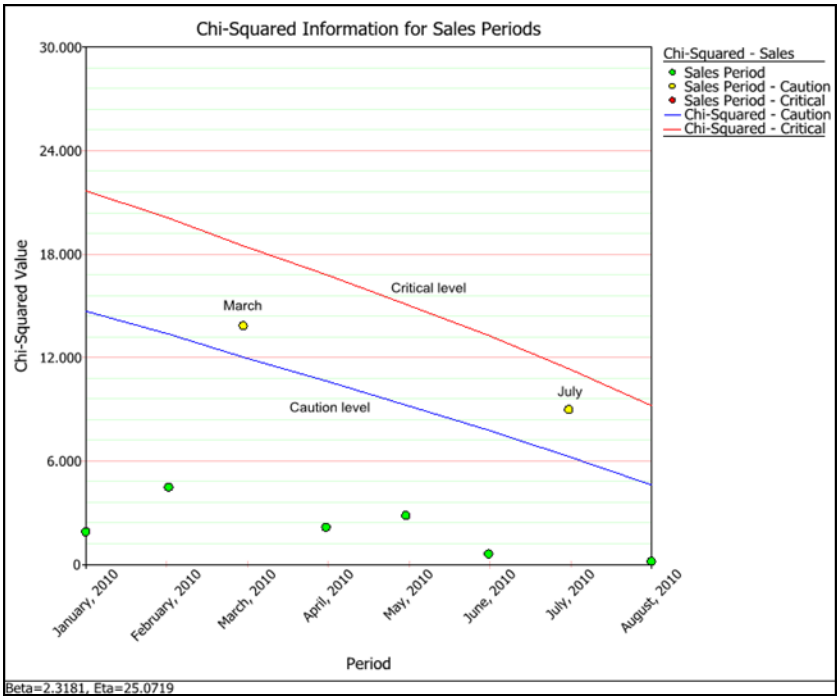


Period	Quantity In-Service	Subset ID
Jan 10	1150	Supplier 1
Feb 10	1100	Supplier 1
Mar 10	1200	Supplier 2
Apr 10	1155	Supplier 1
May 10	1255	Supplier 1
Jun 10	1150	Supplier 1
Jul 10	1105	Supplier 2
Aug 10	1110	Supplier 1

	Feb 10	Mar 10	Apr 10	May 10	Jun 10	Jul 10	Aug 10	Sep 10
Jan 10	2	4	5	7	12	13	16	17
Feb 10		3	4	5	3	8	11	14
Mar 10			2	3	5	7	23	13
Apr 10				2	3	4	6	7
May 10					2	3	3	4
Jun 10						2	3	3
Jul 10							2	12
Aug 10								2

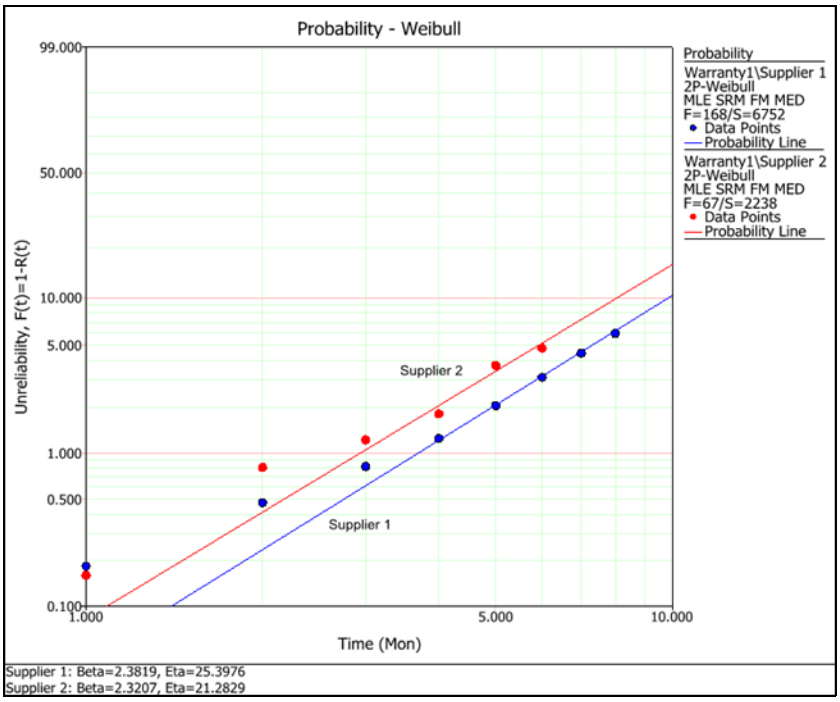
1. On the Main page of the control panel, choose the **2P-Weibull** distribution and the **MLE** parameter estimation method.
2. On the SPC page of the control panel, select the **Calculate chi-squared values** check box. Make sure that the critical value is **0.01** and that the caution value is **0.1**.
3. Select the **Color-code Returns sheet** check box and select the **By sales period** option.
4. Choose **Warranty > Analysis > Calculate** or click the **Calculate** icon on the Main page of the control panel.

   The parameters are estimated to be: beta = 2.3181 and eta = 25.0719. Notice that in the Returns sheet, the March and July sales periods are in yellow. This means that the number of returns for those sales periods is between the caution and critical levels.
5. Click the **Plot** icon. From the Plot Type drop-down list, choose **Chi-Squared - Sales**. As you can see, the data points for March and July are between the caution and critical levels.



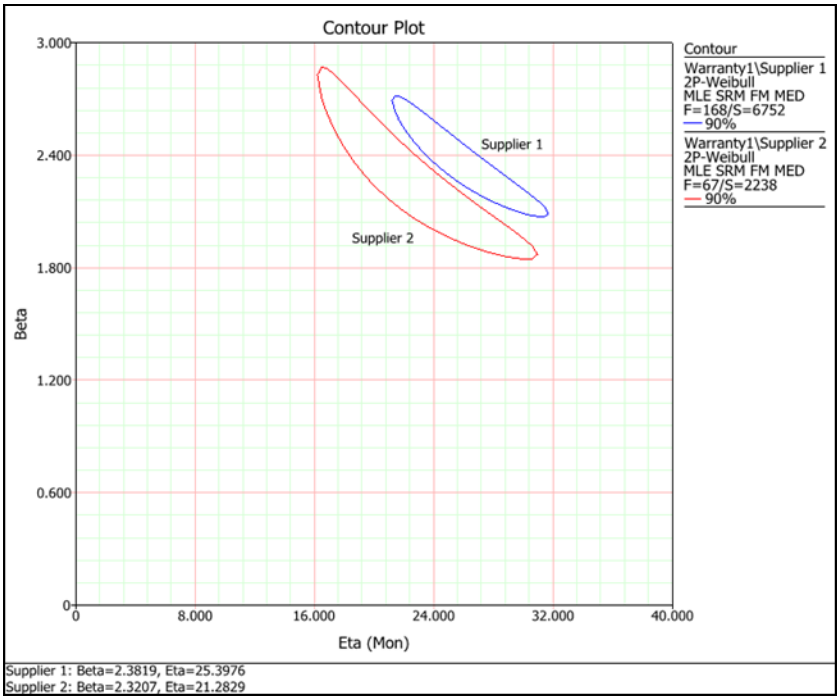
   This implies that the population is not homogeneous and that different subpopulations may exist in the data. One suspected reason for the deviation may be the type of material used by the suppliers. The manufacturer concludes that the data set needs to be analyzed based on their material supplier.
6. To do so, return to the Sales or Returns data sheet. On the Main page of the control panel, select the **Use Subsets** check box. Note that selecting this option will make the SPC feature unavailable. For each subset ID, select the **2P-Weibull** distribution and the **MLE** parameter estimation method.
7. Click the **Calculate** icon to recalculate the parameters. The results show that for Supplier 1, the parameters are beta = 2.3819 and eta = 25.3976; for Supplier 2, the parameters are beta = 2.3207 and eta = 21.2829.

8. Click the **Plot** icon. The following plot shows the probability of failure of the two subsets. As you can see, the material from Supplier 2 tends to have a higher probability of failure compared to the material from Supplier 1.



9. To determine the statistical difference between the two data sets, create a contour plot. To do this, click the **Plot Type** drop-down list on the control panel and choose **Contour Plot**. In the Contour Setup window, select to plot the contours at the 90% confidence level and click **OK**.

   The resulting plot is shown next. As you can see, the two subsets do not overlap at the 90% confidence level. This confirms that the two data sets are significantly different at the 90% confidence level.

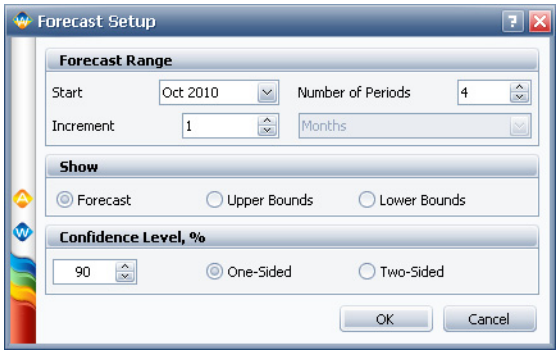


10. The next step in the analysis is to generate a warranty forecast. This will help the manufacturer obtain the expected number of product returns within a specified warranty period.

    Return to the Sales or Returns data sheet. Choose **Warranty > Tools > Forecast** or click the icon on the control panel.



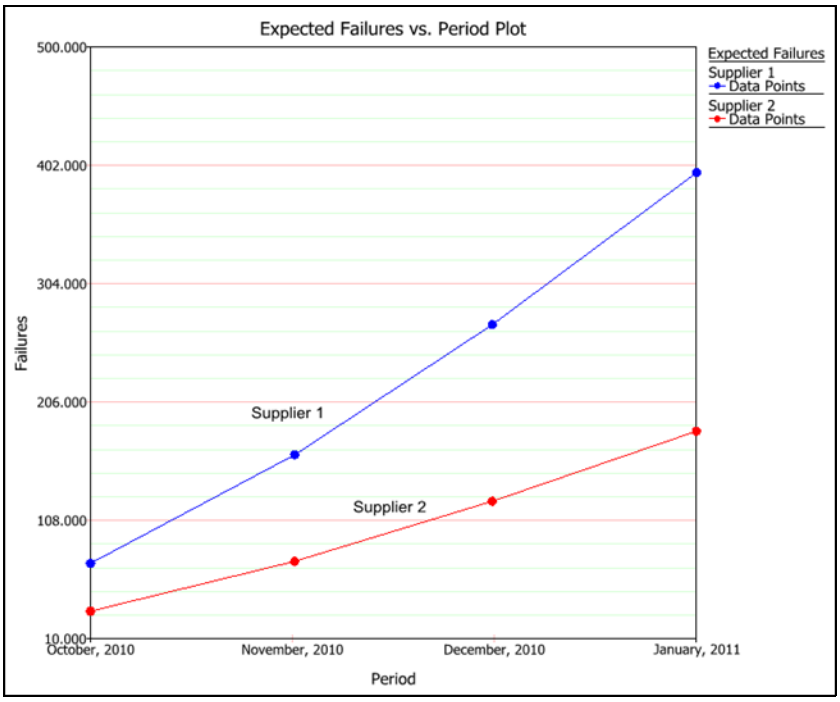
11. In the Forecast Setup window, set the forecast range to start in **October** and end after **4** forecast periods, as shown next. Set the Increment to **1** and click **OK**.



The following figure shows the Forecast sheet with the warranty period starting in October 2010 and ending in January 2011. As you can see, the highest number of product returns is expected to come from the batch that was in-service in March.

	Subset ID	Oct 10	Nov 10	Dec 10	Jan 11
Jan 10	Supplier 1	22	25	28	31
Feb 10	Supplier 1	18	21	24	27
Mar 10	Supplier 2	26	30	35	39
Apr 10	Supplier 1	13	16	19	22
May 10	Supplier 1	11	14	17	21
Jun 10	Supplier 1	7	10	13	16
Jul 10	Supplier 2	7	11	15	19
Aug 10	Supplier 1	2	4	7	9
Total		**105**	**131**	**158**	**184**

12. Click the **Plot** icon on the control panel of the Forecast sheet. On the Plot Type drop-down list, choose **Expected Failures**. The following plot shows the expected number of returned units over the warranty period.



Earlier, we found out that the material from Supplier 2 tends to have a higher probability of failure when compared to the material from Supplier 1. However, the current plot shows that Supplier 1 is expected to have a higher number of total returns within the specified warranty period. Looking back at the Sales sheet, we can see that the manufacturer sold three times more units made with materials from Supplier 1 compared to the number of sold units using materials from Supplier 2. Therefore, more units made with the materials from Supplier 1 will be returned each month due to the high sales volume of those units. This forecast analysis can help the manufacturer plan for warranty costs and service, as well as plan for the number of units to produce using materials from each supplier and decide whether it would be worthwhile to have Supplier 2 increase the reliability of its materials.

11.3 Times-to-Failure Format

If your warranty claims data are already in terms of failure/suspension times, you can use the times-to-failure format to forecast future warranty returns.

11.3.1 Entering Data

Time Units

In the times-to-failure format, you can use any time units appropriate for your data by clicking the **Change Units** icon on the Main page of the folio's control panel.



This opens the Change Units window, which gives you a selection of time units to use. The units that are available in the drop-down list are defined at the database level (*see page 39 in Chapter 2*). When you select new units, the appropriate columns in the data sheet will be automatically configured for the time units you selected, and the new unit will be displayed next to the results in the control panel.

Data Sheet Setup

The times-to-failure format consists of two data sheets: Data and Future Sales. Data entry is similar to entering data in a Weibull++ standard folio. The following picture shows an example of the data sheet. In this example, the values in the **State End Time** column are measured in days.

	Number in State	State F or S	State End Time (Day)	Subset ID
1	100	F	1	Product A
2	90	F	7	Product A
3	195	S	16	Product A
4	85	F	26	Product A
5	70	S	32	Product A
6	144	S	37	Product A

The **Subset ID** column is for logging any pertinent information or comments about the data. You can also use the Subset ID column to categorize subpopulations in your data set. You will be able to choose a distribution appropriate for each subset of data by selecting the **Use Subsets** option on the Main page of the control panel.

Note: The subset ID can be any text up to 30 characters, including spaces. For example, “A _ _ X” is not the same as “A _ X” where “ _ ” is used to designate a space.

The Future Sales sheet, shown next, gives you the option to enter projected sales figures. The information in this sheet is used in the forecast analysis. Enter the number of units to be sold in the **Quantity In-Service** column and enter the anticipated manufacturing/sales/shipment period in the **Time** column.

	Quantity In-Service	Time (Day)	Subset ID
1	150	40	Product A
2	150	50	Product A
3	150	60	Product A
4			
5			
6			

Note: For the column heading in this sheet, the general term *in-service* represents the closest estimate of the time at which each unit started operating in the field. For example, if you have access to both the manufacturing period and the shipping period for a group of parts, you may choose to perform the analysis based on the shipping period because it is a closer indicator of the time the product started operating in the field.

To perform the warranty analysis, go to the Main page of the warranty folio control panel and choose a failure distribution for the data set and a parameter estimation method (you can also choose the parameter estimation settings from the Analysis page of the control panel). Click **Calculate**.

Click anywhere within the **Analysis Summary** area of the control panel to open the Results window, which shows the failure/suspension data and the parameters of the life distribution model. From the Results window, you can edit, copy or print the results.

Because the data set is already in terms of failures and suspensions, you can immediately proceed to generating warranty forecasts. *See Section 11.6 on page 350.*

11.3.2 Example

A manufacturer sold two versions of a product. In addition, they plan to sell more of the product in the next few months. The objective is to analyze the data for both product versions and provide a return forecast for the next 10 months.

The following data sheet shows the data of the product. The return period (State End Time) is measured in months. The manufacturer logs the number of failures and suspensions for each product version at the end of each month.

	Number in State	State F or S	State End Time (Mon)	Subset ID
1	13	F	1	V1
2	1108	S	1	V1
3	17	F	2	V1
4	20	F	3	V1
5	1142	S	3	V1
6	20	F	4	V1
7	1243	S	4	V1
8	27	F	5	V1
9	1133	S	5	V1
10	24	F	6	V1
11	30	F	7	V1
12	1052	S	7	V1
13	17	F	8	V1
14	1074	S	8	V1
15	4	F	1	V2
16	15	F	2	V2
17	1091	S	2	V2
18	5	F	3	V2
19	7	F	4	V2
20	23	F	5	V2
21	13	F	6	V2
22	1147	S	6	V2

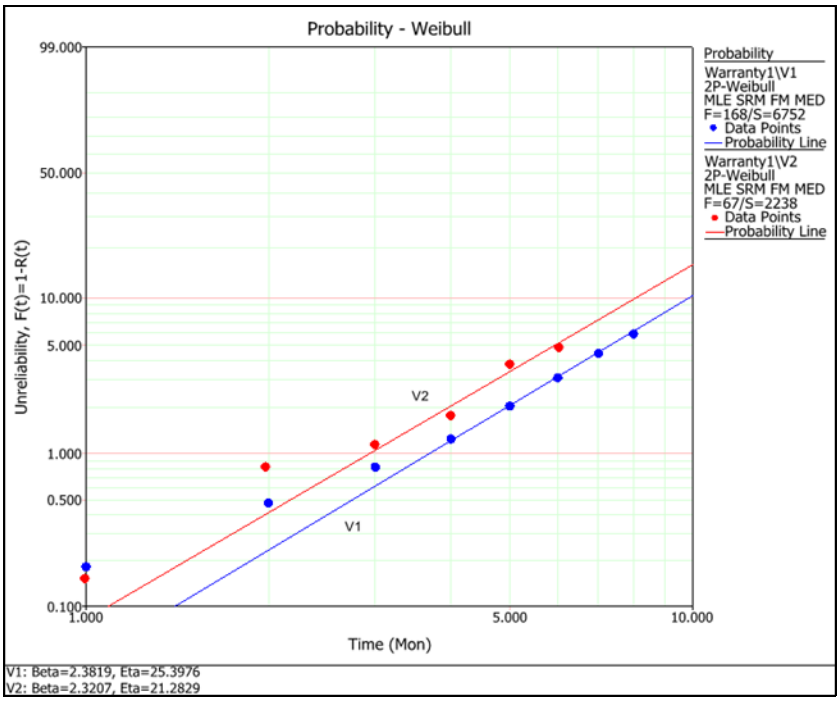
In the Future Sales data sheet, they enter the number of units they plan to ship to stores in the next 9th and 10th months, as shown next.

	Quantity In-Service	Time (Mon)	Subset ID
1	1500	9	V1
2	1500	10	V1
3	1500	9	V2
4	1500	10	V2

1. To analyze each product version, select the **Use Subsets** option on the Main page of the control panel. For both subset IDs, choose the **2P-Weibull** distribution and the **MLE** analysis method. Click **Calculate**.

   The parameters for V1 are estimated to be: beta = 2.3819 and eta = 25.3976; for V2, the parameters are estimated to be: beta = 2.3207 and eta = 21.2829.

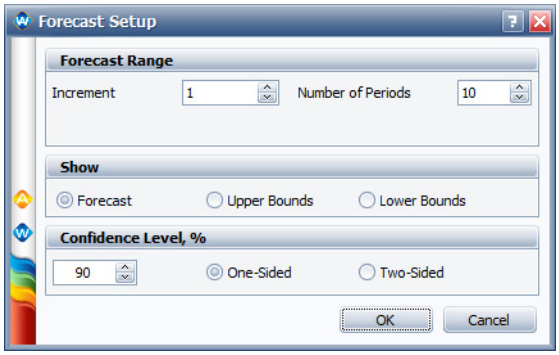
2. On the control panel, click the **Plot** icon. The following plot shows the probability of failure of the two product versions. As you can see, V2 has a higher probability of failure compared to V1.



3. Next, generate a warranty forecast to predict the expected number of product returns. To do this, return to the data sheet and then choose **Warranty > Tools > Forecast** or click the icon on the control panel.



4. In the Forecast Setup window, set the increment to **1** and set the forecast length to **10** periods, as shown next. Click **OK**.

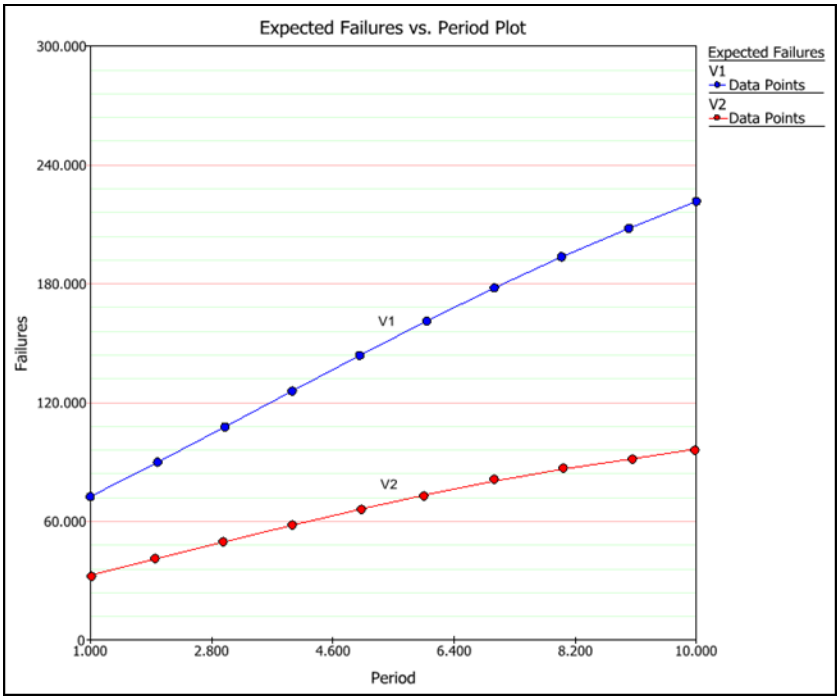


5. On the control panel of the Forecast sheet, select the **Show Subset ID** check box. The following figure shows the results (showing only the first 8 columns).

	Number in State	Current Age	Subset ID	Current Age+1	Current Age+2	Current Age+3	Current Age+4	Current Age+5
1	1108	1	V1	2	4	7	9	12
2	1142	3	V1	7	10	13	16	19
3	1243	4	V1	11	14	17	21	24
4	1133	5	V1	13	16	19	22	25
5	1052	7	V1	18	21	24	27	30
6	1074	8	V1	22	25	28	31	33
7	1091	2	V2	7	11	15	19	23
8	1147	6	V2	26	30	35	39	43
9	1500		V1					
10	1500		V1					
11	1500		V2					
12	1500		V2					
Total				**105**	**131**	**158**	**184**	**210**

The intervals in the warranty period are indicated by the Current Age plus the increment that was specified in the Forecast Setup window. In the sheet shown above, the first row in the data sheet tells us that 1108 (**Number in State**) units were operating in the field when the warranty period started and that these unit will have survived at least 1 month (**Current Age**) of operation. In the 2nd month of operation (**Current Age+1**), 2 units from that batch are expected to fail. After 4 months of operation (**Current Age+3**), 7 more units from the batch are expected to fail, and so on. Note that the last 4 rows of the Forecast sheet will show the expected number of returns from the future sales. (In this example, the returns are expected to start at Current Age+10.)

6. Click the **Plot** icon on the control panel of the Forecast sheet. In the Plot Type drop-down field, choose **Expected Failures**. The following plot shows the trend in the expected number of failures over the next 10 periods. This forecast analysis can help the manufacturer predict the failure behavior of each product version in the field, as well as plan for the warranty costs and service for each product version.



11.4 Dates of Failure Format

If you keep track of the exact calendar dates for sales and returns, you can use the dates of failure format to convert your warranty claims data into failure/suspension data.

11.4.1 Entering Data

Time Units

In the dates of failure format, the inputs and results are in terms of days, where 1 day = 24 hours; however, you have the option to obtain the results based on a different ratio, such as 1 day = 8 hours. The new ratio for the day unit will need to be defined at the database level. *See page 39 in Chapter 2*.

Data Sheet Setup

The dates of failure format consists of three data sheets: Sales, Returns and Future Sales. The figure shown next is an example of the Sales sheet.

	Quantity In-Service	Date In-Service	Subset ID
1	400	1/1/2010	Model A
2	500	1/31/2010	Model A
3	500	5/1/2010	Model A
4	600	5/31/2010	Model A
5	550	6/30/2010	Model A
6	600	7/30/2010	Model A

Note: For the column headings in this sheet, the general term *in-service* represents the closest estimate of the time at which each unit started operating in the field. For example, if you have access to both the manufacturing date and the shipping date for a group of parts, you may choose to perform the analysis based on the shipping date because it is a closer indicator of the time the product started operating in the field.

The Sales sheet contains the following information:

- The **Quantity In-Service** column indicates the number of units that are assumed to have started operating during each period.
- The **Date In-Service** column is for entering the calendar date on which the units are assumed to have started operating.
- The **Subset ID** column is for logging any pertinent information or comments about the data. You can also use the Subset ID column to categorize subpopulations in your data sets. You will be able to choose a distribution appropriate for each subset of data by selecting the **Use Subsets** option on the Main page of the control panel.

Note: The subset ID can be any text up to 30 characters, including spaces. For example, “A _ _ X” is not the same as “A _ X” where “ _ ” is used to designate a space.

The following is an example of the Returns sheet. You will need to enter the **Quantity Returned** and the **Date of Return**. You will also need to enter the **Date In-Service** so the application can calculate how long each unit was in service before it was returned. Each entry in the Date In-Service column in the Returns sheet must have a matching entry in the Date In-Service column in the Sales sheet.

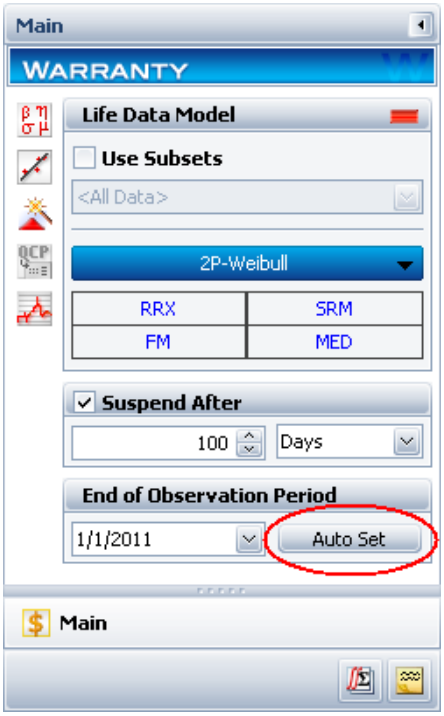
	Quantity Returned	Date of Return	Date In-Service	Subset ID
1	12	1/31/2010	1/1/2010	Model A
2	11	4/1/2010	1/31/2010	Model A
3	7	7/22/2010	5/1/2010	Model A
4	8	8/27/2010	5/31/2010	Model A
5	12	12/27/2010	5/31/2010	Model A
6	3	12/29/2010	6/30/2010	Model A

The Future Sales sheet, shown next, gives you the option to enter projected sales figures. The information in this sheet is used in the forecast analysis. Enter the number of units to be sold in the **Quantity In-Service** column and enter the anticipated manufacturing/sales/shipment date in the **Date In-Service** column.

	Quantity In-Service	Date In-Service	Subset ID
1	500	8/1/2010	Model A
2	500	9/1/2010	Model A
3	500	10/24/2010	Model A
4			
5			
6			

For this type of warranty analysis folio, an additional input is required in order for Weibull++ to convert the sales/return data to failures/suspensions. On the Main page of the control panel, specify the **End of Observation Period**. This date is typically set to the last day the warranty data was collected. It is used to determine the time in service for each unit that had not yet been returned by the end of the observation period. The time in service for a particular suspension unit is equal to the End of Observation Period date minus the Date In-Service of the unit.

To automatically set the end date to the last day the warranty data were collected, click **Auto Set**, as shown next. This may be the latest date that appears on the Sales sheet or the latest date that appears on the Returns sheet, whichever is more recent.



To perform the warranty analysis, go to the Main page of the warranty folio control panel and choose a failure distribution for the data set and a parameter estimation method (you can also choose the parameter estimation settings from the Analysis page of the control panel). Click **Calculate**. Weibull++ automatically converts the warranty data into failure/suspension times and also estimates the parameters of the chosen distribution.

To view the failures/suspension data, click anywhere within the **Analysis Summary** area of the control panel to open the Results window, which shows converted failure/suspension data and the parameters of the life distribution model. From the Results window, you can edit, copy or print the results.

11.4.2 Example

A manufacturer sold two versions of a product. In addition, a new version is planned for future release. The objective is to analyze the data for all versions and provide a return forecast for the next 20 months.

The following data sheet shows the sales data.

	Quantity In-Service	Date In-Service	Subset ID
1	200	1/1/2010	Design A
2	350	3/2/2010	Design A
3	450	4/1/2010	Design A
4	300	6/30/2010	Design A
5	200	8/29/2010	Design A
6	350	10/28/2010	Design A
7	400	1/1/2010	Design B
8	500	1/31/2010	Design B
9	500	5/1/2010	Design B
10	600	5/31/2010	Design B
11	550	6/30/2010	Design B
12	600	7/30/2010	Design B
13	800	9/28/2010	Design B

The following data sheet shows the returns data.

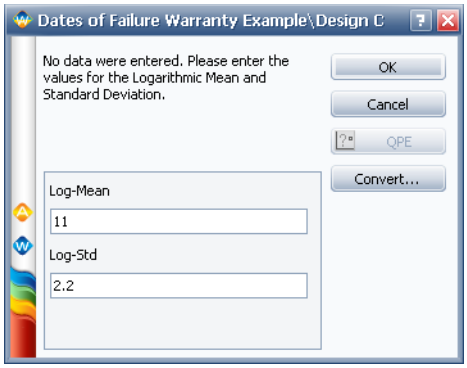
	Quantity Returned	Date of Return	Date In-Service	Subset ID
1	15	1/18/2010	1/1/2010	Design A
2	23	1/26/2010	1/1/2010	Design A
3	16	3/17/2010	1/1/2010	Design A
4	18	5/31/2010	3/2/2010	Design A
5	19	5/31/2010	3/2/2010	Design A
6	20	6/30/2010	3/2/2010	Design A
7	21	7/30/2010	4/1/2010	Design A
8	18	12/27/2010	6/30/2010	Design A
9	19	1/11/2011	8/29/2010	Design A
10	11	1/27/2011	10/28/2010	Design A
11	12	1/31/2010	1/1/2010	Design B
12	11	4/1/2010	1/31/2010	Design B
13	7	7/22/2010	5/1/2010	Design B
14	8	8/27/2010	5/31/2010	Design B
15	12	12/27/2010	5/31/2010	Design B
16	13	1/26/2011	6/30/2010	Design B
17	12	1/26/2011	7/30/2010	Design B
18	14	1/11/2011	9/28/2010	Design B

The future shipments are expected to be as follows.

	Quantity In-Service	Date In-Service	Subset ID
1	400	2/1/2011	Design B
2	500	4/1/2011	Design B
3	550	6/1/2011	Design B
4	2200	8/1/2011	Design C
5	2200	9/1/2011	Design C
6	2200	10/1/2011	Design C

1. The information in the Returns sheet is current as of January 31, 2011. Therefore, on the Main page of the control panel, set the **End of Observation Period** date to **1/31/2011** to indicate that the warranty data collection period ended on that date.
2. To analyze each product, select the **Use Subsets** option on the Main page of the control panel. Next, for all subset IDs, choose the **Lognormal** distribution and the **MLE** analysis method.
3. Click **Calculate**. You will be prompted to enter the parameters of Design C. Because Design C is a future product and no returns have been experienced yet, you will need to define its failure behavior. Information from the

company's test data indicate that Design C follows a lognormal distribution with a log-mean = 11 and log-std = 2.2 days. Enter these values in the input window, as shown next.



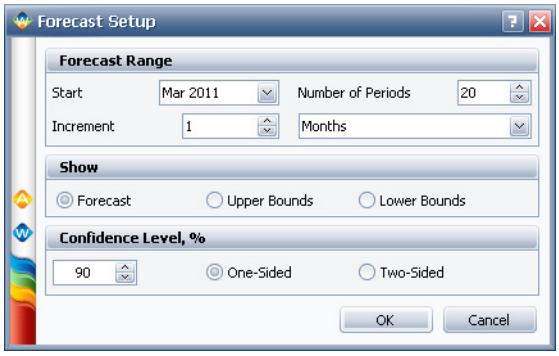
Note: Because the unit of time in the dates of failure format is in days, the parameters that are assigned to a data set must also be in terms of days.

Click **OK**. The results show that for Design A, the parameters are: log-mean = 8.4243 and log-std = 2.2987; for Design B, the parameters are: log-mean = 10.0772 and log-std = 2.2973.

4. Now that all the parameters of the subsets are available, you can proceed to obtaining a forecast of the expected number of failures. Choose **Warranty > Tools > Forecast** or click the icon on the control panel.



5. In the Forecast Setup window, enter a Start date of **3/1/2011** and obtain forecasts for **20** periods, as shown next. Set the increment to once per month, and then click **OK**.

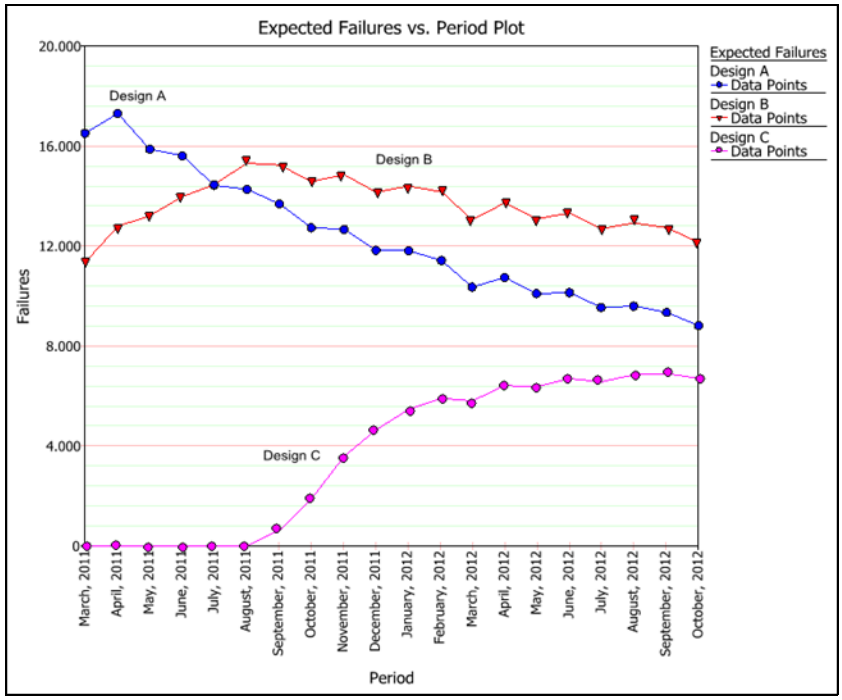


On the control panel of the Forecast sheet, select the **Show Subset ID** check box. The following figure shows the Forecast sheet with the first 4 months of expected product returns. The first row of the Forecast sheet tells us that

200 units of Design A were sold on 1/1/2010 and that 1 of those sold units is expected to fail between February 1, 2011 (the day after the End of Observation Period date) and March 1, 2011.

	Quantity In-Service	Date In-Service	Subset ID	Mar 01, 2011	Apr 01, 2011	May 01, 2011	Jun 01, 2011
1	200	1/1/2010	Design A	1	1	1	1
2	350	3/2/2010	Design A	3	3	2	2
3	450	4/1/2010	Design A	4	4	4	4
4	300	6/30/2010	Design A	3	3	3	3
5	200	8/29/2010	Design A	2	2	2	2
6	350	10/28/2010	Design A	4	4	4	4
7	400	1/1/2010	Design B	1	1	1	1
8	500	1/31/2010	Design B	1	1	1	1
9	500	5/1/2010	Design B	1	1	1	1
10	600	5/31/2010	Design B	2	2	2	2
11	550	6/30/2010	Design B	2	2	2	2
12	600	7/30/2010	Design B	2	2	2	2
13	800	9/28/2010	Design B	2	2	2	2
14	400	2/1/2011	Design B	1	1	1	1
15	500	4/1/2011	Design B			1	1
16	550	6/1/2011	Design B				
17	2200	8/1/2011	Design C				
18	2200	9/1/2011	Design C				
19	2200	10/1/2011	Design C				
Total				28	30	29	30

6. On the control panel of the Forecast sheet, click the **Plot** icon. In the **Plot Type** field, choose **Expected Failures**. The following plot shows the total number of expected failures from each design version over the next 20 months. This forecast analysis can help the manufacturer predict the failure behavior of each product version in the field, as well as plan for the warranty costs and service for each product version.



11.5 Usage Format

In the usage format, the analysis is based on the amount of usage the unit accumulated rather than the amount of time the unit was in the field. There are many applications where usage, not time, affects a product's reliability. In the automotive industry, for example, the majority of failure behavior is based on mileage rather than product age. This approach could also be used in other situations, such as the number of pages for a printer, number of cycles for a washing machine, etc.

This section describes how to work with this format, including:

- Entering data in the usage format - Section 11.5.1 (p. 338)
- Estimating usage - Section 11.5.2 (p. 341)
- Using the Interval Width Estimator - Section 11.5.3 (p. 346)
- An example of a usage format warranty analysis - Section 11.5.4 (p. 348)

11.5.1 Entering Data

Time Units

In the usage format, you can use any usage units (e.g., miles, cycles, etc.) appropriate for your data by clicking the **Change Units** icon on the Main page of the folio's control panel.



This opens the Change Units window, which gives you a selection of units to use. The units that are available in the drop-down list are defined at the database level (*see page 39 in Chapter 2*). Note that in the usage format, selecting a new unit only changes the name of the unit. The data set is not converted to the new unit. The appropriate columns in the data sheet will be automatically configured for the time units you selected, and the new unit will be displayed next to the results in the control panel.

Data Sheet Setup

The usage format consists of three data sheets: Sales, Returns and Future Sales. The figure shown next is an example of the Sales sheet.

	Quantity In-Service	Date In-Service	Subset ID
1	100	1/1/2010	Washing Machine A
2	150	2/1/2010	Washing Machine A
3	200	3/1/2010	Washing Machine A
4	250	4/1/2010	Washing Machine A
5	250	5/1/2010	Washing Machine A
6	250	6/1/2010	Washing Machine A

Note: For the column headings in this sheet, the general term *in-service* represents the closest estimate of the time at which each unit started operating in the field. For example, if you have access to both the manufacturing date and the shipping date for a group of parts, you may choose to perform the analysis based on the shipping date because it is a closer indicator of the time the product started operating in the field.

The Sales sheet contains the following information:

- The **Quantity In-Service** column is for entering the number of units that are assumed to have started operating during each period.
- The **Date In-Service** column is for entering the calendar date on which the units are assumed to have started operating.
- The **Subset ID** column is for logging any pertinent information or comments about the data. You can also use the Subset ID column to categorize subpopulations in your data sets. You will be able to choose a distribution appropriate for each subset of data by selecting the **Use Subsets** option on the Main page of the control panel.

Note: The subset ID can be any text up to 30 characters, including spaces. For example, "A _ _ X" is not the same as "A _ X" where " _ " is used to designate a space.

The following is an example of the Returns sheet, where the usage unit is in terms of cycles. You will need to enter the **Quantity Returned** and the **Usage at Return Date**. You will also need to enter the **Date-In-Service** so the application can calculate how long each unit was in service before it was returned. Each entry in the Date In-Service column in the Returns sheet must have a matching entry in the Date In-Service column in the Sales sheet.

	Quantity Returned	Cycle Usage at Return Date	Date In-Service	Subset ID
1	9	678	1/1/2010	Washing Machine A
2	6	1842	1/1/2010	Washing Machine A
3	5	1623	2/1/2010	Washing Machine A
4	17	2010	3/1/2010	Washing Machine A
5	12	1324	4/1/2010	Washing Machine A
6	3	984	5/1/2010	Washing Machine A

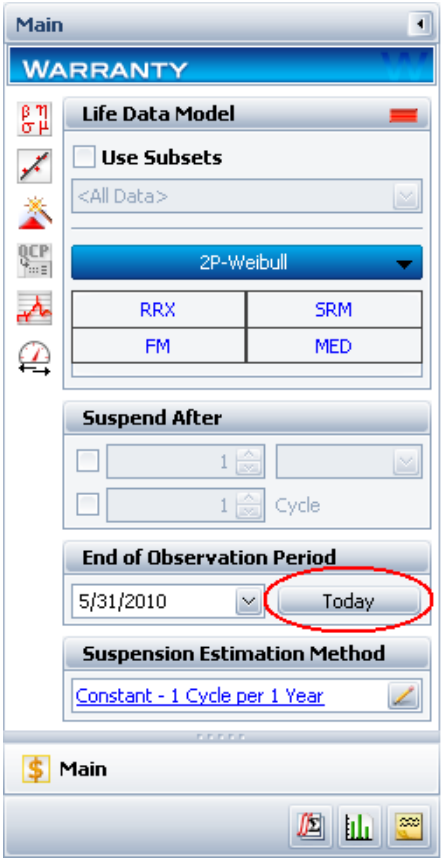
The Future Sales sheet, shown next, gives you the option to enter projected sales figures. The information in this sheet is used in the forecast analysis. Enter the number of units to be sold in the **Quantity In-Service** column and enter the anticipated manufacturing/sales/shipment date in the **Date In-Service** column.

	Quantity In-Service	Date In-Service	Subset ID
1	200	8/1/2010	Washing Machine A
2	200	9/1/2010	Washing Machine A
3	200	10/1/2010	Washing Machine A
4			
5			
6			

For this type of warranty analysis folio, two additional inputs are required in order for Weibull++ to convert the sales/returns data to failures/suspensions:

- On the Main page of the control panel, specify the **End of Observation Period** date. This date is typically set to the last day the warranty data was collected. It is used to determine the time in service for each unit that had not

yet been returned by the end of the observation period (i.e., the suspensions). Click **Today** if you wish to set the end date to the current date, as shown next.



- The next step is to specify how Weibull++ will estimate the amount of usage for units still in the field (suspensions). To view or change the settings, click the link in the **Suspension Estimation Method** area of the Main page or click the **Suspensions** page icon at the bottom of the control panel.



To learn how to specify the suspension estimation method, Section 11.5.2 on page 341.

After you have entered the warranty data and specified the suspension estimation method, perform the warranty analysis by going to the Main page of the warranty folio control panel and choosing a failure distribution for the data set and a parameter estimation method (you can also choose the parameter estimation settings from the Analysis page of the control panel). Click **Calculate**. Weibull++ automatically converts the warranty data into failure/suspension times and also estimates the parameters of the chosen distribution.

To view the failures/suspension data, click anywhere within the **Analysis Summary** area of the control panel to open the Results window, which shows the converted failure/suspension data and the parameters of the life distribution model. From the Results window, you can edit, copy or print the results.

11.5.2 Estimating Usage

Like any other warranty analysis folio in Weibull++, the usage format must convert the sales/returns data to failures/suspensions that can be analyzed with life data analysis techniques. Since the returns information is entered in terms of accumulated usage (e.g., miles, cycles, etc.) rather than time, the failure "times" will be the usage values recorded when the units were returned. However, an additional step is required to estimate the amount of usage accumulated by the units still operating in the field at the end of the observation period (i.e., the suspensions). Two methods are available for you to provide the information required for these calculations:

1. You can define the average amount of usage that any given unit will typically accumulate over a specified period of time (e.g., 500 miles per month, 1,000 cycles per year, etc.).
2. You can define a statistical distribution that reflects the variation in usage patterns among different customers. The information for the usage distribution could come from customer surveys, repair records, built-in devices that record usage data, etc.

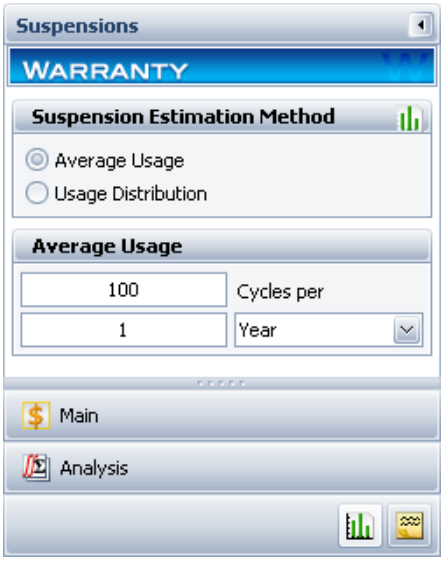
To select a method, click the link in the **Suspension Estimation Method** area of the Main page or click the **Suspensions** page icon at the bottom of the control panel, as shown next.



Both methods are described next.

Performing Estimates Based on Average Usage

- When you choose **Average Usage** on the Suspensions page of the control panel, you will be required to enter the average amount of usage that any given unit typically accumulates over a specified period of time (in days, months or years). As an alternative, you could have the software calculate the average usage based on the time unit conversion factors (*see page 39 in Chapter 2*). For example, the settings shown in the following picture might indicate that the average amount of usage typically accumulated by a washing machine during 1 year of operation is 100 cycles.



- When you click the **Calculate** icon on the Main page of the control panel, Weibull++ will perform the following calculations for each group of units that have the same date in-service:

  1. Calculate the number of units from the sales group that were still operating at the end of the observation period:
     - Number of Units Sold – Number of Units Returned = Number of Suspensions
  2. Calculate the amount of time (in days) that those units had been in service by the end of the observation period:
     - End of Observation Period date – Date In-Service = Days in Service
  3. If necessary, convert the average usage to the daily rate for calculation purposes (e.g., if the average usage is 10,000 miles per year then the daily rate would be approximately 27.3973 miles per day).
  4. Estimate the amount of usage the suspension units may have accumulated by the end of the observation period:
     - Days in Service * Average Usage per Day = Estimated Usage at Time of Suspension

For example, suppose that the Sales sheet records that 50 cars entered service on January 1st while the Returns sheet records the mileage for the 20 units from this sales group that have been returned. If you enter July 1st for the End of Observation Period date and 1000 miles/per year for the Average Usage, the software will calculate the suspensions

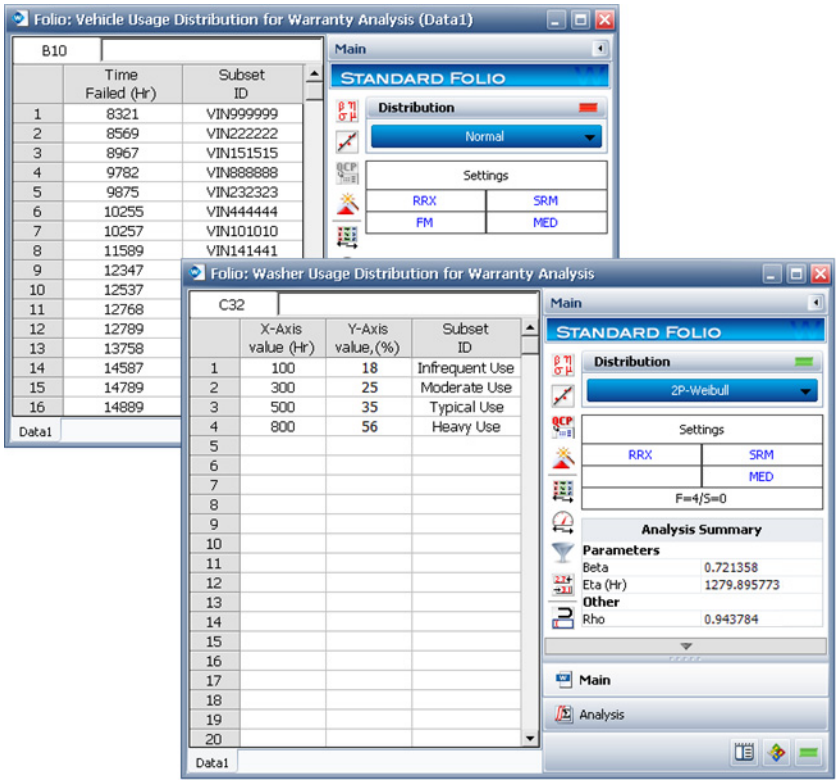
from the January 1st sales group as follows (where the daily rate has been rounded to 4 decimal places for the sake of simplicity, and therefore the estimated usage will not exactly match the value calculated by the software):

- 50 Units Sold – 20 Units Returned = 30 Suspensions
- July 1 – January 1 = 181 Days in Service
- 181 Days in Service * 2.7397 Miles per Day = 495.8857 Miles approximately

When you click the **Show Analysis Summary** button on the Main page of the control panel, the life data analysis data set will include one row with 30 suspensions at "End Time" = 495.8857.

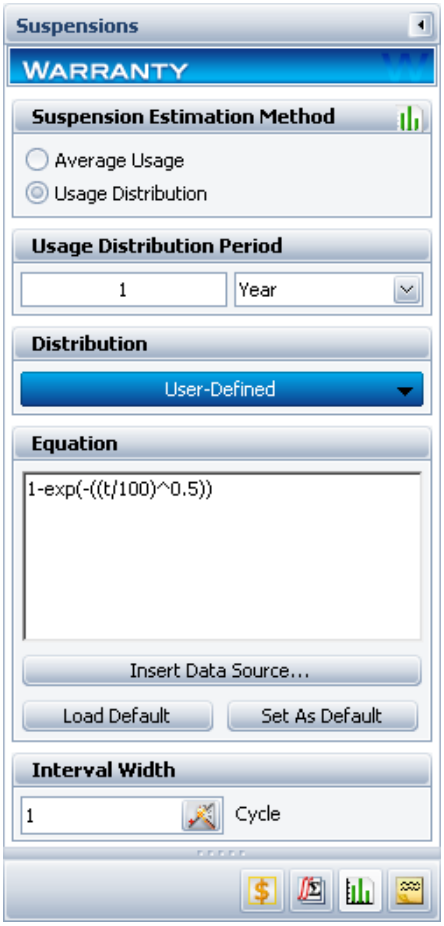
Performing Estimates Based on a Usage Distribution

If you have more specific information about variations in usage patterns between different customers, you may prefer to describe the estimated usage in terms of a statistical distribution instead of an average. There are several different ways that you could obtain a usage distribution. For example, if you have information about the amount of usage accumulated by many different users over a specified period of time, you could analyze the data in a Weibull++ standard folio with a time-to-failure data sheet. Alternatively, if you can define typical usage patterns and estimate the percentage of users who are likely to belong to each group (e.g., 18% of customers use 100 cycles or less, 25% of customers use 300 cycles or less, and so on), you could analyze the data in a Weibull++ standard folio with a free-form (probit) data sheet, as shown next.



- When you choose **Usage Distribution** on the Suspensions page of the control panel, you will be required to define a distribution that reflects the usage amounts that different customers may accumulate during a specified period of time (called the *Usage Distribution Period*).
  - If you choose Weibull, Normal, Lognormal, Exponential, Generalized Gamma, Gamma, Logistic, Loglogistic or Gumbel from the drop-down list, you can simply enter the appropriate parameter(s) for the selected distribution.

- If you choose Mixed Weibull, you can use the **Subpopulation** drop-down list to define parameters and the portion (percentage of the population) represented by each subpopulation.
- If you choose the User-Defined distribution, you can manually enter any equation that describes how customers use your product. You will need to use the variable *t* to represent time or usage, as in the example shown next.



You can also use distributions calculated in standard folios from your current project as data sources for your equation. To insert a data source, click the **Insert Data Source** button and select a calculated standard folio. The software will enter that source into the Equation area. You can insert multiple data sources and combine them (with operators such as + and *) to form new distributions.

To save the equation, click the **Set as Default** button. To automatically re-enter the saved equation in the Equations area, click **Load Default**.

- You will also be required to enter the **Interval Width**. This value is used to divide the usage distribution *pdf* into segments so the software can obtain the probability that any given unit will have accumulated usage at the rate represented by that segment. For example, if the usage distribution represents the number of pages printed per year and you enter 500 for the interval, the software will break the distribution into segments of 0 to 500 pages per year, 501 to 1000 pages per year, 1001 to 1500 pages per year, and so on.

  The appropriate value for this field will depend upon your knowledge of typical product usage levels. The interval width should be selected such that when the warranty data are converted into failure/suspension times, the resulting times-to-suspension will neither be too close together (too few suspension units per interval) or too far apart (too many suspension units per interval). If you find that the failure/suspension data set is not acceptable, you can adjust the interval width and perform the calculation again. To help eliminate some of the guesswork, you

can also use the Interval Width Estimator, which provides an approximation of the interval width based on the number of intervals and suspensions you specify. You can access the tool by clicking the icon in the **Interval Width** area of the control panel.

- When you click the **Calculate** icon on the Main page of the control panel, Weibull++ will perform the following calculations for each group of units that have the same date in-service:

  1. Calculate the number of units from the sales group that were still operating at the end of the observation period:
     - Number of Units Sold – Number of Units Returned = Number of Suspensions
  2. Calculate the amount of time (in days) that those units had been in service by the end of the observation period:
     - End of Observation Period date – Date In-Service = Days in Service
  3. Use the Interval Width to split the usage distribution into segments and calculate the probability that any given unit will fall into each segment:

$$F(x_i) = Q(x_i) - Q(x_{i-1})$$

     where $Q(\ )$ is the cumulative distribution function (*cdf*) of the usage distribution and x represents the intervals used in apportioning the suspensions.
  4. For each segment, calculate the number of suspensions that are expected to have accumulated usage at the rate represented by that segment. (Note that the software applies a correction factor in order to get a whole number of units for each interval.)
     - Number of Suspensions x Percentage for the Segment
  5. If necessary, convert the usage value from the end of each segment to the daily rate for calculation purposes (e.g., if the segment represents users who typically print 201 - 400 pages per month, then the daily rate would be 400/30 or approximately 13.333 pages per day).
  6. For each segment, estimate the amount of usage the suspensions may have accumulated by the end of the observation period:
     - Days in Service * Usage per Day = Estimated Usage at Time of Suspension

For example, suppose that the Sales sheet records that 100 printers entered service on January 1st while the Returns sheet records the number of pages printed for the 20 units from this sales group that have been returned. If you enter December 31st for the End of Observation Period date and the usage distribution indicates that 25% of users are likely to print 50 -100 pages per month, the software will calculate the usage for this segment of the January 1st sales group as follows:

- 100 Units Sold – 20 Units Returned = 80 Suspensions
- December 31st – January 1st = 365 Days in Service
- 80 Suspensions x 25% = 20 Units printing 50 - 100 pages per month (or ~3.33 pages per day)
- 365 Days in Service * 3.33 Pages per Day = 1215.45 Pages approximately

When you click the **Show Analysis Summary** button on the Main page of the control panel, the life data analysis data set will include one row with 20 suspensions at "End Time" = 1215.45. There will be additional rows for the remaining 60 suspensions from the January 1st sales group, which will reflect the different usage amounts calculated based on the other segments of the usage distribution.

11.5.3 Interval Width Estimator

The usage format of the warranty analysis folio converts the amount of usage accumulated by a fielded product into failures/suspensions data so you can perform life data analysis and/or forecast future warranty returns. When you use a statistical distribution to take into account the variations in the amount of product usage, the conversion process becomes more involved. In order to convert the usage data to failures/suspensions data, the software must have a way to estimate the amount of usage accumulated by units still operating in the field at the end of the observation period (i.e., suspensions).

The software uses the concept of *interval width* to estimate the amount of usage accumulated by the suspensions. The interval width is a value that is used to divide the usage distribution into usage rate intervals. For example, if the usage distribution represents the number of pages printed per year, and you specify 500 for the interval width, the software will break the distribution into usage rate intervals of 0 to 500 pages per year, 501 to 1000 pages per year, 1001 to 1500 pages per year, and so on. The software uses the intervals to obtain from the distribution the percentage of customers who use the product at each of the usage rates. The percentages are then used to obtain the probability that any given suspended unit will have accumulated usage at the rate represented by those intervals.

The Interval Width Estimator is a tool that helps you to determine an appropriate interval width by providing a *pdf* graph of the usage distribution and showing you how well the usage values assigned to the suspended units track the *pdf*. Note that this tool only serves as a guide and that the appropriate interval width value will depend upon your own knowledge of typical product usage levels.

Estimating the Interval Width

To use the Interval Width Estimator, you must first define the usage distribution on the Suspensions page of the control panel. Next, open the tool by clicking the icon in the **Interval Width** area on the Suspensions page (note that this area will be displayed only when the warranty analysis folio has been configured for the usage format and only if you have selected to define the usage with a distribution).

The interval width is estimated based on the following inputs:

- In the **End Usage** field, enter the length of the observation period. This also sets the maximum value for the x-axis.
- In the **Number of Intervals** field, set the maximum number of intervals you want to divide the usage distribution into.
- In the **Number of Suspensions** field, set the total number of suspensions that are expected to continue operating in the field at the end of the observation period.

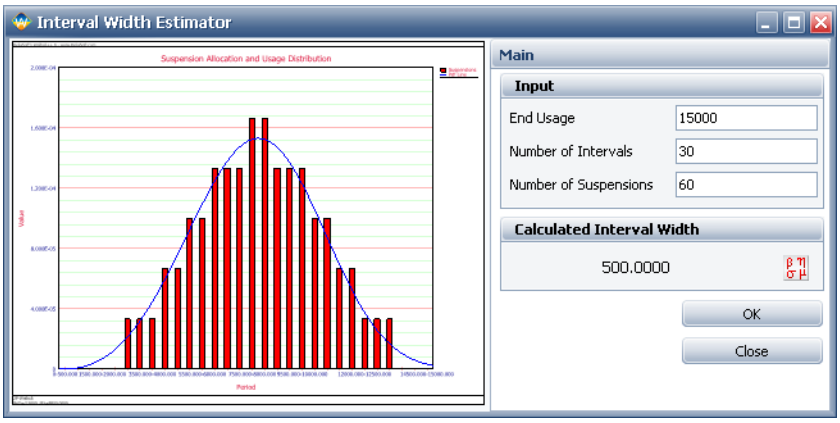
Click the **Calculate** icon to estimate the interval width.



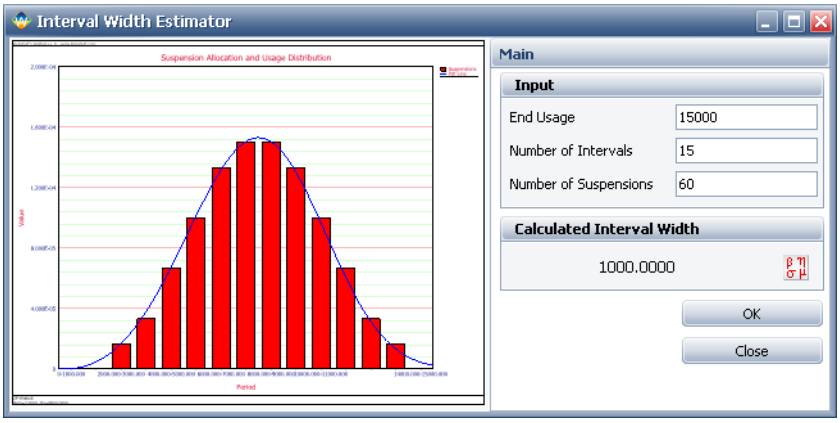
The resulting rate intervals will be plotted in a bar chart and superimposed over the *pdf* of the usage distribution. To show the estimated number of suspensions that accumulate usage at each rate represented by each interval, move the cursor over the bars depicting the intervals.

You can adjust the number of intervals and perform the calculation again to obtain an interval width value that provides a good number of intervals for the usage distribution. The following plots show some examples of how different interval widths may fit the same usage distribution.

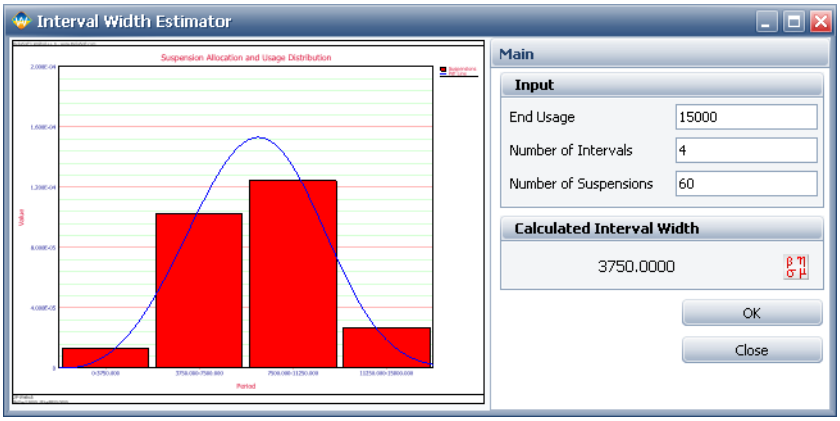
The following picture shows a plot with a narrow interval width, resulting in a *pdf* that is divided into several small segments. This causes the allocated suspensions to track the *pdf* poorly. When the warranty data are converted to failures/suspension data, there may be insufficient accuracy of the usage values of the suspensions (i.e., several intervals with few suspended units in each interval).



The next picture shows a plot with a wider interval width, resulting in a *pdf* that is divided into larger segments. The allocated suspensions track the usage *pdf* well. Therefore, this interval width would be appropriate for the data set.



The following picture shows a plot with a very wide interval width. Again, this causes the allocated suspensions to track the *pdf* poorly. When the warranty data are converted to failures/suspension data, there may be insufficient granularity of the suspensions (i.e., few intervals with a high number of suspended units in each interval).



11.5.4 Example

An automotive manufacturer collected warranty returns and sales data for a vehicle part. The objective is to obtain the percentage of sold vehicles that will need repair under warranty if this particular part has a warranty coverage of 36,000 miles.

The manufacturer has been documenting the mileage accumulation per year for this type of product across the customer base in comparable regions for many years. Based on the data, it was determined that the yearly usage follows a lognormal distribution with a mean = 9.38 and a standard deviation = 0.085. For the interval width, the manufacturer decided to specify an interval of 1,000 miles.

The following is a table of the warranty information for a single year. The information is current as of December 1, 2010.

Sales Month	Units Sold	Quantity Returned	Usage at Time of Return (miles)
12/1/2009	9	1	9072
1/1/2010	13	1	9743
2/1/2010	15	1	6857
3/1/2010	20	1	7651
4/1/2010	15	0	-
5/1/2010	25	1 1 1 1	5083 5990 7432 8739
6/1/2010	19	1	3158
7/1/2010	16	1	1136
8/1/2010	20	1	4646
9/1/2010	19	1	3965
10/1/2010	25	1	3117
11/12010	30	1	3250

1. The first step is to enter the sales and return data in the usage warranty analysis folio, as follows.

	Quantity In-Service	Date In-Service	Subset ID
1	9	12/1/2009	Part A
2	13	1/1/2010	Part A
3	15	2/1/2010	Part A
4	20	3/1/2010	Part A
5	15	4/1/2010	Part A
6	25	5/1/2010	Part A
7	19	6/1/2010	Part A
8	16	7/1/2010	Part A
9	20	8/1/2010	Part A
10	19	9/1/2010	Part A
11	25	10/1/2010	Part A
12	30	11/1/2010	Part A

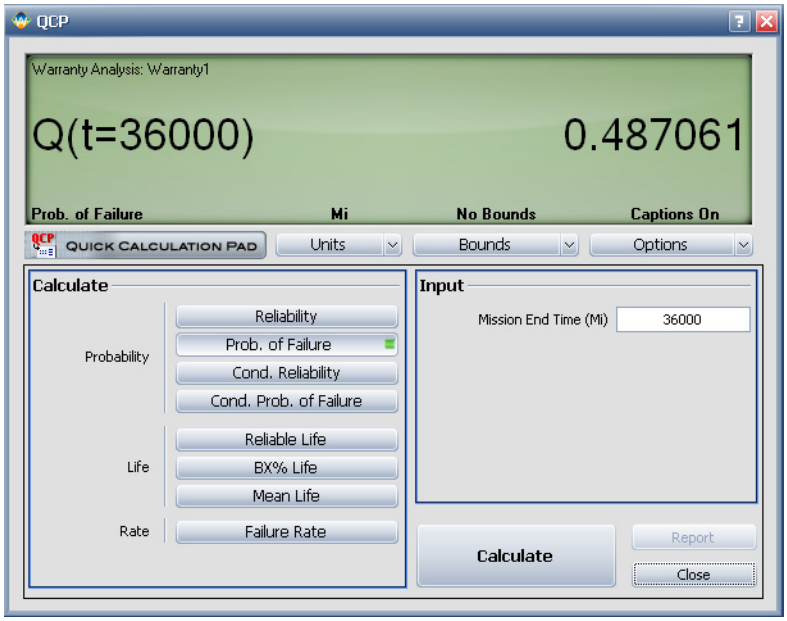
	Quantity Returned	Usage at Return Date	Date In-Service	Subset ID
1	1	9072	12/1/2009	Part A
2	1	9743	1/1/2010	Part A
3	1	6857	2/1/2010	Part A
4	1	7651	3/1/2010	Part A
5	1	5083	5/1/2010	Part A
6	1	5990	5/1/2010	Part A
7	1	7432	5/1/2010	Part A
8	1	8739	5/1/2010	Part A
9	1	3158	6/1/2010	Part A
10	1	1136	7/1/2010	Part A
11	1	4646	8/1/2010	Part A
12	1	3965	9/1/2010	Part A
13	1	3117	10/1/2010	Part A
14	1	3250	11/1/2010	Part A

2. On the Main page of the control panel, set the **End of Observation Period** date to **12/1/2010**.
3. The next step is to set up the usage distribution. On the Suspensions page of the control panel, make the following inputs/selections:
   - Suspension Estimation Method: **Usage Distribution**
   - Usage Distribution Period: **1 year**
   - Distribution: **Lognormal**
   - Parameters: Log-Mean = **9.38**, Log-Std = **0.085**
   - Interval Width: **1000**
4. Next, choose a failure distribution for the vehicle part. On the Main page of the control panel, choose the **Lognormal** distribution and then select **MLE** for the parameter estimation method.
5. Calculate the parameters by choosing **Warranty > Analysis > Calculate** or click the icon on the Main page of the control panel.



   The parameters of the failure distribution are estimated to be log-mean = 10.5281 and log-std = 1.1352.
6. Calculate the probability that the vehicle part will fail under warranty. To do this, click the **QCP** icon on the control panel. In the QCP window, choose to calculate the **Probability of Failure**. Select **Miles** for the time units and enter **36,000** for the mission end time.

Click **Calculate** to obtain the result, as shown next.



The probability of failure when under warranty is estimated to be 48.71%. In other words, an estimated 48.71% of sold vehicles will return for repair under the current warranty policy. Based on this information, the manufacturer may wish to modify the warranty policy and/or work on improving the reliability of the part.

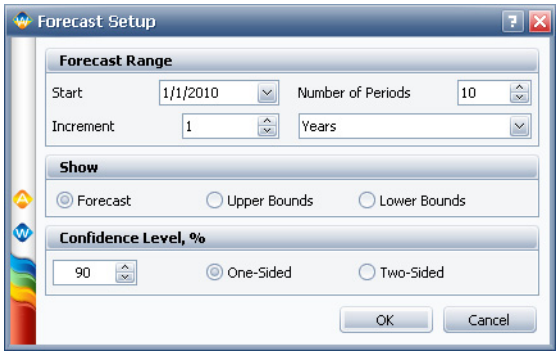
11.6 Forecasting Warranty Returns

Once the warranty data set has been converted to failure/suspension times, the information can be used to predict the number of failures, or warranty returns, in subsequent time periods. Accurate predictions about the quantity of products that will be returned under warranty can provide huge benefits to manufacturing organizations, such as the ability to anticipate customer support needs and correct serious product quality problems in the field before other problems occur.

To generate a forecast, choose **Warranty > Tools > Forecast** or click the icon on the control panel.



The following picture shows the Forecast Setup window.



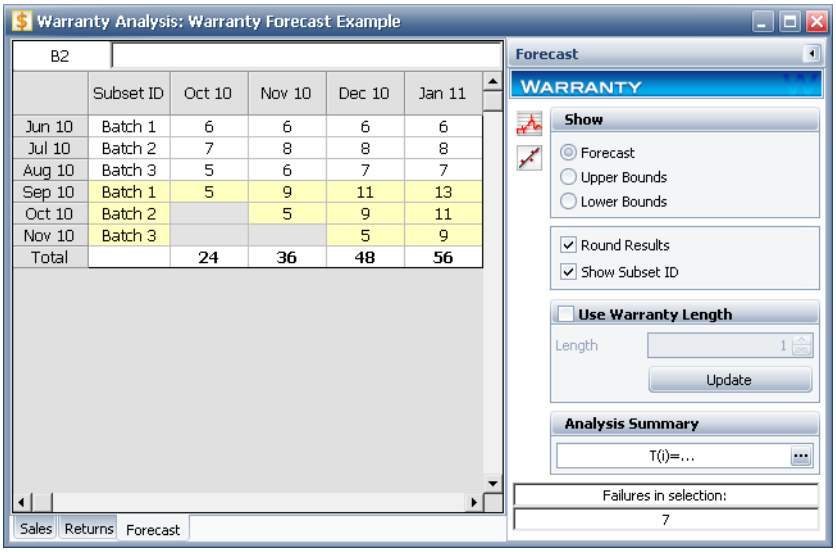
In the Forecast Range area, specify the **Start** date for the forecast and the **Number of Periods** (number of forecasts). You can also specify the length of the forecast period in the **Increment** field. For example, in the figure shown above,

the time unit is in terms of years; therefore, an increment of 1 would mean that 1 period is equivalent to 1 year, while an increment of 2 would mean that 1 period is equivalent to 2 years. Note the following:

- In the Nevada chart format, the forecast period uses the same time units used in the data sheets.
- In the times-to-failure format, the unit of time is not defined; however, it is consistent with the unit of time used in the data sheet.
- In the dates of failure format and the usage format, you can specify the forecast period to be in days, months or years.

You can select whether you want the forecast sheet to display the **Forecast** (i.e., projected returns) or the **Upper Bounds** or **Lower Bounds** of the forecast. You can set the **Confidence Level** for the forecast calculation and specify whether you want to calculate **One-Sided** or **Two-Sided** confidence bounds.

Once you have entered the settings for the forecast, click **OK** to generate a Forecast sheet. The example shown next is a Forecast sheet for the Nevada chart format. The column headers represent the warranty periods and the row headers represent the sales periods. Forecasts for the future sales periods are in the yellow cells.



The Forecast sheet includes a control panel that contains the following settings:

- In the **Show** area, you can select to view the results as the Forecast, as the Upper Bounds of the forecast or as the Lower Bounds of the forecast.
- **Round Results** rounds the estimates to the nearest integer. Note that the column totals are calculated based on the non-rounded results.
- **Show Subset ID** adds a column in the Forecast sheet that displays the corresponding subset ID of the data in the Forecast sheet.
- The **Use Warranty Length** check box allows you to limit the length of the warranty period. By default, the software assumes an infinite warranty period. Limiting the warranty period allows you to disregard the number of returns that are outside the warranty period. For example, a warranty length of 12 months would mean that any units that fail after 12 months of operation are out of warranty, and are therefore not counted in the total number of returns. This is useful when planning for warranty costs, spare parts and other fulfillment needs that fall within a specified period.

  Enter the length of the warranty period in the **Length** field and click **Update**. This option uses the same units you selected for the period increments in the Forecast Setup window.

11.7 Warranty Analysis Plots

You can plot the data in the warranty analysis folio either with or without generating a forecast. The features on the plot sheet are similar to the options available for all other Weibull++ plots. Features that are not applicable to warranty analysis will be hidden or disabled.

In addition to the standard plots in Weibull++, the following plot types are available for warranty analysis:

- **Expected Failures** plot shows the expected number of failures over a period of time. This plot is available for all warranty data formats. Four additional options in the control panel are associated with this plot:
  - **Cumulative Failures** shows the cumulative percentage of failures for each time period. For example, consider 10 units that had failed within a period of 6 months. If 2 units failed during the 1st month, then the percentage of failures is 2/10 = 20%. If 3 more units failed during the 2nd month, then the cumulative percentage of failures is (2+3)/10 = 50%, and so on.
  - **Failures in Percents** shows the failures as percentages of the total number of units originally in-service. Otherwise, the failures will be plotted as raw numbers.
  - **Show Confidence Bounds** shows the upper and lower confidence bounds on the plot.
  - **Include Total Data** is available only when the "Use Subsets" check box was selected on the Main page of the control panel. It shows a plot of the entire data set, along with the selected subsets. This allows you to view the overall trend of the entire data set in addition to the trends exhibited by each subset of data.
- **Chi-Squared Sales** and **Chi-Squared Returns** plot types are available only when using the statistical process control (SPC) feature of the Nevada chart format. These plots give a visual indication of whether the expected number of warranty returns for a certain period is within or outside an expected range.

11.8 Manage Warranty Units in Weibull++

The Manage Warranty Units window allows authorized users to specify which of the configurable "repository time units" are equivalent to the built-in (not configurable) "warranty time units" used in some warranty folio formats.

- To open the Manage Warranty Units window, choose **File > Manage Repository > Manage Warranty Units**.
- To view/modify the current "repository time units" choose **File > Manage Repository > Manage Units**.

The affected warranty folio formats include:

- The **Nevada chart format** when the periods are labeled in terms of days, months or years.
- The **dates of failure format** where the software calculates the number of days between the specified calendar dates.
- The **usage format** where the software can calculate the amount of usage typically accumulated per day, month or year in order to estimate the usage for units still operating in the field.

It is not applicable for the **Nevada chart format** when the periods are labeled with numbers or the **times-to-failure format**. In those cases, you will be prompted to select one of the "repository time units" for all time inputs and calculated results in the folio.

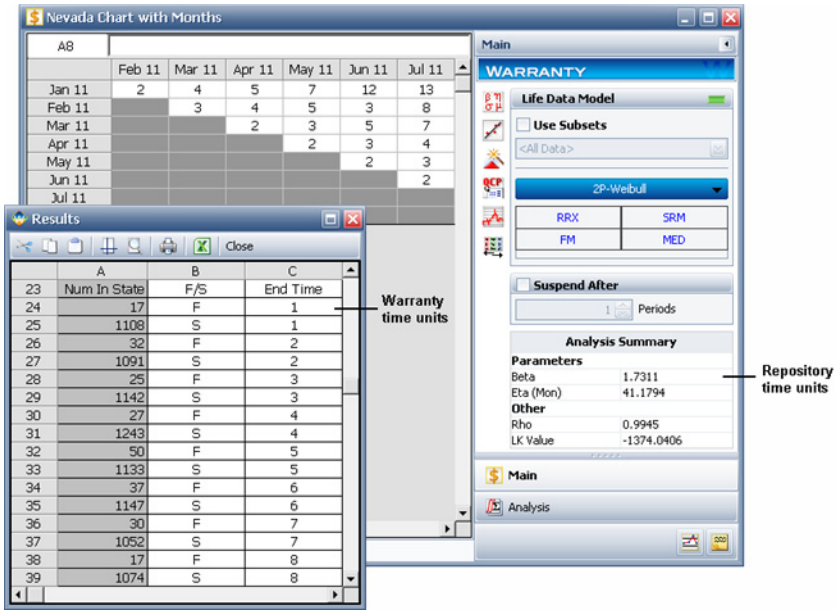
What does this mean?

For each Synthesis repository, 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, page 39 in Chapter 2*).

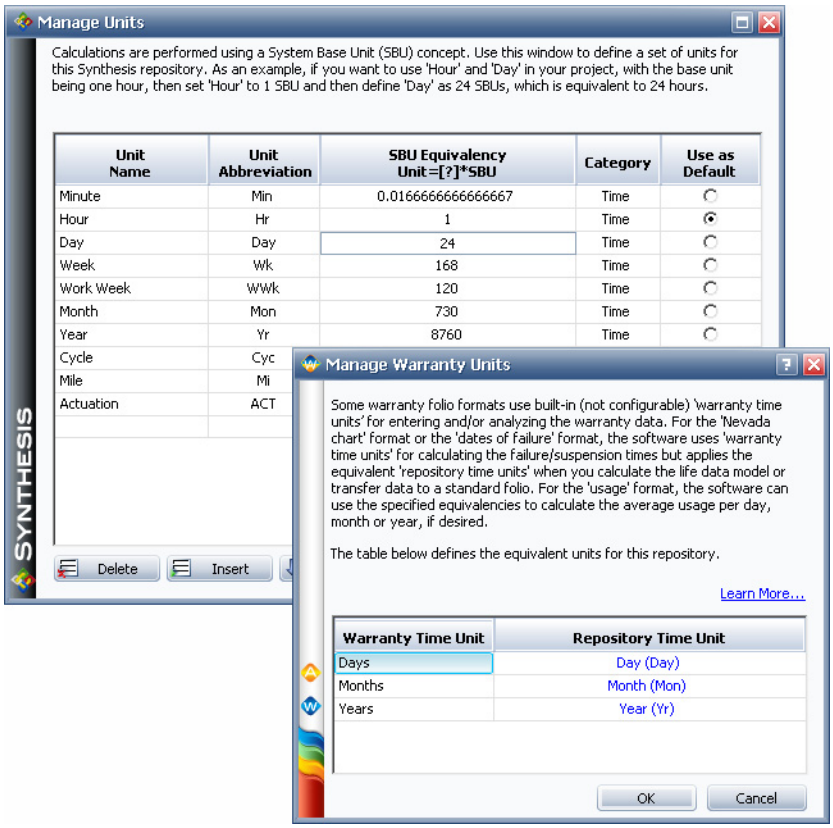
However, it is important to be aware that some warranty folio formats use built-in (not configurable) "warranty time units" for entering sales/returns data and calculating the failure/suspension times. For example, in the **dates of failure format**, if 100 units were shipped on January 1st and 2 units from that shipment were returned on January 15th, the failure times for those returns will be recorded as 14 calendar days. Likewise in the **Nevada chart format**, if you label the periods in terms of months and report that 10 of the 1,000 units shipped in January were returned in July, the failure times will be recorded as 6 calendar months. (Note that the **usage format** is affected differently because the failure/suspension values represent usage rather than time. See below for specific details.)

When you calculate the life data model for the Nevada/dates of failure folios (or when you transfer the data to a Weibull++ standard folio), the software must apply the equivalent "repository time units" so that a) the results will be consistent with other analyses in the same database and b) any tools that utilize time unit conversions (such as the QCP) will be able to perform the conversions properly.

For example, in the following picture, the failure/suspension data in the Results window (which is displayed if you click the Analysis Summary table in the control panel) has been calculated in terms of the "warranty time units" that are appropriate for the sales/returns data entered into the folio (calendar months), while the life data model parameters and QCP results are reported in the equivalent "repository time units" (abbreviated as "Mon" under the default settings).

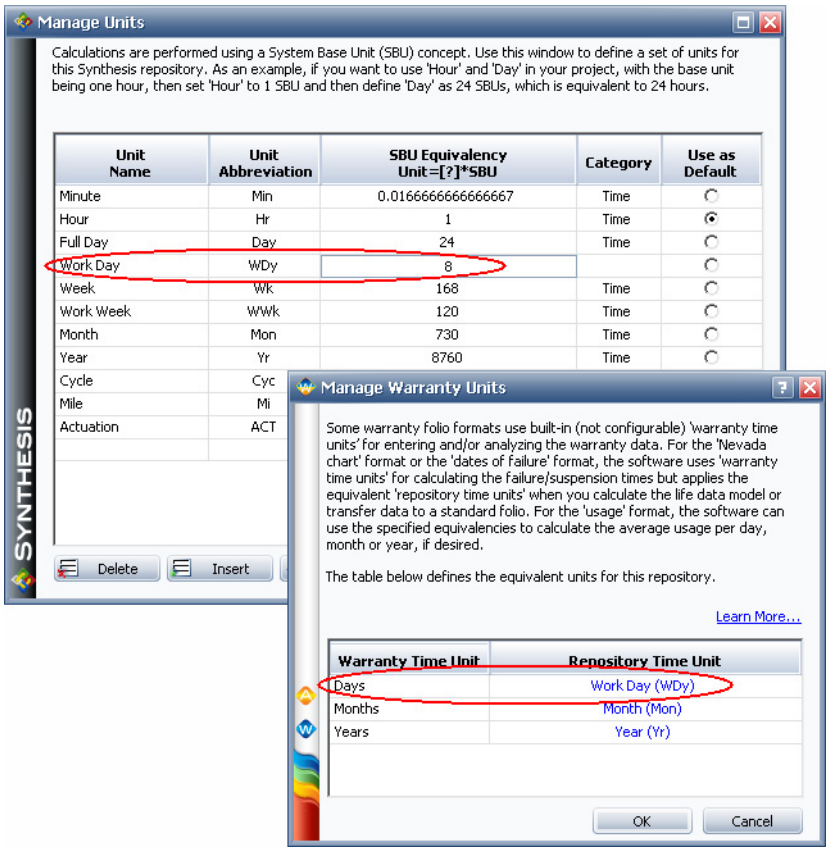


By default, each new database is created with “repository time units” and conversion factors that are equivalent to the built-in “warranty time units” (where a day = 24 hours, a month = 730 hours and a year = 8760 hours), as shown below.



If you choose to modify the configurable time units for any database (and you expect that database users will utilize any of the affected warranty folio formats), you must take this equivalency into account. For example, if you want to give users the choice of working with time inputs/results in terms of a full 24-hour day or an 8-hour work day, you must decide which type of day is appropriate to describe the operating times for warranty returns. Will you assume that a unit shipped on January 1st and returned on January 15th will have accumulated 336 hours of operation (14 days x 24 hours) or 112 hours of operation (14 days x 8 hours)? If the former, then you could simply add the new 8-

hour “Work Day” time unit to the database settings but keep the “warranty time unit” mapped to the original 24-hour “Day.” If the latter, then you could change the “warranty time unit” mapping as shown next.

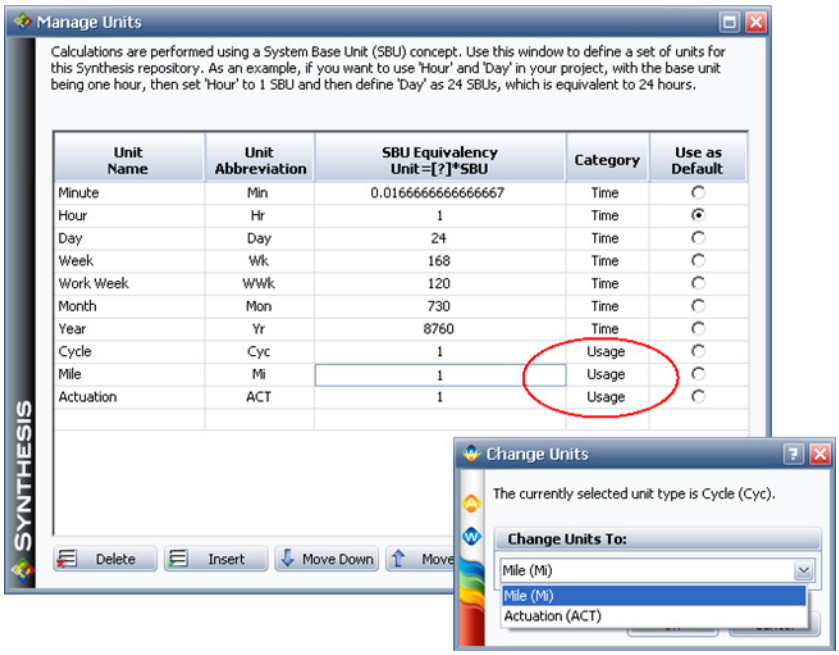


Remember that the failure/suspension data generated by the Nevada chart and dates of failure warranty folios do not get converted based on this mapping. The software simply assigns an appropriate “repository time unit” to the times that were calculated in “warranty time units.”

How are usage format folios affected?

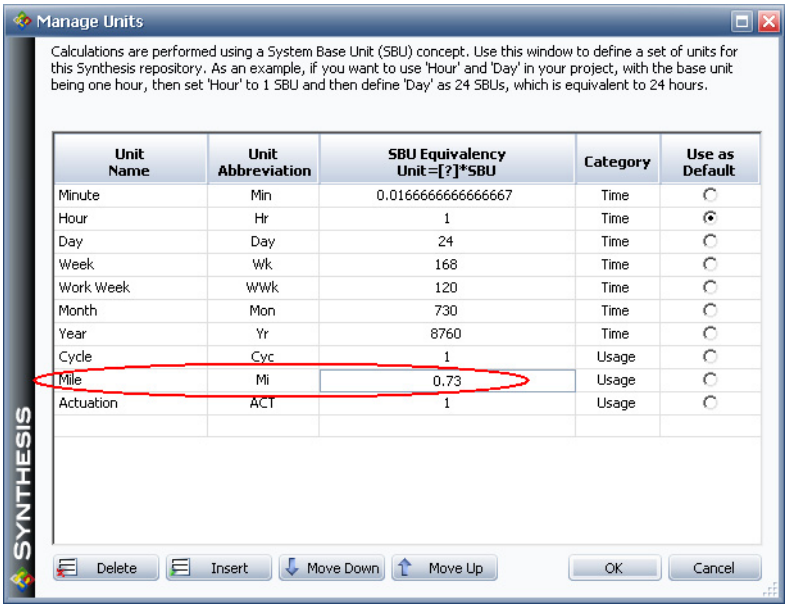
In a usage format warranty folio, you enter the amount of usage (rather than operating time) for the failed units and the software must estimate the amount of usage accumulated by the units that have not failed (suspensions).

- On the **Main** page of the usage format warranty folio's control panel, you can use the **Change Units** icon to select which of the "repository time units" the usage data will be entered in. The options include any of the units that have been assigned to the Usage category in the Manage Units window. The default settings are shown next.

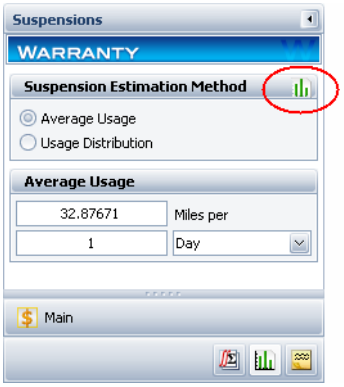


- On the **Suspensions** page of the control panel, you must specify the amount of usage that units typically accumulate per "Day," "Month" or "Year." This refers to the built-in (not configurable) "warranty time units." If you have selected to specify the Average Usage (rather than a Usage Distribution), you can enter the value directly (e.g., 1000 miles per month) or click the **Set Usage** icon to have the software automatically calculate the value based on the conversion factors that have been defined for the equivalent "repository time units."

  For example, suppose that the warranty data has been entered in miles and the "Manage Units" window indicates that typical usage is 1 Mile = .73 Hour.



If you choose **Days** from the drop-down list on the Suspensions page of the control panel, and then click the **Set Usage** icon (circled in the picture below), the software will use the Manage Warranty Units window to determine which of the "repository time units" is equivalent to Days in a warranty folio and then it will use the specified conversion factors to calculate the number of miles. For this example, the calculation would be as follows:



24 Hours / .73 Hour per Mile = 32.8767 'Miles

Another way to think about this is:

- 1 Mile / .73 Hour = 1.3699 Miles per Hour
- 24 Hours x 1.3699 Miles per Hour = 32.8767 Miles

For Months, the calculation would be:

- 730 Hours / .73 Hours per Mile = 1000 Miles

For Years, the calculation would be:

8760 Hours / .73 Hours per Mile = 12000 Miles

Event Log Folios 12

Event logs, or maintenance logs, store information about a piece of equipment's failures and repairs. They provide useful information that can help companies achieve their productivity goals by giving insight about the failure modes, frequency of outages, repair duration, uptime/downtime and availability of the equipment.

WEIBULL++ Only

Some event logs contain more information than others, but essentially event logs capture data in a format that includes the type of event, the date/time when the event occurred and the date/time when the system was restored to operation. The volume of this type of data can become very large and the task of extracting useful and concise information can also be cumbersome. To facilitate the processing of the data, the Weibull++ event log folio is designed to convert your log entries into failure/repair data and perform basic life data analysis. In addition, the folio calculates additional information such as the total uptime/downtime of the equipment and the total number of repair actions performed.

This chapter describes how to use the event log folio, including:

- Setting up the folio and entering data - Section 12.1 (p. 360).
- Using the control panel - Section 12.2 (p. 361).
- An example of event log data analysis - Section 12.3 (p. 364).
- Additional analysis tools, including:
  - Creating a shift pattern - Section 12.4.1 (p. 368).
  - The Systems Setup window - Section 12.4.2 (p. 369).
  - Transferring data to a Weibull++ standard folio - Section 12.4.3 (p. 370).
  - Exporting the analysis results to BlockSim - Section 12.4.4 (p. 372)

12.1 Setting Up the Event Log Folio

To create an event log folio, choose **Insert > Folios > Event Log** or right-click the **Specialized Folios** folder in the current project explorer and choose **Add Event Log** on the shortcut menu.



The following is an example of an event log folio data sheet. In this example, the events are from two cutting machine systems in two parallel production lines.

	System	F=Failure/ E=Event	Date Occurred	Time Occurred	Date Restored	Time Restored	Level 1	Level 2	Level 3	Level 4	OTSF (Y=Yes/N=No)	Description
1	M-1	F	Mar-01-09	11:00:00 AM	Mar-05-09	9:00:00 AM	Electric Saw	Motor			N	Electrical Short
2	M-1	F	Apr-01-09	2:30:00 PM	Apr-03-09	11:00:00 AM	Conveyer	Belt			N	
3	M-1	E	May-03-09	9:00:00 AM	May-06-09	12:00:00 PM	Conveyer	Wheels			N	Lubrication
4	M-1	F	Aug-03-09	3:30:00 PM	Aug-04-09	9:00:00 AM	Electric Saw	Blade			N	Worn
5	M-1	E	Sep-01-09	4:10:00 PM	Sep-02-09	4:00:00 PM	Hydraulic Arm	Motor			N	Inspections
6	M-1	F	Dec-01-09	10:00:00 AM	Dec-02-09	3:00:00 PM	Conveyer	Wheels			N	Bearings Failure
7	M-1	F	Jan-04-10	12:30:00 PM	Jan-07-10	7:20:00 AM	Hydraulic Arm	Hydraulic Lines			N	
8	M-1	F	Feb-01-10	4:00:00 PM	Feb-02-10	7:00:00 AM	Hydraulic Arm	Piston			N	
9	M-1	F	Mar-07-10	4:15:00 PM	Mar-09-10	9:00:00 AM	Conveyer	Belt			N	Unstable
10	M-1	F	Apr-05-10	4:30:00 PM	Apr-06-10	4:45:00 AM	Electric Saw	Blade			N	Crack

Every event log data sheet contains some or all of the following information:

- The **System** column identifies the system that experienced an event. In the sample data sheet, the machines are identified by the labels M-1 and M-2.

  The System column is optional, and it can be displayed by selecting the **Use System** column check box on the Other page of the folio's control panel. If the System column is not used, all events are assumed to have occurred for the same system.

  All occurrence dates in the log must be within the defined observation period for the system. You will enter the start/end dates of the observation period for the system in the System area of the control panel.

- The **F=Failure/E=Event** column indicates whether the occurrence was a failure event (F) or a general event (E). A general event represents an activity that brings the system down but is not directly relevant to the reliability of the equipment, such as preventive maintenance, routine inspections and the like.

  If you choose to include general events (E) in the analysis, you will have the option on the control panel to select whether you want to calculate the failure/repair distributions of the F and E occurrences separately or combined.

- **Date Occurred** and **Time Occurred** indicate the exact date and time of the event.

- **Date Restored** and **Time Restored** indicate the exact date and time the system was restored to operating mode.

- **Levels 1, 2, 3 and 4** indicate the subsystem or component that was responsible for the event. This gives you the flexibility to analyze the failure and repair data by certain levels within the system. For example, in row 1 of the sample data sheet, the failure event was caused by the motor of the electric saw of machine M-1. By entering this level of information in the data sheet, you will have the option to obtain individual failure and repair distributions for the motor (2nd level) and the electric saw (1st level). Note that in order to perform an analysis, you must at least specify a 1st level component for every entry in the log.

- **OTSF** (Operates Through System Failures) allows you to specify whether the component will continue to accumulate age when any failures other than the component's own failure occur. Enter a **Y** in this column to indicate that the component continues to operate through other failures or enter an **N** to indicate that the component is assumed to be failed when the system is failed.

  Note that the entry in this column for a particular component must be consistent with all other events in the data set associated with the component. For example, there are two rows in the sample data sheet with events associated with the electric saw blade (rows 4 and 10), both of those rows must contain the same value (Y or N) in

the OTSF column because the blade either does or does not continue to operate when the system fails due to another type of event.

The OTSF column is optional, and it can be displayed by selecting the **Use OTSF** column check box on the Other page of the folio's control panel. If this column is not used, all components are assumed to be failed when the system is failed.

- The **Description** column is for logging any other pertinent information or comments about the event. The information has no effect on the analysis.

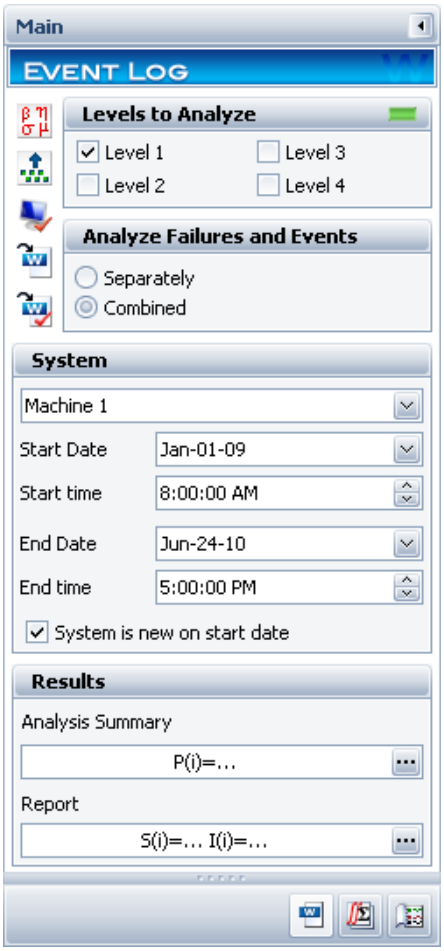
After you have entered the data for your system in the data sheet, the next step is to select the settings for the analysis. If your system does not operate on a 24/7 basis, you may also want to define the shift schedule of the system so that you obtain accurate calculations for the failure and repair times (*see Section 12.4.1 on page 368*).

12.2 Event Log Control Panel Settings

The event log folio control panel allows you to configure the analysis settings for the data sheet and view/access the results. It consists of a Main page, Analysis page and an Other page.

12.2.1 Control Panel Main Page

The Main page of the event log folio control panel contains most of the tools you will need to perform an analysis. As an example, the following picture shows the Main page of the control panel.



The Main page contains the following settings:

- The **Levels to Analyze** area allows you to specify the component level to include in the analysis. At least one level must be selected in order to perform an analysis. For instance, suppose you have a cutting system with an electric saw as the level 1 component and the saw blade as the level 2 component. If you select to analyze both levels 1 and 2, then the software will compute for the total uptime and downtime at each level. In addition:
  - The software will obtain the failure and repair distributions of the electric saw. The analysis of the level 1 component considers the events that affect the electric saw and any other components of the electric saw assembly, including the saw blade.
  - The software will also obtain separate failure and repair distributions for the saw blade. The analysis of the level 2 component considers the events that affect the saw blade and events for any other components of the saw blade (level 3 and 4 components of the saw blade, if any).

  Note that when you select the **Use OTSF** column check box on the Other page of the control panel, the software can analyze only one level at a time.
- The **Analyze Failures and Events** area allows you to select whether you want to obtain the failure/repair distributions of the F and E occurrences jointly or separately. When you select a joint analysis, all general events in the data set will be treated as failures. This affects the calculation for the total uptime/downtime of the component, as well as the failure/repair distributions.
- The **System** area is for specifying the dates/times when you started and stopped collecting the event log data. Note that if you select the **Use System column** check box on the Other page of the control panel, the drop-down list will be enabled. This allows you to select the system name from the drop-down list and enter the start date/time and the end date/time of the observation period of that system. If the **Use System column** check box is not selected, the software assumes that all events are assumed to have occurred for the same system and the drop-down list will be disabled.

  The **System is new on start date** check box indicates that the system has never been used before and that the start date you entered is also the same calendar date that the system started operating. This setting affects how the software classifies data when it converts the log entries to failure/suspension data. If the system is new, then the times to first occurrence of every event are considered to be complete data (F) because the exact time-to-event is known. If the system is not new, then the times to first occurrence of every event are considered to be right censored data (S) because the amount of time that the system operated before the observation period began is not known.

  > **Tip:** As an alternative, you can enter the start date/time and end date/time of the observation period for each system in a separate window by clicking the **Systems Setup** icon on the Main page of the control panel. The larger window makes it easier to enter information for several systems all at once. This option is available only if you have selected the **Use System column** check box on the Other page of the folio's control panel.

  If your system does not operate on a 24/7 basis, you may also want to define the shift schedule of the system so that you obtain accurate calculations for the failure and repair times. *See Section 12.4.1 on page 368*.
- The **Results** area gives you the option to display the following results:
  - To view the failure/repair distributions of each component, click the **Show Analysis Summary (...)** button. The resulting failure/repair distributions are based on the settings you have defined on the Analysis page of the control panel and on the log data in your data sheet.
  - To view a report on the number of repair actions performed, the OTSF settings of each component and other results, click the **Show Report (...)** button.

Folio Tools

The folio tools are arranged on the left side of the Main page of the control panel. These tools are also available from the Event Log tab in the Ribbon.



Calculate converts the log entries into failure/repair times and then fits distributions to the converted data. In addition, it calculates the total uptime and downtime of the components.



Set Shift Pattern opens the Shift Pattern window, which allows you to specify the days and times when the system is operating. *See Section 12.4.1 on page 368*.



Systems Setup is available only if you have selected the **Use System** column check box on the Other page of the folio's control panel. It opens the Systems Setup window, which allows you to record the dates/times when you started and stopped collecting event log data for each system and whether it was new at the specified start date. *See Section 12.4.2 on page 369*.

After the folio has been calculated, three transfer options become available:



Transfer Life Data to New Folio transfers all of the analyzed time-to-failure and time-to-repair data sets to separate data sheets in a new Weibull++ folio.



Transfer Life Data to Selected Folio opens the Transfer Life Data window, which allows you to choose specific data set(s) to transfer to either new or existing folio(s). *See Section 12.4.3 on page 370*.



Transfer to BlockSim is available only if BlockSim is activated on your computer. The Transfer to BlockSim window allows you to export the analysis results as blocks in a new simulation diagram. *See Section 12.4.4 on page 372*.

12.2.2 Control Panel Analysis Page

The Analysis page of the event log folio control panel allows you to specify how the software will fit a distribution to the failure data and the repair data. It contains the following settings:

- If you select the **Use quick defaults** check box, all the other options will be unavailable and one of the following actions will be applied:
  - When the failure/repair data set contains at least two unique failure times, the software uses the 2-parameter Weibull distribution and the MLE parameter estimation method to analyze the data.
  - When the failure/repair data set contains one failure time, the software uses the 1-parameter exponential distribution and the MLE parameter estimation method to analyze the data.
- If you select the **Prefer RRX** if sufficient data option, the software uses the RRX parameter estimation method when the failure/repair data set has at least two unique failure times; otherwise, the MLE method is used.
- If you select the **Always use MLE** option, the software always uses the MLE parameter estimation method.
- The **Distributions to Consider** area allows you to select one or more of the listed distributions to be considered for the data. The software evaluates the fit of each selected distribution to the data set and displays in the results the one that provides the best fit.

12.2.3 Control Panel Other Page

The Other page of the event log folio control panel contains optional settings that allow you to format the event log data sheet.

- The **Times to Failure/Repair In** area allows you to select the unit of time to be used when the software converts the log entries to failure/repair data.
- The **Time Format** and **Date Format** areas allows you to select the formatting options for displaying the date and time information in the data sheet and in the results.
- The **Other** area allows you to include either an OTSF (Operates Through System Failures) column or a System column in the data sheet, or both.
  - If the **Use OTSF** column check box is selected, you will have the option to specify in the data sheet whether a component continues to accumulate age when any failures other than that component's own failure occurs. If the check box is not selected, all components are assumed to be failed when the system is failed.
  - If the **Use System** column check box is selected, you will have the option to enter events for multiple systems in the data sheet. If the check box is not selected, all events are assumed to have occurred for the same system.
- The **Comments** area allows you to enter notes or other text that will be saved with the folio.

To save your current settings for all new event log folios, click the **Set as Default** button. To automatically apply the saved settings to any existing event log folio, click **Load Default**.

12.3 Event Log Data Analysis Example

Consider a simple system comprising two components of interest for analysis purposes: a motor and a pump. When the motor fails, the pump also stops operating and the entire system is shut down for repairs. However, when the pump fails, the motor continues to run. The system operates from 8 AM to 5 PM seven days a week.

The following table shows the maintenance log. System monitoring started on January 1, 2010 at 12 PM and stopped on March 18, 2010 at 1 PM. The system had been in operation for some time before the log began but information for events prior to January 1 is not available.

	Event Type	System Failed		System Restored		Component
		Date	Time	Date	Time	
1	Failure	Jan-02-10	4:00 PM	Jan-02-10	7:49 PM	Pump
2	Maintenance	Jan-09-10	8:30 AM	Jan-09-10	10:43 AM	Motor
3	Failure	Jan-10-10	9:13 AM	Jan-10-10	7:48 PM	Motor
4	Inspection	Jan-12-10	3:26 PM	Jan-12-10	6:46 PM	Pump
5	Failure	Jan-13-10	4:56 PM	Jan-13-10	5:21 PM	Pump
6	Maintenance	Jan-15-10	1:16 PM	Jan-15-10	4:39 PM	Motor
7	Failure	Jan-20-10	1:38 PM	Jan-21-10	7:15 PM	Pump
8	Failure	Jan-25-10	10:32 AM	Jan-27-10	10:47 PM	Motor
9	Cleaning	Jan-28-10	11:31 AM	Jan-28-10	12:00 PM	Pump
10	Failure	Feb-02-10	2:38 PM	Feb-02-10	7:11 PM	Motor
11	Inspection	Feb-08-10	3:51 PM	Feb-08-10	8:22 PM	Pump
12	Failure	Feb-12-10	4:42 PM	Feb-13-10	9:59 AM	Pump
13	Cleaning	Feb-17-10	2:47 PM	Feb-17-10	7:13 PM	Motor
14	Maintenance	Feb-25-10	4:31 PM	Feb-25-10	5:00 PM	Pump

	Event Type	System Failed		System Restored		Component
		Date	Time	Date	Time	
15	Failure	Feb-28-10	9:00 AM	Feb-28-10	3:10 PM	Motor
16	Failure	Mar-01-10	10:16 AM	Mar-01-10	10:43 AM	Pump
17	Inspection	Mar-02-10	3:41 PM	Mar-02-10	9:11 PM	Pump
18	Failure	Mar-12-10	8:46 AM	Mar-12-10	9:20 PM	Motor
19	Failure	Mar-13-10	4:45 PM	Mar-13-10	5:13 PM	Pump
20	Maintenance	Mar-15-10	9:36 AM	Mar-15-10	10:02 PM	Motor

The objective is to analyze each component and obtain the failure and repair distributions for all recorded events.

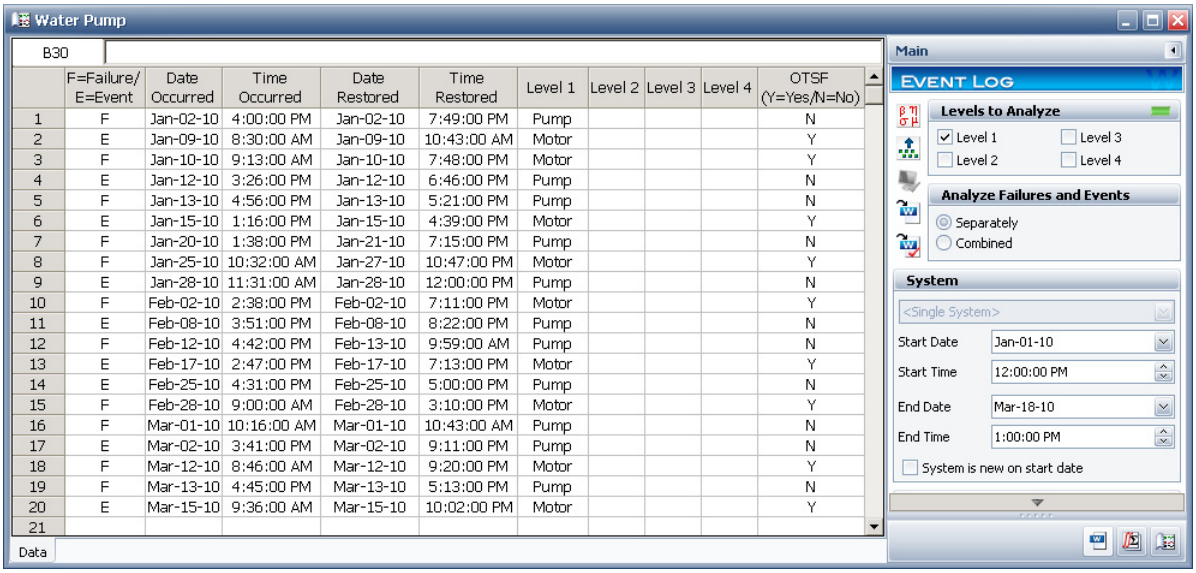
1. The first step is to set up the event log folio for data entry. On the Other page of the control panel, select the **Use OTSF column** check box. Clear the check box for the **Use System column** and then select your preferred format for the date and time.
2. On the Analysis page of the control panel, select the **Prefer RRX if sufficient data** option and select all three distributions for consideration. Do this for both the failure and repair distributions.
3. Next, fill out the data sheet with the event log data.
   - In the "F=Failure/E=Event" column, mark all the events that are not considered to be actual failures (i.e., maintenance, cleaning and inspections) with an **E**. Use **F** for the failures.
   - In the "OTSF" column, mark all the events attributed to the motor component with a **Y** because it continues to operate despite a failed pump. The pump, however, will stop accumulating age if the motor fails. In this case, mark all the events attributed to the pump with an **N**.
4. Once data entry is complete, go to the Main page of the control panel and make the following selections/inputs:
   - In the **Levels to Analyze** area, select **Level 1**.
   - In the **Failures and Events** area, select the **Analyze Separately** option so that we get separate results for the failures and the general events.
   - In the **System** area, enter the date and time the system monitoring started, as well as the date and time the system monitoring ended. Clear the check box for **System is new on start date**.
5. Next, enter the shift schedule of the system by choosing **Event Log > Actions and Settings > Set Shift Pattern** or by clicking the icon on the control panel.



6. In the Shift Pattern window, specify 8:00 AM to 5:00 PM shifts seven days a week, as shown next. Click **OK**.

Shift Number		Monday	Tuesday	Wednesday	Thursday	Friday	Saturday	Sunday
Shift 1	Start Time	8:00:00 AM	8:00:00 AM	8:00:00 AM	8:00:00 AM	8:00:00 AM	8:00:00 AM	8:00:00 AM
	End Time	5:00:00 PM	5:00:00 PM	5:00:00 PM	5:00:00 PM	5:00:00 PM	5:00:00 PM	5:00:00 PM

The following figure shows the completed data sheet and control panel setup.

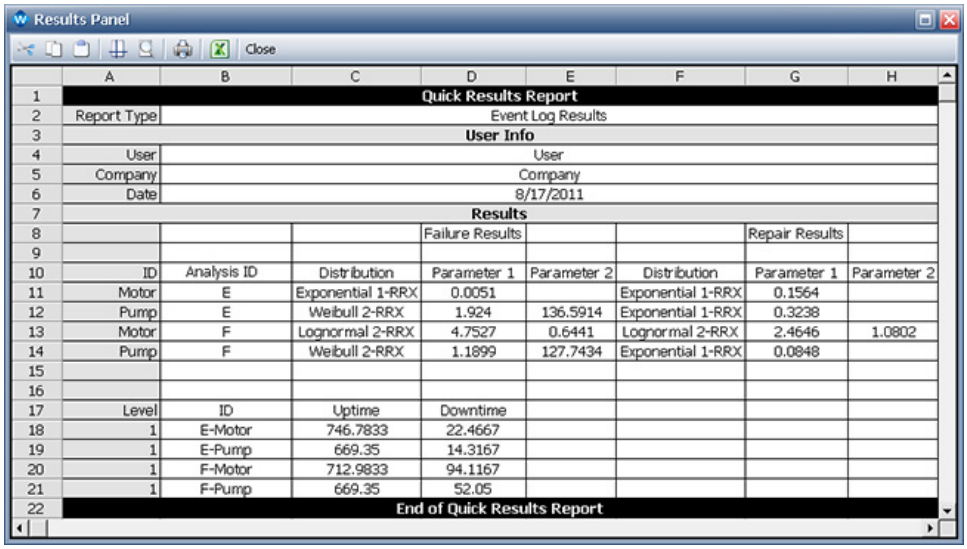


	F=Failure/ E=Event	Date Occurred	Time Occurred	Date Restored	Time Restored	Level 1	Level 2	Level 3	Level 4	OTSF (Y=Yes/N=No)
1	F	Jan-02-10	4:00:00 PM	Jan-02-10	7:49:00 PM	Pump				N
2	E	Jan-09-10	8:30:00 AM	Jan-09-10	10:43:00 AM	Motor				Y
3	F	Jan-10-10	9:13:00 AM	Jan-10-10	7:48:00 PM	Motor				Y
4	E	Jan-12-10	3:26:00 PM	Jan-12-10	6:46:00 PM	Pump				N
5	F	Jan-13-10	4:56:00 PM	Jan-13-10	5:21:00 PM	Pump				N
6	E	Jan-15-10	1:16:00 PM	Jan-15-10	4:39:00 PM	Motor				Y
7	F	Jan-20-10	1:38:00 PM	Jan-21-10	7:15:00 PM	Pump				N
8	F	Jan-25-10	10:32:00 AM	Jan-27-10	10:47:00 PM	Motor				Y
9	E	Jan-28-10	11:31:00 AM	Jan-28-10	12:00:00 PM	Pump				N
10	F	Feb-02-10	2:38:00 PM	Feb-02-10	7:11:00 PM	Motor				Y
11	E	Feb-08-10	3:51:00 PM	Feb-08-10	8:22:00 PM	Pump				N
12	F	Feb-12-10	4:42:00 PM	Feb-13-10	9:59:00 AM	Pump				N
13	E	Feb-17-10	2:47:00 PM	Feb-17-10	7:13:00 PM	Motor				Y
14	E	Feb-25-10	4:31:00 PM	Feb-25-10	5:00:00 PM	Pump				N
15	F	Feb-28-10	9:00:00 AM	Feb-28-10	3:10:00 PM	Motor				Y
16	F	Mar-01-10	10:16:00 AM	Mar-01-10	10:43:00 AM	Pump				N
17	E	Mar-02-10	3:41:00 PM	Mar-02-10	9:11:00 PM	Pump				N
18	F	Mar-12-10	8:46:00 AM	Mar-12-10	9:20:00 PM	Motor				Y
19	F	Mar-13-10	4:45:00 PM	Mar-13-10	5:13:00 PM	Pump				N
20	E	Mar-15-10	9:36:00 AM	Mar-15-10	10:02:00 PM	Motor				Y
21										

7. Next, calculate the results by choosing **Event Log > Analysis > Calculate** or by clicking the icon on the control panel.



8. To view the failure and repair distributions, click the **Show Analysis Summary (...)** button on the control panel. The following report shows the results.



	A	B	C	D	E	F	G	H
1	Quick Results Report							
2	Report Type	Event Log Results						
3	User Info							
4	User	User						
5	Company	Company						
6	Date	8/17/2011						
7	Results							
8				Failure Results			Repair Results	
9								
10	ID	Analysis ID	Distribution	Parameter 1	Parameter 2	Distribution	Parameter 1	Parameter 2
11	Motor	E	Exponential 1-RRX	0.0051		Exponential 1-RRX	0.1564	
12	Pump	E	Weibull 2-RRX	1.924	136.5914	Exponential 1-RRX	0.3238	
13	Motor	F	Lognormal 2-RRX	4.7527	0.6441	Lognormal 2-RRX	2.4646	1.0802
14	Pump	F	Weibull 2-RRX	1.1899	127.7434	Exponential 1-RRX	0.0848	
15								
16								
17	Level	ID	Uptime	Downtime				
18	1	E-Motor	746.7833	22.4667				
19	1	E-Pump	669.35	14.3167				
20	1	F-Motor	712.9833	94.1167				
21	1	F-Pump	669.35	52.05				
22	End of Quick Results Report							

In this example, rows 11 to 14 display the failure and repair distributions for each component due to each type of event. Rows 18 to 21 display the total uptime and downtime for each component due to each type of event.

9. Close the Results window. To view a report on the number of repair actions performed and other results, click the **Show Report (...)** button on the control panel as shown next.

Results

Close

	A	B	C	D	E	F	G	H	I	J
1	**Level: 1 Identifier: F-Pump**									
2	Type: Failure. Operates through other failures: No.									
3	Observed Data Points					Repair Actions				
4	7					6				
5	*Analysis Summary*									
6	Failure Analysis - Assumed Distribution: Weibull - Analysis Method: RRX									
7	Repair Analysis - Assumed Distribution: Lognormal - Analysis Method: RRX									
8										
9	**Level: 1 Identifier: E-Motor**									
10	Type: Event. Operates through other failures: No.									
11	Observed Data Points					Repair Actions				
12	5					4				
13	*Analysis Summary*									
14	Failure Analysis - Assumed Distribution: Weibull - Analysis Method: RRX									
15	Repair Analysis - Assumed Distribution: Exponential - Analysis Method: RRX									
16										
17	**Level: 1 Identifier: F-Motor**									
18	Type: Failure. Operates through other failures: No.									
19	Observed Data Points					Repair Actions				
20	6					5				
21	*Analysis Summary*									
22	Failure Analysis - Assumed Distribution: Lognormal - Analysis Method: RRX									
23	Repair Analysis - Assumed Distribution: Lognormal - Analysis Method: RRX									
24										
25	**Level: 1 Identifier: E-Pump**									
26	Type: Event. Operates through other failures: No.									
27	Observed Data Points					Repair Actions				
28	6					5				
29	*Analysis Summary*									
30	Failure Analysis - Assumed Distribution: Weibull - Analysis Method: RRX									
31	Repair Analysis - Assumed Distribution: Exponential - Analysis Method: RRX									

10. Close the Results window. You can perform further analysis by transferring the failure/repair data from the event log folio to a Weibull++ standard folio. To do this, choose **Event Log > Transfer Life Data > Transfer Life Data to New Folio** or click the icon on the control panel.



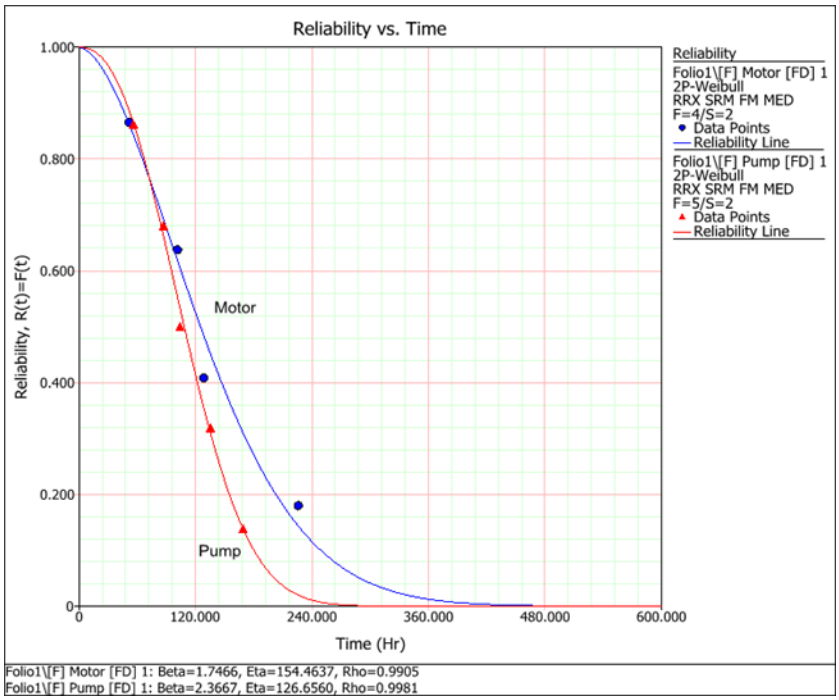
This automatically creates a new Weibull++ standard folio within the project. The folio will contain separate data sheets for the failure and repair data. The following data sheet shows the failure data for the motor component.

	State F or S	Time to F or S (Hr)	Subset ID
1	S	78.21666667	[F] Motor [FD]
2	F	128.5333333	[F] Motor [FD]
3	F	51.63333333	[F] Motor [FD]
4	F	226	[F] Motor [FD]
5	F	101.6	[F] Motor [FD]
6	S	50	[F] Motor [FD]

[F] Pump [FD] 1 | [F] Pump [RD] 1 | [E] Motor [FD] 1 | [E] Motor [RD] 1 | [F] Motor [FD] 1 | [F] Motor [RD] 1 | [E] Pump [FD] 1 | [E] Pump [RD] 1

The **[F]** indicates an analysis of the failures, while **[E]** indicates an analysis of the general events (note that a combined analysis of F and E events would display **[C]**). The **[FD]** indicates that the data set is for the failure distribution, while **[RD]** indicates that the data set is for the repair distribution.

You can visually compare the reliability of the components by fitting a distribution to the failure data and then generating an overlay plot. For example, the following overlay plot shows the failure data for the motor and for the pump, both analyzed using the 2-parameter Weibull distribution.



12.4 Folio Tools

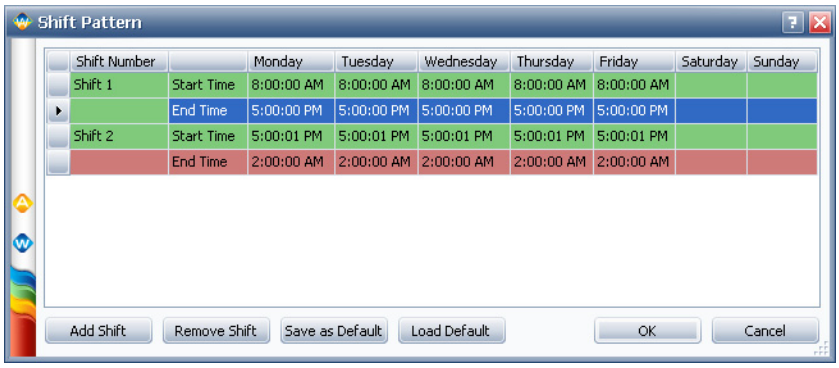
12.4.1 Set Shift Pattern

One of the functions of the Weibull++ event log folio is to convert your log entries into failure/repair times. The failure times are obtained by calculating the date/time between the last repair and the date/time the new failure occurred. The repair times are obtained by calculating the difference between the date/time of occurrence and the date/time of restoration. In the case of equipment that does not run on a 24/7 basis, the Shift Pattern feature allows the analysis to take into consideration the periods when your system is not in use. This ensures accurate calculations for the failure and repair times.

To create a shift pattern, choose **Event Log > Action and Settings > Set Shift Pattern** or click the icon on the Main page of the control panel.



The shift schedule you create applies to all of the systems you have identified in your data sheet. In other words, all the systems follow the same shift schedule. The following picture shows an example.



The buttons at the lower left side of the window allow you to add or remove shifts. To save the current shift pattern as the default for new event log folios, click **Save as Default**. To load the default shift pattern settings for any existing event log folio, click **Load Defaults**.

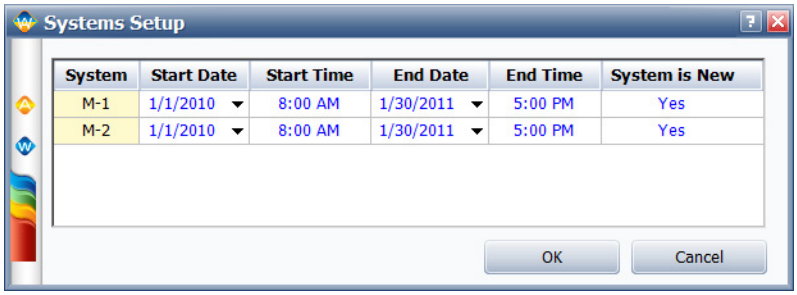
12.4.2 Systems Setup

The Systems Setup window is available only if you have selected the **Use System column** check box on the Other page of the folio's control panel. This window allows you to record the dates/times when you started and stopped collecting event log data for each system and whether it was new at the specified start date. The information you provide is used by the software to obtain the time to first occurrence of the failure (F) and general (E) events of every unique system.

To open the Systems Setup window, click the icon on the Main page of the control panel.

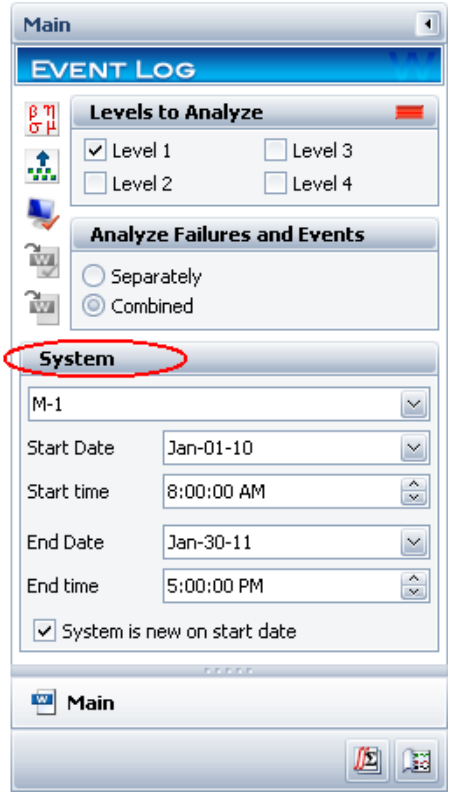


The following picture shows an example of a setup for two machines, M-1 and M-2.



The **System is New** column indicates that the system has never been used before and that the start date you entered is also the same calendar date that the system started operating. If the system is new, then the times to first occurrence of every event are considered to be complete data (failures) because the exact time-to-event is known. If the system is not new, then the times to first occurrence of every event are considered to be right censored data because the amount of time that the system operated before the observation period began is not known.

Note that the information you provide in the Systems Setup window will be automatically copied to the **System** area on the Main page of the control panel and vice versa.



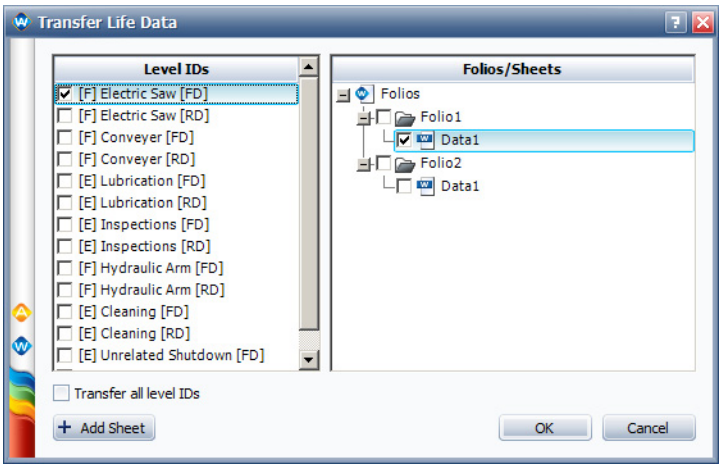
12.4.3 Transfer Life Data

You can transfer the time-to-failure and time-to-repair data from an event log folio to a new or existing Weibull++ standard folio. To do this, choose **Event Log > Transfer Life Data > Transfer Life Data to Selected Folio** or click the icon on the Main page of the control panel.



The following example shows the Transfer Life Data window for a system with three 1st level components. The **[F]** indicates an analysis of the failures, while **[E]** indicates an analysis of the general events (note that a combined

analysis of F and E events would display **[C]**). The **[FD]** indicates that the data set is for the failure distribution, while **[RD]** indicates that the data set is for the repair distribution.



Transferring Data Sets

This utility provides the option to transfer one selected data set, or to transfer all data sets together.

To transfer a data set:

1. On the left panel, select a data set.
2. On the right panel, specify where the data will be transferred. There are three ways to do this:
   - If you want to overwrite an existing data sheet with the selected data, select the check box for the specific data sheet.
   - If you want to create a new data sheet, select the name of the folio where you want to insert the new sheet and click **Add Sheet**.
   - If you want to create a new folio, select the **Folios** header at the top of the panel and click **Add Folio**.
3. Click **OK** to transfer the data.

To transfer all data at once:

1. Select the **Transfer all level IDs** check box.
2. On the right panel, select the check box for the folio where the data will be transferred. Note that the utility will create new data sheets in the folio for each data set. If the folio already contains data sheets that have the same name as the data to be transferred, you can choose to overwrite the data in those sheets by selecting the **Replace sheets with same names** check box.
3. If you want to create a new folio instead, click **Add Folio**.
4. Click **OK** to transfer the data.

12.4.4 Transfer to BlockSim

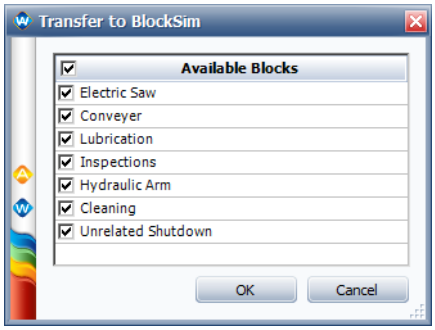
If you have the Synthesis version of ReliaSoft's system availability and maintainability analysis software, BlockSim, activated on your computer, you can export the analysis results as blocks in a new BlockSim simulation diagram. Each block represents a 1st level component, where the block's reliability model is based on the component's failure distribution and the corrective task is based on the component's repair distribution. (*See also Universal Reliability Definitions (URD) on page 184 in Chapter 5*.)

The current versions of Weibull++ and BlockSim are integrated via the Synthesis Platform®. This means that when you transfer your Weibull++ data, the same database will be automatically opened in the BlockSim application. The Weibull++ data will then be stored in a new BlockSim simulation diagram that is in the same project as your Weibull++ data.

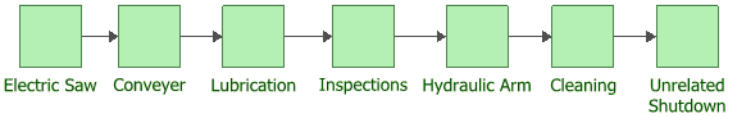
To transfer your Weibull++ analysis results, first calculate the parameters of the data set, and then choose **Event Log > Transfer Life Data > Transfer to BlockSim**.



In the Transfer to BlockSim window, select which components you wish to transfer and click **OK**. The selected components will be transferred to BlockSim as blocks that are connected in a reliability-wise series configuration, as shown in the following example. You can then edit the reliability block diagram in the BlockSim application, if desired.



Transfer to BlockSim window in Weibull++



Reliability block diagram in BlockSim

Recurrent Event Data Analysis (RDA) Folios

13

WEIBULL++ Only

In life data analysis, it is assumed that the components being analyzed are *non-repairable*; that is, they are either discarded or replaced upon failure. When analyzing the failure behavior of non-repairable components, the data points are typically either times-to-failure or times-to-suspension. For a group of non-repairable units coming from a single population, the time-to-failure of one unit in the sample does not affect the time-to-failure of other units in the sample. Therefore, the lifetimes of non-repairable systems are considered to be independent and identically distributed (i.i.d).

On the other hand, for complex systems such as automobiles, computers, aircraft, etc., it is likely that the system will be repaired (not discarded) upon failure. Failures are recurring events in the life of a *repairable system*, and data from such a system are obtained by recording the age of the system at the time when each failure occurred. This type of data is known as *recurrent event data*.

The failure behavior of a repairable system is dependent on that system's history of repairs; therefore, traditional life data analysis methods such as the Weibull distribution are not appropriate because those methods treat every failure event as identical and independent from the previous one. In order to analyze recurrent event data, Weibull++ includes a choice of two methods: non-parametric and parametric analysis.[1]

The following sections describe how to perform a non-parametric RDA in Weibull++

- Setting up a non-parametric RDA folio and evaluating the results - Section 13.1.1 (p. 374).
- Example of a non-parametric RDA - Section 13.1.2 (p. 377).

The following sections describe how to perform a parametric RDA:

- Setting up a parametric RDA folio - Section 13.2.1 (p. 379).
- How to use the parametric RDA folio control panel and select the settings for the analysis - Section 13.2.2 (p. 380).
- Perform reliability calculations via the RDA Quick Calculation Pad (QCP) - Section 13.2.3 (p. 383).
- Generate RDA plots - Section 13.2.4 (p. 384).
- Example of a parametric RDA - Section 13.2.5 (p. 385).

1. The ReliaWiki resource portal has more information on recurrent event data analysis at: http://www.ReliaWiki.org/index.php/Recurrent_Event_Data_Analysis. For general information on parametric vs. non-parametric analysis, please read "*Data Analysis and Reporting*" at: http://www.Weibull.com/hotwire/issue44/hottopics44.htm.

13.1 Non-Parametric RDA Folio

The non-parametric RDA folio is based on the mean cumulative function (MCF). This analysis provides a plot of the MCF to illustrate the average number of recurring failures of a system, or a group of systems, over a given period of time (or distance, cycles, etc.). The MCF can be used to evaluate whether the number of failures increases or decreases over time, to predict the future number of failures or to compare different data sets from different designs, operating conditions, production periods, etc.[2]

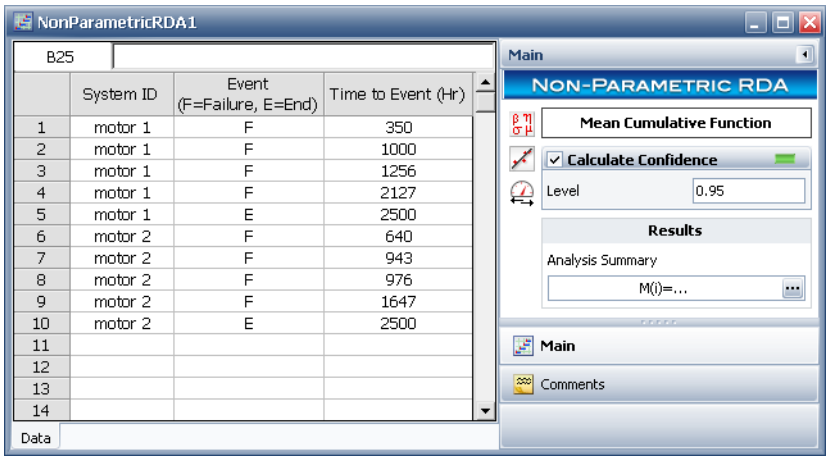
13.1.1 Setting Up the Folio and Evaluating Results

Setting Up the Folio

To create a non-parametric RDA folio, choose **Insert > Folios > Non-Parametric RDA** or right-click the **Specialized Folios** folder in the current project explorer and choose **Add Non-Parametric RDA** on the shortcut menu.



The following figure shows an example of the non-parametric RDA folio data sheet.



The data sheet contains the following information:

- The **System ID** column identifies the system that experienced an event. In the sample data sheet, two units of the same type of motor are being tracked. The resulting mean cumulative function (MCF) will depict the average behavior of the systems.
- The **Event** column describes the type of event. An **F** indicates a failure and an **E** indicates that the observation period for the unit has ended. Each unique system in the data sheet must have a single end event. If you do not enter an end event for a system, then the end event is assumed to be equal to the last failure time for that system.

2. The ReliaWiki resource portal has more information on the MCF at: http://www.ReliaWiki.org/index.php/Non-Parametric_Recurrent_Event_Data_Analysis.

- The **Time to Event** column indicates the age of the system when the event occurred. For each unique system, the time of the end event (E) must be equal to or greater than the last failure time of that system. The time of the end event is also known as the *censoring age*, because the product's failure or repair history beyond that time is unknown.
- The **Calculate Confidence** option on the control panel calculates the upper 1-sided and lower 1-sided confidence bounds of the MCF at a specified confidence level. Specify the desired confidence level in the **Level** input field. Selecting this option also automatically includes the confidence bounds in the plot of the analysis.[3]

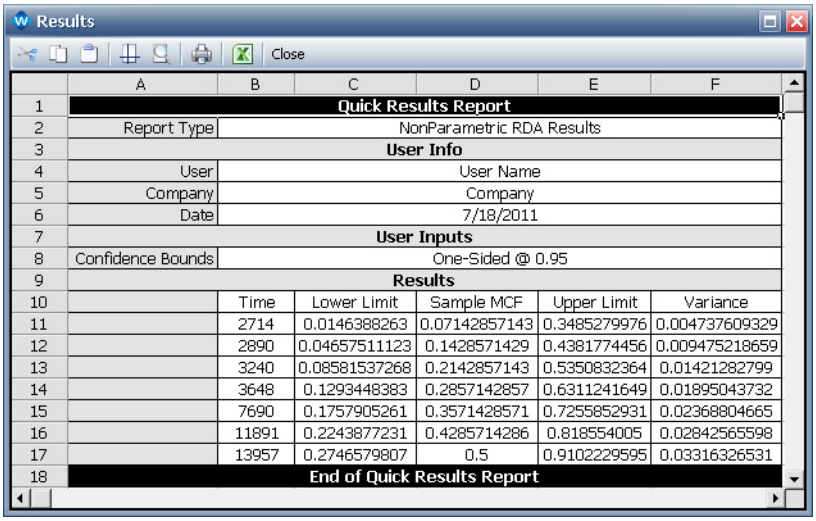
Evaluating Results

To analyze the data set, choose **Recurrent Event Data > Analysis > Calculate** or click the icon on the control panel.



Note that this will also sort the data set by system ID and then by time to event.

To view the results, click the **Show Results (...)** button to open the Results window, as shown next.



	A	B	C	D	E	F
1	**Quick Results Report**					
2	Report Type	NonParametric RDA Results				
3	**User Info**					
4	User	User Name				
5	Company	Company				
6	Date	7/18/2011				
7	**User Inputs**					
8	Confidence Bounds	One-Sided @ 0.95				
9	**Results**					
10		Time	Lower Limit	Sample MCF	Upper Limit	Variance
11		2714	0.0146388263	0.07142857143	0.3485279976	0.004737609329
12		2890	0.04657511123	0.1428571429	0.4381774456	0.009475218659
13		3240	0.08581537268	0.2142857143	0.5350832364	0.01421282799
14		3648	0.1293448383	0.2857142857	0.6311241649	0.01895043732
15		7690	0.1757905261	0.3571428571	0.7255852931	0.02368804665
16		11891	0.2243877231	0.4285714286	0.818554005	0.02842565598
17		13957	0.2746579807	0.5	0.9102229595	0.03316326531
18	**End of Quick Results Report**					

The **Time** column displays the system ages, sorted in chronological order, that were entered for the failure events. The **Sample MCF** column shows the average number of failures at each particular age.

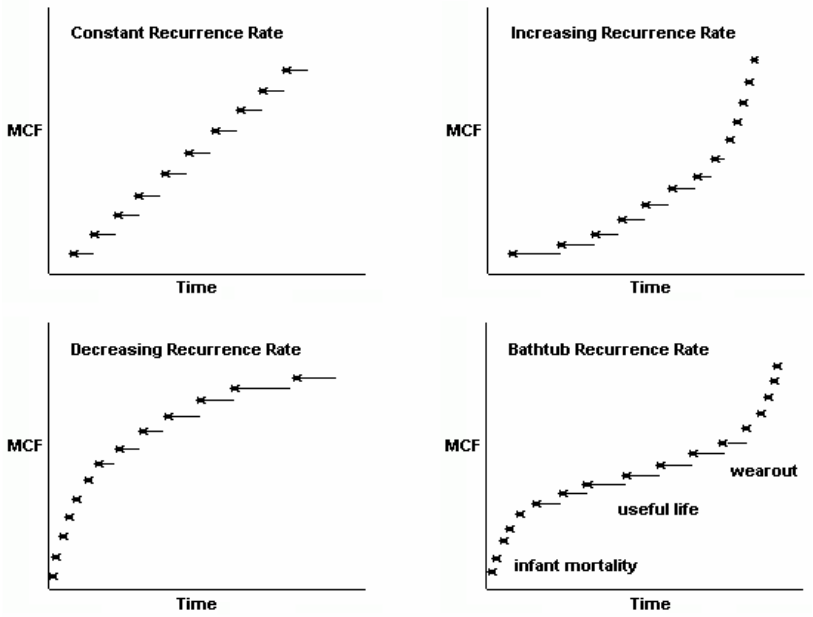
The **Variance** column indicates the amount of uncertainty in the MCF value. This is also used to obtain the 1-sided confidence bounds that are displayed in the **Lower Limit** and **Upper Limit** columns. These three columns are available only if you have selected the **Calculate Confidence** check box in the control panel and specified a confidence level.

The MCF values and the confidence bounds can also be displayed in an MCF plot in order to visualize the behavior of the recurring failures. To view the MCF plot, choose **Recurrent Event Data > Analysis > Plot**, or click the icon on the control panel.



3. The ReliaWiki resource portal provides more information on confidence bounds at: http://www.ReliaWiki.org/index.php/Confidence_Bounds.

The following plots show some examples of the different MCF shapes.



- **Constant Recurrence Rate** indicates that the units are exhibiting what is usually referred to as *useful life*, which is a stable period during which the failures occur at a rate that is neither increasing or decreasing with time.
- **Increasing Recurrence Rate** is a curve that is concave up and the recurrence rate of the failures increases (the data points are becoming horizontally closer) as the population ages. This is typical of units with *wearout* problems. It could also indicate that the system maintenance is degrading with time.
- **Decreasing Recurrence Rate** is a curve that is concave down and the recurrence rate of the failures decreases (the data points are becoming horizontally farther apart) as the population ages. This is typical of units exhibiting a period of *infant mortality* problems. It could also indicate that the system maintenance is improving with time.
- **Bathtub Recurrence Rate** is a curve that starts as concave down, and then becomes concave up. This behavior is typical of units that first experience infant mortality problems and then exhibit a useful life period. As the age increases, the units start to enter the wearout stage.

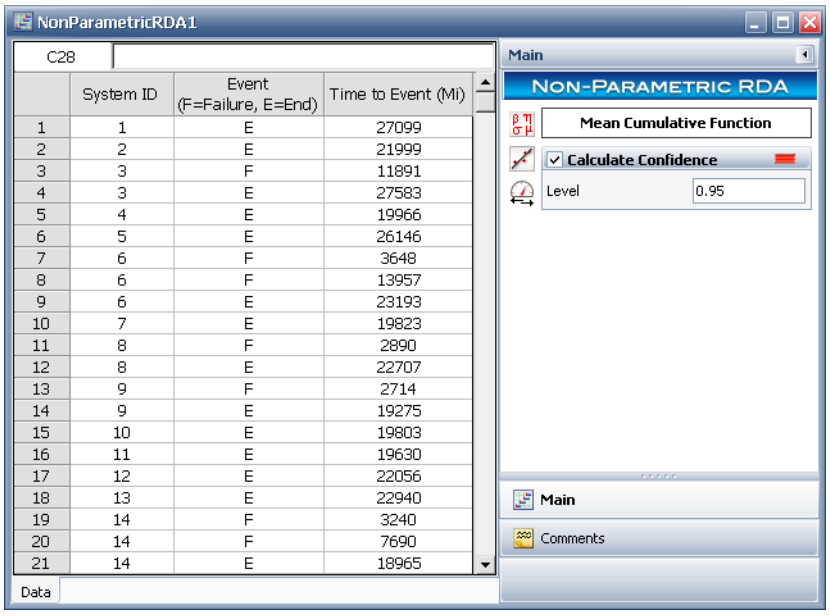
13.1.2 Non-Parametric RDA Example

A car manufacturer wants to make projections about the expected cumulative number of transmission repairs by 24,000 miles for 100 cars of the same model. The manufacturer conducted a preproduction road test and tracked the transmission repairs in a sample of 14 cars.

The following table shows the data.[4]

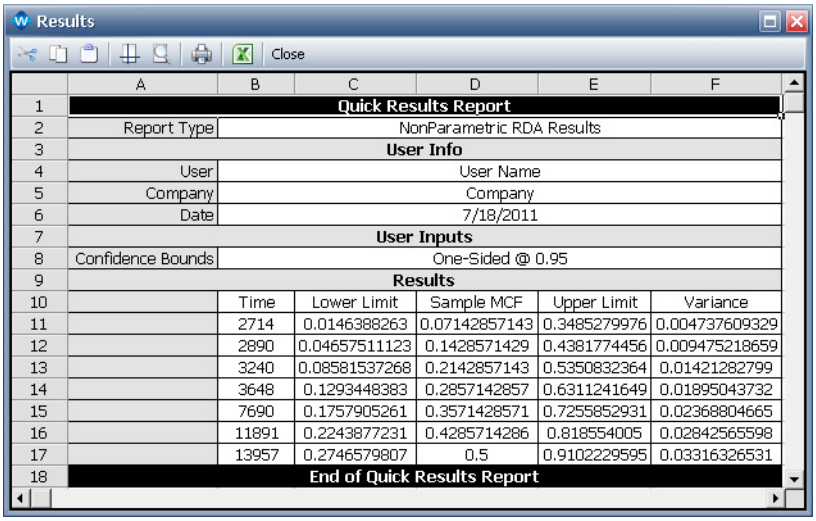
Car ID	Mileage at 1st Repair	Mileage at 2nd Repair	Last Observed Mileage
1	-	-	27099
2	-	-	21999
3	11891	-	27583
4	-	-	19966
5	-	-	26146
6	3648	13957	23193
7	-	-	19823
8	2890	-	22707
9	2714	-	19275
10	-	-	19803
11	-	-	19630
12	-	-	22056
13	-	-	22940
14	3240	7690	18965

1. The first step is to enter the data into a non-parametric RDA folio data sheet, as shown next.



4. Adapted from the example that appears on page 4 in *Recurrent Events Data Analysis for Products Repairs, Disease Recurrences, and Other Applications*, by Wayne B. Nelson, 2003.

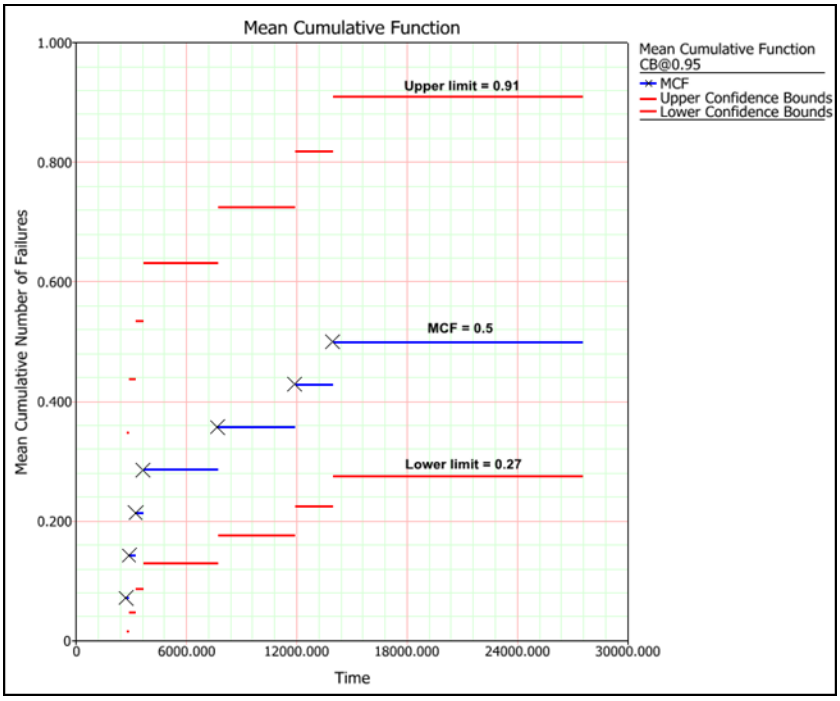
2. Select to calculate the confidence bounds, enter a confidence level of **0.95** and click **Calculate**.
3. Click the **Show Results (...)** button to open the Results window, as shown next.



	A	B	C	D	E	F
1	**Quick Results Report**					
2	Report Type	NonParametric RDA Results				
3	**User Info**					
4	User	User Name				
5	Company	Company				
6	Date	7/18/2011				
7	**User Inputs**					
8	Confidence Bounds	One-Sided @ 0.95				
9	**Results**					
10		Time	Lower Limit	Sample MCF	Upper Limit	Variance
11		2714	0.0146388263	0.07142857143	0.3485279976	0.004737609329
12		2890	0.04657511123	0.1428571429	0.4381774456	0.009475218659
13		3240	0.08581537268	0.2142857143	0.5350832364	0.01421282799
14		3648	0.1293448383	0.2857142857	0.6311241649	0.01895043732
15		7690	0.1757905261	0.3571428571	0.7255852931	0.02368804665
16		11891	0.2243877231	0.4285714286	0.818554005	0.02842565598
17		13957	0.2746579807	0.5	0.9102229595	0.03316326531
18	**End of Quick Results Report**					

The results indicate that for the period between 2,714 and 2,890 miles, the estimated mean cumulative number of repairs per car is 0.0714. For the period between 2,890 and 3,240 miles, the estimate is 0.1429, and so on.

4. To view a plot of the MCF results, close the Results window and click the **Plot** icon on the control panel. The following plot shows the MCF and the confidence bounds. The system ages are on the x-axis and the values for the mean cumulative number of repairs per car are on the y-axis.



The x's in the plot represent the system ages that were entered for the failure events. As you can see, the MCF value of 0.5 applies to the period that is between 13,957 to 27,583 miles (the latest system age entered for failure event F and the latest observed censoring age entered for end event E). Therefore, for a fleet of 100 cars, the

expected number of transmission repairs by 24,000 miles is 50 (i.e., 0.5 x 100). This estimation can be useful for predicting the future number of failures, estimating repair cost and planning for spare parts.

Notice that the MCF data points on the plot grow horizontally farther apart as the population ages. If a smooth curve were drawn through the x's on the plot, the curve would be concave down, indicating a decreasing recurrence rate. This means that the rate of repairs decreases as the cars accumulate more miles. This could also mean that the cars have a manufacturing defect and the recurrence rate will continue to decrease until all defects have been repaired.

13.2 Parametric RDA Folio

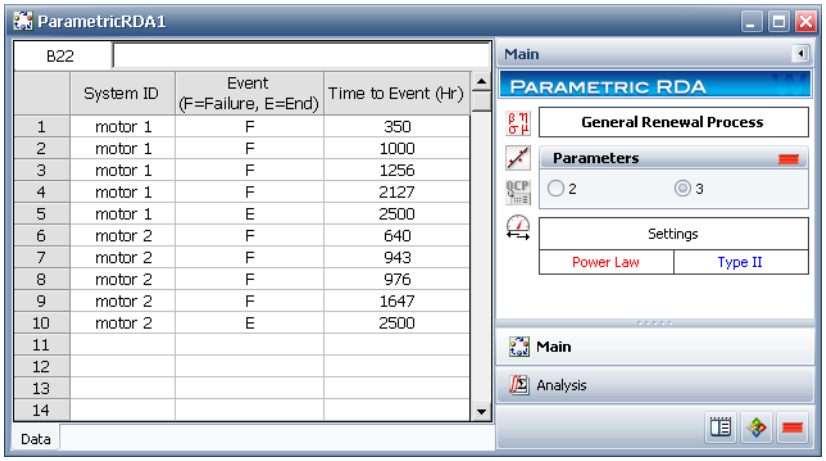
The parametric RDA folio uses the General Renewal Process (GRP) model to analyze the failure behavior of a repairable system. The GRP analysis method takes into account the effectiveness of repairs on the condition of the system. For example, a repair may bring the system to an as-good-as-new condition, to an as-bad-as-old condition or to some stage in between. If the system is only partially rejuvenated after the repair, then this may affect how the system fails in the future. Over time, the recurrence rate of failures may remain constant, increase or decrease. The GRP model allows you to analyze the failure behavior of a partially restored system over time so you can obtain estimates such as the cumulative number of failures, mean time between failures (MTBF) and failure intensity.[5]

13.2.1 Setting up the Folio

To create a parametric RDA folio, choose **Insert > Folios > Parametric RDA** or right-click the **Specialized Folios** folder in the current project explorer and choose **Add Parametric RDA** on the shortcut menu.



The following picture shows an example of a parametric RDA folio data sheet.



The data sheet contains the following information:

- The **System ID** column identifies the system that experienced an event. In the sample data sheet, two units of the same type of motor are being tracked. The results, such as the mean time between failures (MTBF) and failure intensity, will depict the average behavior of the systems.
- The **Event** column describes the type of event. An **F** indicates a failure event and an **E** indicates that the observation period for the unit has ended. Each unique system in the data sheet must have a single end event. If you do not enter an end event for a system, then the end event is assumed to be equal to the last failure time for

5. The ReliaWiki resource portal has more information on the GRP model at: http://www.ReliaWiki.org/index.php/Parametric_Recurrent_Event_Data_Analysis.

that system. In addition, the data set must have at least three failures in order for the software to perform the calculations.

Notice that the repair times of the motors are not recorded in the data sheet. This is because the GRP model assumes that the repair times are negligible and only the effects of the repairs on the condition of the system are of concern. Therefore, each data point in the data sheet can be treated as a single fail-and-repair process.

- The **Time to Event** column indicates the age of the system when the event occurred. For each unique system, the time of the end event (E) must be equal to or greater than the last failure time of that system. The time of the end event is also known as the *censoring age*, because the product's failure or repair history beyond that time is unknown.

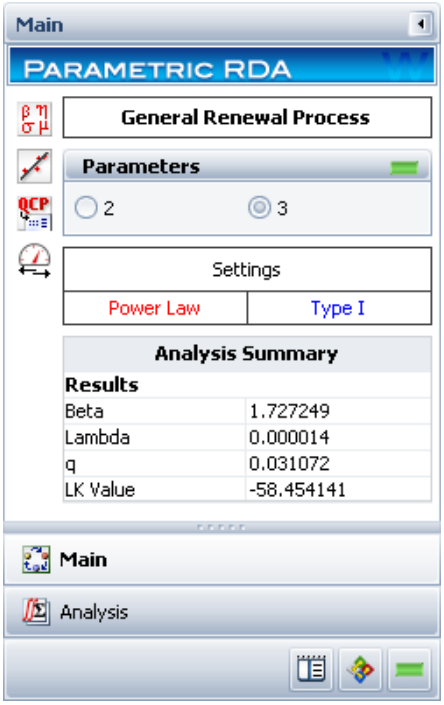
13.2.2 Parametric RDA Folio Control Panel Settings

The parametric RDA folio control panel allows you to configure the analysis settings for the data sheet and view/access the results. It consists of two pages: the **Main** page, which contains most of the tools you will need to perform the analysis, and the **Analysis** page, which includes the settings that are also displayed on the Main page, along with any additional settings that may be applicable for the current data sheet.

13.2.2.1 Control Panel Main Page

The GRP model uses the concept of *virtual age* to mathematically capture the effect of the repairs on the subsequent rate of occurrence of failure (i.e., failure intensity). The rate of occurrence of failure over time is modeled by the power law function. This means that the GRP analysis method has three parameters, two of which are used to describe the power law function and a third for estimating the virtual age of a system.

The Main page of the parametric RDA control panel is shown next.



The following settings are available:

- The **Parameters** area allows you to select how you want to calculate the parameters of the GRP model. If you select **2** parameters, you will be prompted to specify a value for the virtual age parameter based on your own knowledge about the effectiveness of the repairs (this may be the *q* or RF parameter, depending on your default setting). If you select **3** parameters, the software will calculate the virtual age parameter based on the data, giving you an indication of the effectiveness of the repairs as reflected by the data set.
- The **Settings** area displays "Power Law" to indicate that the power law function will be used in the analysis. This setting cannot be changed; however, you have the option to select which virtual age model will be used in the analysis. Click the option displayed in blue text on the control panel to toggle between the available virtual age models.
- The **Analysis Summary** area displays the calculated results. You can click anywhere within the Analysis Summary area to open the Results window, which displays the calculated results in a worksheet. From the Results window, you can edit, copy or print the results.

Folio Tools

The folio tools are arranged on the left side of the Main page of the control panel:



Calculate estimates the parameters of the GRP model, based on the current data set. This tool is also available by choosing **Recurrent Event Data > Analysis > Calculate**.



Plot creates a new sheet in the folio that provides a choice of applicable plot types. In parametric RDA folios, this includes cumulative failures vs. time, failure intensity vs. time, etc. *See Section 13.2.4 on page 384*. This tool is also available by choosing **Recurrent Event Data > Analysis > Plot**.



QCP opens the Quick Calculation Pad, which allows you to calculate results based on the analyzed data sheet, such as the number of failures and mean time between failures (MTBF). *See Section 13.2.3 on page 383*. This tool is also available by choosing **Recurrent Event Data > Analysis > Quick Calculation Pad**.



Change Units opens the Change Units window, which allows you to change the units for the time values in the current data sheet. *See page 113 in Chapter 3*.

13.2.2.2 Control Panel Analysis Page

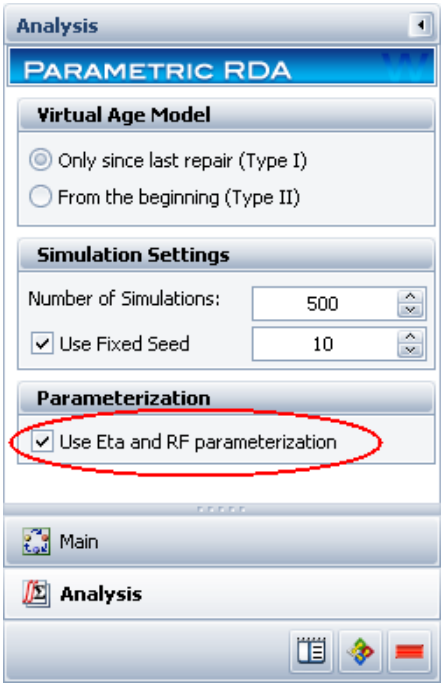
The following settings are available on the Analysis page of the parametric RDA control panel:

- The **Virtual Age Model** area allows you to select one of two ways that the GRP model can calculate the virtual age after each repair:[6]
  - **Only since last repair (Type I)** assumes that the repairs will remove some portion of the damage that has accumulated since the last repair.
  - **From the beginning (Type II)** assumes that the repairs will remove a portion of all the damage that has accumulated since the system was new.

  Since it may not be feasible to classify real-world repair situations into one of these two abstract categories, the virtual age type is usually selected based on what provides a better statistical fit for the data. The model fit can be assessed by using the likelihood (LK) value that is displayed in the **Analysis Summary** area on the control panel, where the higher value indicates the better fit.

6. The ReliaWiki resource portal explains the mathematical concept behind the two models at: http://www.ReliaWiki.org/index.php/Imperfect_Repairs. (Note that the article is written for ReliaSoft's BlockSim software, but the theory remains the same.)

- The **Simulation Settings** are for the Monte Carlo simulation. In general, the application uses the failure times supplied by the data set in order to calculate virtual age. However, when extrapolating for results that are outside of the range of observations, the failure times are unknown. Therefore, unless the virtual age parameter $q = 1$, there are no analytical solutions available for metrics such as total failure number and failure intensity. In this case, the solution can be obtained only through Monte Carlo simulation.
  - In the **Number of Simulations** field, specify the number of data points to generate in the simulation.
  - The **Use a Fixed Seed** option is an optional setting that allows you to specify a starting point from which random numbers are generated in the simulation. The same random numbers and, therefore, the same simulation results will be generated when the same seed value and number of data points are used.
- The **Parameterization** option allows you to select how you want the parameters to be calculated. By default, the software calculates for the beta, lambda and q parameters. However, if you want to publish the model and use it in BlockSim, then you need to express the power law parameter, lambda, in terms of the Weibull parameter, eta, and express the virtual age parameter, q, in terms of the RF parameter (which users of BlockSim will recognize as the *restoration factor* of a maintenance action).[7] To do this, select the check box in the **Parameterization** area shown next.



The relationship between the power law parameter, lambda (λ), and the Weibull parameter, eta (η), can be described as:

$$\eta = \frac{1}{\lambda^{1/\beta}} \quad \text{or} \quad \lambda = \frac{1}{\eta^{\beta}}$$

The relationship between the q and RF parameters is described as:

$$RF = 1 - q$$

7. For more information on ReliaSoft's BlockSim software, please see http://BlockSim.ReliaSoft.com.

The RF parameter indicates the degree to which the condition of a system will be restored after each repair. Parameter *q* indicates the opposite. A value between 0 and 1 indicates the extent of the repair, where:

- An RF = 1 indicates that the system will be "as good as new" after the repair. This is also called *perfect repair*.
- An RF value between 0 and 1 indicates that the system will be "better than old but worse than new" after the repair. This is also called *imperfect repair*.
- An RF = 0 indicates that the system will be "as bad as old" (i.e., no improvement) after the repair. This is also called *minimal repair*.

13.2.3 QCP Calculations for Parametric RDA

The parametric RDA folio includes a Quick Calculation Pad (QCP) for computing useful metrics. You can access the QCP by choosing **Recurrent Event Data > Analysis > Quick Calculation Pad** or by clicking its icon on the Main page of the control panel.



To perform a calculation, select the appropriate option and enter any required inputs in the **Input** area, then click **Calculate**. *For more detailed information on how to use the QCP in general, see page 219 in Chapter 6*.

The following calculations are available for parametric RDA folios:

Failure Intensity

- **Cumulative Failure Intensity**

  Calculates the average rate of occurrence of failures from time = 0 up to a specified period of time. Enter the time period in the **Time** field. For example, a cumulative failure intensity of 0.15 for a time period of 100 hours means that, on average, 0.15 failures occurred every hour over the 100-hour period.

- **Instantaneous Failure Intensity**

  Calculates the rate of occurrence of failures over a small interval *dt* that begins at a specified time. Enter the time in the **Time** field. For example, an instantaneous failure intensity of 0.15 at 100 hours means that, over the next small interval *dt* that begins at 100 hours, the rate of occurrence of failures is 0.15 failures per hour.

MTBF (Mean Time Between Failures)

- **Cumulative MTBF**

  Calculates the average MTBF from time = 0 up to a specified end time. Enter the end time in the **Time** field. For example, a cumulative MTBF of 5 hours from 0 to 100 hours means that, on average, the time between failures was 5 hours over the 100-hour period.

- **Instantaneous MTBF**

  Calculates the average MTBF over a small interval *dt* that begins at a specified time. Enter the time in the **Time** field. For example, an instantaneous MTBF of 5 hours at 100 hours means that, over the next small interval *dt* that begins at 100 hours, the average MTBF will be 5 hours.

> **Note:** MTBF is used as a metric for the analysis of repairable components. On the other hand, the term *mean time to failure* (MTTF) is used as a metric for *non-repairable* systems, where components are discarded or replaced upon failure. To analyze non-repairable system data in Weibull++, use the Weibull++ standard folio.

Failures

- **Cumulative Number of Failures**

  Calculates the cumulative number of failures to be expected for a specified interval of 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 be estimated to fail 10 times between 100 and 500 operating hours.

- **Failures by System**

  Calculates the cumulative number of failures to be expected for each system for a specified interval. The start time of the interval is equivalent to the time when the observation period for the system ended (indicated by an E in the folio data sheet). If you did not enter an end time for a system, then the start time of the interval is assumed to be equal to the last failure time for that system. Enter the duration of the interval in the **Mission Additional Time** field and then click **Calculate** to open the Results window, which shows the cumulative number of failures for each system, as well as the overall number of failures.

Probability

- **Conditional Reliability**

  Calculates the probability of the system successfully completing a mission, given that it may or may not have successfully completed a previous mission. Enter the start time of the new mission in the **Mission Start Time** field and enter the duration of the new mission in the **Mission Additional Time** field.

  For example, a system failed three times during its first three years of operation. The probability that the system will operate for an additional six months without failure, or the conditional reliability for one half of a year given that the system is three years old, may be 30%.

- **Conditional Probability of Failure**

  Calculates the probability that the system will fail a mission, given that it may or may not have successfully completed a previous mission. Enter the start time of the new mission in the **Mission Start Time** field and enter the duration of the new mission in the **Mission Additional Time** field.

  For example, a system failed three times during its first three years of operation. The probability that the system will fail in the next six months, or the conditional probability of failure for one half of a year given that the system is three years old, may be 70%.

13.2.4 Parametric RDA Folio Plots

Weibull++ includes several plot types you can use to visualize the results of your parametric RDA analysis. You can create plots by choosing **Recurrent Event Data > Analysis > Plots** or by clicking the icon on the Main page of the control panel.



The features on the plot sheet are similar to the options available for all other Weibull++ plots. Features that are not applicable to parametric RDA analysis will be hidden or disabled. *To learn how to customize a plot, see Chapter 15*.

The following is a description of the plots types that can be created in a parametric RDA folio:

- **Cumulative Number of Failures** plots the failure times in the x-axis and the cumulative number of failures in the y-axis. The points represent the actual failure times in the data set and the line represents the expected cumulative number of failures based on a simulation using the calculated model parameters. This gives you an indication of how the number of failures is increasing over time.
- **Cumulative Failure Intensity vs. Time** plots the average rate of occurrence of failures over a period of time. The points represent the cumulative failure intensity calculated at each failure time in the data set and the line

represents the cumulative failure intensity based on a simulation using the calculated model parameters. This shows how the rate of occurrence of failures increases, decreases or remains constant for that period of time.

- **Instantaneous Failure Intensity vs. Time** plots the rate of occurrence of failures over several instances of time. The line represents the instantaneous failure intensity based on a simulation using the calculated model parameters. This plot may be used to show how the rate of occurrence of failures has changed at a particular point in time.
- **Conditional Reliability vs. Time** and **Conditional Unreliability vs. Time** shows the reliability or probability of failure over intervals of time. There are two options:
  - **Vary Mission Time** shows how the reliability or probability of failure would vary over different mission times if the system operated at a specific **Start Time**. The start time is an assumed value for the system age at the beginning of the mission. For example, a start time = 100 hours means that the origin of the plot corresponds to a system with a total age of 100 hours after a mission of 0 hours. Therefore, an x-axis value of 500 mission hours on that plot would correspond to a system with a total age of 600 hours after a mission of 500 hours.
  - **Vary Start Time** shows how the reliability or probability of failure would vary over different start times if the system operated within a specific **Mission Time**. The mission time is an assumed value for additional age a system will accumulate during the mission. For example, a mission time = 100 hours means that the origin of the plot corresponds to a system with a total age of 100 hours after a mission of 100 hours. Therefore, an x-axis value of 500 hours on that plot would correspond to a system with a total age of 600 hours after a mission of 100 hours.

13.2.5 Parametric RDA Example

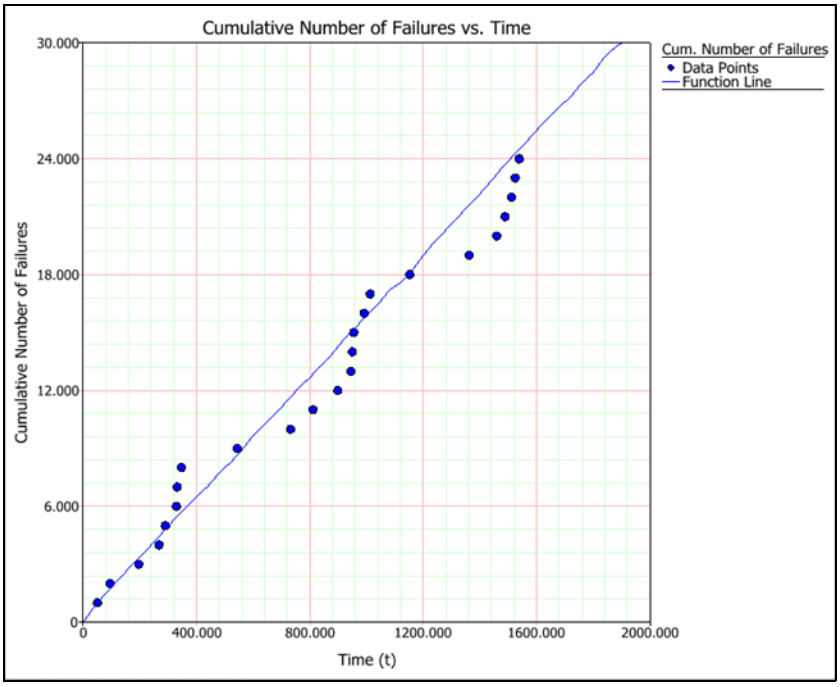
An analyst recorded the successive failures of the air-condition unit of an aircraft. The objective is to obtain the expected number of failures that may occur by 1,800 hours. The following data sheet shows the failure logs of the unit between 0 to 1,539 hours of observation. The observation period ended at the time of the last failure.

	System ID	Event (F=Failure, E=End)	Time to Event (Hr)
1	Unit A	F	50
2	Unit A	F	94
3	Unit A	F	196
4	Unit A	F	268
5	Unit A	F	290
6	Unit A	F	329
7	Unit A	F	332
8	Unit A	F	347
9	Unit A	F	544
10	Unit A	F	732
11	Unit A	F	811
12	Unit A	F	899
13	Unit A	F	945
14	Unit A	F	950
15	Unit A	F	955
16	Unit A	F	991
17	Unit A	F	1013
18	Unit A	F	1152
19	Unit A	F	1362
20	Unit A	F	1459
21	Unit A	F	1489
22	Unit A	F	1512
23	Unit A	F	1525
24	Unit A	F	1539
25	Unit A	E	1539

1. The first step is to determine which virtual age model provides the best statistical fit for the given data. On the Main page of the control panel, select **3** for the parameters settings and then calculate the results for each of the virtual age models. Use the default settings for the simulation.

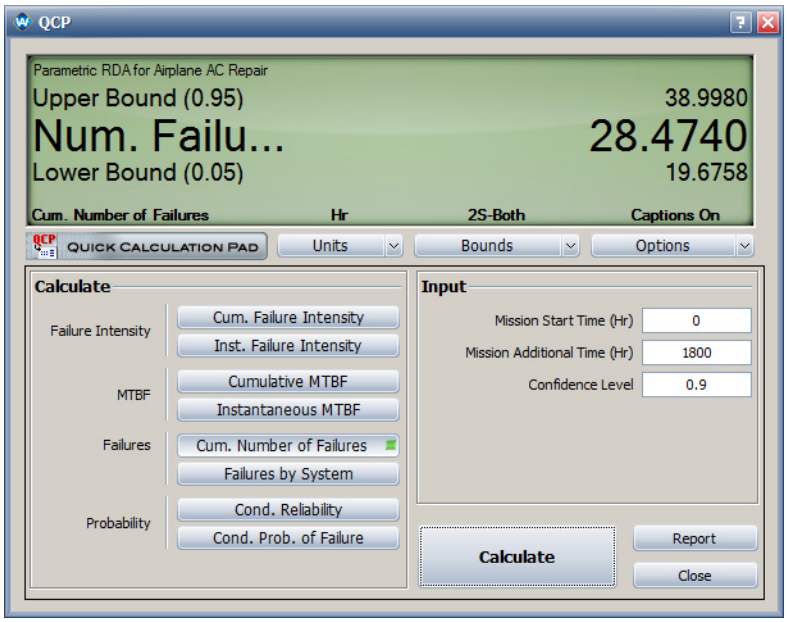
   The LK value for the Type I model is -123.6347 while the LK value for the Type II model is -123.5964. Therefore, for this data set, the Type II model may provide the best fit (i.e., have a slightly larger LK value). The parameters of the Type II GRP model are estimated to be beta = 0.8276, lambda = 0.0389 and q = 0.2758.

2. Click the **Plot** icon to visualize how the number of failures increases over time, as shown next.



3. Next, predict the number of failures expected to occur by 1,800 hours. Click the **QCP** icon on the control panel. In the QCP, choose to calculate the **Cumulative Number of Failures** with **two-sided** confidence bounds. Select **Hour** for the time units and then make the following inputs:
   - Mission Start Time: **0**
   - Mission Additional Time: **1800**
   - Confidence Level: **0.9**

   Click **Calculate**. The cumulative number of failures in 1,800 hours is 28.4740 or about 28 failures, and the number may be as high as about 39 failures, as shown next.



Plots 14

Plots are available in standard folios and all specialized folios except the Weibull++ Event Log folio, as well as in diagrams, in the Non-Linear Equation Fit Solver and in SimuMatic. In all cases, these plots can be viewed by clicking the **Plot** icon located on the main page of the control panel. You can also choose **Life Data > Analysis > Plot** (in the case of Weibull++ folios), **ALTA > Analysis > Plot** (in the case of ALTA folios) or **Diagrams > Analysis > Plot** (in the case of diagrams).



If you click the **Plot** icon before the data sheet or diagram has been calculated or analyzed, it will automatically be calculated or analyzed and then the data will be plotted in the plot sheet. You can add more plot sheets to a standard folio by choosing **[Life Data/ALTA] > Folio Sheets > Insert Additional Plot**.



These additional plot sheets can also function as overlay plots to place multiple data sheets from the current folio on a single plot.

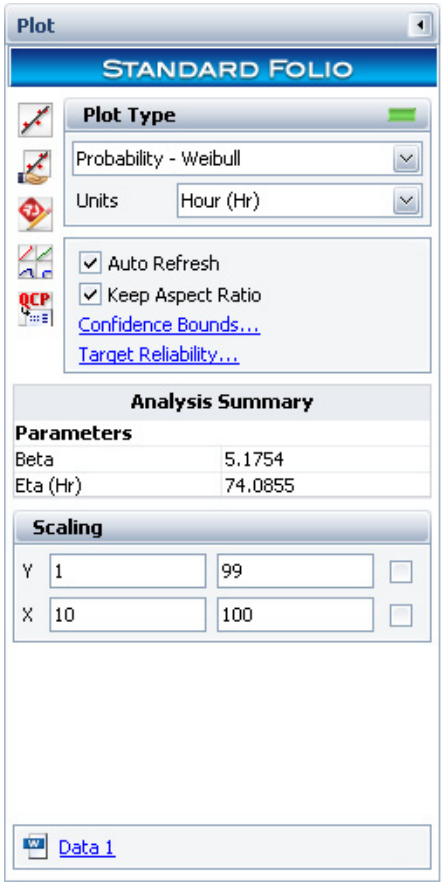
Note that the **Plot** icon on the plot tab control panel refreshes the plot. If the data set has been changed since the data sheet or diagram was last calculated or analyzed, you must recalculate the data before refreshing the plot.) The status light on the control panel indicates when the plot is out of sync with the data sheet. 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 > Plot Actions > Show/Hide Plot Items**, except in the case of SimuMatic plots, which have display options on the plot sheet control panel. *See Section 14.3 on page 393*.

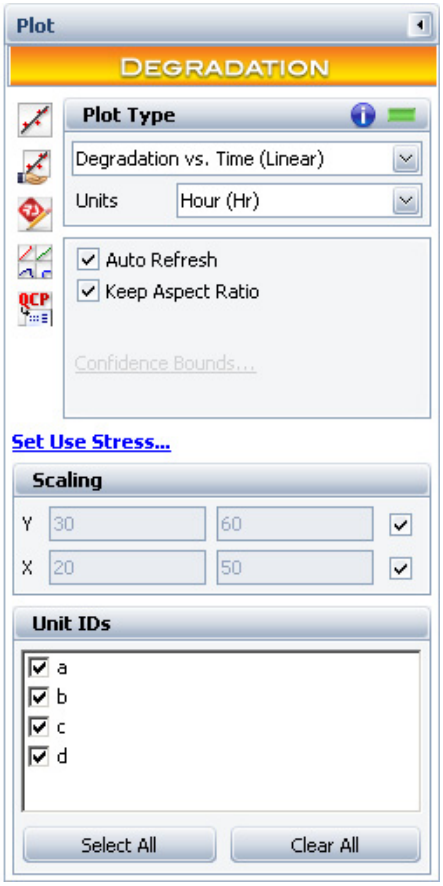
Note: You can copy the plot graphic to the Clipboard for use in other locations. Choose **Plot > Copy Plot Graphic** and select an image type. You can copy the plot as a bitmap image or as a metafile image optimized for pasting into either a Synthesis spreadsheet (e.g., analysis workbook) or an external 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. *See page 587 in Chapter 30*.

14.1 Plot Sheet Control Panel

The control panel in the plot sheet allows you to control the options that go along with the plot. The options available in the control panel vary depending on the type of folio or tool that the plot is associated with.

The control panels shown below are from a Weibull++ standard folio and an ALTA degradation analysis folio, and illustrate most of the available options, all of which are presented next. A table showing the available options for each folio or tool follows.

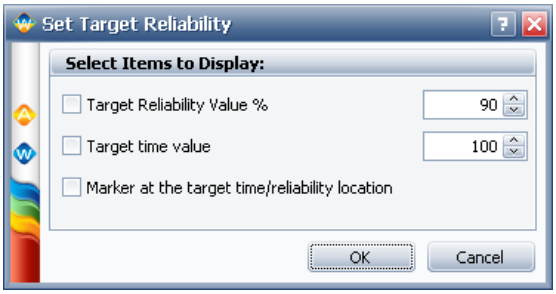




Note: Unless otherwise stated, all options apply to both Weibull++ and ALTA.

- 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 data sheet. *For more information about the available plot types, refer to the specific topic covering the data sheet type or diagram*.
- **Units** allows you to choose which units you want the plot to show from the drop-down menu.
- **Auto Refresh** automatically updates the plot to reflect any changes that have been made. If not selected, you must click the **Redraw Plot** icon to refresh the display.

- **Keep Aspect Ratio** keeps the proportion of the horizontal size to vertical size constant when you resize the plot. Note that changing this option will also change it in the Page Setup window for the plot. *See page 581 in Chapter 29*.
- **Logarithmic Axes** plots the function on a log-log paper.
- **Confidence Bounds** opens the Confidence Bounds Setup window, which allows you to specify the type of confidence bounds you would like to show on your plot. *See Section 14.2 on page 392*.
- **Target Reliability** opens the Set Target Reliability window, as shown below. This window allows you to specify whether and how the target reliability will be shown on the plot.



  - **Target Reliability Value, %** allows you to enter a reliability value in percent for the target point. If the check box is selected, this value will be displayed on the plot as a horizontal line.
  - **Target time value** allows you to enter a time value for the target point. If the check box is selected, this value will be displayed on the plot as a vertical line.
  - **Marker at the target time/reliability location** if selected, the target point (the intersection of the target reliability value and the target time value) will be circled on the plot.
- **Life Lines** ALTA Only

  Opens the Specify Life Lines window, which allows you to enter up to 10 unreliability values for each additional life line you would like to see plotted on life vs. stress plots. *See Section 14.5 on page 395*.
- **Contours Setup** WEIBULL++ Only

  This option is available only for Contour plots. It opens the Contours Setup window, which allows you to select the contour lines to be shown, the confidence levels represented by the lines and the resolution. If an error in calculating the selected contour lines is encountered, this button will turn red. Use the Contours Setup window to adjust the settings. *See Section 14.4 on page 394*.
- **Adjust MLE points on plot line** WEIBULL++ Only

  This option is available only if the data sheet uses the MLE analysis method. If selected, it adjusts the y-axis plotting positions so that the data points are plotted directly on the MLE line (i.e., at the position where the failure time intersects the MLE line). The solution of the MLE line is obtained from the likelihood function and not from the plotting positions of the points; therefore, this plot setting does not affect the parameter estimates.
- **Analysis Summary** displays calculated results in the Results window. The location of this option varies depending on the folio you are working with.
- **Set Use Stress** ALTA Only

  Opens the Use Stress Level window, which allows you to enter a use stress value for the selected stress type. *See page 260 in Chapter 7*.

- **Scaling**
  - **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 application will automatically choose appropriate values for the range.
- **Unit IDs** is available only for Degradation vs. Time plots in degradation folios. It allows you to select the Unit ID(s) to be shown on the plot. To refresh the plot to show the currently selected ID(s), click the **Redraw Plot** icon.
  - **Select All** selects all the Unit ID(s) to be placed on the plot.
  - **Clear All** clears all selected Unit ID(s).

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, line styles and point styles. *See Chapter 15*.



RS Draw launches ReliaSoft Draw, which allows you to annotate your plot and view your plot in greater detail. *See Chapter 16*.



Side-by-Side Plots adds a side-by-side plot to the project, which allows you to view different plots in a single window, and associates the current folio and data sheet with the side-by-side plot. If no side-by-side plots exist in the project, selecting this option creates one. Otherwise, a window will appear which allows you to select an existing side-by-side plot to associate the data sheet with or to create a new one. *See Section 14.7 on page 397*.



QCP opens the Quick Calculation Pad (QCP), which allows you to obtain reliability results based on the currently active folio. *See page 219 in Chapter 6*.

The following table displays the available Weibull++ plot sheet control panel options for each folio or tool.

	Standard Folio	Warranty	Degradation	Non-Parametric LDA	Non-Parametric RDA	Parametric RDA	Diagram	Non-Linear Solver	SimuMatic	Test Design Folios
Refresh Plot	X	X	X	X	X	X	X	X	X	X
Plot Setup	X	X	X	X	X	X	X	X	X	X
Edit with RS Draw	X	X	X	X	X	X	X	X	X	X
Export Plot Graphic	X	X	X	X	X	X	X	X	X	X
Side-by-Side Plots	X	X	X							
QCP	X	X	X			X				
Plot Type	X	X	X			X	X			
Show Results				X	X					
Auto Refresh	X	X	X			X	X		X	X
Keep Aspect Ratio	X	X	X	X	X	X	X	X	X	X
Confidence Bounds	X	X	X			X	X		X	
Target Reliability	X	X	X							
Analysis Summary	X	X	X			X				
Scaling Area	X	X	X	X	X	X	X	X	X	X
Unit/Subset IDs			X							
Save Plot									X	
Get Failure from Folio										X

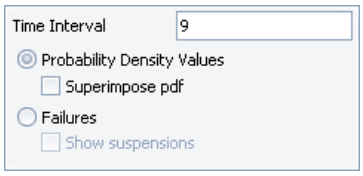
The following table displays the available ALTA plot sheet control panel options for each folio or tool.

	Standard Folio	Degradation	Stress Profiles	SimuMatic
Refresh Plot	X	X	X	X
Plot Setup	X	X	X	X
Edit with RS Draw	X	X	X	X
Export Plot Graphic	X	X	X	X
Side-by-Side Plots	X	X		
QCP	X	X		
Plot Type	X	X		
Auto Refresh	X	X		X
Keep Aspect Ratio	X	X	X	X
Confidence Bounds	X	X		X
Life Lines	X	X		
Target Reliability	X	X		
Analysis Summary	X	X		
Set Use Stress	X	X		
Scaling Area	X	X	X	X
Unit/Subset IDs		X		
Save Plot				X

F/S Histogram Options

WEIBULL++ Only

When you are working with a failures/suspensions histogram plot in Weibull++, the following additional options are available in the Plot Sheet control panel.



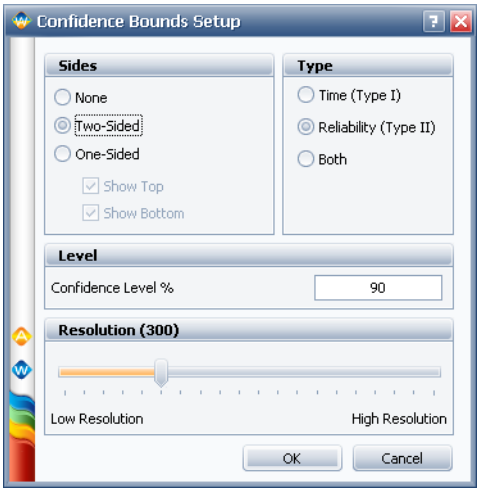
- **Time Interval** allows you to specify the length of the intervals used on the plot.
- Select one of the following options:
  - **Probability Density Values** to display the probability density of the failures. In addition, the following additional option will become available.
    - Select **Superimpose pdf** to superimpose the probability density function line on the bars.
  - **Failures** to plot the failures as raw numbers. In addition, the following additional option will become available.
    - Select **Show suspensions** to display the suspensions on the plot.

14.2 Setting Confidence Bounds

You can include and specify the confidence bounds on plots associated with standard folios, degradation folios, warranty analysis folios and parametric RDA folios 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 not available if the data type is free-form data.

- **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. Note that not all options are available with every plot.
  - **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 Probability plots and Unreliability vs. Time plots
  - **Both** displays both types.

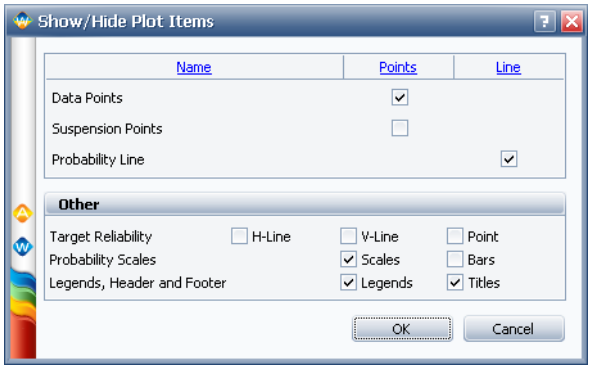
- **Bounds On** ALTA Only

  This option is available only for the life vs. stress plot type. It allows you to select to display confidence bounds on the scale parameter, mean line or both.

- **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.

14.3 Show/Hide Plot Items Window

For all plot sheets except those associated with SimuMatic, the Show/Hide Plot Items window allows you to select the data you want displayed on the plot. The appearance of this window will vary depending on the plot type and the data type used, but it will function in the same way regardless.

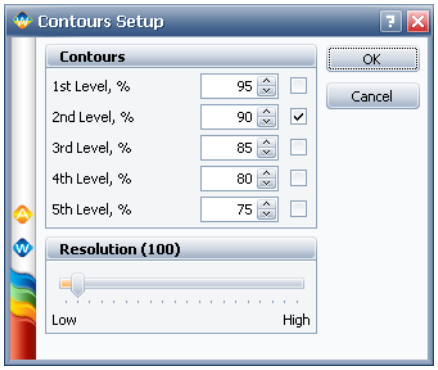


- **Subset** displays the available data sheets and/or plot items.
- **Name** displays each item (e.g., data points, function line, contour, etc.) available for display on the plot.
- **Points/Line/Bars/Slices** allows you to select how the values for the item are displayed (e.g., remove certain slices from a pie chart, choose to display intervals using points, lines or both, etc.). Clicking the header of this column will toggle between selecting all options in the column and clearing all options in the column.
- **Other**
  - **Target Reliability** if the target reliability option is available for the plot, the following options allow you to choose how it will be displayed.
    - **H-Line** displays the horizontal line through the target reliability value.
    - **V-Line** displays the vertical line through the target reliability value.
- **Point** displays the target reliability value as a point.
  - **Probability Scales** displays the probability scales on the plot.
    - **Bars** encloses the probability scales in bars.
  - **Legends, Header and Footer**
    - **Legends** displays all legends on the plot.
    - **Titles** displays the Header and Footer titles on the plot.

14.4 Contour Plots

WEIBULL++ Only

Contour Plots are available only for the 2- and 3-parameter Weibull, normal and lognormal distributions in Weibull++. The first time you select a **Contour Plot** from the **Plot Type** drop-down list, the Contour Setup window will appear. This window allows you to select: a) the number of contour lines to be shown, b) the confidence level represented by each selected line and c) the resolution used to draw the lines.



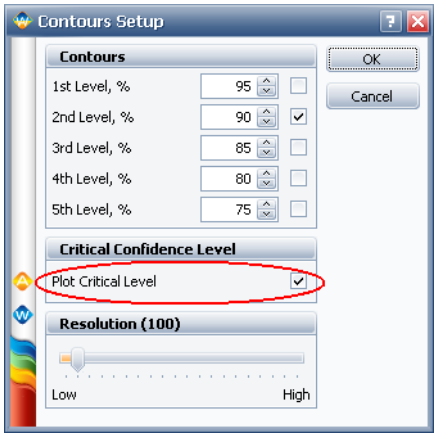
For existing contour plots, you can access the Contours Setup window by clicking the **Contours Setup** link on the control panel of the plot sheet.

To change the level of detail at which the contours are plotted (and the smoothness of the lines), adjust the **Resolution** slider. You can click and drag the slider or, for fine adjustments, click the slider and then use the arrow keys. The number of points plotted to create the lines is displayed beside the field name. Note that higher resolutions will take longer to plot.

If an error in calculating the selected contour lines is encountered, the Contour Setup window will expand to include an error notification at the bottom of the window. In addition, the **Contours Setup** link on the plot sheet control panel will turn red.

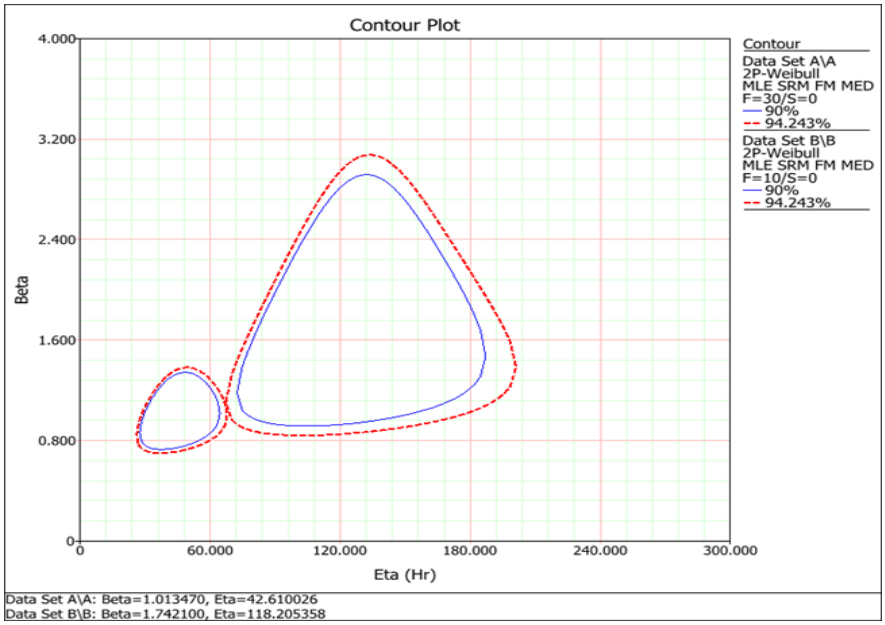
Comparing Data Sets

In Weibull++, contour plots are typically used to compare data sets. For two contour plots that are superimposed in an overlay plot, the **Plot Critical Level** check box will be available, as shown next. This allows you to calculate and display the minimum confidence level at which the two contour plots intersect.



At any confidence level below the critical level, the contour plots will not overlap and there will be a statistically significant difference between the data sets at those levels.

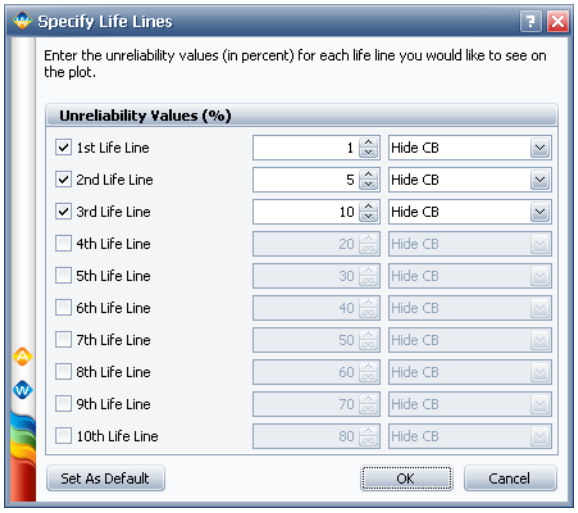
The following example shows two contour plots in an overlay plot. The solid contour lines represent the 90% confidence level while the dashed contour lines represent the critical confidence level, which is calculated to be 94.243%. (The appearance of the lines in the example were modified via the Plot Setup to make the plot easier to interpret.) Note that due to the calculation resolution and plot precision, the contour lines at the calculated critical level may appear to overlap or have a gap.



14.5 Specifying Life Lines

ALTA Only

You can specify additional life lines to be plotted on life vs. stress plots in ALTA by clicking the **Life Lines** link on the plot sheet control panel. The Specify Life Lines window will open.



This window allows you to define up to ten additional life lines to be shown on life vs. stress plots. For each life line, enter the unreliability value. You can enter values manually or use the up and down arrows beside each field to increase/decrease the value. You can keep previously entered values without plotting them by simply clearing the appropriate check boxes. You can select to show or hide confidence bounds for each life line.

To use your current settings by default on life vs. stress plots, click the **Set As Default** button. You can reset these settings by clicking **Reset Application Settings** on the Reset Settings page of the Application Setup. *See page 601 in Chapter 30.*

14.6 Overlay Plots

Overlay plots give you the ability to display results from multiple data sheets in a single plot. This allows you to easily compare different data sets, analysis methods or distributions. For example, you may wish to show the reliability plots of two product designs in the same plot or compare a simulation-based data set with actual data obtained from fielded products.

You can compare any data set from Weibull++ and ALTA standard folios, warranty folios and degradation folios. Note that you can add data sets from parametric RDA folios to an overlay plot but you can compare them only with other parametric RDA data sets. You can select up to a maximum of 20 data sheets to add to the same plot.

Before using an overlay plot, you must first calculate the parameters of the data sets. Once that is completed, you can add an overlay plot to either the project or the folio:

- To add an overlay plot to a project, choose **Insert > Reports and Plots > Overlay Plot**.



  All the calculated data sheets in the project will be available for inclusion in the overlay plot.

- To add an overlay plot to a Weibull++ or ALTA standard folio, choose **[Life Data/ALTA] > Folio Sheets > Insert Additional Plot**.



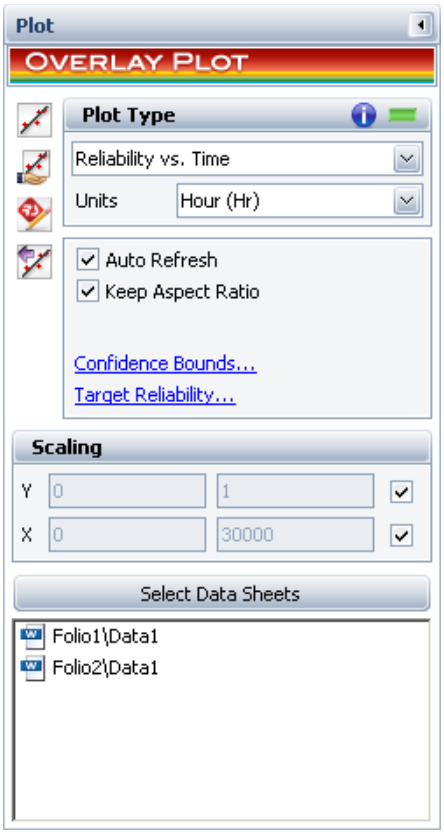
  Only the calculated data sheets in the current folio will be available for inclusion in the overlay plot.

What's Changed? In previous versions of the software, overlay plots were known as MultiPlots.

14.6.1 Overlay Plot Control Panel

The control panel in an overlay plot is similar to the control panel in standard plots, except for two things:

- The **Select Data Sheets** button gives you the ability to add or remove data sets from the plot.
- Probability plots, contour plots and ALTA stress plots are available only if the data sets have been calculated with the same distribution. This is because the scales or axes in those types of plots vary for different types of distributions.



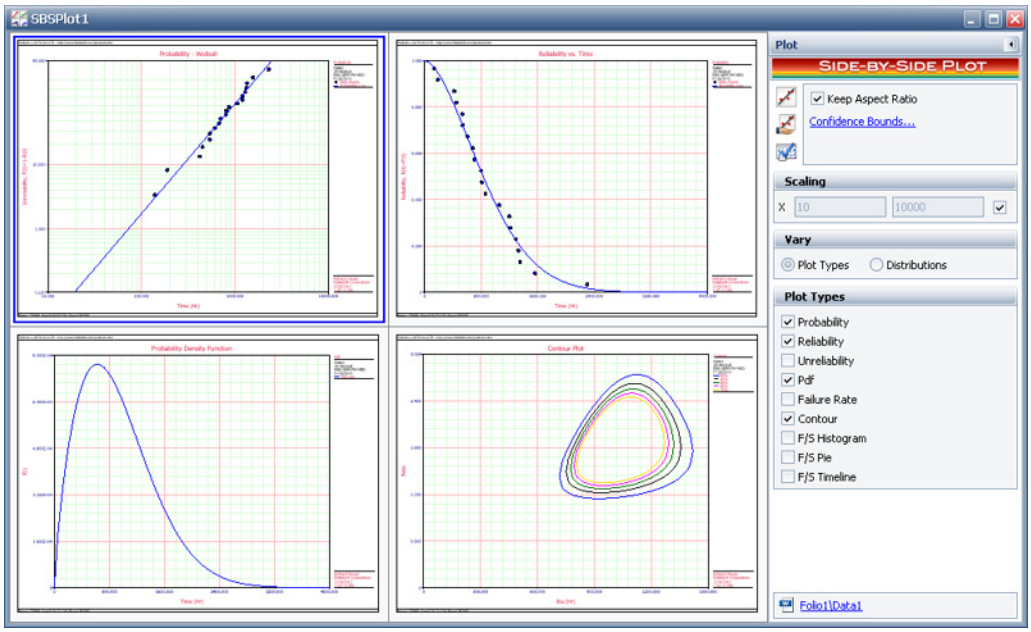
14.7 Side-by-Side Plots

Side-by-side plots allow you to display different plots for a single data set all in a single window for easy comparison. In Weibull++, you can select to plot the data set using multiple distributions or plot types. In ALTA Standard and ALTA PRO, you get a third option to plot the data set using multiple life-stress relationship models.

To add a side-by-side plot to a project, choose **Insert > Reports and Plots > Side-by-Side Plot** or click the icon on the control panel of the folio plot sheet.



To view a single plot in greater detail, double-click the plot. You can double-click the plot again to return to the side-by-side view. The following picture shows an example of a side-by-side plot in Weibull++. Note that any cosmetic changes you make to a plot will be applied to all plots in the side-by-side view.



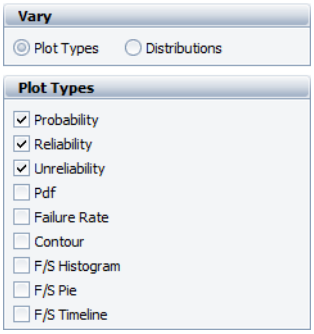
14.7.1 Side-by-Side Plots Control Panel

The control panel in a side-by-side plot is similar to the control panel in standard plots, except for two things:

- You can select which data sheet to plot by clicking **Select Data Sheet** on the control panel.



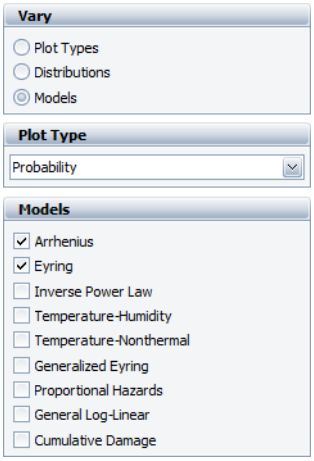
- You have two choices for displaying the plots. You can either:
  - Display several plot types for the selected data set. To do this, select the **Plot Types** option in the **Vary** area of the control panel, and then select the check box for each plot type of interest, as shown in the following example for Weibull++. (Note that for degradation and warranty folios, the plots are based on the extrapolated data set.)



  - Display several instances of the same plot type, where each instance shows the data set fitted with a different life distribution/model. To do this, select the **Distributions** option in the **Vary** area of the control panel. Select one plot type, and then select the check box for each distribution you wish to plot. Note that regardless of your

choices, the side-by-side plot will display only those distributions that can be successfully calculated. You will see notification messages for any unsuccessful calculations.

ALTA includes a third option to vary the **Models**, as shown in the example below. If you select to vary the models, each plot will use the life distribution currently used in the selected data sheet in combination with the selected models. Similarly, if you select to vary the distributions, each plot will use the life-stress relationship currently used in the selected data sheet in combination with the selected life distributions.



14.8 3D Plots

For standard folios, ReliaSoft's 3D Plot utility allows you to visualize 3-dimensional functions, such as a surface plot of the likelihood function. You can rotate the plot both vertically and horizontally to examine it from any angle.

- In Weibull++, the 3D plot is a surface plot of the likelihood function. It plots the likelihood values (LK values) with respect to the estimated parameters of a distribution. The shape of the plot may help you to evaluate the amount of variability in the parameter estimates. A small surface (tall, tower-like peak) suggests that there is less uncertainty in the parameter estimates at the maximum LK value, while a large surface suggests that there is a large uncertainty in the parameter estimates. You can create a 3D plot of the likelihood function for any of the available distributions in Weibull++ except for 1-parameter Weibull, mixed Weibull, Bayesian-Weibull or Weibull using competing failure modes distribution.

  To create a 3D Plot for a Weibull++ folio, choose **Life Data > Analysis > 3D Plot**.



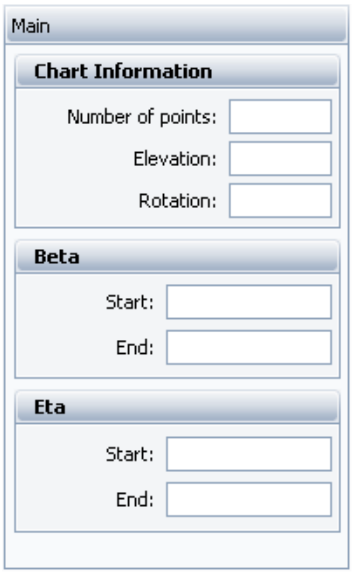
- In ALTA, the 3D plot provides a choice of several functions that you can plot with respect to time and stress, in addition to the likelihood function plot. This includes the *pdf*, failure rate, reliability and unreliability functions. The shape of the plot may help you to evaluate how the function is influenced by time and stress.

  To create a 3D Plot for an ALTA folio, choose **ALTA > Analysis > 3D Plot**.

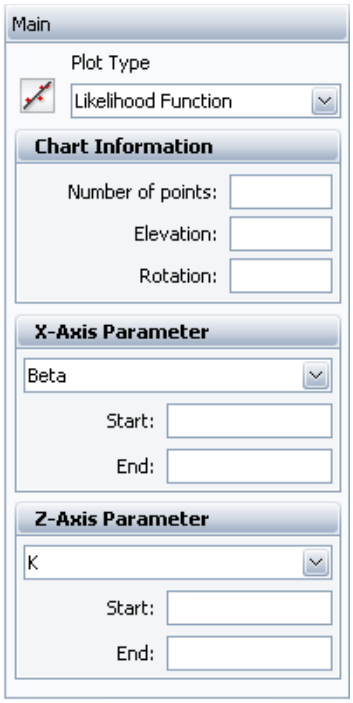


14.8.1 3D Plot Control Panel

The 3D Plot window contains a control panel, which allows you to customize the axes of the plot. In the pictures below, the Weibull++ control panel is shown on the left and the ALTA control panel is shown on the right.



Weibull++ Version



ALTA Version

The control panel contains the following settings:

- **Plot Type** ALTA Only

  Select which plot type you want to view in the 3D plot sheet from the drop-down list. The available 3D plots include the likelihood function plot, the *pdf* plot, the failure rate plot, the reliability plot and the unreliability plot.

- **Chart Information** allows you to customize the resolution, elevation and rotation of the plot.
  - **Number of points** sets the resolution for the plot. You must have at least 10 points; however, 40 or more points will result in a slow re-draw.
  - **Elevation** measures the rotation in degrees (0°-90°) about the vertical plane.
  - **Rotation** measures the rotation in degrees (0°-360°) about the horizontal plane.

- **Axis Parameters** ALTA Only

  Allows you to select the parameters that you want the x-axis and the z-axis to represent and to modify the starting and ending values of the axes. The parameters available in the drop-down box will depend on the distribution used to calculate the parameters in the current data sheet.
  - **Start** shows the low (starting) value on the axis.
  - **End** shows the high (ending) value on the axis.

- **Axes** WEIBULL++ Only

  Allows you to modify the starting and ending values of the plot. The starting and ending values of the axes that appear here will depend on the distribution used to calculate the parameters.

  - **Start** shows the low (starting) value on the axis.
  - **End** shows the high (ending) value on the axis.

  For Weibull++ folios, the following table shows the possible values for each of the distributions.

Distribution	Values
Weibull	Beta, Eta
Normal	Mean, Std
Lognormal	Log-Mean, Log-Std
Exponential	Lambda, Gamma
G-Gamma	Mean, Std
Gamma	Log-Mean, Log-Std
Logistic	Log-Mean, Log-Std
Loglogistic	Log-Mean, Log-Std
Gumbel	Log-Mean, Log-Std

14.8.2 Rotating the Chart

The software provides two methods for rotating the chart picture. You can:

- Enter exact values in the **Elevation** and/or **Rotation** fields on the control panel.

  or

- Hold down the **CTRL** key and point to the plot area will change the pointer to a four-headed arrow,  . When the pointer looks like this, clicking in the plot area will cause a transparent outline of the plot area to be displayed. You can drag this outline to change the rotation (i.e., the horizontal viewing angle) and/or the elevation (i.e., the vertical viewing angle) of the plot. The **Elevation** and/or **Rotation** fields on the control panel will be updated with the new values when you release the pointer.

14.8.3 Alter Parameters - 3D Plot

While working on a 3D Plot, you can choose **Alter Parameters** to open the Alter Parameters window and change the settings. This allows you to experiment or visualize different solutions on the current data set. The altered parameter(s) are not saved in the folio when you close the 3D Plot window.

14.8.4 Saving and Opening Saved 3D Plots

- Saving an image

  You can save an image by choosing **Save As** and saving the file in one of four formats: as a 3D Graphic file (*.rlg), as a Windows metafile (*.wmf), as a JPEG file (*.jpeg) or as a bitmap file (*.bmp). If you save the file as *.wmf, *.jpeg or *.bmp you can then use the file in other applications. If you save the file as a *.rlg you can open it in the 3D Plot Viewer window at a later time.

- Opening an image

  You can open an image by choosing **Open** and selecting an existing *.rlg file.

  > **Note:** While you can open an *.rlg file that was created in another folio, you should only change the Elevation and Rotation values either manually or by using the pointer. If you change any other value the plot will refresh and show the values based on the current data sheet, not the saved plot's values.

- Changing the chart settings

  You can change the chart settings in two ways. The chart settings include the placement of the titles and legends, the appearance of the axes and data used to create the chart.

  - Choose **Show Chart Designer** to open the Chart Designer window and change the chart's properties. *See page 546 in Chapter 27.*
  - Choose **Edit Chart Data** to open the Edit Chart Data window in the Chart Designer window, which allows you to modify the data, rows, columns and labels in the data grid of the chart.

14.8.5 3D Plot Ribbon

File



Open opens a previously saved 3D Graphic file (*.rlg).



Save As saves the current plot with a new filename. The plot can be saved as a 3D Graphic file (*.rlg), as a Windows metafile (*.wmf), as a JPEG file (*.jpeg) or as a bitmap file (*.bmp).

- **Save Default View Style** sets the current plot settings as the default settings. Each time 3D Plot is opened, the plot will be viewed in this setting.
- **Restore RS Default View Style** restores the plot settings back to the original settings when the application was installed.

Edit



Copy copies the current plot to the Clipboard as a Windows metafile (*.wmf).



Paste takes a graphic in Windows bitmap (*.bmp) or Windows metafile (*.wmf) format from the Clipboard and places it in the selected element's backdrop or fill.



Page Setup allows you to specify printing options. *See page 581 in Chapter 29.*



Print Preview allows you to preview the plot before it is sent to the printer. *See page 585 in Chapter 29*.



Print Plot opens the Graphic Print window, which allows you to send current plot to the printer.

Actions



Redraw Plot re-draws the plot to reflect the current display specifications.



Alter Parameters opens the Alter Parameters window, allowing you to experiment or visualize different solutions on the current data set. The altered parameters are not saved in the folio when you close the 3D Plot window.



Show Chart Designer opens open the Chart Designer window, which allows you to modify various elements of the plot. *See Chapter 27*.



Edit Chart Data opens the Edit Chart Data window in the Chart Designer window, which allows you to modify the data, rows, columns and labels in the data grid of the selected chart.

3D Plot Quick Access Toolbar

The 3D Plot quick access toolbar includes the following options:



Open opens a previously saved 3D Graphic file (*.rlg).



Save As - 3D Plot saves the current plot as a 3D Graphic file (*.rlg), as a Windows metafile (*.wmf), as a JPEG file (*.jpeg) or as a bitmap file (*.bmp). You can also click the drop-down arrow to either save the current view as the default view style or restore the plot settings back to the original settings when the application was installed.



Print sends the current document to the printer.



Redraw Plot - 3D Plot redraws the plot to reflect the current display specifications.

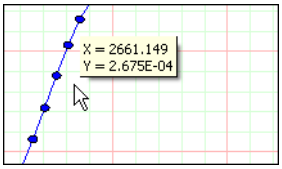


Alter Parameters - 3D Plot Toolbar opens the Alter Parameters window, allowing you to experiment or visualize different solutions on the current data set. The altered parameter(s) are not saved in the folio when you close the 3D Plot window.

14.9 Tips on Working with Plots

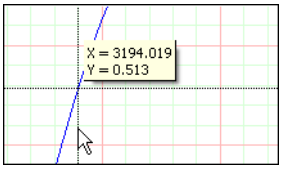
There are several additional commands that you can use when you are working with plots.

- 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.



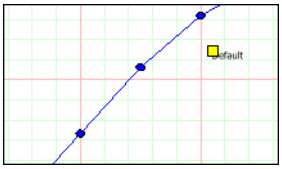
  You can release the **SHIFT** key once the coordinates are displayed. The coordinates will continue to be displayed as long as you hold down the mouse button. You can move the cursor around in the plot and the coordinates will change to reflect its current location.

- If the cursor is on a plot line (e.g., the probability line, confidence bounds, etc.), pressing **SHIFT** and clicking the line will track the line when you move the cursor. A crosshair will be displayed showing the current location on the line. As you move the cursor 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.

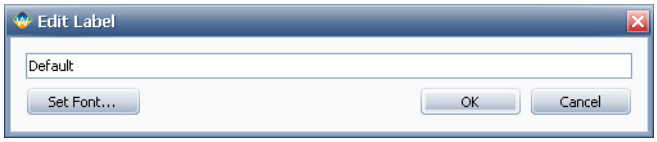


  Pressing **CTRL+SHIFT** and clicking the line does the same thing, but X is calculated based on Y.

- 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.



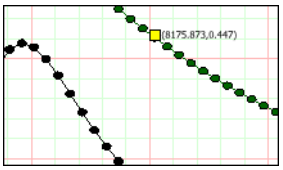
  The yellow box at the upper left corner of the label indicates that the label is selected. When the label is selected, you can edit the label text directly in the plot and can also click the box and drag it to a new location, if desired. Clicking elsewhere in the plot will de-select the label. Double-click the label to open the Edit Label window, as shown next.



  In the Edit Label window, you can specify the text for the label. Clicking **Set Font** opens the Font window, which allows you to specify the font, size, style, color and orientation used for the label.

  Custom labels, including the coordinate labels presented next, are not available in side-by-side plots.

- To add a label that displays coordinates to the plot, position the cursor 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 cursor will change from an arrow to a pointing hand when you hover over an element for which this is possible. Note that the element you clicked is automatically selected on the Plot Items page of the Plot Setup. *See page 416 in Chapter 15*.

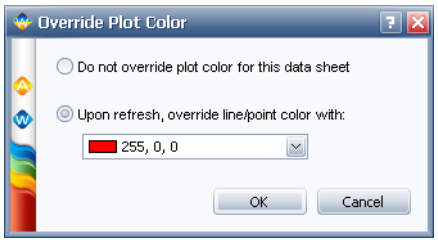
14.10 Override Plot Color

The Override Plot Color setting allows you to ignore the default colors of the plotted line and points, and apply a color that you specify. This setting is activated in the source data sheet of the plot. This means that when you enable the setting for a particular data sheet, it applies only to the plots that you create for that data sheet and not the plots for all other data sheets that may be within the same folio.

To enable the setting, choose **[Life Data/ALTA/Degradation] > Format and View > Override Plot Color**.



This opens the Override Plot Color window, as shown next.



When you select the override setting, only the colors of the lines and points are affected. Other settings such as the line style and thickness, or the point size and shape are based on the settings that you have specified in the Plot Items page of the Plot Setup window. *See page 416 in Chapter 15*.

Plot Setup 15

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.

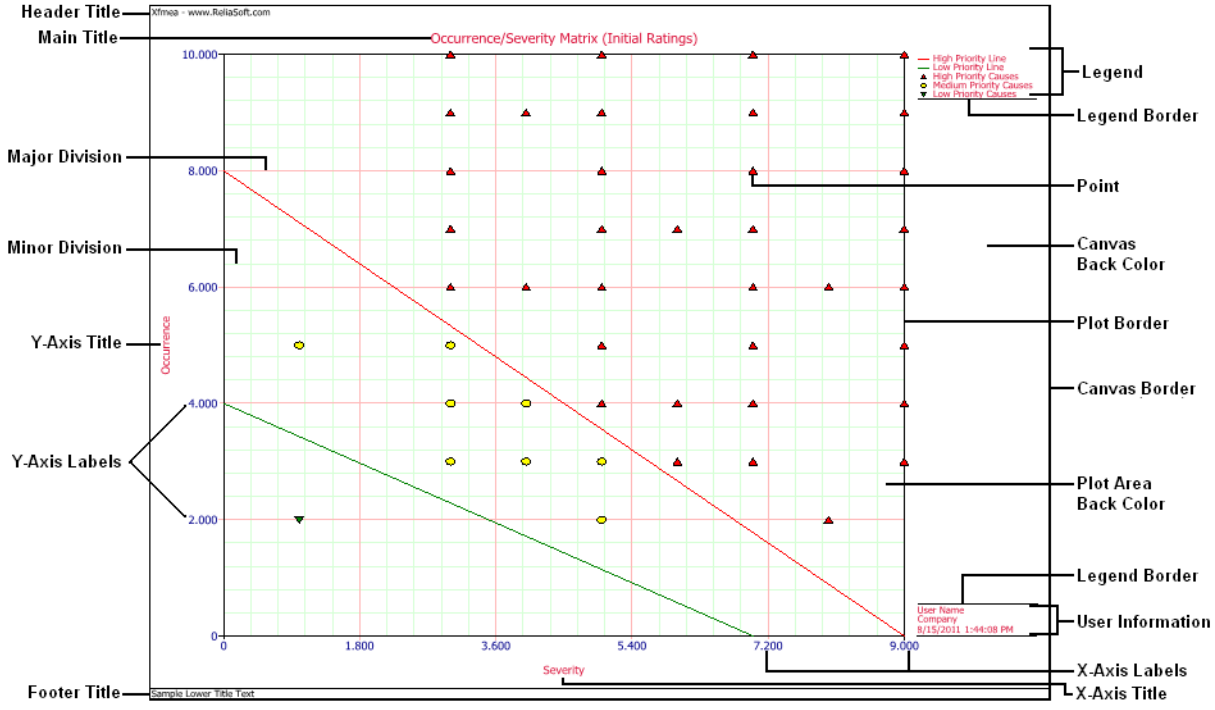
Open the Plot Setup window by clicking the **Plot Setup** icon on the Plot control panel.



In addition, clicking an element (e.g., point, line, etc.) in the plot will open the Plot Setup window.

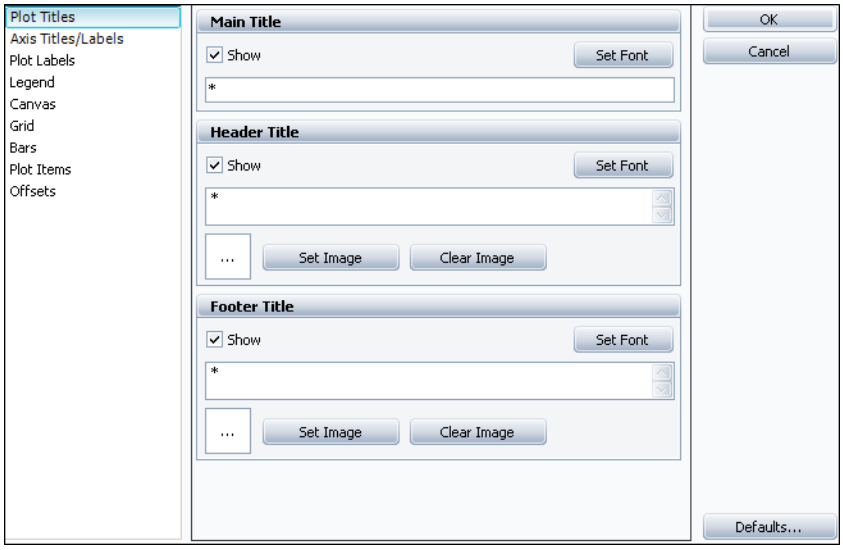
Each time you open the Plot Setup window, the changes that you make will apply only to the current plot unless you specify otherwise. To specify settings to be used as defaults for all subsequent plots or to re-apply default settings to the current plot, click the **Defaults** button to open the Plot Defaults window. *See Section 15.11 on page 418*.

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 applications.) You can change these settings on the pages of the Plot Setup window.



15.1 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.

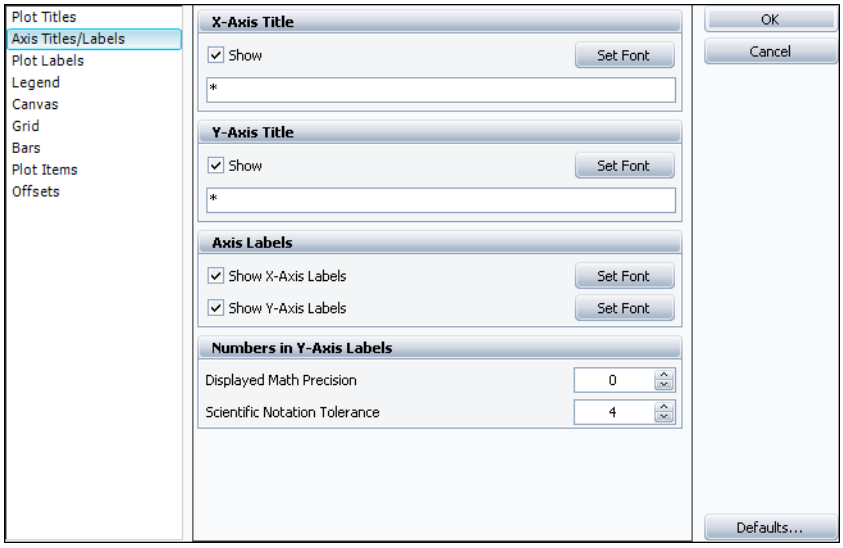
Each asterisk (*) represents the default title text. For the main title, this is defined on the Titles Text page of the Plot Defaults window. (*See Section 15.11.1 on page 419*.) If you want to append additional text to the default title, keep the asterisk and add the additional text before or after 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**.

15.2 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**

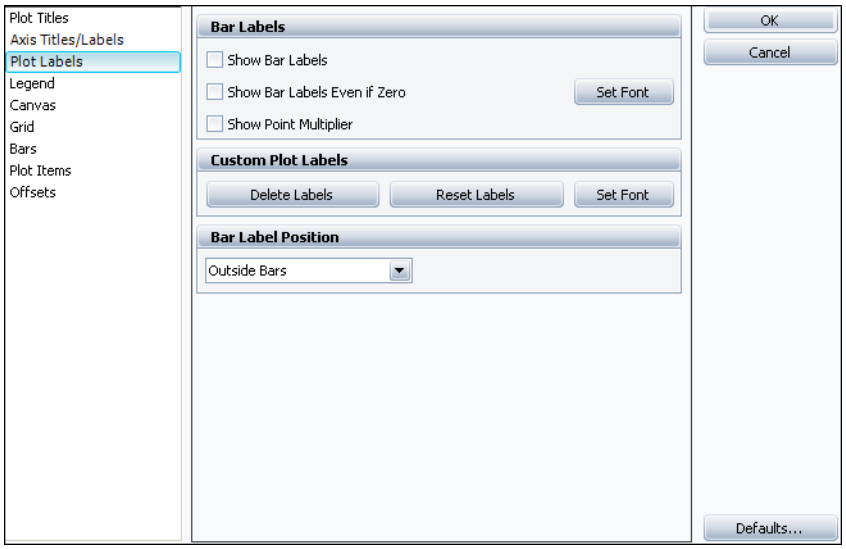
  Use the options in this area to configure the mathematical precision (number of decimal places) and scientific tolerance of the values shown on the axes.

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.

15.3 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.

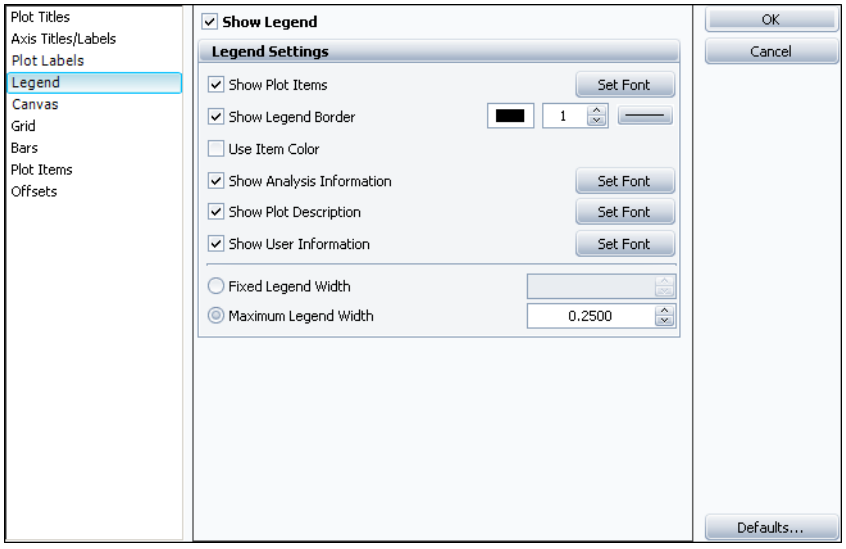
- **Bar Labels** (available only for bar charts)
  - To display the y-axis value of each bar with the bar, select the **Show Bar Labels** check box. (You can set the bar style using the **Bar Orientation** field on the Bars page of the Plot Setup. *See Section 15.7 on page 414*.)
  - **Show Bar Labels 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. *See page 404 in Chapter 14*.
  - **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.

15.4 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.

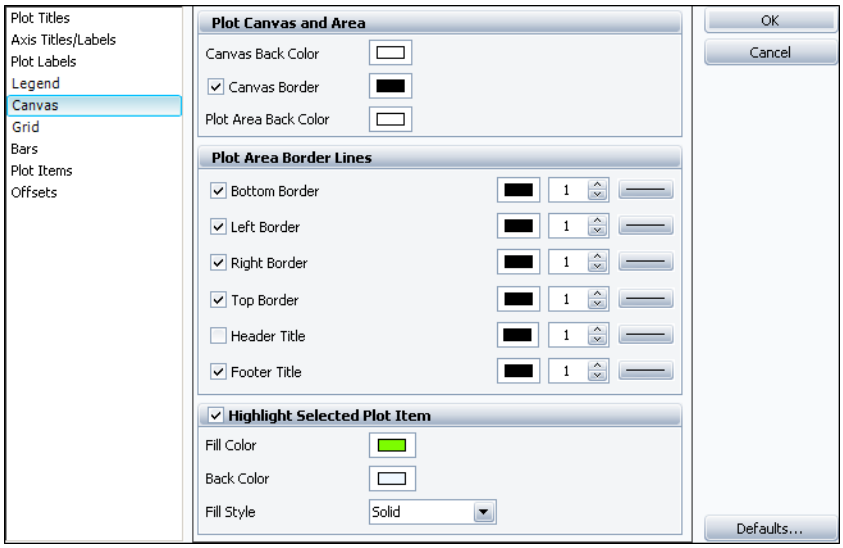
- To display the text of the legend in the same color as the lines, points, bars and/or slices on the plot, select the **Use Item Color** check box. If not selected, the text will be red.
- **Show Analysis Information** if selected, the legend will display information about the analysis shown on the 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. *See page 61 in Chapter 2*.)

- 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.

15.5 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.

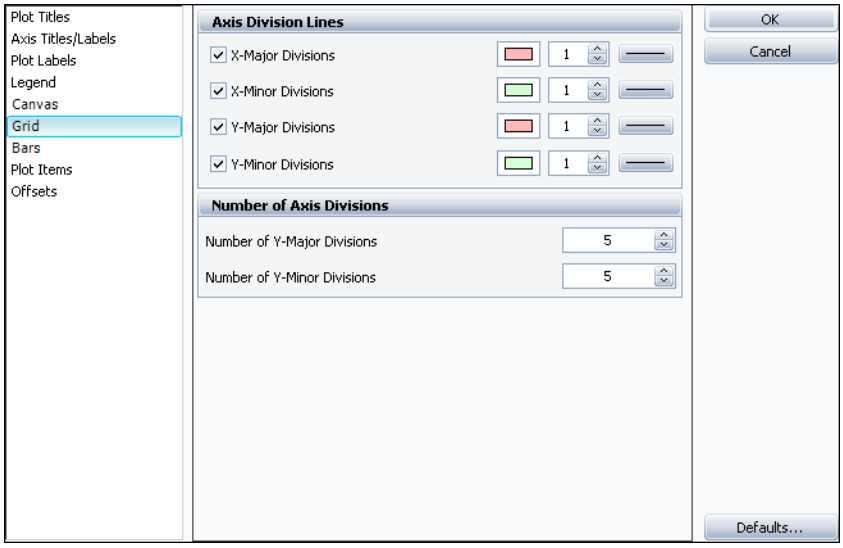


- **Plot Canvas and Area**
  - **Canvas Back Color** allows you to choose the color for the area outside the plot (i.e., the area containing the titles, labels, legend, etc.).
  - **Canvas Border** allows you to determine if the border around the plot field will be displayed and, if so, to set the color.
  - **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.

15.6 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.



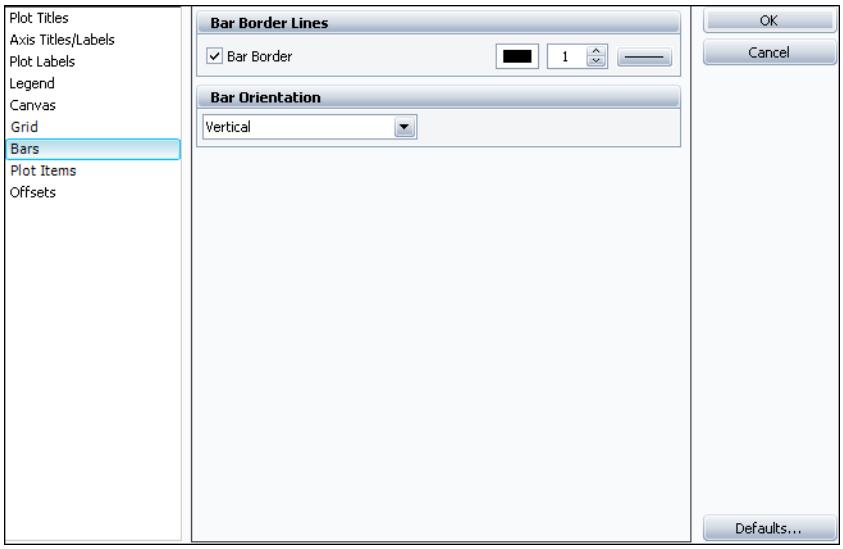
The options on this page will vary depending on the plot style you are working with. All available options are presented below.

- **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.

15.7 Bars Page

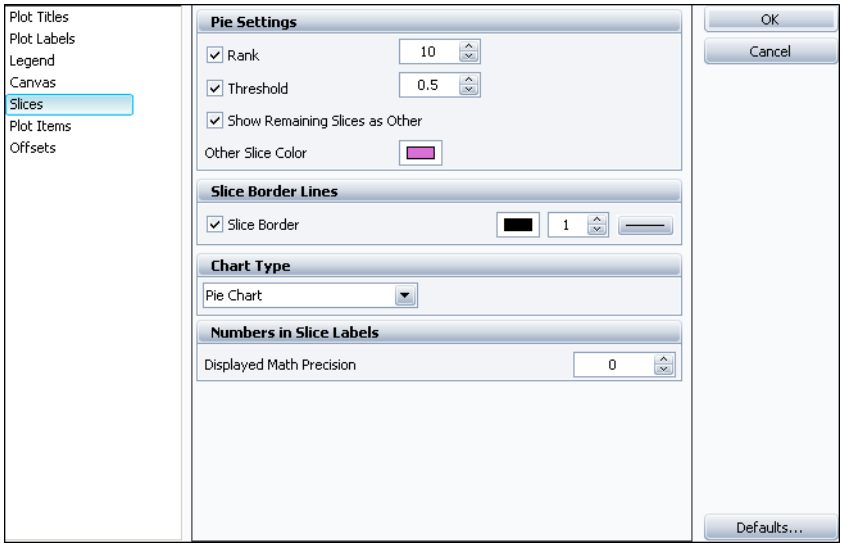
The Bars page is available only when you are working with a bar chart. It allows you to customize the appearance of the bars in the plot.



- **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.

15.8 Slices Page

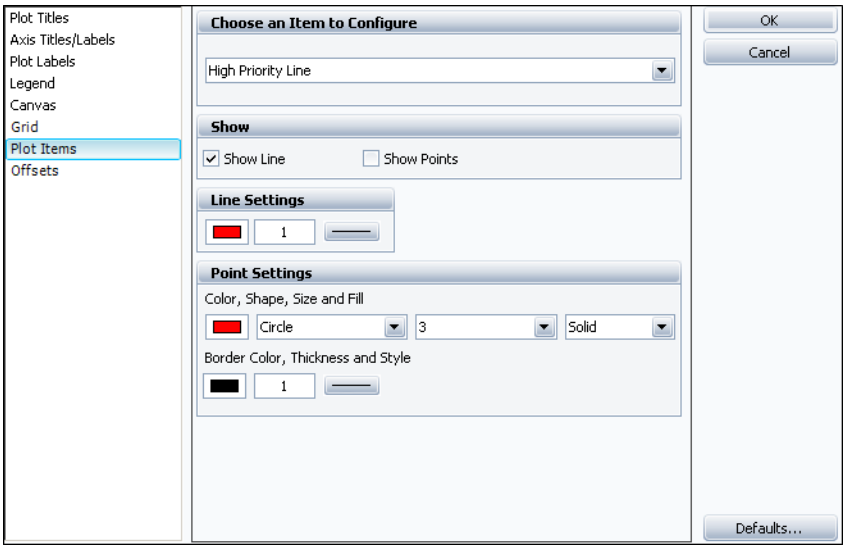
The Slices page is available only when you are working with a pie chart. It allows you to customize the appearance of the slices in the plot.



- **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.

15.9 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.

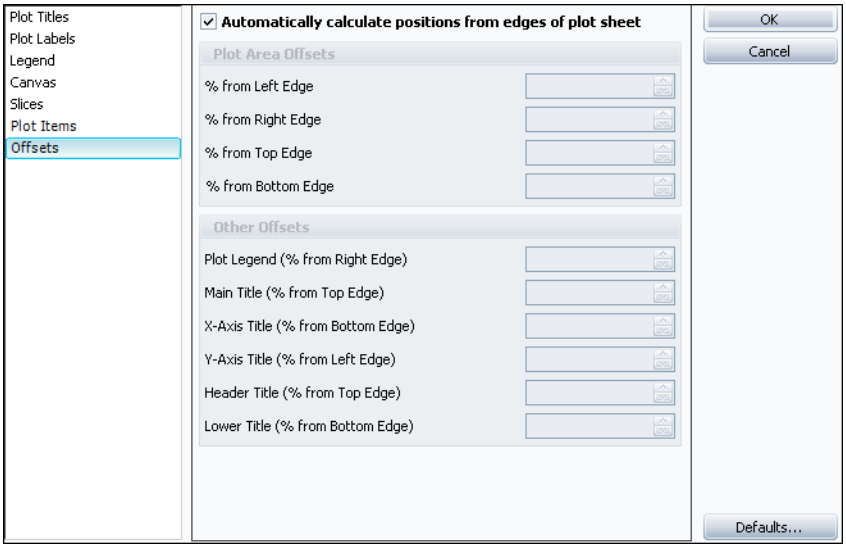
> **Tip:** If you find that the software does not use the default plot settings when you create or refresh a plot, then the Override Plot Color setting may be in effect. To turn off the setting, go to the source data sheet of the affected plot and choose **[Life Data/ALTA/Degradation] > Format and View > Override Plot Color**.

- **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.

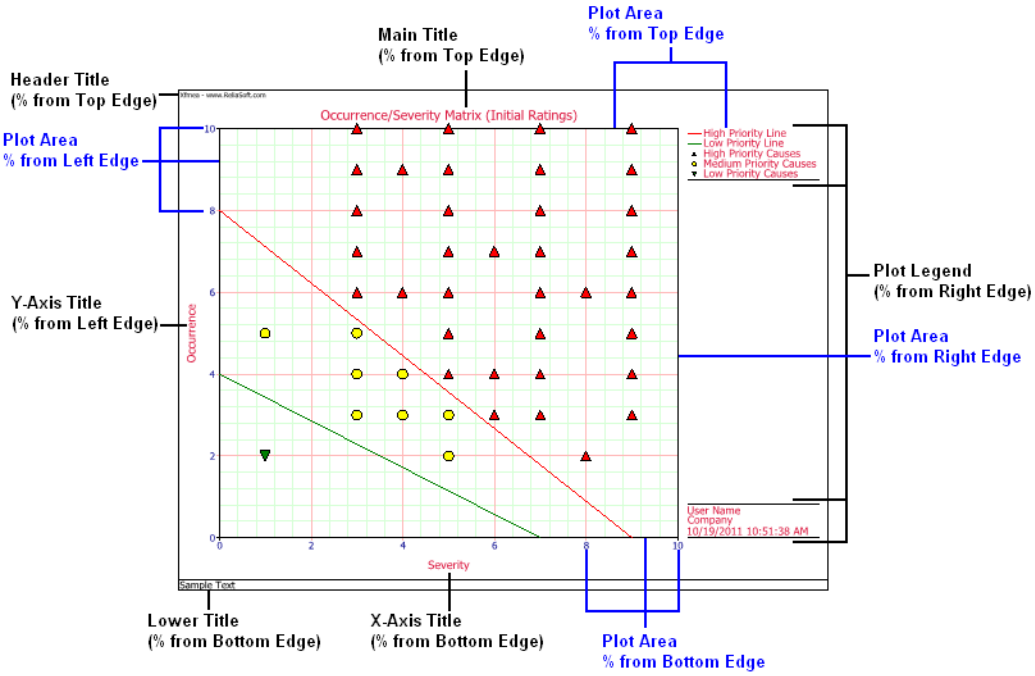
15.10 Offsets Page

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 applications.)



15.11 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 15.11.1 on page 419*.
- 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 15.11.2 on page 420*.
- 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

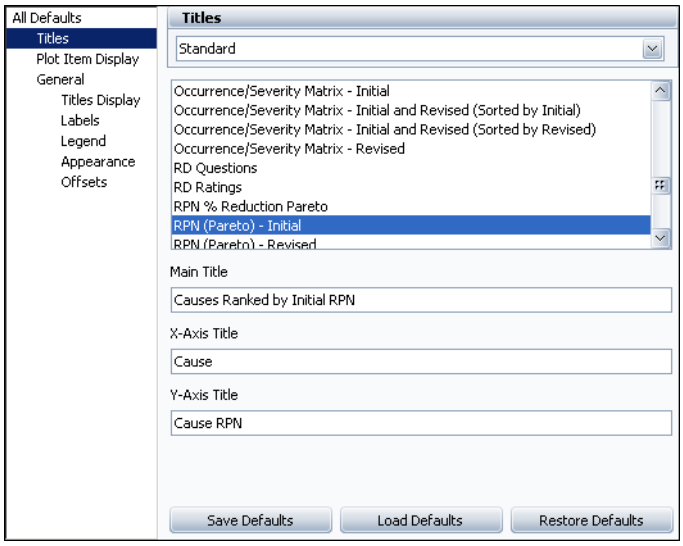
Canvas and Grid page of the Plot Defaults window will save the settings from the Canvas, Grid, Bars and Slices pages).

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.

15.11.1 Plot Defaults: Titles Page

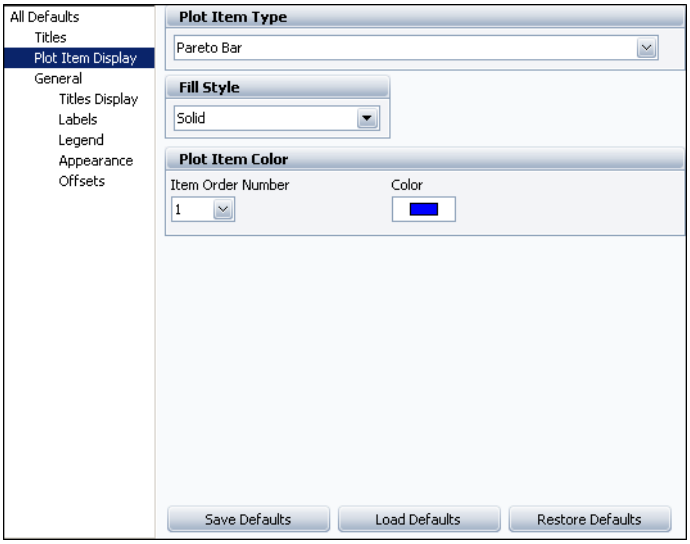
The Titles Text page of the Plot Defaults window allows you to customize default plot titles for each plot type.



- **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.

15.11.2 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.



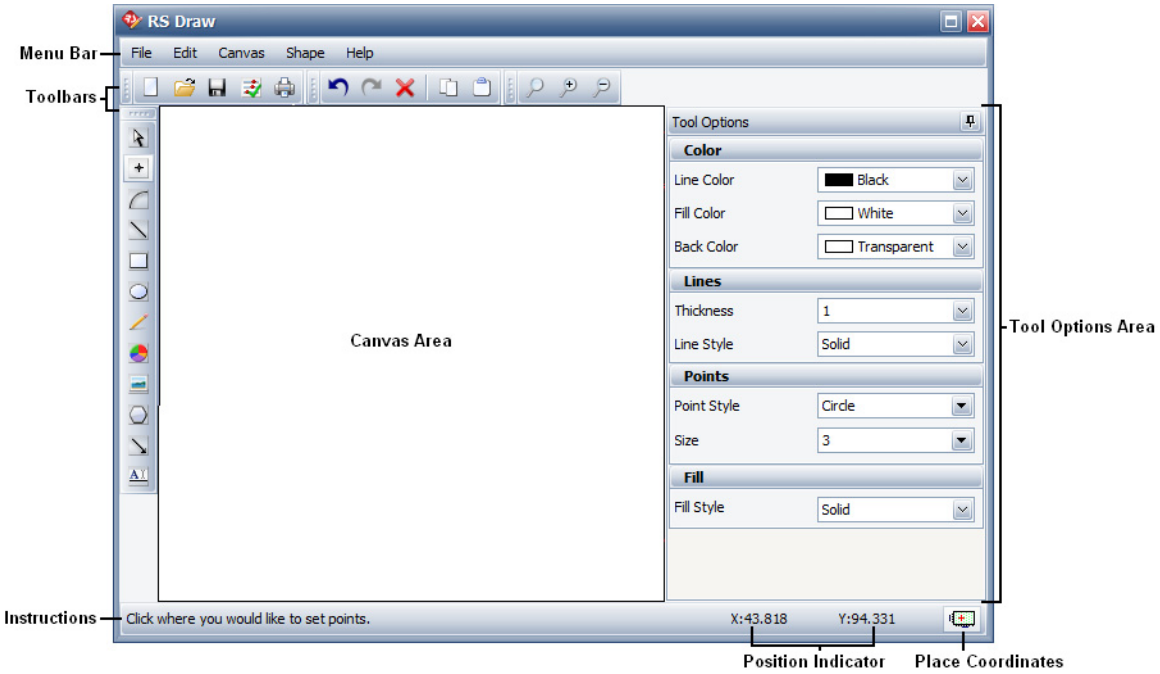
Plot Item Type allows you to select the style of plot that you want to specify default settings for. The settings you specify are retained in the background when you choose a new item in this list, so you can specify settings for each type and save all of them, if desired.

The options on this page are identical to those on the Plot Items page of the Plot Setup window for the corresponding plot style.

Tip: If you find that the software does not use the default plot settings when you create or refresh a plot, then the Override Plot Color setting may be in effect. To turn off the setting, go to the source data sheet of the affected plot and choose **[Life Data/ALTA/Degradation] > Format and View > Override Plot Color**.

ReliaSoft Draw 16

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.



16.1 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.



16.2 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.

16.2.1 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.

16.2.2 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.

16.2.2.1 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.

16.2.3 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.

16.2.4 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**.

16.2.5 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.

16.3 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 16.3.5 on page 426*. The toolbars and their icons are shown and explained next.

16.3.1 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.

16.3.2 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.

16.3.3 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.

16.3.4 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 16.5 on page 427*. 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.

16.3.5 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.

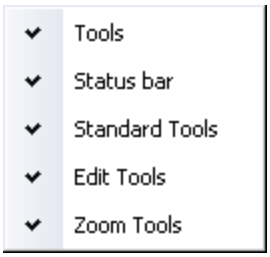
16.3.5.1 Moving Toolbars

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.



16.3.5.2 Hiding and Showing Toolbars

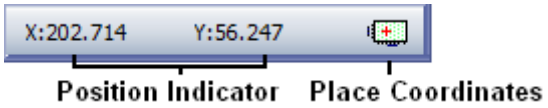
You can choose to display or hide any of RS Draw's toolbars by right-clicking inside the grey toolbar area. A Toolbar shortcut menu like the one shown next will appear.



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.

16.4 Additional Tools

Two additional tools are located on the bottom of the RS Draw window.



- **Position Indicator** indicates the exact position of the mouse 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.

16.5 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**.

16.5.1 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.

16.5.2 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.

16.5.3 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.

16.5.4 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.

16.5.5 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.

16.5.6 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.

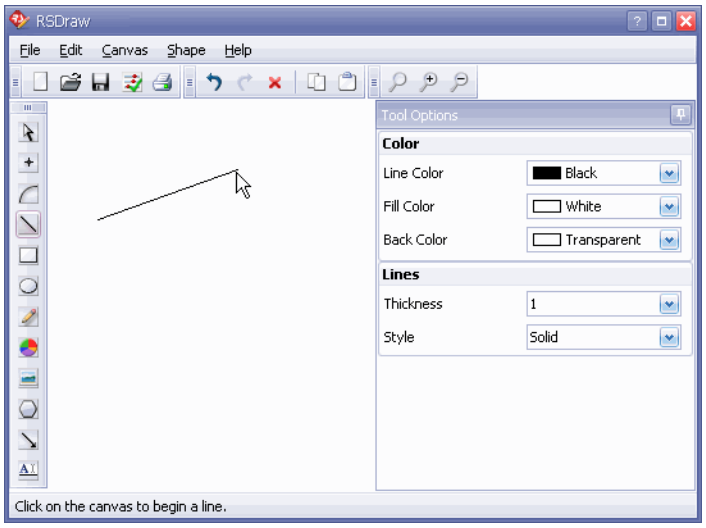
- **Back Color** opens the Color Palette, which allows you to choose the background color for the selected object from the drop-down menu. **Transparent** indicates that there is no background color.

16.6 Working with Objects: Examples

16.6.1 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.



16.6.2 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 16.2.2.1 on page 423*.

16.6.2.1 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 cursor 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.

16.6.2.2 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 cursor. When you are satisfied with the new size, release the button.

To move the rectangle, drag the central handle to the appropriate place.

16.7 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**.

Monte Carlo and SimuMatic 17

In life data analysis and accelerated life testing analysis, the reliability engineer will typically select a model to fit data obtained from testing or usage in the field. However, in some situations, it is useful to generate simulated data sets containing values that are distributed according to a specified life distribution or model. For example, simulated data could be used to:

- Test different warranty and maintenance strategies.
- Perform risk analysis.
- Obtain simulation-based confidence bounds.
- Analyze probabilistic design models.
- Design reliability tests.
- Compare different parameter estimation methods.
- Evaluate the impact of different censoring schemes.

With Weibull++ and ALTA, you can use Monte Carlo simulation to produce data sets based on various user inputs, such as distribution type, distribution parameters and sample size. To create a data set, the software uses the *cdf* (cumulative distribution function) of the relevant distribution or model to solve for time given an unreliability value chosen from a uniform random distribution. The process is repeated with new random unreliability values until the desired number of data points is obtained.

Two utilities are available for producing simulated data:

The Monte Carlo utility (which comes in a Weibull++ version and an ALTA version) uses Monte Carlo simulation to generate a single data set containing values that are distributed according to a specified life distribution or user-defined model. The data set is then automatically placed in a standard folio, where it can be analyzed like any other data set. SimuMatic (which also comes in a Weibull++ version and an ALTA version) generates a large number of data sets using Monte Carlo simulation. It then analyzes the group of data sets as a whole. For example, you can use SimuMatic to find the average reliability at a given time for a thousand simulated data sets.

This chapter describes how to use these utilities, including:

- Using the Weibull++ version of the Monte Carlo tool:
  - Generating data based on a user-defined equation - Section 17.1.1 (p. 434).
  - A user-defined equation example - Section 17.1.2 (p. 435).
- Using the ALTA version of the Monte Carlo tool - Section 17.2 (p. 438).

- Using SimuMatic, including:
  - Generating data with Weibull++ SimuMatic - Section 17.3.1 (p. 442).
  - Generating data with ALTA SimuMatic - Section 17.3.2 (p. 443).
  - Using the SimuMatic folio - Section 17.3.3 (p. 444).
  - Transferring data to a Weibull++ standard folio - Section 17.3.4 (p. 447).
  - Designing reliability tests - Section 17.3.5 (p. 449).
  - An example of using the Weibull++ SimuMatic utility - Section 17.3.6 (p. 451).
  - An example of using the ALTA SimuMatic utility - Section 17.3.7 (p. 453)

17.1 Weibull++ Monte Carlo Utility

WEIBULL++ Only

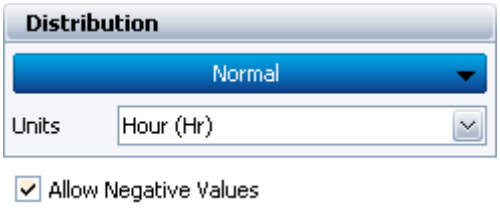
The Weibull++ Monte Carlo utility uses Monte Carlo simulation to generate a single data set containing values that are distributed according to a specified life distribution or user-defined model. The software uses the *cdf* (cumulative distribution function) of the relevant distribution to solve for time given an unreliability value chosen from a uniform random distribution. The process is repeated with new random unreliability values until the desired number of data points is obtained. The data set is then automatically placed in a Weibull++ standard folio, where it can be analyzed like any other data set.

To access the Weibull++ Monte Carlo utility, choose **Home > Tools > Weibull++ Monte Carlo**.



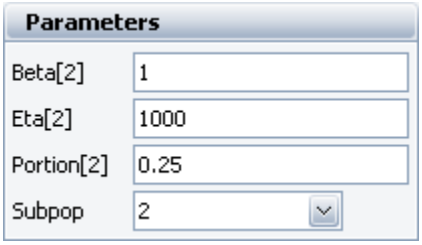
The setup window will appear. Follow the steps outlined below to generate the data set:

1. On the Main tab of the window, select a lifetime distribution, enter its required parameter values and select the time units (e.g., hours) for the failures/suspensions in the data set. In addition, please note the following:
   - If the estimated values of the distribution parameters are not available, you can use the **Quick Parameter Estimator** button to solve for them.
   - With the **User-Defined** option, you can manually enter an equation that relates different random variables. In the **Equation** area, use *R* to insert a uniform random variable. If desired, you can also click the **Insert Data Source** button to generate simulated values based on distributions calculated in standard folios from your current project. *See Section 17.1.1 on page 434*.
   - If you select the normal, logistic or generalized gamma distributions, an additional option to allow the generation of negative time values will become available. When generating data to simulate a test, negative failure times can be used to simulate failures that occur before a unit's testing period begins, such as failures due to manufacturing defects or damage that is incurred during shipping.



   For all other predefined distributions (e.g., Weibull, lognormal, etc.), the utility will generate only positive values.

- If you select a mixed Weibull distribution, you will need to specify parameters for each subpopulation.
  - To specify the parameters for a subpopulation, select the subpopulation in the **Subpop** drop-down list, then enter the parameter values. Each input field will have a number in brackets to indicate which subpopulation that field is applicable to.



  - The **Portion** field is for the portion (or percentage, entered as a decimal) of the total data set represented by the selected subpopulation. Thus, the sum of the portion values for all subpopulations must be equal to 1. *For more information about the Mixed Weibull analysis, see page 233 in Chapter 6*.

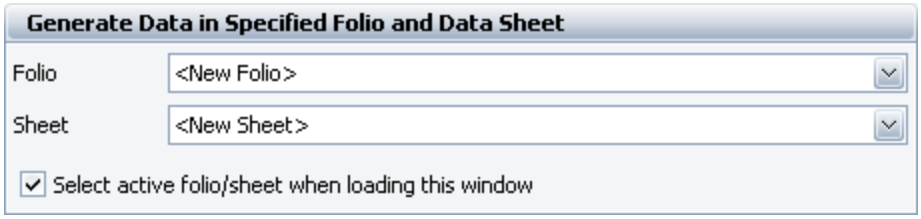
2. Select the appropriate option on the Censoring tab:
   - **No censoring**: The generated data set will contain only exact failure times (i.e., it will contain no right censored, interval or left censored data).
   - **Right censoring after specific number of failures**: You specify the number of failures, *n*, that the generated data set will contain in the **Number of Failures** field. After the data are generated, the data is sorted. The first *n* data points will be marked as failures while the remaining data points will be marked as suspensions at a time equal to the time of the *n*th failure. When used with SimuMatic, this option allows you to simulate a failure-terminated test (i.e., a test that ends after a specified number of failures occur).
   - **Right censoring after a specific time**: You specify a duration in the **Time** field. All simulated values that do not exceed this time will be marked as failure times. Values that exceed this time will be considered suspensions at the specified time (i.e., units that had not failed by the end of the test). When used with SimuMatic, this option allows you to simulate a time-terminated test (i.e., a test that ends after a specified time).
   - **Random censoring**: You provide percentages to specify what proportions of your total data points will consist of right censored, interval and left censored data. The remaining data points will be exact failure times. Thus, the sum of the three percentages entered cannot exceed 100%. This option can be useful for exploring how different kinds of uncertainty in your data can influence the results of an analysis.

     For example, with the settings shown in the following picture, the software will generate a set of data points, where 20% of the data are right censored, 20% are interval censored, 20% are left censored and the remaining 40% are complete data (i.e., exact failure times).



Note: With random censoring, the software will censor your data using a uniform distribution. If this option is used to generate multiple data sets in SimuMatic, the percentages you enter will apply to the total number of generated data points, not the number of data points within each data set. Thus, if you select to have 50% of your generated data consist of suspensions (right censored data), half of all your data points will be suspensions, though some data sets may still contain more suspensions than others.

3. On the Settings tab, specify how you want to generate the data set and where you want the data to be stored.
   - Select the **Use Seed** check box if you would like to set a consistent starting point from which the random numbers will be generated. Using the same seed value and keeping all other settings the same will allow you to replicate your results.
   - In the **Math Precision** field, enter the number of decimal places you wish to use for each simulated data point.
   - In the **Number of data points** field, enter the number of data points you wish to generate.
   - In the **Folio** and the **Sheet** drop-down lists, choose where to put the simulated data.
     - If you choose **<New Folio>** the software will create a new folio to place the data into. If you choose **<New Sheet>** the software will create a new sheet in a new or existing folio, depending on what you chose in the **Folio** drop-down list.
     - Click the **Active** button to select the folio and data sheet that were active when the utility was opened. If you would like the utility to always select the active folio and data sheet, select the check box labeled **Select active folio/sheet when loading this window**.



After the simulation is set up, click **Generate** to create a data set according to your specifications. A Weibull++ standard folio containing the data will appear.

17.1.1 User-Defined Equations

The Weibull++ Monte Carlo utility includes the option to generate data sets based on user-defined equations. This can be used for a wide variety of applications in risk analysis, probabilistic design and other areas.

Entering a User-Defined Equation

When you select **User-Defined** in the **Distribution** area on the Main tab of the Monte Carlo utility, the **Parameters** area will be replaced with an **Equation** field.

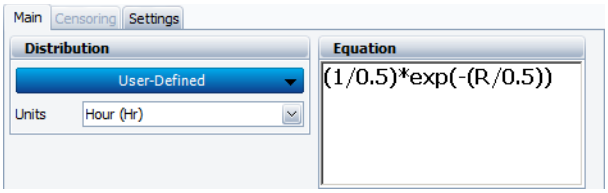
To use the tool, enter an equation with at least one random variable. There are two ways to do that:

- **Use the letter R to indicate a random variable**

  If you use the letter *R* to indicate a random variable in the user-defined equation, the utility will generate random values for the variable based on a uniform distribution. For example, suppose you want to generate a data set based on the following exponential model, where $\beta = 0.5$.

$$f(x, \beta) = \frac{1}{\beta} \times e^{-(x/\beta)}$$

Substituting the value of $\beta = 0.5$ and using the variable R to represent x, you would enter the following equation:

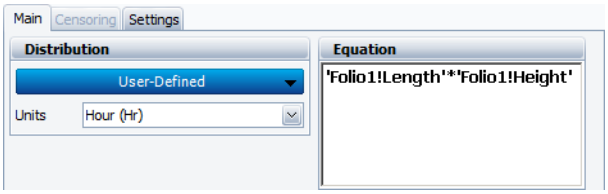


Note: A new uniform random variable is chosen for each time *R* appears in the user-defined equation. For example, if you enter **R^2**, the software will compute the square of one uniform random number. But if you entering **R*R**, the software will compute the product of two different uniform random numbers.

- **Draw a random number from a distribution calculated in a standard folio**

  If you want to obtain a random number from the distribution in an existing standard folio, click the **Insert Data Source** button and then select the calculated data sheet from the list.

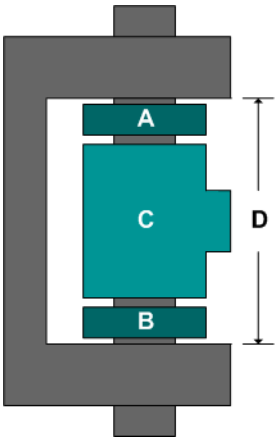
  For example, suppose you want to generate a data set that contains a range of possible area measurements (area = length x height), where the length is obtained by drawing a random number from the distribution computed in one data sheet (called "Folio1!Length") and the height is obtained by drawing a random number from the distribution computed in another (called "Folio1!Height"). Your user-defined equation would look like the following:



17.1.2 User-Defined Equation Example

This example demonstrates how to use the Monte Carlo utility to perform an analysis with a user-defined equation.

A company manufactures hinges that are made up of four components A, B, C and D. The next figure shows a schematic of the hinge's assembly.

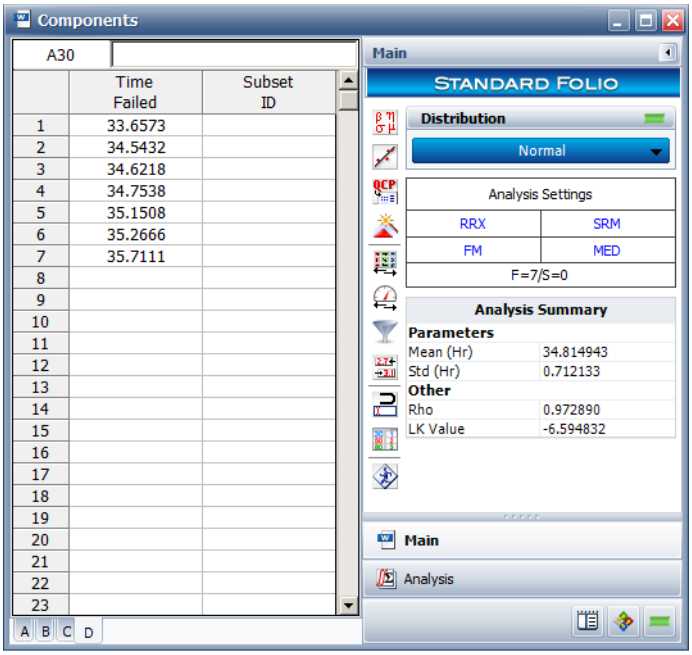


The manufacturer wants to determine the percentage of hinges that would fall out of specifications. Specifically, the manufacturer wants to estimate the probability that (A+B+C) will be greater than D. For that purpose, data sets were

collected for the dimensions of each of the manufactured components. Of each of the components A, B, C and D, seven units were taken from the assembly line and the following measurements were recorded.

Dimensions for A (cm)	Dimensions for B (cm)	Dimensions for C (cm)	Dimensions for D (cm)
2.0187	1.9795	30.4216	33.6573
1.9996	2.0288	29.9818	34.5432
20.167	1.9883	29.9724	34.6218
2.0329	2.0327	30.192	34.7538
2.0233	2.0119	29.9421	35.1508
2.0273	2.0354	30.1343	35.2666
1.984	1.9908	30.0423	35.7111

The first step in this analysis is to model the dimensions of each of the parts using the recorded data. To do this, create a Weibull++ standard folio configured for times-to-failure. Name this folio "Components." Then enter the dimension measurements into the new folio with a separate data sheet for each component. Next, analyze each data set assuming a normal distribution and RRX as the analysis method. Below is the "Components" folio with the analysis results for component D displayed.



After the parameters for each data set are calculated, Monte Carlo simulation can be performed to estimate the probability that (A+B+C) will be greater than D. Choose **Home > Weibull++ Monte Carlo**. The Weibull++ Monte Carlo tool will appear.

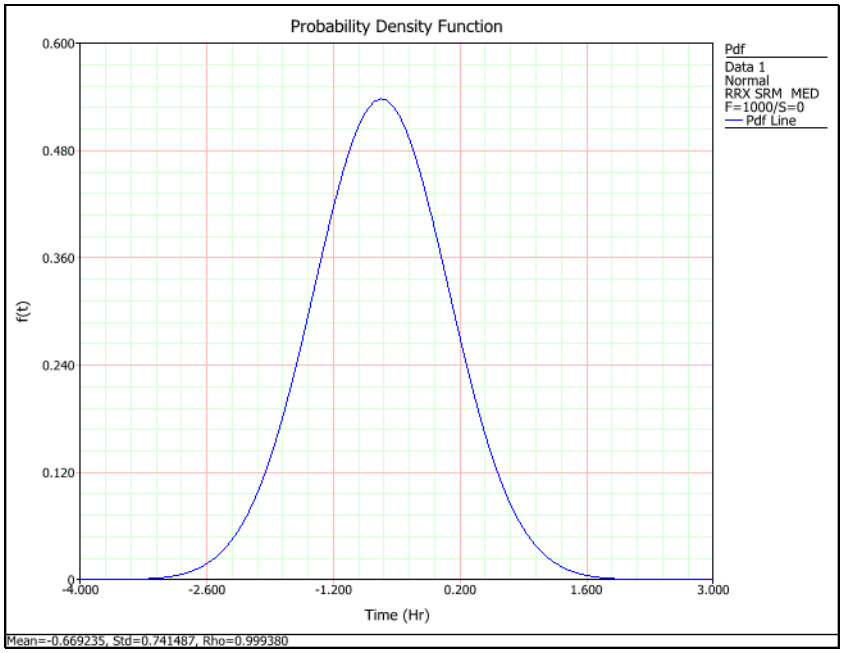
In the **Distribution** area on the Main tab, choose **User-Defined** for the distribution and click the **Insert Data Source** button to insert the first data sheet from the "Components" folio. Click the button again to insert the second data sheet, and repeat until all four data sheets are entered. Edit the user-defined equation so it appears as shown next.

'Components!A' + 'Components!B' + 'Components!C' - 'Components!D'

In the **Data Points** area on the Settings tab, enter **1000** for the number of data points. Choose **<New Folio>** and **<New Sheet>** from the drop-down lists in the **Put the data points in** area. Then click **Generate** to create the simulated data.

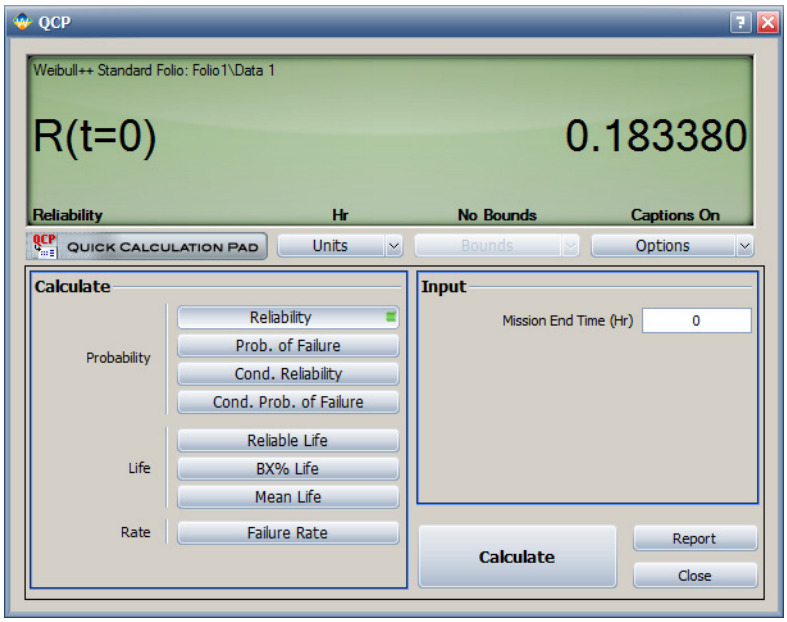
In the new data sheet that contains the simulated data, calculate the parameters, assuming a normal distribution and RRX as the analysis method.

The following is a *pdf* plot of the simulated data for (A+B+C-D).



The probability that (A+B+C) > D is equal to the probability that (A+B+C-D) > 0. Thus, the area under the curve of the above pdf from time = 0 to infinity—where "time" is actually the value of (A+B+C-D)—will tell us how likely it is that the product will fall out of specifications (i.e., the likelihood that A+B+C will be greater than D).

Use the QCP to calculate the probability, as shown next. (Note that we will use the **Results as Reliability** option in the QCP, even though we are finding the likelihood of falling out of specifications, since the calculation we wish to perform is equivalent to finding the reliability of a randomly generated data set at time = 0.)



The results show that the probability that (A+B+C) will be greater than D is about 18.34%. Note that your results may vary because the data were obtained through simulation.

17.2 ALTA Monte Carlo Utility

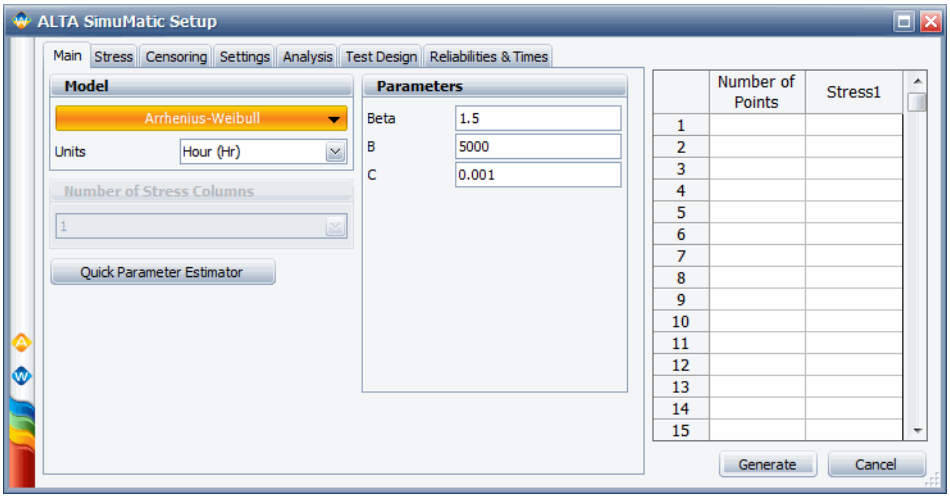
ALTA Only

The ALTA Monte Carlo utility uses Monte Carlo simulation to generate a single data set containing values that are distributed according to a specified model. The software uses the *cdf* (cumulative distribution function) of the relevant model to solve for time given an unreliability value chosen from a uniform random distribution. The process is repeated with new random unreliability values until the desired number of data points is obtained. The data set is then automatically placed in a ALTA standard folio, where it can be analyzed like any other data set.

To access ALTA's Monte Carlo utility choose **Home > Tools > ALTA Monte Carlo**.

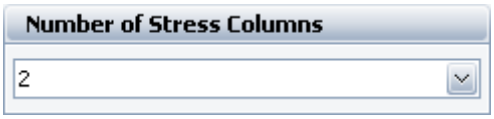


The setup window will appear as shown next.

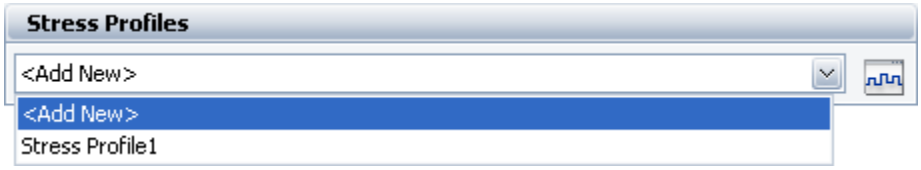


Follow the steps outlined below to set up the simulation:

1. On the Main tab of the window, select a model, enter its required parameter values and select the time units (e.g., hours) for the failures/suspensions in the data set. In addition, please note the following:
   - If the estimated values of the distribution parameters are not available, you can use the **Quick Parameter Estimator** button to solve for them.
   - If you select a proportional hazards (PPH), general log-linear (GLL) or cumulative damage (CD) model, you will have the option to change the number of stress columns in the data sheet. When this option is available, the **Number of Stress Columns** area, shown next, will be enabled.



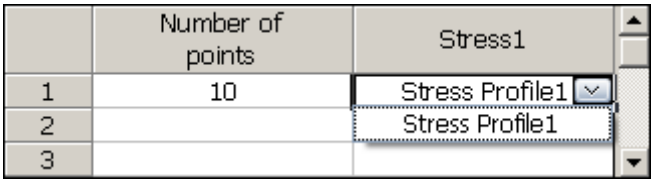
   Every additional stress column represents an additional stress type. So, for example, a data sheet with two stress columns would be used to simulate data obtained from a two-stress test. The different rows would then be used to represent different stress value combinations for the two stresses.
   - If you select a cumulative damage (CD) model, then you will have the option to create a new time-dependent stress profile or open an existing one from this window. You can create a new profile by choosing **<Add New>** from the **Stress Profiles** drop-down list and then clicking the **Create/View Profile** icon. You can open an existing profile by choosing it from the drop-down list (as shown next) and then clicking the icon.



2. On the Stress tab, enter the appropriate **Use Level** stress value for each stress type. This is the stress level that the product is expected to operate under during normal use conditions. If you have selected to generate a data set with more than one stress type, multiple input fields will be available, where the first input field corresponds to Stress 1, the second to Stress 2, and so forth.
   - If you select a general log-linear (GLL) or cumulative damage (CD) model, you must also select an appropriate option from the **Transformation** drop-down list for each stress.
     - Use the **None X=V** transformation for stress types associated with the exponential life-stress relationship (LSR). This transformation is commonly used for indicator variables (e.g., 0 = on/off and 1 = continuous operation).
     - Use the **Reciprocal X=1/V** transformation for stress types associated with the Arrhenius LSR. This transformation is commonly used for thermal stresses.
     - Use the **Logarithmic X=ln[V]** transformation for stress types associated with the inverse power law LSR. This transformation is commonly used for non-thermal stresses.
3. In the data sheet on the right side of the window, specify how many data points you want generated for each stress level, and specify the values for each stress level.
   - For example, if you wanted to generate 30 data points, 10 at 3 different stress levels, you could enter the following.

	Number of points	Stress1	Stress2
1	10	348	3
2	10	378	3
3	10	378	5

   Each row in the data sheet represents a stress level, and every stress column represents a stress type. For a multi-stress model, the stress level is the combination of stress values that will be applied to each group of tested units. Thus, the first row of the above table would configure the utility to generate 10 data points from 2 stresses, one with a value of 348 and another with a value of 3. The second row would generate an additional 10 data points from a different combination of stress values, and so on.
   - If you select a cumulative damage (CD) model, you will be able to assign time-dependent stress profiles to the stress columns of the data sheet. Click a cell in a stress column and a drop-down list will appear. This list will include all validated stress profiles in your current project.



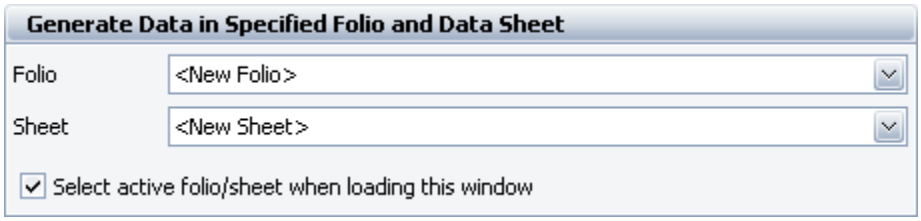
4. Select the appropriate option on the Censoring tab:
   - **No censoring**: The generated data set will contain only exact failure times (i.e., it will contain no right censored, interval or left censored data).
   - **Right censoring after specific number of failures**: You specify the number of failures, *n*, that the generated data set will contain in the **Number of Failures** field. After the data are generated, the data is sorted. The first *n* data points will be marked as failures while the remaining data points will be marked as suspensions at a

time equal to the time of the *n*th failure. When used with SimuMatic, this option allows you to simulate a failure-terminated test (i.e., a test that ends after a specified number of failures occur).

> **Note:** To illustrate how this censoring is performed, suppose you chose to generate 20 data points (e.g., 10 at one stress level and 10 at another) and entered **10** in the **Number of Failures** field. The software would first generate 20 time values. Then the lowest 10 values across all the stress levels would be marked as failure times. The remaining 10 values would be marked as suspensions.

- **Right censoring after a specific time**: You specify a duration in the **Time** field. All simulated values that do not exceed this time will be marked as failure times. Values that exceed this time will be considered suspensions at the specified time (i.e., units that had not failed by the end of the test). When used with SimuMatic, this option allows you to simulate a time-terminated test (i.e., a test that ends after a specified time).

5. On the Settings tab, specify how you want to generate the data set and where you want the data to be stored.
   - Select the **Use Seed** check box if you would like to set a consistent starting point from which the random numbers will be generated. Using the same seed value and keeping all other settings the same will allow you to replicate your results.
   - In the **Math Precision** field, enter the number of decimal places you wish to use for each simulated data point.
   - In the **Data Points** field, enter the number of data points you wish to generate.
   - In the **Folio** and the **Sheet** drop-down lists, choose where to put the simulated data.
     - If you choose **<New Folio>** the software will create a new folio to place the data into. If you choose **<New Sheet>** the software will create a new sheet in a new or existing folio, depending on what you chose in the **Folio** drop-down list.
     - Click the **Active** button to select the folio and data sheet that were active when the utility was opened. If you would like the utility to always select the active folio and data sheet, select the check box labeled **Select active folio/sheet** when loading this window.



After the simulation is set up, click **Generate** to create a data set according to your specifications. An ALTA standard folio containing the data will appear.

17.3 SimuMatic

SimuMatic (which comes in a Weibull++ version and an ALTA version) generates a large number of data sets using Monte Carlo simulation. It then analyzes the group of data sets as a whole. For example, you can use SimuMatic to find the average reliability at a given time for a thousand simulated data sets. This section includes information on how to generate and analyze data using SimuMatic.

17.3.1 Generating Data with Weibull++ SimuMatic

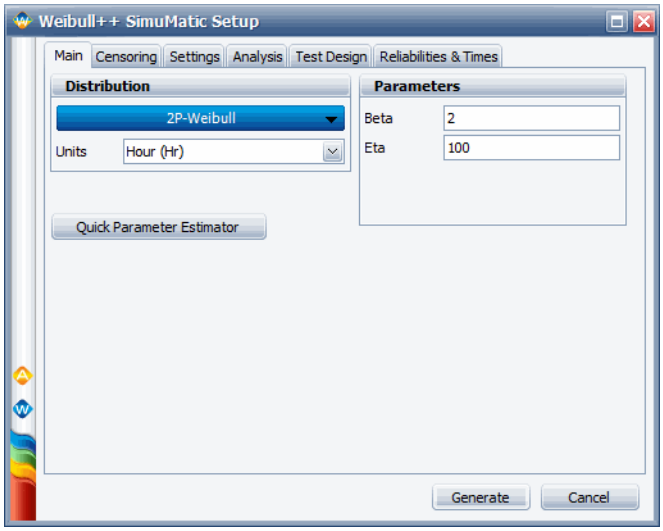
WEIBULL++ On

Weibull++ SimuMatic generates multiple data sets using Monte Carlo simulation (the same method used by the Monte Carlo utility discussed above). It then analyzes each data set individually (e.g., to find the reliability at a specified time) and the group of data sets as a whole (e.g., to find the average reliability at a specified time across the data sets).

To access Weibull++ SimuMatic, choose **Insert > Tools > Weibull++ SimuMatic**.



The setup window will appear as shown next.



Follow the steps outlined below to generate and analyze the simulated data sets.

- Configure the Main, Censoring and Settings tabs of the SimuMatic setup window in the same way you would for the Monte Carlo utility (*see steps 1 through 3 described in Section 17.1 on page 432*). This will determine how each data set is generated.
- On the Settings tab, enter the number of data sets you wish to generate in the **Number of data** sets field. In most cases, you will want this number to be reasonably large (e.g., **1000**).
- On the Analysis tab, select how you want to calculate the parameters of the simulated data sets.
  - Choose the appropriate option from the **Analysis Method** drop-down list. There are three options to select from: rank regression on X (RRX), rank regression on Y (RRY) and maximum likelihood estimation (MLE).

- Choose the appropriate option from the **Rank Method** drop-down list. The available options are median ranks (MED) and Kaplan-Meier (K-M).
  - Select the **Use RRM for interval** check box to use the ReliaSoft ranking method (RRM) for interval data. If this check box is cleared or if there is no interval data, then the software will use the standard ranking method (SRM).
- Enter a confidence level percentage in the **Confidence Bounds on Plot** area. If you choose to create a plot of the data points after you create them, the software will use this value to display confidence bounds on the plot. For example, if you enter **80**, the software will display the 80% two-sided confidence bounds on time, reliability or both, depending on which option you select on the control panel of the Weibull++ SimuMatic folio's plot sheet.

- On the Test Design tab, select the **Calculate test plan results** check box if you wish to use SimuMatic to design or evaluate a reliability test. The software will then analyze the data sets in order to estimate the time at which the product would have a specified reliability (i.e., the *demonstrated life*). This value can help you determine the combination of test time and sample size that will be adequate to demonstrate a required reliability target at a specified confidence level.

> **Note:** Assuming that all the generated data sets have calculated parameters, the software will calculate the demonstrated life by taking each data set and solving for the time at which a product with that data set's distribution would have the specified reliability. It will then order these time values to find the time value that corresponds to the specified confidence level. For example, if you enter **85** for the target reliability and **90** for the confidence level, the software will, for each data set, solve for the time at which a product with that data set's distribution would have a reliability of 85%. It would then order the calculated time values and locate the value in the 10th percentile, which is reported as the 90% lower one-sided confidence bound for the demonstrated time.

- On the Reliabilities and Times tab, enter up to 10 reliability values (as decimals less than 1) and up to 10 time values. SimuMatic will calculate the time for every given reliability, and it will calculate the reliability for every given time.
  - For example, if you enter **0.5** in the **Reliabilities** area of this tab, the data sheets in the SimuMatic folio will display the estimated time when the reliability will be equal to 50%. If you enter **100** in the **Times** area of this page, the reliability at time 100 will be displayed.
- Click **Generate** to create and analyze the data. You can view the analysis results in the SimuMatic folio. *See Section 17.3.3 on page 444*.

17.3.2 Generating Data with ALTA SimuMatic

ALTA Only

ALTA SimuMatic generates multiple data sets using Monte Carlo simulation (the same method used by the Monte Carlo utility discussed above). It then analyzes each data set individually (e.g., to find the reliability at a specified time) and the group of data sets as a whole (e.g., to find the average reliability at a specified time across the data sets).

To access ALTA SimuMatic, choose **Insert > Tools > ALTA SimuMatic**.



The setup window will appear. Follow the steps outlined below to generate and analyze the simulated data sets.

- Configure the Main, Censoring and Settings tabs of the SimuMatic setup window in the same way you would for the Monte Carlo utility (*see steps 1 through 5 described in Section 17.2 on page 438*). This will determine how each data set is generated.
- On the Settings tab, enter the number of data sets you wish to generate in the **Number of data sets** field. In most cases, you will want this number to be reasonably large (e.g., **1000**).

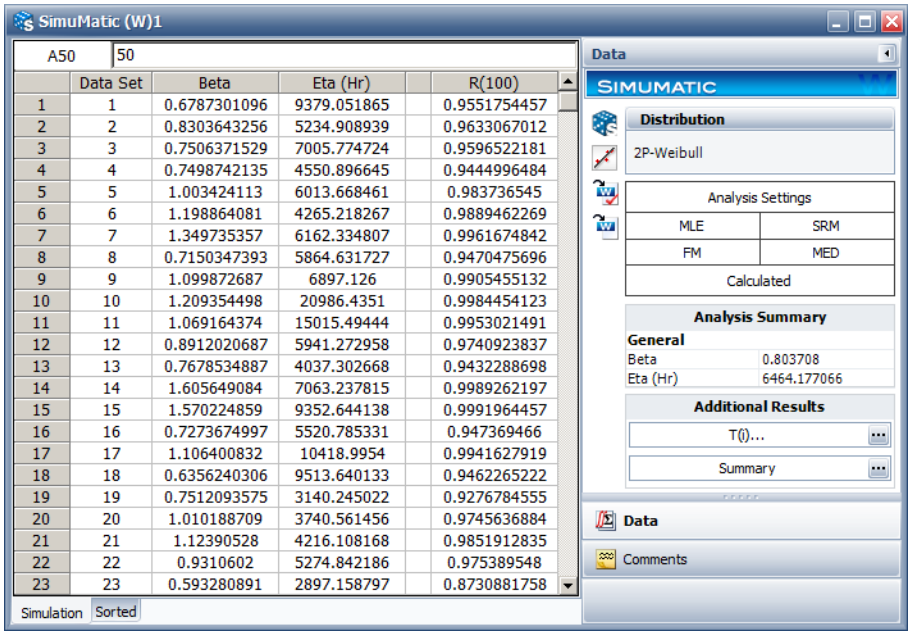
- On the Analysis tab, enter a confidence level percentage in the **Confidence Bounds on Plot** area. If you choose to create a plot of the data points after you create them, the software will use this value to display confidence bounds on the plot. For example, if you enter **80**, the software will display the 80% two-sided confidence bounds on time, reliability or both, depending on which option you select on the control panel of the ALTA SimuMatic folio's plot sheet.
- On the Test Design tab, select the **Calculate test plan results** check box if you wish to use SimuMatic to design or evaluate a reliability test. The software will then analyze the data sets in order to estimate the time at which the product would have a specified reliability (i.e., the *demonstrated life*). This value can help you determine the combination of test time and sample size that will be adequate to demonstrate a required reliability target at a specified confidence level.

> **Note:** The software calculates the demonstrated time by taking each data set and solving for the time at which a product described by that data set's model would have your specified reliability. It then orders these time values to find the time value that corresponds to your specified confidence level. For example, if you enter **85** for the target reliability and **90** for the confidence level, the software will, for each data set, solve for the time at which a product described by that data set's model would have a reliability of 85%. It would then order the calculated time values and locate the value in the 10th percentile, which is reported as the 90% lower one-sided confidence bound for the demonstrated time.

- On the Reliabilities and Times tab, enter up to 10 reliability values (as decimals less than 1) and up to 10 time values. SimuMatic will calculate the time for every given reliability, and it will calculate the reliability for every given time.
  - For example, if you enter **0.5** in the **Reliabilities** area of this tab, the data sheets in the SimuMatic folio will display the estimated time when the reliability will be equal to 50%. If you enter **100** in the **Times** area of this page, the reliability at time 100 will be displayed.
- Click **Generate** to create and analyze the data. You can view the analysis results in the SimuMatic folio.

17.3.3 SimuMatic Folio

The SimuMatic folio gives you access to the data sets generated by the SimuMatic utility and to the results of analyses on those data sets. The Weibull++ version of the folio is shown next.

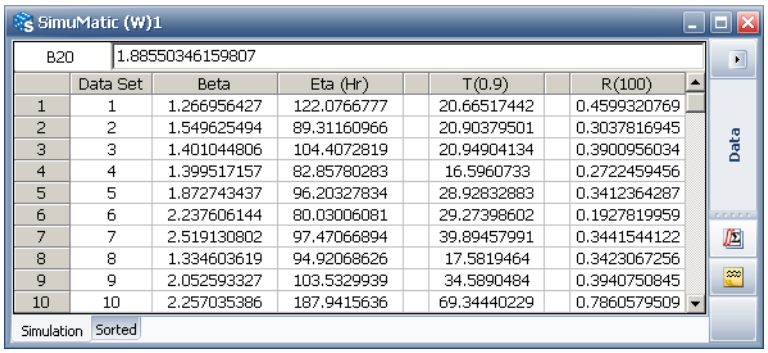


This section includes information on various elements of this folio, including:

- The Simulation sheet
- The Sorted sheet
- The control panel
- The Plot sheet

17.3.3.1 Simulation Sheet

The Simulation sheet shows the parameters calculated for each data set. It also contains a separate column for each of the calculations that you specified on the Reliabilities and Times tab of the setup window. The following example shows the results for the first ten data sets created with SimuMatic.


SimuMatic (W)1

B20 1.88550346159807

	Data Set	Beta	Eta (Hr)	T(0.9)	R(100)
1	1	1.266956427	122.0766777	20.66517442	0.4599320769
2	2	1.549625494	89.31160966	20.90379501	0.3037816945
3	3	1.401044806	104.4072819	20.94904134	0.3900956034
4	4	1.399517157	82.85780283	16.5960733	0.2722459456
5	5	1.872743437	96.20327834	28.92832883	0.3412364287
6	6	2.237606144	80.03006081	29.27398602	0.1927819959
7	7	2.519130802	97.47066894	39.89457991	0.3441544122
8	8	1.334603619	94.92068626	17.5819464	0.3423067256
9	9	2.052593327	103.5329939	34.5890484	0.3940750845
10	10	2.257035386	187.9415636	69.34440229	0.7860579509

Simulation Sorted

Data

The sheet contains the following columns:

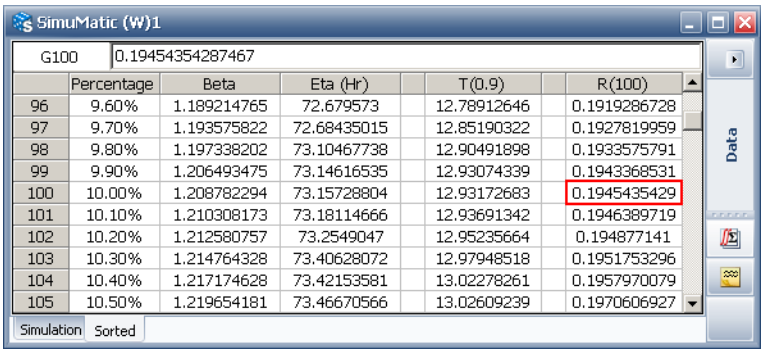
- **Beta** and **Eta (Hr)** are parameter columns. Parameter columns show the estimated parameters for each data set based on your selected distribution and analysis method. For parameters with an associated unit of measurement (e.g., the Weibull shape parameter eta), the unit will be indicated in the column header.
- **T(0.9)** is a time value column. It displays the time value calculations based on the calculated parameters and a reliability value you entered in the Reliabilities and Times tab of the setup window. In this example, it shows the time at which the product's reliability is expected to be 0.9, or 90%.
- **R(100)** is a reliability value column. It displays the reliability calculations based on the calculated parameters and a time value you entered in the Reliabilities and Times tab of the setup window. In this example, it shows the reliability value at time = 100.

Note: "N/A" will appear in the parameter columns if there was an error in calculating a particular data set. This is usually an indication that there were not enough failure times in the data set to calculate the parameters.

17.3.3.2 Sorted Sheet

In the Sorted sheet, the calculated values from the Simulation sheet are sorted from least to greatest in order to show the confidence bounds. Assuming all the simulated data sets have calculated parameters, the Percentage column will display the percentage of data sets that are equal to or less than a given data set. So, for example, if you wished to

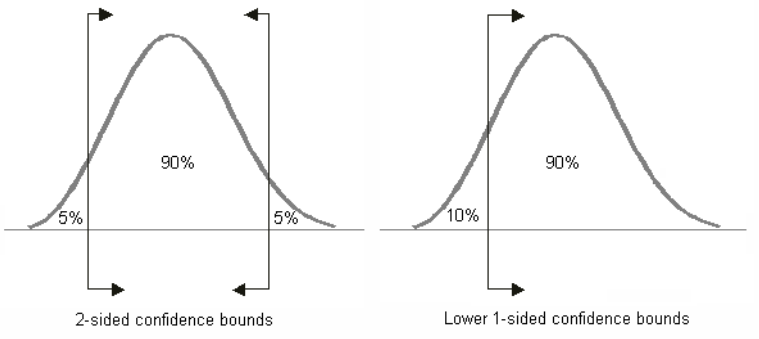
obtain the 90% lower one-sided confidence bound of R(100), you would look up the value of R(100) that corresponds to 10% (i.e., 100% - 90%).



	Percentage	Beta	Eta (Hr)	T(0.9)	R(100)
96	9.60%	1.189214765	72.679573	12.78912646	0.1919286728
97	9.70%	1.193575822	72.68435015	12.85190322	0.1927819959
98	9.80%	1.197338202	73.10467738	12.90491898	0.1933575791
99	9.90%	1.206493475	73.14616535	12.93074339	0.1943368531
100	10.00%	1.208782294	73.15728804	12.93172683	0.1945435429
101	10.10%	1.210308173	73.18114666	12.93691342	0.1946389719
102	10.20%	1.212580757	73.2549047	12.95235664	0.194877141
103	10.30%	1.214764328	73.40628072	12.97948518	0.1951753296
104	10.40%	1.217174628	73.42153581	13.02278261	0.1957970079
105	10.50%	1.219654181	73.46670566	13.02609239	0.1970606927

If you wished to obtain the 90% two-sided confidence bounds of R(100), you would look up the values that correspond to 5% (for the lower bound) and 95% (for the upper bound).

This concept is summarized in the figure shown below.



17.3.3.3 SimuMatic Control Panel

The SimuMatic control panel and its components are described next.

- The **Distribution** area displays the type of distribution that was used to generate each data set.
- The **Analysis Settings** area displays what analysis settings were used to analyze the simulated data.
- The **Analysis Summary** area displays the parameters that were used to generate each data set.
- In the **Additional Results** area of the control panel:
  - The **T(i) (...)** button opens a report that displays each data set in its own column or group of columns. Each calculated data set is separated from the others with an empty data sheet column. This report will contain multiple sheets when there are too many data points to fit onto one sheet.

    Note: Data sets without enough data points to fit to a model (i.e., those with an “N/A” in the parameter columns of the SimuMatic's data sheet) will not be shown in this report.

  - The **Summary (...)** button opens a report that displays the settings used to generate the data. It also displays information about the variation of the parameter values and, if you entered a target reliability on the Test Design tab of the setup window, it will display the results as well.

The following tools are available on the SimuMatic control panel:



SimuMatic Setup allows you to change your simulation settings and replace your current simulated data sets with new ones.



Plot generates a probability plot of all the data sets.



Transfer Parameters to Selected Folio is available only for Weibull++. It allows you to transfer the calculated parameter values for each simulated data set to an existing Weibull++ standard folio, with one data sheet for each parameter of the model that was used to generate the data. When you click this icon, a window will appear asking whether you want the data sheets to be configured for free-form or times-to-failure data. Then the Transfer Life Data window will appear, where you can select the existing folio that you wish to transfer the calculated parameters to.



Transfer Parameters to New Folio is available only for Weibull++. It creates a new Weibull++ standard folio and transfers the calculated parameter values for each simulated data set to that folio, with one data sheet for each parameter of the model that was used to generate the data. After you click this icon, a window will appear asking whether you want to transfer the data to a data sheet configured for free-form or times-to-failure data.

17.3.3.4 Plot Sheet

The following options on the plot sheet are unique to SimuMatic plots. They allow you to control which elements will be displayed on the plot. *For more general information on plot sheets, see Chapter 14*.

- **Simulation Lines** plots the time vs. unreliability line for every simulated data set.
- **True Parameter Line** plots the time vs. unreliability line defined by the parameters of the distribution that was used to generated the simulated data. (These parameters are visible on the control panel of the Simulation sheet.)
- **Median Line** plots the time vs. unreliability line defined by the median parameters of all generated data sets.
- **Average Line** plots the time vs. unreliability line defined by the average parameters of all generated data sets.
- **CB on Reliability** and **CB on Time** show confidence bounds on the plot. The percentile of the confidence bounds is determined by what you entered for **Confidence Bounds on Plot** on the Analysis tab of the setup window.
  - For example, if you entered 90% in the setup window and selected **CB on Time**, then the displayed confidence bounds would mark the simulated time values at the 5th and 95th percentile for every given reliability value.
- **Target** is available only if you have specified a target reliability on the Test Design tab of the setup window. This option marks the target reliability as a point at the intersection of a horizontal line drawn at the target unreliability and a vertical line drawn at the target time.

17.3.4 Transferring Data to a Weibull++ Standard Folio

WEIBULL++ On

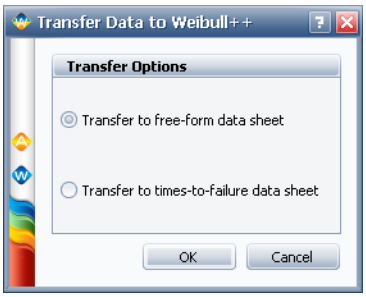
The Weibull++ version of SimuMatic allows you to transfer the calculated parameter values for each simulated data set to a new or existing Weibull++ standard folio, with one data sheet for each parameter of the model that was used to generate the data. Uncalculated data sets (i.e., those showing "N/A" in the parameter column of the SimuMatic folio) will be ignored.

17.3.4.1 Transferring Data to a New Folio

To transfer data to a new folio, choose **SimuMatic(W) > SimuMatic > Transfer Life Data to New Folio** or click the icon on the control panel.



The window shown next will appear.



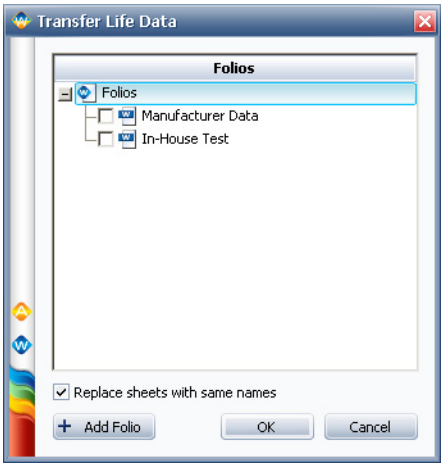
Select whether the data will be transferred to a free-form or times-to-failure data sheet. After you click **OK**, a new standard folio will be created containing the simulated data in the selected format.

17.3.4.2 Transferring Data to an Existing Folio

To transfer data to an existing folio, choose **SimuMatic(W) > SimuMatic > Transfer Life Data to Selected Folio** or click the icon on the control panel.



After you select whether the data will be transferred to a free-form or times-to-failure data sheet, the Transfer Life Data window will appear, as shown next.



Select the folio that the data will be transferred to.

- If the **Replace sheets with same names option** is selected, then the software will replace any data sheets that have the same name as the data to be transferred. For example, if a selected folio already has a Beta sheet and you will be transferring beta values calculated in SimuMatic, then the software will replace the data in the current Beta sheet with the new beta values. If this option not selected, then the software will always create a new sheet in the selected folio.
- Click the **Add Folio** button to automatically generate a blank Weibull++ standard folio and display it in the window.

17.3.5 Designing Reliability Tests with SimuMatic

One of the applications of SimuMatic is simulating the outcome from a particular test design that is intended to demonstrate a target reliability. You can specify various factors of the design, such as the test duration (for a time-terminated test), number of failures (for a failure-terminated test) and sample size. By running the simulations you can assess whether the planned test design can achieve the reliability target. Depending on the results, you can modify the design by adjusting these factors and repeating the simulation process—in effect, simulating a modified test design—until you arrive at a modified design that is capable of demonstrating the target reliability within the available time and sample size constraints.

Tip: SimuMatic is useful for designing tests where there will be enough failure times observed during the test to analyze the data and fit a life distribution (e.g., two or more observed failures are required for most 2-parameter distributions). To design a test that demonstrates the target reliability with minimal "allowable failures" (e.g., a zero-failure test), use the Reliability Demonstration Test Design tool. *See page 470 in Chapter 19*.

Follow the steps outlined below to use SimuMatic's test planning results to design a reliability life test. The calculations described here can also be used for other types of what-if analyses designed to explore the life estimates that can be obtained from different types of data sets.

- You must be able to provide reasonable assumptions concerning your product's underlying life distribution. Specify these assumptions on the Main tab of the SimuMatic setup window.
- Enter a sample size in the **Number of points** field on the Settings tab. If you are solving for the sample size, start with a large value. You can later repeat the simulation with smaller values until you arrive at an acceptable test plan.
  - In the Weibull++ version, this will be entered on the Settings tab.
  - In ALTA, you will use the data sheet to enter a separate sample size for each stress level that will be applied during the test.
- On the Censoring tab, select an appropriate censoring scheme.
  - If you are designing a test that will end only after all units fail, select **No censoring**. All simulated time values will be considered failure times.
  - If you are designing a time-terminated test (i.e., a test that will end at a specified time), select **Right censoring after a specific time** and enter an estimated test duration in the **Time** field. All simulated time values that exceed this limit will be considered suspensions at the specified time (i.e., units that had not failed by the end of the test). If you are solving for the required test duration, enter a large estimate, and then you can repeat the simulation with smaller values until you arrive at an acceptable test plan.

    Note: A short censoring time may create many data sets with too few failures to estimate an underlying life distribution. These data sets will display "N/A" for the parameters on the SimuMatic folio's data sheet. This can be due to an insufficient sample size or a test termination time that is too short. As a rule of thumb (and for 2-parameter distributions) the combined sample size and test duration should be sufficient to observe three or more failures. In other words, if you have a sample size of 10 then the test duration should be greater than the product's B30 life (i.e., the time at which unreliability = 30%).

  - If you are designing a failure-terminated test (i.e., a test that will end after a specified number of failures occur), select **Right censoring after a specific number of failures** and enter a value in the **Number of Failures** field. The tool will simulate a test that ends after this number of failures, with the surviving units

marked as suspensions. If you are solving for the number of failures needed, enter a large estimate, and then you can repeat the simulation with smaller values until you arrive at an acceptable test plan.

> **Tip:** You can also choose the **Random censoring** option if you wish to explore the results that can be obtained from data sets that contain certain types of uncertainty. In this type of analysis, you can use the T1, T2 and DELTA values to estimate whether you'll be able to use the data set to demonstrate a required reliability target and evaluate how much the demonstrated life varies from the intrinsic reliability.

- On the Settings tab, make sure an appropriate number of data sets (e.g., 1,000) has been specified in the **Number of Data Sets** field. Lower numbers may lead to less accurate results.
- Select the appropriate analysis settings on the Analysis tab.
- On the Test Design tab, select to **Calculate test plan results** and enter the **Target Reliability** you wish to demonstrate along with the **Lower 1-Sided Confidence Level** at which you want to demonstrate it.
- Click **Generate** to create and analyze the simulated data.
- On the control panel, click the **Summary (...)** icon inside the **Additional Results** area to open the Results window. The **Test Planning Results** area of the report displays your test design inputs (reliability requirement, confidence level and sample size) along with the following results:
  - **Expected Life (T1)** is the lower one-sided confidence bound on the reliable life (i.e., the time for a given reliability) calculated based on the specified distribution/parameters. This value represents the intrinsic reliability of a product with the specified distribution/parameters. So if this value does not meet your reliability goal, then it is not possible to demonstrate the desired reliability for a product with your assumed life distribution, regardless of the test time or sample size that is employed.
  - **Demonstrated Life at Given Confidence (T2)** is the lower one-sided confidence bound on the reliable life calculated based on the simulated data sets. Specifically, SimuMatic calculates the time given reliability for each data set and then sorts the results to obtain the value associated with the specified confidence level percentile. You can then compare this value to the time for which you want to demonstrate the reliability (e.g., you might want to demonstrate that the product will have a reliability of 95% at 1,000 hours, at the 90% confidence level).
    - If the demonstrated value is greater than your reliability goal, then SimuMatic predicts that the specified test plan would demonstrate the target. For example, if you wish to demonstrate that the product has a reliability of 95% at 1,000 hours and T2 = 1,200, then SimuMatic predicts that the test plan would demonstrate the target.
    - If the demonstrated value is less than your reliability goal, then SimuMatic does not predict that your test plan would demonstrate the target. For example, if you wish to demonstrate that the product has a reliability of 95% at 1,000 hours and T2 = 800, then SimuMatic does not predict that the test plan would demonstrate the target.
  - **Ratio of Expected Life/Demonstrated Life (DELTA)** is a measure of the difference between the intrinsic reliability of your product (T1) and the life demonstrated by analyzing the simulated data (T2). This value is calculated by dividing the expected life by the demonstrated life. For example, if your test design would demonstrate a reliability of 90% at 50 hours (demonstrated life), and your product has an intrinsic reliability of 90% at 100 hours (expected life), then DELTA = 2.
  - **Expected Test Duration (T3)** is the average of the last failure time from each of the simulated data sets. In most contexts, this value provides an indication of how long you would need to run a test in order to demonstrate the reliable life calculated in T2. In the case of a time-terminated test, you can compare this value against the specified censoring time. If the censoring time (i.e., planned test time) is greater than T3, you may

choose to repeat the simulation with smaller test times to see if it's feasible to demonstrate the target with a less costly test plan. In addition, please note the following:

- T3 is an average of all the test runs.
- T3 is calculated based on the simulated data sets, and therefore it can never be greater than the specified censoring (test) time. If you set a test time that is too small, T3 may be identical to the censoring time but the test plan is still not adequate unless the demonstrated life (T2) also meets the requirement.

- If the results indicate that you will not be able to demonstrate the target reliability, or if they indicate that the current test plan uses more samples or more test time than necessary to demonstrate the target, you can keep repeating the simulation with gradual adjustments until you arrive at an optimum plan.

Note that because the results are obtained through Monte Carlo simulation, you may arrive at slightly different answers each time you run the analysis.

17.3.6 Weibull++ SimuMatic Example

WEIBULL++ On

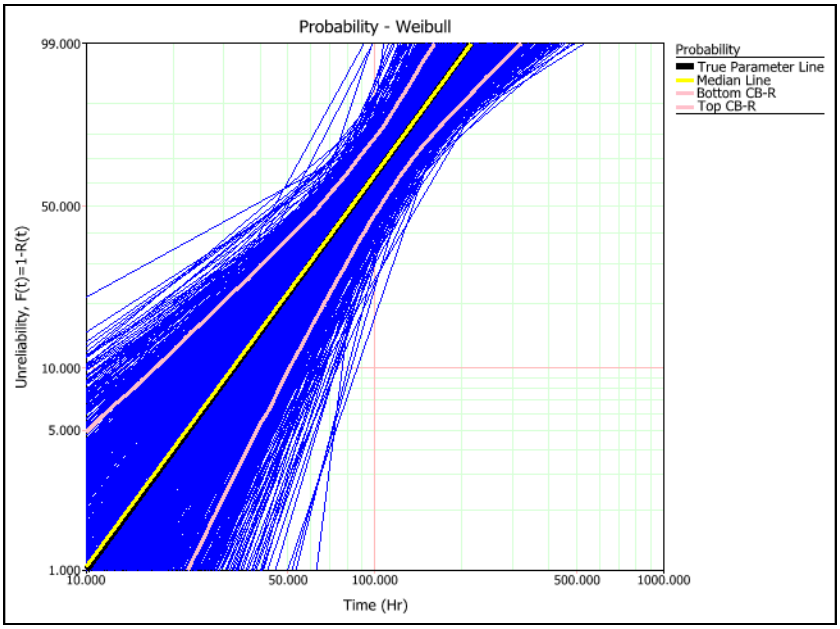
This example uses Weibull++ SimuMatic to compare different parameter estimation methods for a sample of 10 units following a Weibull distribution where beta = 2 and eta = 100, with all units tested to failure.

Follow the steps outlined below to see the results of an analysis that uses the rank regression on X (RRX) parameter estimation method.

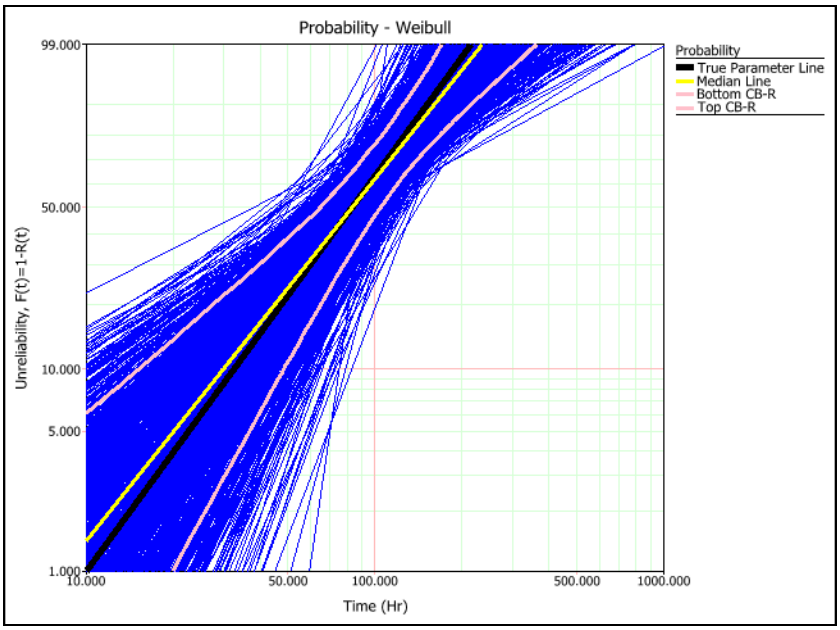
- Add a new Weibull++ SimuMatic window to the project by choosing **Insert > Tools > Weibull++ SimuMatic**.
- On the Main tab, select the **2P-Weibull** distribution from the **Distribution** drop-down list, and select **Hour (Hr)** for the time unit. Enter **2** in the **Beta** field and **100** in the **Eta** field.
- On the Censoring tab, select **No censoring**.
- On the Settings tab, enter **1000** in the **Number of data sets** field and enter **10** in the **Number of points** field.
- On the Analysis tab, choose **Maximum Likelihood (MLE)** from the **Analysis Method** drop-down list. This method will be used to estimate the parameters of the generated data sets.
- Also on the Analysis tab, choose **Median Ranks** from the **Rank Method** drop-down list and clear the **Use RRM for interval data** check box. Enter **90** in the **Confidence Level** field.
- Click **Generate** to create and analyze the data. A SimuMatic folio will appear.
- Click **Plot** on the control panel. You will be taken to the Plot page of the folio. In the **Show** area of the control panel, make sure the following options are selected.
  - **Simulation Lines**
  - **True Parameter Line**
  - **Median Line**
  - **CB on Reliability**
- Click **Plot** again to refresh the plot display all the selected lines.

To compare RRX to the rank regression on Y (RRY) and maximum likelihood estimation (MLE) methods, repeat the steps above, but choose a different option in the **Analysis Method** drop-down list on the Analysis tab. Then compare the three plots.

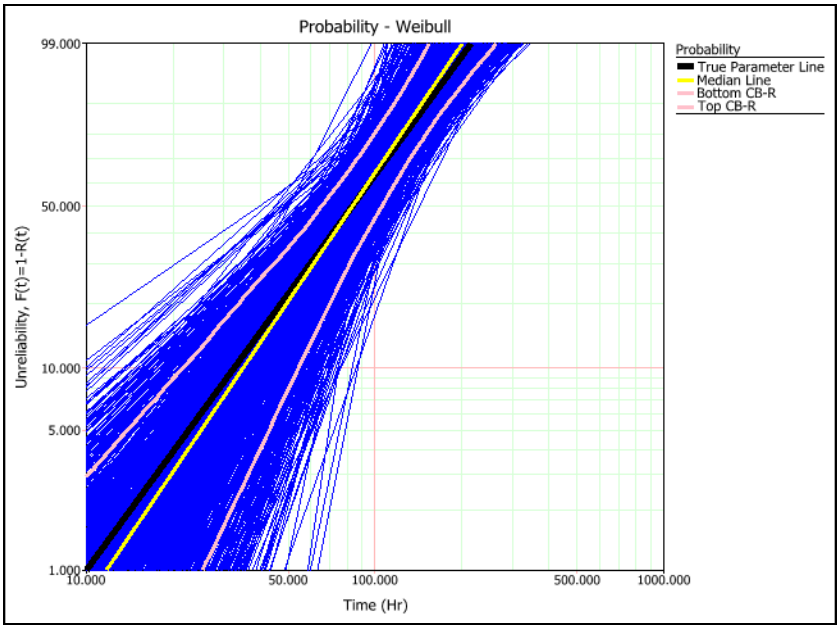
The plots for the three methods are shown below:



RRX Method



RRY Method



MLE Method

The thick line running through the middle of the plot shows the time vs. unreliability defined by the parameters of the distribution that was used to generated the simulated data. The thinner median line shows the time vs. unreliability defined by the median parameters of all generated data sets. By comparing these two lines, you can estimate how close the estimated parameters would be to the product's assumed parameters (which you provided in the setup). The plots above show that RRX would provide the least deviation (i.e., "bias") from the truth for this sample size and data type.

The lines marking the lower and upper confidence bounds on reliability allow you to determine how closely grouped together the parameter estimation results are. The plots above show that MLE would provide results with less variation than any of the other analysis methods for this sample size and data type.

Thus given our relatively small sample size and uncensored data, RRX is shown to be best in terms of bias, while MLE is shown to be best in terms of variation (the median line and true parameter line overlap almost perfectly).

17.3.7 ALTA SimuMatic Example

ALTA Only

This example uses ALTA SimuMatic to verify whether a proposed test plan could be used to demonstrate, with 95% confidence, that a product has a reliability of at least 90% at 650 hours at the use stress level of 300 K.

The engineers have prior experience with a similar design, and as a result they expect the product to have an unreliability of 40% at 7,000 hours under normal conditions. At 340 K, they expect the product to follow a Weibull distribution, where eta = 500 hours. The life-stress relationship is assumed to be Arrhenius, and for all stress levels the life distribution is assumed to be Weibull, where beta = 1.5.

The proposed plan is to test 20 units at 325 K and 15 units at 340 K. All units will be tested to failure. Follow the steps outlined below to determine whether this test plan will meet the reliability goal:

- Add a new ALTA SimuMatic window to the project by choosing **Insert > Tools > ALTA SimuMatic**. Then, on the Main page of the ALTA SimuMatic Setup window that appears, select the **Arrhenius-Weibull** model and enter **1.5** for the beta parameter and **Hour (Hr)** for the time unit.

- Because we do not have the values of B and C available, we will use the Quick Parameter Estimator to solve for those parameters and transfer them to the setup window. Click the **Quick Parameter Estimator** button to open the utility. The **Arrhenius-Weibull** model and **Hour** units should be automatically selected in the **Model** area.
  - In the **Quantification Method** area, choose **Unreliability and a Parameter**. Enter the life-stress values from the assumptions given above.
  - In the **Point #1** area, enter **7000** for the time and **0.4** for the unreliability.
  - In the **Solve for Parameter** area, select **Eta** and then enter **1.5** in the **Beta** field.
  - In the **Use Stress Level** area, enter **300** for the use stress level.
  - Click the **Next>** button to enter the accelerated life estimate.
  - On the second page, enter **340** for the stress level and **500** for the characteristic life (i.e., eta).
  - Click **Next>** to see the calculated the B and C parameters, which are shown to be 7871.5321 and 4.4903E-08, respectively. Then click the **Update** button to transfer the parameters to the SimuMatic Setup window.
- On the Stress tab of the SimuMatic window, enter the use stress level of **300** K.
- On the Censoring tab, select **No censoring**.
- On the Settings tab, set the **Number of data sets** to **1000**. Use the default values in the other fields.
- On the Test Design tab, enter **90** in the **Target Reliability** field and **95** in the **Lower 1-Sided Confidence Level** field.
- In the data sheet on the right side of the window, enter the number of units that will be tested for each stress level, as shown next.

	Number of points	Stress1
1	20	325
2	15	340

- Click **Generate** to create and analyze the simulated data sets.
- In the SimuMatic folio, click **Summary** in the **Additional Results** area of the control panel to open the Results window. T1 confirms that the product's intrinsic reliability is high enough to meet the target reliability. Also, because T2 in the **Test Planning Results** area is greater than the time at which the engineers want to demonstrate the target reliability, SimuMatic predicts that the given test plan is acceptable. The engineers may then choose to decrease the number of points and repeat the simulation to determine whether a more efficient plan is possible. Note that your results may vary because the data were obtained through Monte Carlo simulation.

39	**Test Planning Results**	
40		Required test time
41	Expected Test Duration (T3)	800
42		Demonstrated Reliability
43	Demonstrated Life at Given Confidence (T2)	772.2144
44	Target Reliability (R)	90%
45	1-Sided Lower Confidence Level (CL)	95%
46		Variation from True Value
47	Expected Life (T1)	2488.4971
48	Ratio of Expected Life/Demonstrated Life (DELTA)	3.2225

Diagrams 18

If you have ReliaSoft's BlockSim software installed on your computer, the shared Synthesis Platform makes it possible to use a model (i.e., a shared Synthesis resource) that is based on a Weibull++ or ALTA standard folio to define any of the probabilities in BlockSim's reliability block diagrams (RBDs) and fault trees. For convenience, a simplified version of BlockSim's analytical RBD tool is also directly integrated within the Weibull++/ALTA interface.

To create a diagram directly within Weibull++/ALTA, choose **Insert > Tools > Diagram** or right-click the **RBDs** folder in the current project explorer and choose **Add Diagram** on the shortcut menu.



For every standard block that you add to this type of diagram, you will be prompted to select a calculated data sheet from a Weibull++ or ALTA standard folio that the block will represent. You can then use connectors, node blocks and/or junction blocks to build an RBD, calculate the exact system reliability equation for the system, and obtain plots and results based on that equation.

Note: If the analysis in the data sheet changes, the reliability characteristics of the block will be updated automatically, and you can double-click the block to open the associated data folio.

One typical application for this tool is for performing complex failure modes analysis. As explained on page 240 in Chapter 6, the CFM analysis method that is built-in to the Weibull++ standard folio assumes a series reliability model and uses only one type of distribution to analyze all the modes. If the situation is more complex, an RBD analysis method must be used instead. This applies to any or all of the following circumstances:

- The analysis involves more than four failure modes.
- The failure modes are described by different life distributions (e.g., Modes A and B follow a Weibull distribution but Mode C follows a lognormal).
- The relationship between the failure modes does not follow a series configuration. For example, if more than one failure mode must occur together in order for the component to fail, a parallel configuration must be used for the analysis and this is referred to as "complex failure modes analysis" rather than "competing failure modes analysis."

This chapter explains how to build and analyze diagrams in Weibull++/ALTA, including:

- Building a diagram - Section 18.1 (p. 456).
- Using the diagram control panel - Section 18.2 (p. 464).
- Using the Equation Viewer - Section 18.3 (p. 465).
- Using the diagram plot sheet - Section 18.4 (p. 468).
- Changing the appearance of a diagram - Section 18.5 (p. 468).

18.1 Building a Diagram

The diagram sheet is where you will build the reliability block diagram that represents the reliability-wise configuration of your system. There are three types of items available for use in diagrams: standard blocks, node blocks, and junction blocks. These are presented in the following sections.

18.1.1 Standard Blocks



Standard blocks represent data sheets in Weibull++ and ALTA standard folios within the project. These blocks enable you to represent each data set individually in the diagram.

You can add standard blocks to the diagram by choosing **Diagrams > Diagram Tool > Add Block**. This will open a window that allows you to select the standard folio and data sheet that you want the block to represent. You can also add a standard block by dragging the data sheet of interest from the Data Sheets page of the control panel into the diagram. In both cases, the list of available data sheets includes any data sheet in any Weibull++ or ALTA standard folio, as long as it has been calculated and is not using one of the competing failure modes distributions.

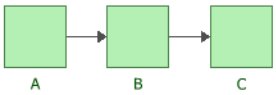
Double-clicking a standard block or choosing **Diagrams > Format & Style > Edit Block > Block Properties** opens the standard folio and data sheet represented by the block.

Standard blocks are named with the name of the data sheet they represent.

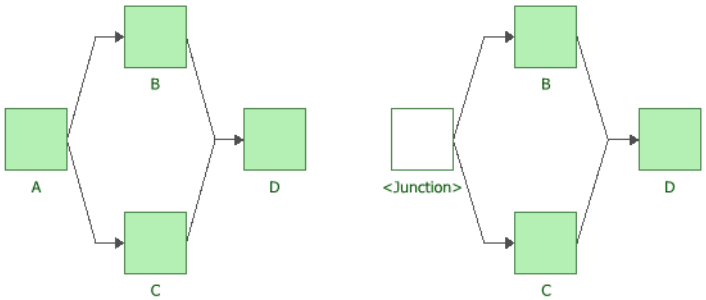
If a block represents a data sheet that has been changed and has not been calculated or a data sheet that has since been deleted, the block is marked with a red flag.

Using only standard blocks, you can define series configurations, parallel configurations and combinations of the two.

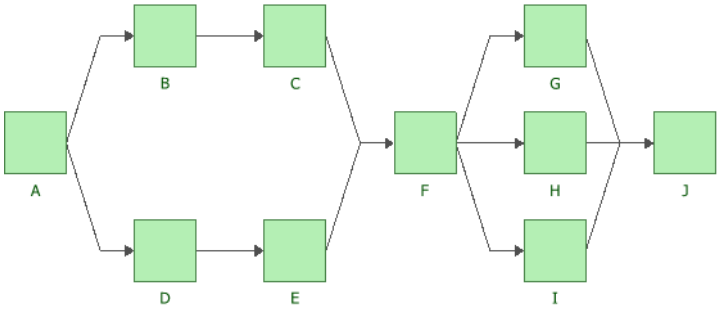
In a series configuration, all items must work for the system to work. In the RBD shown next, the system fails if either Block A, Block B or Block C fails.



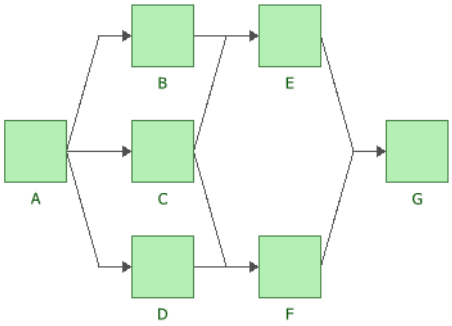
In a parallel configuration, items are considered to be redundant. In the RBDs shown next, the system will continue to function if either Block B or Block C fails.



You can combine series and parallel configurations in a single RBD, as shown next.



You can also use the software 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.

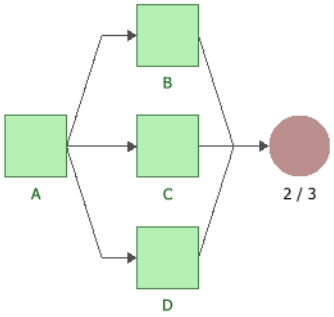


18.1.2 Node Blocks

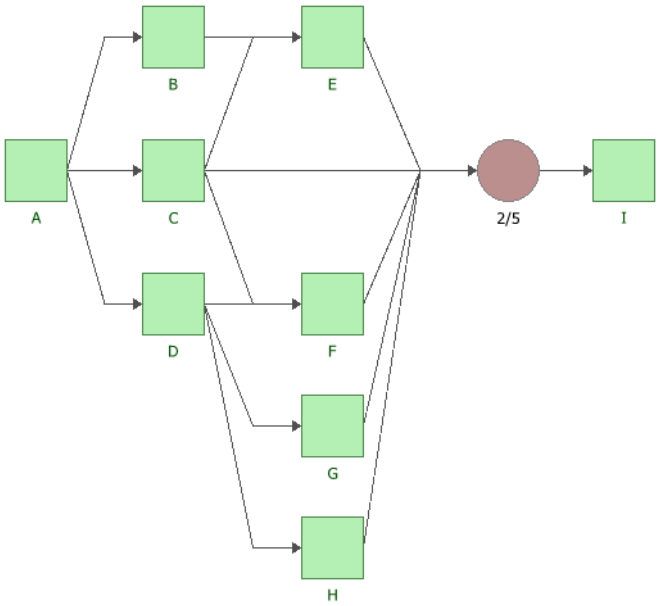


Node blocks are like switches that diagram paths move through. A node allows you to model an alternative form of redundancy known as *k-out-of-n redundancy.* A k-out-of-n node block can have *n* paths leading into it and requires that *k* out of *n* paths must function for the system to function. There must be at least one path leading into a node; therefore, a node block cannot be placed as the starting block for a diagram.

The configuration in the figure shown next displays a k-out-of-n node block where either Block B, Block C or Block D can fail without causing system failure, but if two of the blocks fail, the system will fail.

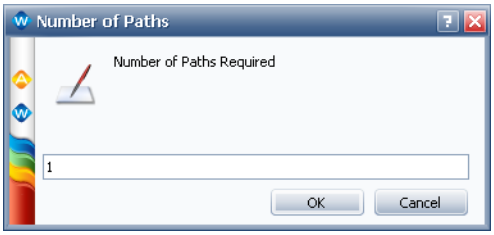


The figure shown next displays another k-out-of-n node block, this time used in a complex configuration. If more than three of the five blocks leading to the node block fail, the system will fail.



You can add node blocks to the diagram by choosing **Diagrams > Diagram Tools > Add Node**.

Double-clicking the node block or selecting **Diagrams > Format & Style > Edit Block > Block Properties** opens the properties window for the node block, which allows you to specify the number of paths that must reach the node block in order for the system to succeed.



18.1.3 Junction Blocks



Junction blocks are non-failing blocks. Because diagrams must flow from a single starting point to a single ending point, you may need to use junction blocks as starting and/or ending blocks. (*Diagram analysis constraints are presented in Section 18.1.6 on page 461*.) They can also be used anywhere within the diagram that you need a block that does not fail (i.e., does not affect the system reliability).

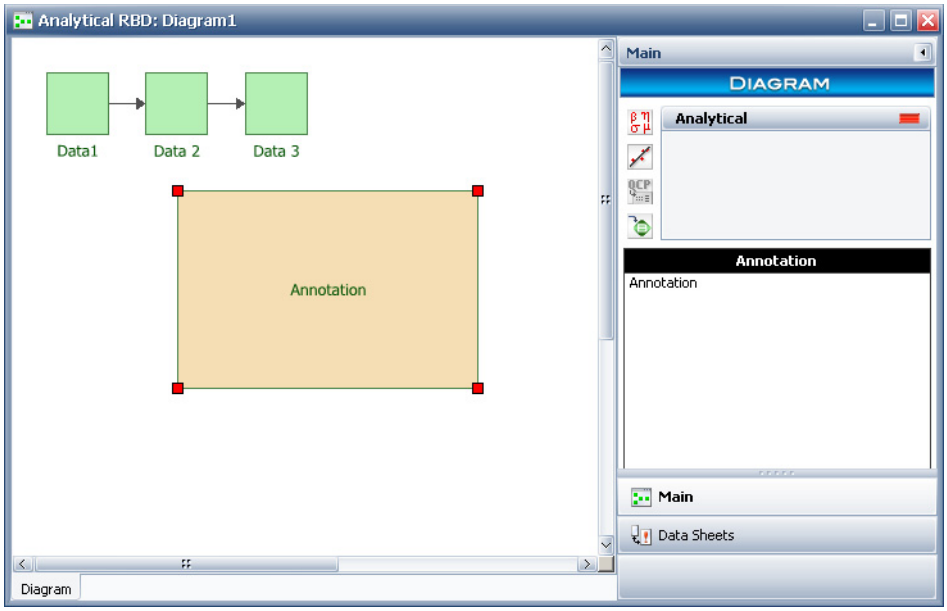
You can add junction blocks to the diagram by choosing **Diagrams > Diagram Tools > Add Junction**.

Junction blocks do not have properties, so double-clicking the junction block or selecting it and choosing Dia**grams > Format & Style > Edit Block > Block Properties** will cause a message to be displayed indicating that the block does not have properties.

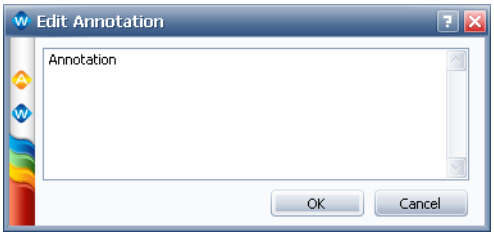
18.1.4 Annotations

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 **Diagrams > Diagram Tools > Add Annotation**. The annotation will appear in the diagram, as shown next.



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 **Diagrams > Format & Style > Edit Block > Block Properties**.



18.1.5 Connecting Blocks in Diagrams

Connectors in diagrams 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:

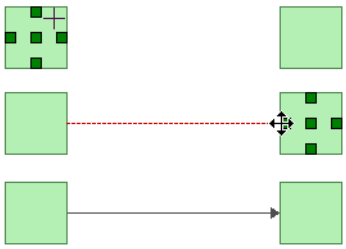
- Choose **Diagrams > Diagram Tools > Connect Blocks**.
- Hold down the **ALT** key while selecting the source block and then drag a line to the destination block.

In all methods, the cursor 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. (*For more information, see Section 18.5 on page 468*.) 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.



To stop adding relationships and return the cursor to its normal mode, right-click the diagram or clear the Co**nnect Blocks** option (i.e., by choosing the command again).

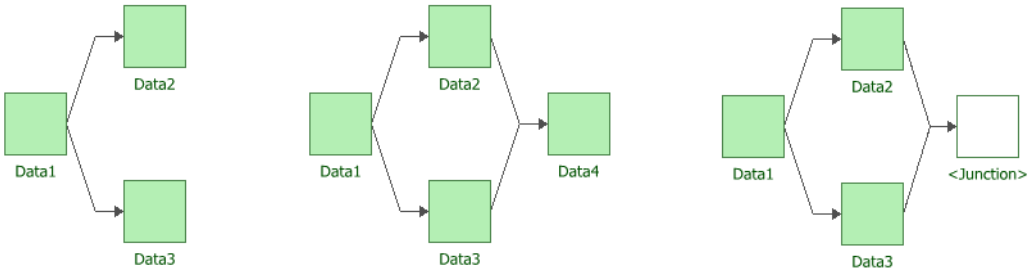
Tip: If you do not right-click the diagram or clear the **Connect Blocks** option to return the cursor to its normal mode, you will not be able to perform certain other activities in the software, such as moving or deleting blocks. If you are experiencing difficulties with the program, make sure that the cursor is in its normal mode.

Note that existing connectors can be dragged to different destination blocks, if needed.

18.1.6 Diagram Analysis Constraints

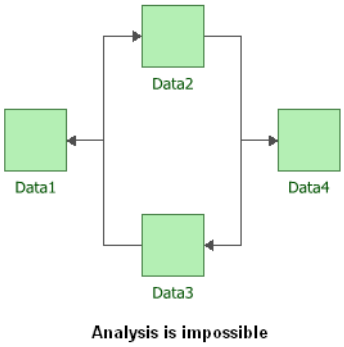
In order to permit analysis, all RBDs created in the software must conform to the following constraints:

- A diagram must contain at least one block.
- If a diagram contains more than one block, all blocks in the diagram must be connected.
- 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. Junction blocks can be used as starting and/or ending blocks that possess no reliability characteristics.

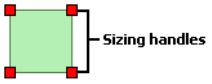
- Circular references, as shown below, are not allowed. That is, there must be a clear path towards the end of the RBD and the RBD cannot loop back on itself.

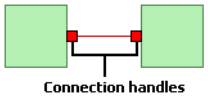
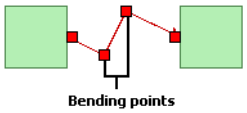


18.1.7 Keyboard and Mouse Combinations

This section describes the various keyboard and mouse key combinations that you can use while working with diagrams. 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.

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.
Resizing Objects	
Resize a block	 Select the block then click one of the sizing handles and drag it to the desired size.

If you want to do this	Do this
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 18.5 on page 468*.	 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**.
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).

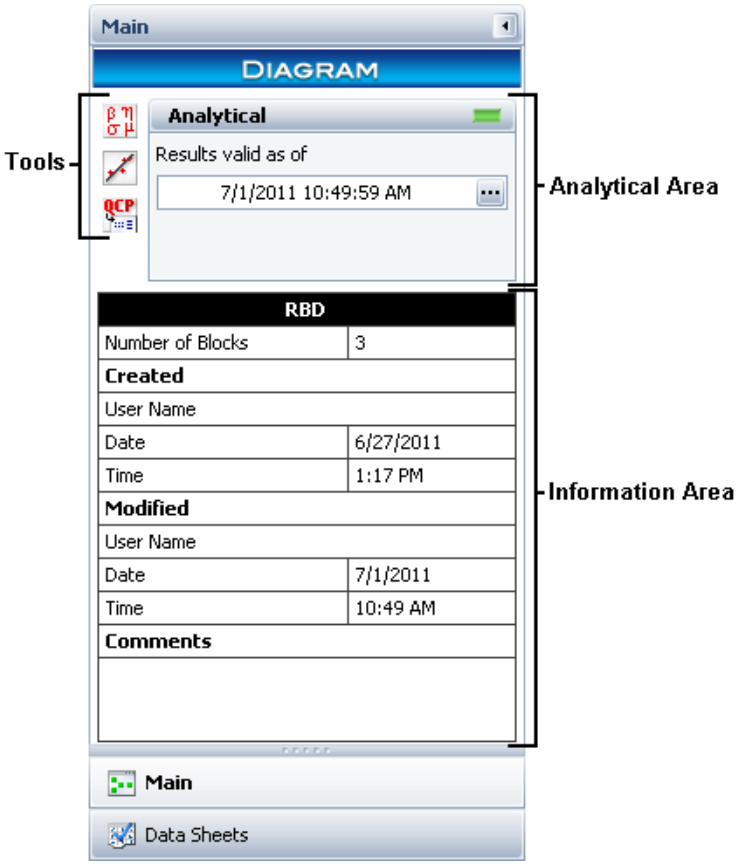
If you want to do this	Do this
Other	
Zoom in or out	Press **CTRL** and rotate the mouse wheel.
Exit the Format Painter or Connect Blocks modes	Press **ESC**.
Create a copy of a block and move the copy to the desired location.	Press **CTRL**+ **SHIFT** and click the block then drag to the desired location.

18.2 Diagram Control Panel

The diagram control panel allows you to analyze and plot the diagram. It also displays the status and information about the diagram and its blocks. It consists of two pages: Main page and Data Sheets page.

Control Panel Main Page

The Main page of the diagram control panel is shown next.



The Main page includes the following settings:

- The **Analytical** area displays the status and results of the analysis. If the light is green, as shown above, 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 or changed, a connector is removed, etc.).You can click the **(...)** button to display the algebraic solution for the diagram in the Equation Viewer. *See Section 18.3 on page 465*.
- The **Information** area displays information for the diagram or for the selected block in the diagram. If a block is not selected in the diagram, the Information area will display the number of blocks (i.e., standard blocks, node blocks and/or connector blocks) and/or annotations in the current diagram, as well as the user name, date and time for the diagram's creation and last modification. You can also click the **Comments** title to add comments on the diagram that will be displayed here.

  If a block is selected in the diagram, information for the selected item will appear in this area. This information will vary depending on the type of item that is selected. The Information area for a standard block is shown next.

Standard Block	
Data Sheet	Folio1\Data1
Failure Distribution	2P-Weibull

  The **Data Sheet** field displays the folio and data sheet that the block represents, and the Failure Distribution field displays the distribution used to calculate the data sheet. Note that for a junction block, no information is displayed. For a node block, the **Paths In** field displays the number of required paths.
- The **Tools** area 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 Quick Calculation Pad (QCP), which allows you to obtain reliability results based on the algebraic solution for the diagram. Clicking the **QCP** icon before the diagram has been analyzed will automatically perform the analysis and then open the QCP.

Control Panel Data Sheets Page

The Data Sheets page of the control panel contains a list of all of the data sheets in the project that are available for use in the diagram. This includes any data sheet in any Weibull++ or ALTA standard folio, as long as it has been calculated and is not using one of the competing failure modes distributions.

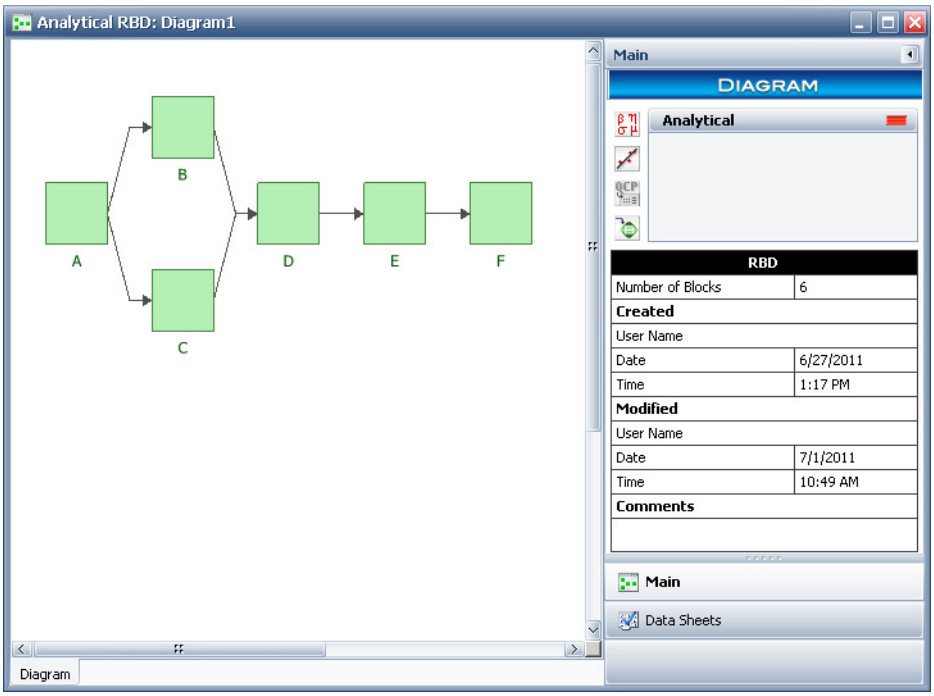
18.3 Equation Viewer

When you analyze a diagram, the software analyzes the relationships of the components and puts the reliability-wise configuration into a mathematical formula. The result of this system analysis is used to obtain reliability calculations and plots. The Equation Viewer displays the mathematical formulation of the system relationships in addition to a legend with the failure distribution for each block in the diagram.

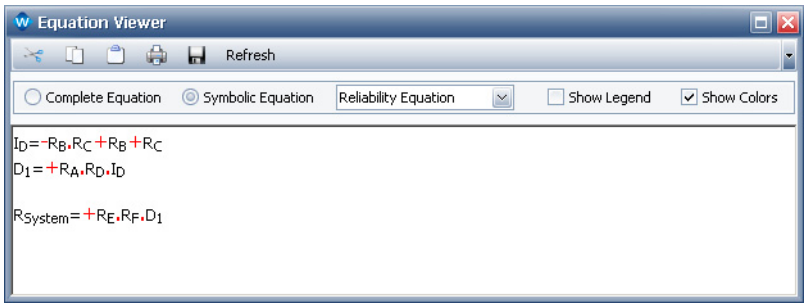
To open the Equation Viewer, click the **Show Algebraic Solution (...)** icon on the control panel.

In the Equation Viewer, 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, the symbolic reliability equation is:



This would be read as follows:

$$I_D = -R_B \cdot R_C + R_B + R_C \tag{1}$$

$$D_1 = +R_A \cdot R_D \cdot I_D \tag{2}$$

$$R_{System} = +R_E \cdot R_F \cdot D_1 \tag{3}$$

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:

$$R_{System} = +R_E \cdot R_F \cdot \left(+R_A \cdot R_D \cdot I_D\right) \quad (4)$$

Now Eqn. (4) contains the token ID, which is defined in Eqn. (1). Substituting this into Eqn. (4) yields:

$$R_{System} = +R_E \cdot R_F \cdot \left(+R_A \cdot R_D \cdot \left(-R_B \cdot R_C + R_B + R_C\right)\right) \quad (5)$$

When you use the complete equation view, the software 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, the software 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.

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.

The following icons are available in the Equation Viewer's toolbar:



Cut cuts the selected text to the Clipboard. Data stored on the Clipboard can be pasted into this and other applications.



Copy copies the selected text to the Clipboard. Data stored on the Clipboard can be pasted into this and other applications.



Paste pastes the contents of the Clipboard to the current control.



Print sends the current document to the printer. *See Chapter 29*.



Save saves the equation to a rich text file (*.rtf).

If you have made any changes to the Equation Viewer's display settings (i.e., the type of equation to be displayed, the display of the legend and/or whether to use color-coding in the window), click the **Refresh** button to update the contents of the window.

18.4 Diagram Plot Sheet and Control Panel

You can plot the data of the analysis by choosing **Diagrams > Analysis > Plot** or clicking the **Plot** icon on the Main page of the control panel. Note that clicking the **Plot** icon before the parameters have been calculated will automatically calculate the parameters and then plot the data. The scaling, setup, exporting and confidence bounds settings are similar to the options available for all other software plot sheets.

The following plot types are available for diagrams:

- **Reliability vs. Time** plot shows the reliability values over time, capturing trends in the product's failure behavior.
- **Unreliability vs. Time** plot shows the probability of failure of the product over time.
- **pdf Plot** shows a histogram of the probability density function of the data over time, allowing you to visualize the distribution of the data set.
- **Failure Rate vs. Time** plot shows the failure rate function of the product over time.

Note: Confidence bounds will not be available if any block in the diagram represents an ALTA data sheet calculated using the cumulative damage model.

18.5 Diagram Appearance

For diagrams in a Synthesis application, there are several ways to customize the appearance to fit your particular preferences and needs. Depending on the application, this may include:

- An RBD, fault tree or flowchart in BlockSim/RENO.
- An integrated RBD in Weibull++/ALTA.
- An FMEA block diagram, process flow diagram or cause and effect diagram in Xfmea/RCM++/RBI.

For any of these situations, you have the ability 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. 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).
- Save the Diagram Style settings as **reusable diagram skins**.
- Use the **Block Style** window to customize properties for one or more selected blocks.
- Use the **Connector Style** window to customize properties for one or more selected connectors.

The commands for these features are located on the diagram's **Format** tab.

See Diagram Appearance Help for information on the features available for changing the diagram appearance. (Note that if you do not currently have Internet access, this link will not work, but you can browse to the local copy of the help file by opening the Help\Diagrams\diagram_appearance_help.htm file in your Synthesis application directory.)

Test Design Folios 19

Weibull++ and ALTA include test design tools that provide ways to design reliability tests as well as evaluate and compare proposed test designs. Three of these tools can be accessed by choosing **Insert > Add New Project Items > Add Test Design**.



The Test Design Assistant will appear and ask you to select one of the following:

- The **Reliability Demonstration Test** tool solves for values associated with a specified test design, such as required test time, required sample size, demonstrated reliability and confidence level - Section 19.1 (p. 470).
- The **Expected Failure Time** plot determines what failure times you can expect in a test with a given sample size. Because you can display actual failure times on the plot, it is especially useful for determining whether a test is proceeding as expected - Section 19.2 (p. 476).
- The **Difference Detection Matrix** shows how much test time is needed to detect a statistically significant difference in life between two product designs when the data are obtained from reliability life tests and then analyzed with a Weibull++ or ALTA standard folio - Section 19.3 (p. 480).

Please note that in addition to the tools that are discussed in this chapter, the software includes two other tools that can assist you in designing reliability life tests:

- The **ALTA Test Plan** utility recommends a test plan for accelerated life testing data analysis. The recommended plan will include information about what stress levels to use in the test, as well as what sample sizes to use for each stress level. *See Chapter 24*.
- **SimuMatic**, which comes in a Weibull++ version and an ALTA version, allows you to use simulation-based methods to experiment with various testing scenarios. For example, you could try to improve a given test plan by running a simulation of the test with a smaller sample size and seeing whether the more conservative plan would still allow you to meet your test goals. *See page 449 in Chapter 17*.

19.1 Reliability Demonstration Test Design

In a zero-failure reliability demonstration test (RDT), the engineer aims to demonstrate a specified target metric (e.g., reliability at a specific time with a given confidence level) by testing a specified number of units for a predetermined time. If no failures occur, then the target metric is demonstrated. This method has been adapted for scenarios where the target metric can be demonstrated even if some failures occur, as long as a specified number of *allowable failures* is not exceeded. For example, in a demonstration test where the number of allowable failures is 2, the target metric is demonstrated if no more than 2 failures occur during the test.[1]

ReliaSoft's RDT tool can assist the user in designing a demonstration test by solving for various values related to the test, such as sample size, required test time, the demonstrated reliability and the confidence level at which the target reliability will be demonstrated.

Tip: This tool could determine, for example, that if you assume your units follow a Weibull distribution with a shape parameter of 2 and you tested 10 units for 150 hours and none failed during the test, then you would demonstrate a 90% reliability at 100 hours with 90% confidence. It does not, however, calculate the probability that one or more units might actually fail during the test. If you wish to estimate whether a failure is likely to occur, one option is to use the Expected Failure Time Plot to estimate the first failure time based on the assumed distribution and parameters.

19.1.1 Using the RDT Tool

To use the RDT tool, follow the steps outlined below:

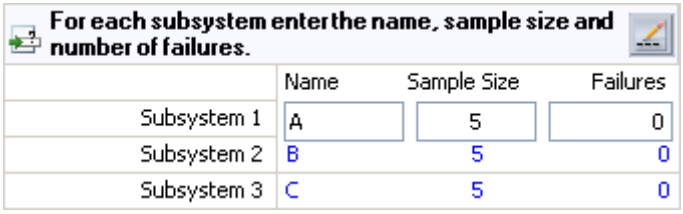
- Select the appropriate test design option from the **Test Design Method** drop-down list on the control panel and enter all the required inputs. When you change the test design option, the required inputs on the RDT sheet will change. The four options are the following:
  - The **Parametric Binomial** option determines what test duration or sample size will be required to demonstrate either reliability at a specific time or a specified mean time to failure. To use this option, you must provide an estimate of the underlying life distribution's shape parameter.

    Note: You can get the parameters and time units from a calculated data sheet in the current project by clicking the **Get Distribution** icon on the control panel.

    - If you are demonstrating the reliability at a specified time, enter the time in the **At this Time** field. If you used the **Get Distribution** option, this value will initially be set to the time at which a product with the calculated life distribution would have the reliability that you specified.
    - If you are demonstrating the mean time to failure, enter the mean time in the **Demonstrate this MTTF** field. If you used the **Get Distribution** option, this value will initially be set to the MTTF that a product with the calculated life distribution would have.
    - You can also create a table that displays a range of test duration values as a function of sample size and number of allowable failures. The table can also display a range of required sample size values as a function of test time and number of allowable failures. After you create this table, you can also create a plot displaying the same information. This provides a quick way to consider many possible test plan scenarios without having to perform each calculation individually. *See Section 19.1.3 on page 473*.

1. If more than the number of allowable failures occurs, then the reliability demonstration test failed and the target metric was not demonstrated. The engineers may choose to continue the test as a reliability life test, recording the exact failure/suspension times and then analyzing the data with standard life data analysis methods. Note that although mathematically it is possible to design a new test with more allowable failures and a longer duration to demonstrate the same target metric, it is unlikely that the metric will be demonstrated by the new test if the original demonstration test failed.

- The **Non-Parametric Binomial** option does not require a life distribution. It solves for the demonstrated reliability, confidence level or sample size that is associated with a specified test design.
  - The use of this option assumes that the time at which the reliability is demonstrated is equal to the specified test time multiplied by the specified acceleration factor. For example, if the acceleration factor is 1, then the time at which reliability is demonstrated equals the test duration. If the acceleration factor is 2, then the time at which the reliability is demonstrated is double the test duration, and so on.
- The **Exponential Chi-Squared** option determines the total accumulated test time that will be required to demonstrate a specified metric.
  - The total accumulated test time is equal to the total amount of time experienced by all of the units on test. For example, if the total accumulated time is 100 hours, then you could test 1 unit for 100 hours, or you could test 2 units for 50 hours each, or you could test 5 units for 1 hour each and 1 unit for 95 hours.
  - This option is based on the chi-squared distribution, and it is only for products that have an assumed exponential life distribution (i.e., products with a constant failure rate).
- The **Non-Parametric Bayesian** option solves for the demonstrated reliability or confidence level that can be expected from a specified test design. It can also solve for the sample size needed to demonstrate a target reliability at a specified confidence level. To use this option, you must have some prior information with which you can estimate the product's reliability.
  - With this option, you will need to specify whether the source of prior information is expert opinion (i.e., engineering experience) about the reliability of the entire system or prior testing at the subsystem level. This is done in the **Prior information about the failure rate behavior** area of the RDT sheet.
    - If you have experience regarding the system as a whole, choose **Expert opinion on reliability**. Then, enter worst case, expected and best case estimates of the product's reliability.
    - If you have prior test data for subsystems that compose your system, then choose **Prior tests at the subsystem level**. In the **Number of subsystems** drop-down list, choose the number of different subsystems that you will enter data for. After you select the number of subsystems, click the **Edit Bayesian Subsystems** icon on the control panel to open a window you can use to enter the prior test data. Alternatively, you can click the appropriate cell on the RDT sheet to enter the information directly into the sheet, as shown next.



  - Like the non-parametric binomial option, the use of the non-parametric Bayesian option assumes that the time at which the reliability is demonstrated is equal to the specified test time multiplied by the specified acceleration factor.

The following chart shows the information that each method can provide:[2]

	Metric to demonstrate:	Can solve for:
Parametric Binomial	The reliability at a specific time or Mean time to failure (MTTF)	The test time for a specified sample size or The sample size for a specified test time
Non-Parametric Binomial	Reliability at test time (if acceleration factor = 1)	The demonstrated reliability or The associated confidence level or The necessary sample size
Exponential Chi-Squared	The reliability at a specific time or Mean time to failure (MTTF)	The total accumulated test time
Non-Parametric Bayesian	Reliability at test time (if acceleration factor = 1)	The demonstrated reliability or The associated confidence level or The necessary sample size

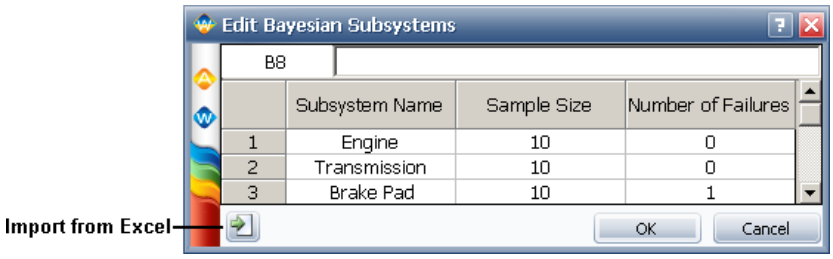
- In the **Units** field of the control panel, define the units of time that will be used in the test.
- If you selected to solve for sample size, select whether sample sizes will be displayed as integers in the **Display Options** area.
- In the **Acceleration Factor** field of the control panel, enter the acceleration factor associated with the stress level that will be used in the test, if applicable.
  - The acceleration factor is obtained by dividing the product's life at the use stress level by its life at the accelerated stress level to be used in the test, if applicable. For example, if the product has a life of 100 hours at the use stress level, and it is being tested at an accelerated stress level which reduces its life to 50 hours, then you would enter 2 in the **Acceleration Factor** field. If you are testing units under normal operating conditions, then the acceleration factor is 1.
- Click **Calculate** to solve for your selected value.

19.1.2 Edit Bayesian Subsystems

The Edit Bayesian Subsystems window is used to enter prior test data for subsystems that compose your system. When you select to use the non-parametric Bayesian test design method with the Reliability Demonstration Test (RDT) tool, this prior data is used to estimate the demonstrated reliability or confidence level that you can expect from a test's results, or the sample size that must be used in a test.

2. The ReliaWiki resource portal has more information on these methods at: http://www.ReliaWiki.org/index.php/Reliability_Test_Design.

To open the window from the RDT tool, select **Non-Parametric Bayesian** from the **Test Design Method** drop-down list on the control panel. Then click the **Edit Bayesian Subsystems** icon. The window will appear as shown next.



- The **Subsystem Name** column is for the name or description of the subsystem that was tested.
- The **Sample Size** column indicates how many units were used in the subsystem's test.
- The **Number of Failures** column indicates how many units failed during the test.
- The **Import from Excel** icon allows you to import the first three columns of an Excel sheet into the table.

The software assumes a series configuration of these subsystems (i.e., the failure of any one subsystem is sufficient for the whole system to fail) and uses this subsystem-level information to estimate the expected value and variance of the reliability of the entire system.

19.1.3 Parametric Binomial Table and Plot

When you use the parametric binomial option of the RDT tool, you can also create a table and plot based on the target metric and life distribution that you specified on the RDT sheet. Depending on what you select to solve for, the table will display a range of test duration values as a function of sample size and number of allowable failures, or it will display a range of required sample size values as a function of test time and number of allowable failures. The table and plot provide quick ways to consider many possible test plan scenarios without having to perform each calculation individually.

Follow the steps outlined below to create the table.

- Open the RDT tool and solve for required test time or sample size using the parametric binomial test design option. The table will use the target reliability and life distribution that you specified to produce its results.
- Click the **Show RDT Table** icon (shown next) on the control panel.



- The Test Design Table page will appear with an empty data sheet. In the **Solve for** area, select which value you wish to solve for.
  - **Test time for given sample size** solves for the test time given a range of sample sizes. When you select this option, the **Sample Size Range** area will require that you enter a starting sample size, an ending sample size and an increment value by which the sample will increase in the table.
  - **Sample size for given test time** solves for sample size given a range of test times. When you select this option, the **Test Time Range** area will require that you enter a starting test time, an ending test time and an increment value by which the test time will increase.
  - For either option, you must enter starting, ending and increment values for the number of allowable failures in the **Number of Failures Range** area. A demonstration test will fail to demonstrate the target reliability if the

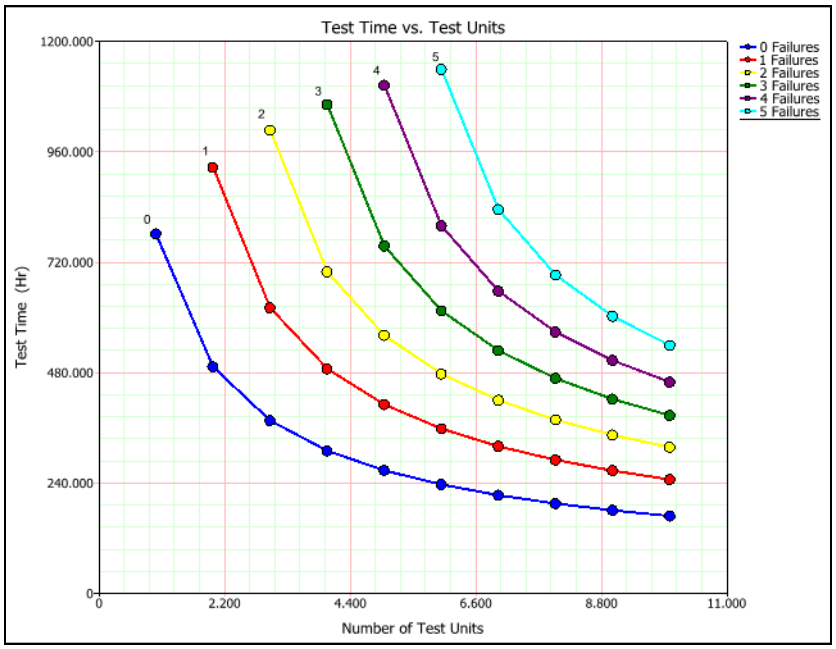
number of failures exceeds this number. Thus, if you are designing a zero-failure test, then the test will demonstrate the target reliability only if no failures occur.

- Click **Calculate** to create the table.

The data sheet shown next displays a parametric binomial table. For example, the second cell in the table tells us that if you used a test time of about 621 hours and no more than one failure occurred during the test, then you would need a sample size of 3 to demonstrate the target metric.

	Sample Size	Test Time for 0 Failures	Test Time for 1 Failures	Test Time for 2 Failures	Test Time for 3 Failures	Test Time for 4 Failures	Test Time for 5 Failures
1	1	781.673161	N/A	N/A	N/A	N/A	N/A
2	2	492.423235	926.142189	N/A	N/A	N/A	N/A
3	3	375.78926	620.969185	1006.917969	N/A	N/A	N/A
4	4	310.2072	488.54706	699.156752	1062.62632	N/A	N/A
5	5	267.328461	410.635486	560.930125	755.263045	1104.921369	N/A

To view a plot of the table results, click **Redraw Plot**. Each colored line in the plot corresponds to a specific number of failures; this depicts how the allowed number of failures influences the test time and the required sample size.



19.1.4 Example

This section presents an example of how to use the Reliability Demonstration Test (RDT) tool to calculate the test time needed to show a specified reliability at a specified confidence level in a zero-failure demonstration test. A parametric binomial table will also be created which displays a range of test duration values as a function of sample size and number of allowable failures.

A reliability engineer is asked to plan a test for a component whose failure behavior is governed by a Weibull distribution, where beta = 1.5. The goal of the test is to show, with 90% confidence, that the component has a reliability of at least 90% at 100 hours. The component will be tested at normal use conditions. 10 units are available for testing. Due to resource constraints, it is decided that the target metric will be demonstrated in a zero-failure test. The engineer's task is to determine the minimum test time needed to demonstrate this reliability in such a test.

Follow the steps outlined below to determine the required test time.

1. Open the tool by choosing **Insert > Tools > Test Design**. In the Test Design Assistant, select **Reliability Demonstration Test Design** and click **OK**.
2. Select **Parametric Binomial** from the **Test Design Method** area of the control panel.
3. In the **Input** area of the control panel, select **Hour (Hr)** for the units to be used for all time inputs and results (e.g., hours, days, miles, etc.). Since the component will be tested at the use stress level, enter 1 for the acceleration factor.
4. Provide the required inputs for the RDT sheet, as shown next.

Design a reliability demonstration test	
What metric would you like to demonstrate?	
Metric	Reliability value at a specific time
Demonstrate this reliability (%)	90
With this confidence level (%)	90
At this time (Hr)	100
Assume the failure rate behavior is governed by this distribution	
Distribution	2P-Weibull
With this Beta	1.5
Solve for this value	
Value	Required test time
With this sample size	10
With a maximum of this many failures	0

5. Click the **Calculate** icon to solve for the required test time. A **Results** area will appear at the bottom of the data sheet.

Results	
Test time per unit (Hr)	**168.4064**

The results show a test time of at least 168.4064 hours is required for the specified test plan. In other words, if 10 units are tested for 168.4064 hours at normal use conditions and none fail, then the test will demonstrate, with 90% confidence, that the component has a reliability of at least 90% at 100 hours.

6. To generate a parametric binomial table, click the **Show RDT Table** icon.



7. In the **Solve for** area, select **Test time for given sample size** so the table shows test duration values as a function of sample and number of allowable failures.
8. In the **Sample Size Range** area, enter a range of sample sizes of 1 through 10 with an increment of 1.
9. In the **Number of Failure Range** area, enter a range of 0 to 5 with an increment of 1.
10. Click the **Calculate** icon. The table will appear as shown next.

	Sample Size	Test Time for 0 Failures	Test Time for 1 Failures	Test Time for 2 Failures	Test Time for 3 Failures	Test Time for 4 Failures	Test Time for 5 Failures
1	1	781.673161	N/A	N/A	N/A	N/A	N/A
2	2	492.423235	926.142189	N/A	N/A	N/A	N/A
3	3	375.78926	620.969185	1006.917969	N/A	N/A	N/A
4	4	310.2072	488.54706	699.156752	1062.62632	N/A	N/A
5	5	267.328461	410.635486	560.930125	755.263045	1104.921369	N/A
6	6	236.7324	358.097647	477.402142	614.436587	798.976461	1139.079356
7	7	213.612424	319.731849	419.95805	527.90453	656.926023	834.606628
8	8	195.41829	290.319111	377.421361	467.549564	568.518449	692.045156
9	9	180.660633	266.854187	344.377928	422.40046	506.404075	602.598257
10	10	168.406378	247.690451	317.848284	387.028287	459.458203	539.18805

19.2 Expected Failure Time Plot

The Expected Failure Time plot provides a visual depiction of what failure times you can expect to observe when you implement a particular test plan. If you perform the test and enter the actual failures as they're observed, you can use the plot to monitor whether the test is proceeding as planned, which could provide an early warning that adjustments may be needed.[3]

Tip: You can also use this plot to determine whether a specified reliability demonstration test will exceed the allowable number of failures. For example, if you have designed a zero-failure test and wish to determine whether zero failures will occur during the test, you could examine the lower confidence bound of the first predicted failure for that test. This information will help you determine whether a failure is likely to occur before the end of the test.

19.2.1 Using the Expected Failure Time Plot

Generating an Expected Failure Time Plot

Follow the steps outlined below to generate a plot.

- Enter the sample size, two-sided confidence level and acceleration factor in the **Plot Setup** area of the control panel.
  - The acceleration factor is obtained by dividing the product's life at the use stress level by its life at the accelerated stress level to be used in the test, if applicable. For example, if the product has a life of 100 hours at the use stress level, and it is being tested at an accelerated stress level which reduces its life to 50 hours, then you would enter 2 in the **Acceleration Factor** field. If you are testing units under normal operating conditions, then the acceleration factor is 1.
- Define the expected life distribution for your product and then choose the time units to be used in the calculations. To enter the parameters, click the arrow in the **Expected Failure Model** area.
  - A window will appear where you can select a life distribution and enter its parameters. If the estimated values of the distribution parameters are not available, you can use the Quick Parameter Estimator (QPE) to solve for them. *See page 194 in Chapter 5*.

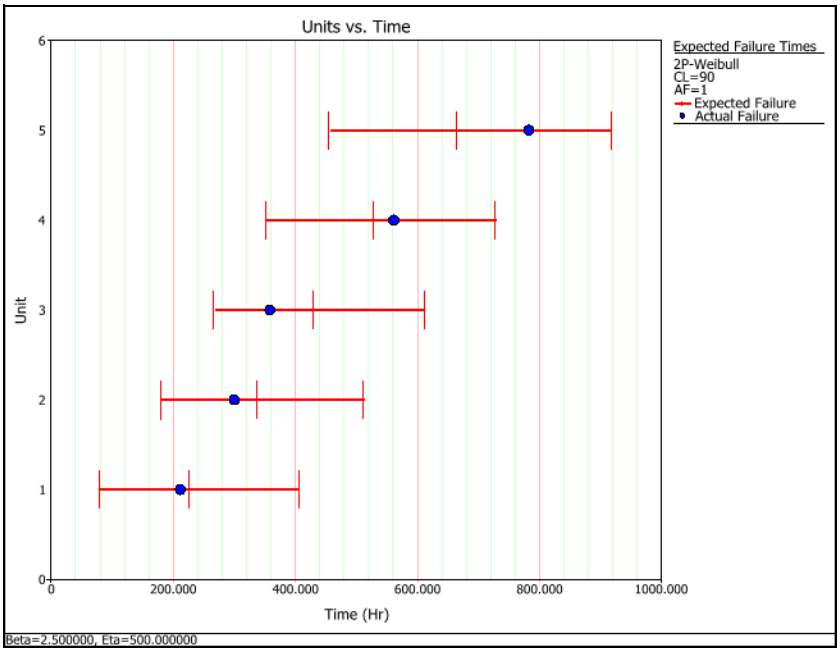


  - In the same window, select the time units (e.g., hours) that will be used in the test. Then click **OK** to close the window and generate the plot.

Note: You can also get the parameters and time units from a calculated data sheet in the current project by clicking the **Get Failure Model** icon on the control panel. After you select the data sheet, click the **Redraw Plot** icon to generate the plot.

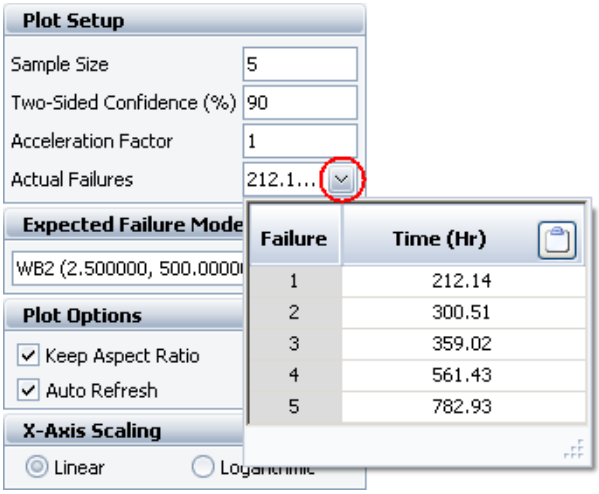
3. The ReliaWiki resource portal has more information on the Expected Failure Time Plot at: http://www.ReliaWiki.org/index.php/Reliability_Test_Design.

- The plot will show the predicted failures times for each test unit with a margin of error corresponding to the confidence level you entered in the **Plot Setup** area.



The three vertical tick marks on each line display the lower confidence bound, median value and upper confidence bound for the expected failure time of a tested unit.

- To display actual failure times as points that can be compared against the estimated intervals, click the arrow in the **Actual Failures** box. A table will appear where you can enter the recorded failure times, as shown next. The entered times will be automatically sorted from least to greatest.



> **Tip:** If you have a column of data copied to the Clipboard (e.g., from a data sheet), you can click the **Clipboard** icon at the top of the Time column to paste the data into the table.

You can view a calculated value by pointing to the relevant vertical tick mark on the plot. Alternatively, click **Results (...)** in the **Expected Failure Time** area of the control panel to view all calculated values at once.

Additional Control Panel Options

The control panel also includes the following options:

- In the **Plot Options** area:
  - **Keep Aspect Ratio** maintains the ratio of the horizontal size of the plot graphic to the vertical size when you resize the plot.
  - **Automatically Refresh** will automatically update the plot whenever any of the inputs change. If this option is cleared, you will have to click the **Redraw Plot** icon to update the plot.
- In the **X-Axis Scaling** area, select the **Linear** check box to use a linear scale for time. Select the **Logarithmic** check box to use a logarithmic scale.
- In the **Scaling** area:
  - The **X** and **Y** boxes show the minimum and maximum values for the x- and y-axes. To change these values, clear the check box beside the value range. If it is selected, the application will automatically choose appropriate values for the range.
- The **Plot Setup** icon opens the Plot Setup window, which allows you to customize most aspects of the plot including the titles, line styles and point styles. *See Chapter 15*.
- The **RS Draw** icon launches ReliaSoft Draw, which allows you to annotate your plot and view your plot in greater detail. *See Chapter 16*.
- The **Export Plot** icon opens the Save As window, which allows you to save the current plot graphic in one of the following formats: *.jpg, *.gif, *.png or *.wmf.

19.2.2 Example

This section presents an example of how to use the Expected Failure Time plot to monitor an in-progress test.

A reliability engineer is asked to perform a reliability life test for a component whose underlying life distribution is estimated to be a Weibull distribution, where beta = 2.5 and eta = 500 hours. This estimation is uncertain, however, and the engineer has been asked to monitor the test and report to management if the failure times suggest that the estimated life distribution is incorrect. Ten units will be tested under normal operating conditions.

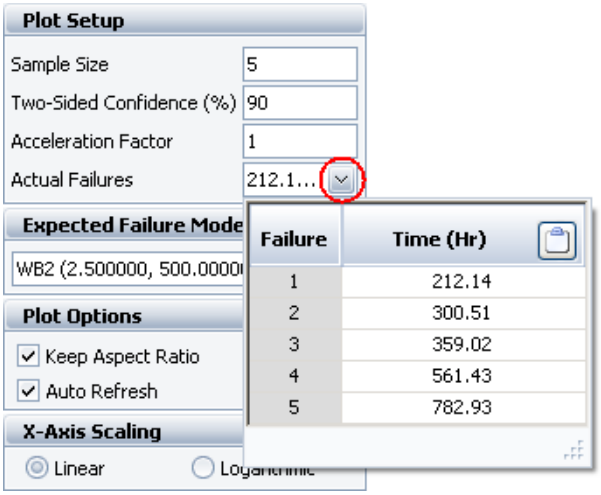
Follow the steps outlined below to view two-sided 80% confidence bounds for the failure times of each unit in the test.

1. Open the Test Design Assistant by choosing **Insert > Tools > Test Design**.

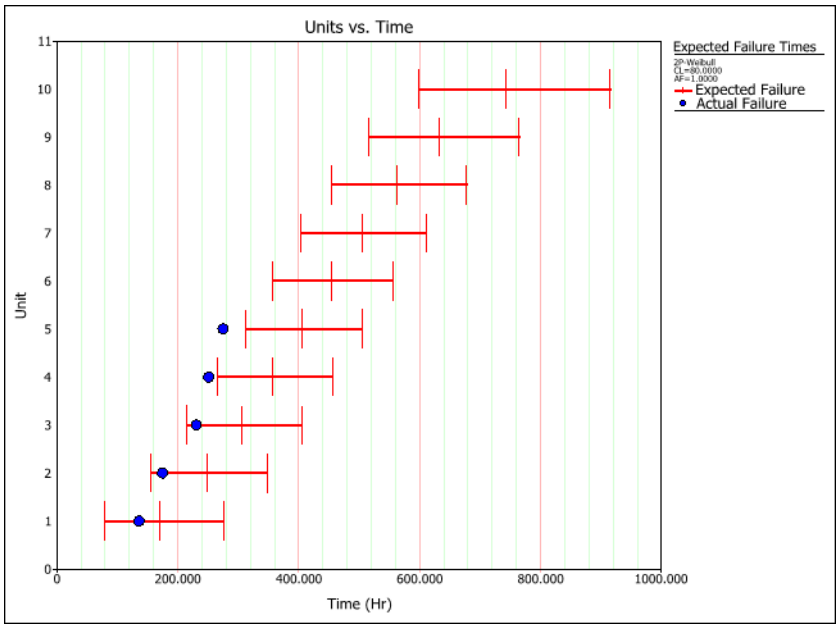


2. In the Test Design Assistant, select **Expected Failure Times Plot** and click **OK**. The Expected Failure Time window will appear.
3. In the **Plot Setup** area of the control panel, enter **10** for the sample size. Then enter **80** for the two-sided confidence level. Because the units will be tested under normal operating conditions, enter **1** for the acceleration factor.
4. Enter the life distribution of the product by clicking the arrow in the **Expected Failure Model** area of the control panel. In the window that appears, select **2P-Weibull** for the distribution. Then enter the distribution's parameters and select **Hour** for the time unit.
5. Then click the **Plot** icon to calculate the expected failure times. A plot will show the predicted failure times for the ten units with 80% two-sided confidence bounds on each prediction.

6. To add actual failure times to the plot, click the arrow in the **Actual Failures** field to enter the failure times (shown next) as they are observed during the test. The entered times will be automatically sorted from least to greatest, and they will be displayed as points on top of the lines that represent the expected failure times.



The following plot shows the first five actual failures displayed with the ten expected failure times.



At this point, the engineer would inform management that the observed failure times indicate that the estimated life distribution may be inaccurate.

19.3 Difference Detection Matrix

The Difference Detection Matrix calculates how much test time is required before it is possible to detect a statistically significant difference in the mean life or BX% life of two product designs by analyzing the data from a reliability life test. You can use this tool to evaluate different test plans in order to choose the one that will be most efficient to compare the reliability of the two designs. To use the matrix, you must provide various inputs—including the sample sizes that will be used to test each design and the life distributions that describe each design's life behavior.[4]

19.3.1 Using the Difference Detection Matrix

Reading the Detection Matrix

In this section, we will look at an example detection matrix and discuss how to read it. The next section explains how to set up the matrix. We will assume that a sample size of 10 will be used to test both product designs.

		Design 1 Mean Life					
		500	1000	1500	2000	2500	3000
Design 2 Mean Life	500	0	4	4	4	4	4
	1000	4	0	0	3	4	3
	1500	4	0	0	0	2	3
	2000	4	3	0	0	0	0
	2500	4	3	3	0	0	0
	3000	4	3	3	0	0	0

0	Difference cannot be detected using current setup, existing sample size, and test time of <= 6000 (Hr).
1	Difference can be detected with 6000 (Hr) of testing.
2	Difference can be detected with 4500 (Hr) of testing.
3	Difference can be detected with 3000 (Hr) of testing.
4	Difference can be detected with 1500 (Hr) of testing.

Because the column title in the above matrix is "Design 1 Mean Life," we know that the metric being compared is mean life (rather than BX% life). The column headers represent possible mean life values for Design 1, and the row headers represent possible mean life values for Design 2. We'll begin by examining the cell in the second column and the first row of the matrix, which corresponds to a Design 1 mean life of 1,000 hours and a Design 2 mean life of 500 hours.

The number inside this cell is 4, and the legend below the matrix tells us that a 4 means a difference between the two designs can be detected, but only after at least 1,500 hours of testing. In other words, one thing the matrix tells us is that if Design 1 has a mean life of 1,000 hours and Design 2 has a mean life of 500 hours, then the tests we use to compare these designs will need a duration of at least 1,500 hours in order for the analysis of collected data to show that there is, at the specified confidence level, a statistically significant difference in mean life between the two designs.

Tip: You can click a cell in the matrix to see the difference in the compared metric that was calculated for both designs based on the associated test time. For example, if a cell in the matrix shows that a difference in mean life can be detected with 2,000 hours of testing, clicking the cell will show the mean life calculations (including two-sided confidence intervals) that would be obtained for each design if you analyzed the test data after 2,000 hours of testing. (If only one of the designs would produce enough failures to fit to a distribution during the given test time, no intervals will be shown.)

4. The ReliaWiki resource portal has more information on the Difference Detection Matrix at: http://www.ReliaWiki.org/index.php/Reliability Test Design.

Setting Up the Detection Matrix

Follow the steps outlined below to set up the detection matrix. All the settings are located on the control panel.

- In the **Metric to Compare** area, select whether you want to compare **Mean Life** or **BX% Life**. If you select to compare the BX% life, you must enter a value in the associated input field (e.g., a BX% life of 10 indicates that the software will compare the time when 10% of units will have failed). Also, specify the confidence level that will define when a difference between the specified designs is statistically significant. Then specify the time units (e.g., hours) that will be used for the inputs and results.
  - The software calculates two-sided confidence intervals (at the level you specified in the **Confidence (%)** field) on the specified life metric for both designs, when possible. A difference is considered detected at a given test time when the two calculated confidence intervals do not overlap, or when only one of the designs produces enough failures to fit to a distribution.
- In the **Design 1** and **Design 2** areas, enter the sample sizes of the tests that will be used to compare the designs and define the distributions for each design's failure behavior.
- In the **Reliability Metric Setup** area, configure the rows and columns in the table by first entering the **Max Metric Time** value, which is the highest metric value (mean life or BX% life, depending on your selected metric) that the software will consider in the comparison. Then enter a **Metric Increment** to specify the difference between the displayed metric values in each column and each row. This value is also used as the minimum metric value that will be considered in the comparison.
  - For example, if you entered **1000** in the **Max Metric Value** field and **200** in the **Metric Increment** field, then your matrix would have five columns and five rows, each starting with a value of 200 and moving up in increments of 200 until reaching the maximum value of 1000.

		Design 1 Mean Life				
		200	400	600	800	1000
Design 2 Mean Life	200	0	4	4	4	4
	400	4	0	0	3	3
	600	4	0	0	0	2
	800	4	3	0	0	0
	1000	4	3	2	0	0

- The matrix indicates whether a difference in the selected reliability metric will be detected after various test times. These are the test times shown in the legend under the matrix. In the **Test Times Matrix Setup** area, specify the test times that will be evaluated.
  - To specify how many test times to evaluate, enter a value in the **Number of Test Times** field. For example, if you enter **4** in this field, then the software will evaluate will evaluate 4 different test times. The software cannot evaluate more than 10 test times.
  - If the **Calculate Test Times** check box is cleared, you will be able to click the arrow inside the **Test Time** field to open a table where you can enter the specific times. For example, if you enter **500** as one of the test times, then the matrix will determine whether 500 hours (assuming your selected time unit is hours) is sufficient to detect a difference in the selected life characteristic. The number of rows available in this table is determined by what you entered in the **Number of Test Times** field. Note that upon exiting the table, the times in the table will be sorted in descending order.
  - If the **Calculate Test Times** check box is selected, the software will automatically enter test times from 1,500 to 6,000 in the **Test Time** field.

- Click the **Calculate** icon to validate your settings and generate the matrix.
  - If you wish to export the matrix as a Microsoft Excel file, click the **Send to Excel** icon.



  - You can also change the color used in the matrix by clicking the **Set Matrix Color** icon.



    Your selected color will be used to indicate when the shortest test time provided in the **Test Time Matrix Setup** area is sufficient to detect a difference in the specified life characteristic. Darker versions of that color will be used to indicate when longer test times are required.

19.3.2 Example

This section presents an example of how to use the Difference Detection Matrix to estimate the test time required in a reliability life test to demonstrate that a new product design will have a higher mean life than an older design.

A manufacturer plans to implement a new product design that is expected to improve the product's mean life. The current design is expected to have a mean life of 200 hours, and the new design is expected to have a mean life of 300 hours. A reliability engineer is asked to determine, with 90% confidence, the minimum amount of test time that would be required to determine that there has been an improvement in mean life with the new design. No more than 400 hours will be available for testing, and only 10 units of each design will be available for testing.

The engineer also has some information about the life distributions for both designs. They each have a Weibull life distribution, and the beta parameter for the original design is 4.2, while the beta parameter for the new design is 3.

Follow the steps outlined below to estimate the test time required to detect a difference in mean life.

1. Open the detection matrix by choosing **Insert > Tools > Test Design**. In the Test Design Assistant, select **Difference Detection Matrix** and click **OK**.
2. In the **Metric to Compare** area of the control panel, select **Mean Life** and enter **90** for the confidence level.
3. Choose **Hour** for the units that will be used for all time inputs and results.
4. In the **Design 1** area, enter **10** for the sample size. Design 1 is the original design, so select **2P-Weibull** for the distribution and enter **4.2** for the beta value.
5. In the **Design 2** area, enter **10** for the sample size. Design 2 is the new design, so select **2P-Weibull** for the distribution and enter **3** for the beta value.
6. The matrix will display a range of possible mean life values for both designs. The range begins with 0 and increases by an increment of your choice. In the **Reliability Metric Setup** area, enter **500** for the maximum metric time. Then enter **100** for the metric increment. The range of mean life values displayed for both designs will thus be 100, 200, 300, 400 and 500.
7. In the **Test Time Matrix Setup** area, enter **4** for the number of test times that will be evaluated. The software will automatically enter these test times. To view them, clear the **Calculate Test Times** check box, then click the arrow in the **Test Time (Hr)** box. A table will appear where you can view and modify the test times that will be evaluated. In the table, enter **400** for Test Number 1, **350** for Test Number 2, **300** for Test Number 3 and **250** for Test Number 4.

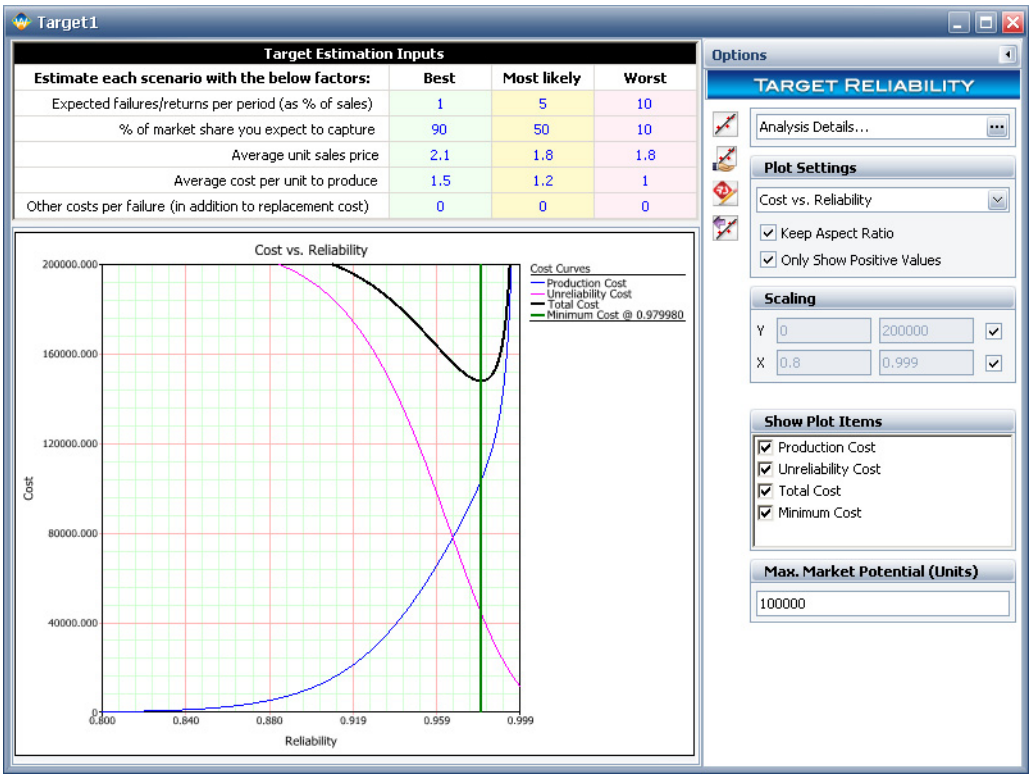
8. Click the **Calculate** icon to generate the matrix, which will appear as shown next.

Design 2 Mean Life \ Design 1 Mean Life	100	200	300	400	500
100	0	4	4	4	4
200	4	0	3	3	3
300	4	3	0	0	0
400	4	3	0	0	0
500	4	3	1	0	0

According to the above matrix, if Design 1 has a mean life of 200 and Design 2 has a mean life of 300, then a statistically significant improvement in mean life can be detected between the two after analyzing the data of a 300-hour reliability life test.

Target Reliability Tool 20

Deciding on a reliability goal/target involves trade-offs. This is because higher reliability typically correlates with higher production costs, lower warranty costs and higher market share. With Weibull++'s Target Reliability tool, you can generate plots that help you visualize and estimate a target reliability that will minimize cost, maximize profit and/or maximize your return on an investment in improving the product's reliability.[1]



The Target Reliability tool is divided into a target estimation sheet and control panel. The top part of the sheet is the **Target Estimation Inputs** area, where you provide the best, most likely and worst case estimates for various factors involved in the production and sale of the product. The bottom part of the sheet contains the plot that you select to generate from your inputs.

The control panel contains settings for the type of plot that will be displayed, as well as other plot settings and icons. It also contains an input field for the product's maximum market potential and, when the R3OI vs. reliability plot is selected, an input field for the expected initial investment.

---

1. The ReliaWiki resource portal has more information on the Target Reliability tool at: http://www.ReliaWiki.org/index.php/Target_Reliability_Tool.

20.1 Generating a Plot

Follow the steps outlined below to generate a plot.

- Open the tool by choosing **Insert > Tools > Add Target Reliability**.



- In the **Target Estimation Inputs** area, enter the estimated best, worst and most likely scenarios. Each scenario includes all of the following factors.

> **Note:** A better scenario involves an improvement in both reliability and ROI. Thus the production costs in the best case scenario must be higher than that of the worst case scenario. This is because improving reliability requires increasing spending on factors such as facilities, higher quality materials, etc.

  - **Expected failures/returns per period (as % of sales)**
    The number of failures or returns that are expected during a given period, given as a percentage of sales. For example, if the best case scenario is one where 1% of the sold units are expected to fail or be returned within a year, then you would enter **1** under the **Best** heading for this factor.
  - **% of market share that you expect to capture**
    The percentage of the market share that you expect to capture with your product. For example, if the worst case scenario is one where you expect to capture 40% of the market, then you would enter **40** under the **Worst** heading for this factor.
  - **Average unit sales price**
    The average sales price per unit. For example, if the most likely scenario is one where the average sales price per unit is $20, then you would enter **20** under the **Most Likely** heading for this factor.
  - **Average cost per unit to produce**
    The average cost of producing a single unit. For example, if the best case scenario is one where it costs, on average, $2 to produce a single unit, then you would enter **2** under the **Best** heading for this factor.
  - **Other costs per failure (in addition to replacement cost)**
    The average failure cost per unit, ignoring the cost of replacing a returned unit. For example, if the worst case scenario involves a cost of $5 per failed unit, $3 of which are due to the cost of replacing the failed unit, then you would enter **2** (the remaining cost) under the **Worst** heading for this factor.
- Select the kind of plot you want to generate in the **Plot Settings** area of the control panel. The following four plots are available:
  - The **Cost vs. Reliability** plot displays total cost as a function of target reliability. The total cost is the sum of the *production cost*, which is the cost involved in manufacturing the product to achieve a target reliability, and the *unreliability cost*, which is the cost incurred as a result of failures/returns and lost sales. By default, the plot displays curves for the production and unreliability costs and displays a vertical line marking the lowest point of the total cost curve.
  - The **Profit vs. Reliability** plot displays profit as a function of target reliability. Profit is calculated by subtracting the production cost, warranty cost and other failure-related costs from the estimated total sales revenue. By default, the plot displays a vertical line marking the highest point of the profit curve.
  - The **R3OI vs. Reliability** plot uses ReliaSoft's Reliability Return on Investment (R3OI) equation to calculate expected returns on an initial investment that will affect reliability. The R3OI is calculated by subtracting the initial investment from the expected increase in sales revenue and dividing the result by the initial investment.

This plot displays the return on investment as a function of the reliability achieved through that investment. By default, the plot displays a vertical line marking the highest point of the R3OI curve.

  - If you choose **R3OI vs. Reliability**, then you must enter your expected initial investment in the **Initial Investment** field at the bottom of the control panel.
- Near the bottom of the control panel, enter the maximum number of units that could be sold in a given period in the **Max. Market Potential (Units)** field.
- The **Show Plot Items** area will display a list of curves and vertical lines that can be displayed for the plot you selected in the **Plot Settings** area. To remove a line/curve from the current plot display, clear the associated check box.
- Click the **Plot** icon on the control panel to generate the plot.

Tip: You can hide the **Target Estimation Inputs** area and expand the plot to fill the entire sheet by double-clicking the plot. To show the area and shrink the plot to its original size, simply double-click the plot again. See the descriptions below for other plot commands that are available in this control panel. For other general tips, see Chapter 14 on page 404.

20.2 Control Panel

The Target Reliability Tool control panel contains the following options:

- The **Results** button displays the equations that are used to generate the plots. The values of A and B for each equation are calculated from the inputs provided in the **Target Estimation Inputs** area of the target estimation sheet.
- In the **Plot Settings** area:
  - **Keep Aspect Ratio** maintains the ratio of the horizontal size of the plot graphic to the vertical size when you resize the plot.
  - **Only Show Positive Values** will include only positive values for the y-axis when that axis has a minimum value of at least zero.
- In the **Scaling** area:
  - The **X** and **Y** boxes show the minimum and maximum values for the x- and y-axes. To change these values, clear the check box beside the value range. If it is selected, the application will automatically choose appropriate values for the range.
- The **Plot Setup** icon opens the Plot Setup window, which allows you to customize most aspects of the plot including the titles, line styles and point styles. *See Chapter 15*.
- The **RS Draw** icon launches ReliaSoft Draw, which allows you to annotate your plot and view your plot in greater detail. *See Chapter 16*.
- The **Export Plot** icon opens the Save As window, which allows you to save the current plot graphic in one of the following formats: *.jpg, *.gif, *.png or *.wmf.

Tests of Comparison 21

Weibull++ and ALTA provide two tools designed for statistical comparison of data sets:

- The **Life Comparison** tool calculates the probability that the times-to-failure of one population will be better or worse than the times-to-failure of the second. This allows you to compare life distributions from two alternate designs, manufacturers, lots, assembly lines, etc. in order determine whether the units in the first group will outlast the units in the second.
- The **Stress-Strength** comparison tool allows you to determine the probability of failure based on the probability of stress exceeding strength. The strength data set could be actual data that represents the strength of the material (i.e., minimum applied stress to cause failure) and the stress data set could be the actual stress levels that the material will experience under use conditions.

Both tools are based on the same underlying probability equation, and both folios function in a similar way. This chapter describes them both in the following sections:

- Folio setup - Section 21.1 (p. 490)
- Control panel settings - Section 21.2 (p. 491)
- An example of a life comparison test - Section 21.3 (p. 496).
- An example of a stress-strength test - Section 21.4 (p. 498).
- Using the Target Reliability Parameter Estimator - Section 21.5 (p. 502).

What's Changed? In previous versions of Weibull++ and ALTA, the results of the tests were not saved to a folio. In the Synthesis version, the software now creates a folio for saving your results and also provides a plot for visualizing the resulting probability function. The folio is saved under the **Tools** heading in the current project explorer.

21.1 Folio Setup

To add a life comparison folio to a project, choose **Insert > Tools > Life Comparison** or right-click the **Tools** folder in the current project explorer and choose **Add Life Comparison** on the shortcut menu.



To add a stress-strength folio to a project, choose **Insert > Tools > Stress-Strength** or right-click the **Tools** folder in the current project explorer and choose **Add Stress-Strength** on the shortcut menu.



When you add the folio to a project, you will be asked to select two data sheets to compare. Note that the data sheets must have already been analyzed (i.e., the parameters of the distribution have been calculated) in order for you to select them.

After you select the data sheets, the software will automatically plot the distributions of the data sets and estimate the probability value. You can then use the control panel settings in the folio to perform additional analyses or change how you want the probability value to be calculated.

21.1.1 Probability Equation

The life comparison test uses the following equation to evaluate the probability of failure based on the probability that the life of the second data set is greater than or equal to the life of the first data set:[1]

$$P\left[t_2 \geq t_1\right] = \int_0^{\infty} f_1(t) \cdot R_2(t)dt \qquad (1)$$

where $f(t)$ is the *pdf* of data set 1 and $R(t)$ is the reliability function of data set 2. The evaluation is based on whether this probability is less than or greater than 0.5. The result is interpreted as follows:

- If P = 0.5, then the lives of both data sets are equal.
- If P > 0.5, then the life of data set 2 exceeds the life of data set 1. For example, if P = 0.8, then data set 2 is better than data set 1 with an 80% probability.
- If P < 0.5, then the life of data set 1 exceeds the life of data set 2. For example, if P = 0.10, then data set 1 is better than data set 2 with a 90% probability (1 - 0.10 = 0.9).

1. Brown, Gerald G. and Rutemiller, Herbert C. "Evaluation of P[X>=Y] When Both X and Y are from Three-Parameter Weibull Distributions." IEEE Transactions in Reliability, R-22, no. 2 (1973).

For the stress-strength test, Eqn. (1) may be expressed as:

$$P[Stress \geq Strength] = \int_0^{\infty} f_{strength}(x) \cdot R_{stress}(x)\,dx \tag{2}$$

The expected reliability is calculated as:

$$R = P[Stress \leq Strength] = \int_0^{\infty} f_{stress}(x) \cdot R_{strength}(x)\,dx \tag{3}$$

For cases where the limits of one of the distributions are truncated to a specified range, Eqn. (3) is adjusted as:

$$R = P[X_1 \leq X_2] = \frac{1}{F_1(U) - F_1(L)} \int_L^U f_1(x) \cdot R_2(x)dx \tag{4}$$

where: $L \leq X_1 \leq U$, and X_1 = Stress, X_2 = Strength.

For data sets that are calculated in an ALTA standard folio, the comparison is performed at the given use stress levels of both data sets; therefore, Eqn. (1) can also be expressed as:

$$P[t_2 \geq t_1] = \int_0^{\infty} f_1(t, V_{use1}) \cdot R_2(t, V_{use2})dt \tag{5}$$

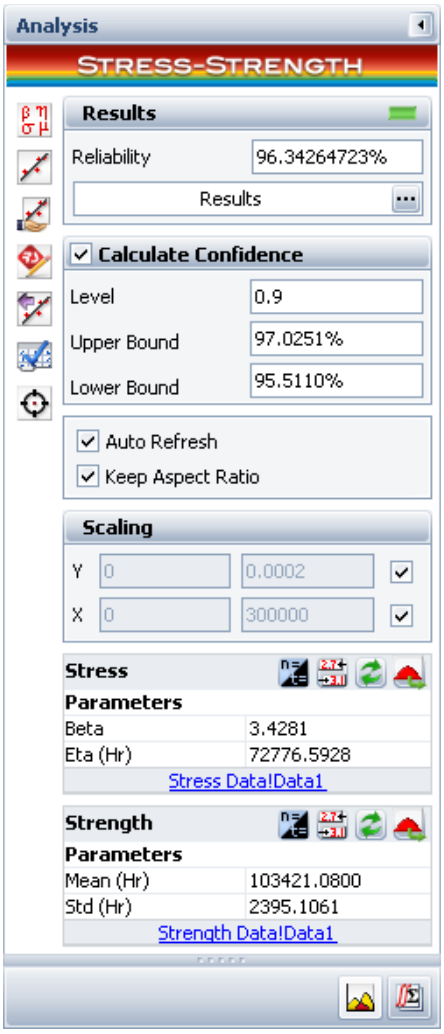
21.2 Control Panel Settings

The life comparison and stress-strength folios share the same control panel settings. The icons arranged vertically on the left side of the panel allow you to perform a variety of tasks related to the comparison (e.g., calculate the folio, select different data sheets, etc.), while the icons arranged horizontally perform tasks for each data set.

The control panel consists of two pages: the **Analysis** page, which contains most of the tools you will need to perform the analysis and view/access the results, and the **Setup** page, which contains options for setting how you want the probability value to be computed.

21.2.1 Control Panel Analysis Page

The Analysis page contains most of the tools you will need to perform the analysis and view/access the results. The following example shows the Analysis page of the stress-strength folio's control panel.

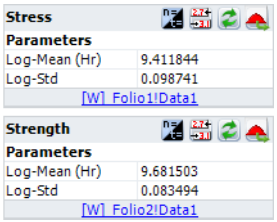


The following settings are available for both the life comparison and stress-stress folios:

- The **Results** area shows the calculated probability value.
  - In the life comparison folio, the value represents the probability that the life of the second data set exceeds the life of the first data set (**Probability X1<X2**).
  - In the stress-strength folio, the value represents the **Reliability** of the product (i.e., the probability that the product can withstand the applied stress). This can also be displayed as the **Probability of Failure** if you select the check box on the Setup page of the control panel.

  To display a report of the results, click the **Show Results (...)** button. The Results window allows you to edit, copy or print the results.
- The **Calculate Confidence** check box gives the option to compute the two-sided confidence bounds of the probability value. Enter the desired confidence level (in decimal) in the **Level** field, and then click the **Calculate** icon on the control panel to display the results in the **Upper Bound** and **Lower Bound** fields.

- The following settings affect how the plot is displayed:
  - The **Auto Refresh** check box allows you to automatically refresh the plot if any of the data sets, inputs or settings are modified
  - The **Keep Aspect Ratio** check box allows you to maintain the ratio of the horizontal size to the vertical size of the plot graphic when you resize the plot sheet.
  - The **Scaling** area shows the minimum and maximum values for the X and Y axes. You can change these values if the **Auto Scale** check box beside the value range is cleared. If it is selected, the application will automatically choose appropriate values for the range based on the current data.
- The control panel also displays the values of the parameters of the distribution for each data set. For the stress-strength comparison test (as shown in the picture below), the first data set represents the **Stress** and the second data set represents the **Strength**. For the life comparison test, these are displayed as **Set 1** and **Set 2**.



To see a summary report of the analysis for a data set, click anywhere within the parameters table. To open the standard folio, click the blue link that displays the name of the folio and data sheet.

Folio Tools

The folio tools are arranged on the left side of the control panel and in the areas dedicated to the selected data sheets (e.g., the **Stress** area in the picture previously shown).



Calculate calculates the probability value.

- In the life comparison folio, the value represents the probability that the life of the second data set exceeds the life of the first data set.
- In the stress-strength folio, it calculates the reliability of the product (i.e., the probability that the product can withstand the applied stress). This can also be displayed as the probability of failure if you select the check box on the Setup page of the control panel.



Redraw Plot updates the plot to reflect any changes that have been made.



Plot Setup opens a window that allows you to customize most aspects of the plot including the titles, line styles and point styles. *See Chapter 15*.



RS Draw launches ReliaSoft Draw, which allows you to annotate your plot and view your plot in greater detail. *See Chapter 16*.



Export Plot Graphic opens a window that allows you to save the current plot graphic in one of the following formats: *.jpg, *.gif, *.png or *.wmf.



Select Data Sheet opens a window that allows you to select the data sheets to compare.



Target Reliability Parameter Estimator opens the Target Reliability Parameter Estimator window, which allows you to explore how you could attempt to change one of the distribution parameters in order to meet a

specified target reliability. The plot and probability value will be based on the altered distributions and therefore may be appropriate only within the context of your specific scenario. *See Section 21.5 on page 502*.



Reliability Demonstration Test opens a window that allows you to solve for values associated with a specified test design, such as required test time, required sample size, demonstrated reliability and confidence level. *See page 470 in Chapter 19*.



Alter Parameters (also called Alter Parameters w/o Recalculation) allows you to manually alter the value of one (or more) parameters, while keeping the original values of the rest of the parameters and the variance/covariance matrix the same. The LK value, the solution line on plots and all subsequent analyses will be based on the modified set of parameters.



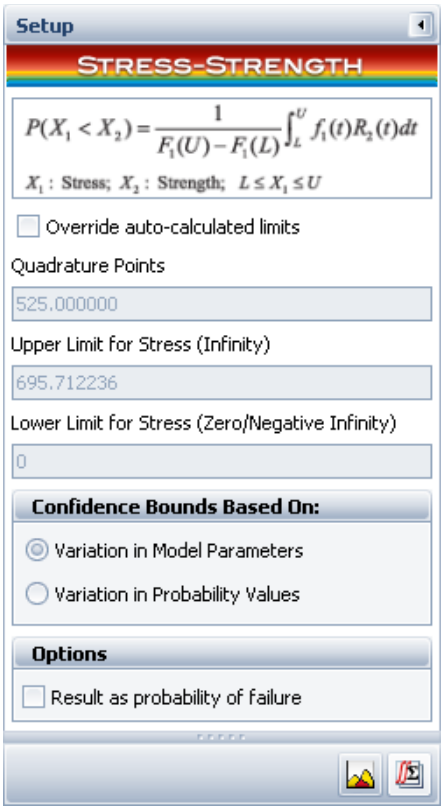
Refresh Parameters if you have manually modified the parameters of the data set, this option allows you to revert to the parameters calculated from the original data source.



Transfer Parameters allows you to transfer those modified parameters to the original data source if you have manually modified the parameters of the data set. All subsequent analysis made on the data source, such as calculations made via the QCP and other tools, are based on the modified parameter values. Therefore, the analysis may be appropriate only within the context of your specific scenario.

21.2.2 Control Panel Setup Page

The Setup page of the control panel contains options for setting how you want the probability value to be computed. The following example shows the Setup page of a stress-strength folio.



The Setup page allows you to configure three settings:

- Truncate the Integral

  The limits of the integral in the probability equation are from zero to infinity (*see Section 21.1.1 on page 490*). You can change the limits by selecting the **Override auto-calculated limits** check box. If you select this option, the equation shown on the control panel (as shown in the picture above) will be used to obtain the result.

  The ability to change the upper and lower limits of the integral gives you the option to limit your analysis to specific areas of interest. When you use a finite upper limit, the test excludes the outcomes where both products are expected to operate longer than the time specified by the upper limit. When you specify a lower limit, the test excludes the outcomes where both products are expected to fail before the time specified by the lower limit. For example, you might wish to set the lower limit to a time other than zero in order to exclude the burn-in period from the comparison. Likewise, you could set a specific time for the upper limit if you want to consider only the behavior that occurs while the product is under warranty.[2]

  The **Quadrature Points** field allows you to set the number of intervals (evaluation points) that the integral is divided into. Increasing the quadrature points increases the accuracy of the approximation but at the expense of increased calculation time.

- Select a confidence bounds calculation method

  The comparison tools provide two methods for calculating the confidence bounds, depending on whether the parameters of the distributions were estimated from actual data or based on engineering knowledge or existing references. A brief description of each method is provided next.[3] (Note that confidence bounds calculations are not possible when one or both data sheets is an ALTA standard folio that uses stress profiles.)

  - **Variation in Model Parameters**. Select this option if the parameters of both distributions are estimated from actual data. Parameters that are estimated from actual data sets have associated uncertainties that the software can use to calculate the variance and confidence bounds on the estimated probability. Note that this method is not supported if one or both of the data sets meets any of the following conditions. In these cases, use the "Variation in Probability Values" option.
    - The data sheet has no data set (i.e., has fixed parameters).
    - The data set is from a free-form data sheet.
    - The data set is from an ALTA standard folio that uses the cumulative damage model.
    - The data set was not analyzed using Fisher Matrix confidence bounds.
    - The distribution has parameters that were modified via the Alter Parameters tool or Target Reliability Parameter Estimator.
  - **Variation in Probability Values**. Select this method if the parameters of either distribution are fixed values. The software will use the variations in the estimated probability values to obtain the variance and confidence bounds.

- Display the result as the probability of failure.

  The **Result as probability of failure** check box is available only in stress-strength folios. It allows you to display the probability of failure instead of the reliability.

---

2. For more information about how the limits of the integral affect the calculations, please read "*Truncation of the Integral Used in the Comparison Wizard in Weibull++*" at: http://www.weibull.com/hotwire/issue125/relbasics125.htm. (Note that this article uses Weibull++ 7, but the theory remains the same.)

3. For an in-depth discussion on both calculation methods, the ReliaWiki resource portal provides more information at: http://www.ReliaWiki.org/index.php/Stress-Strength_Analysis.

21.3 Life Comparison Example

A product is being redesigned to improve its reliability. The following tables show the times-to-failure data of each design. The objective is to determine whether the redesign improved the product's reliability.

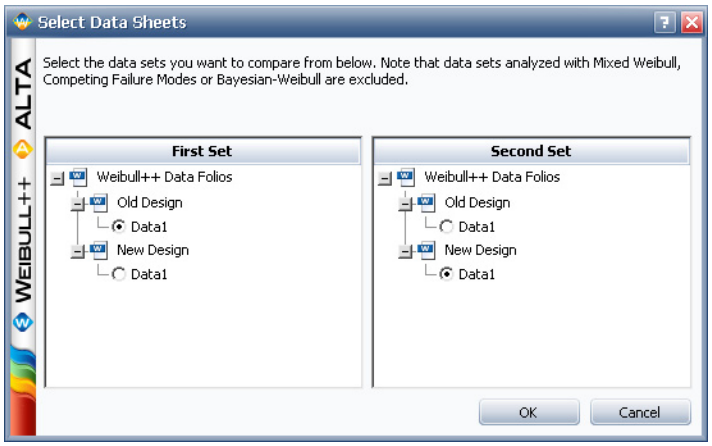
Failure Times for the Old Product Design (Hr)					
2	2	3	4	6	9
9	11	17	17	19	21
23	28	33	34	34	37
38	40	45	55	56	57
67	76	90	115	126	197

Failure Times for the New Product Design (Hr)				
15	32	61	67	75
116	148	178	181	183

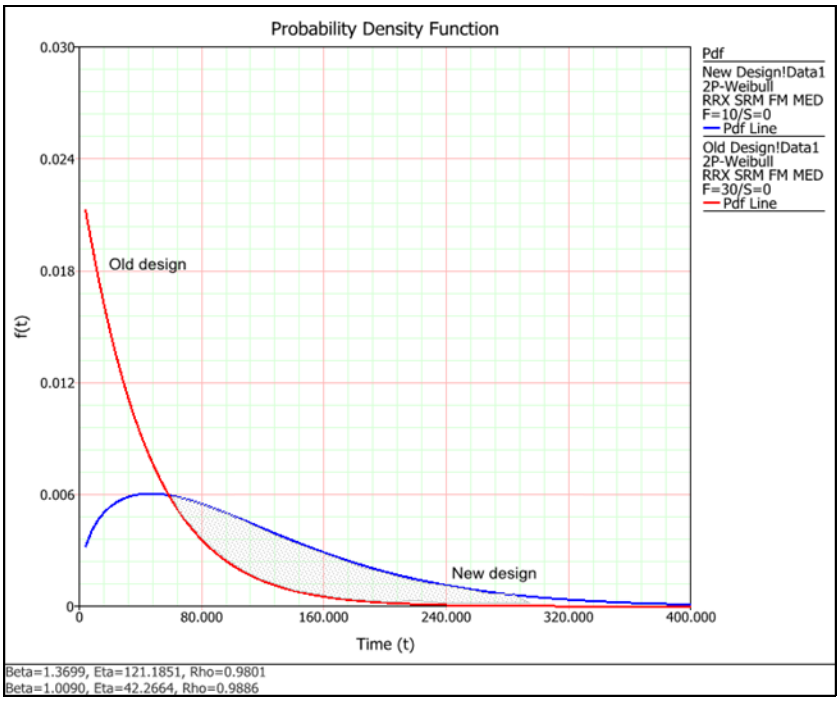
1. The first step in the analysis is to enter the data in two separate Weibull++ standard folio data sheets. Calculate the parameters for both data sheets by using the 2P-Weibull distribution and the RRX analysis method.
2. Next, add a life comparison folio to the project by choosing **Insert > Tools > Life Comparison**.



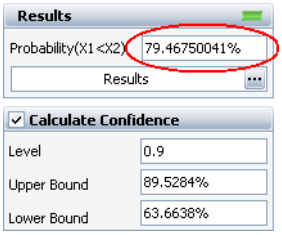
3. Select the data sheets for the two designs, as shown next. Click **OK**.



The result shows the following plot (with an area shaded to show the region where the life of the new design exceeds the life of the old design).



The result on the control panel indicates that the new design will last longer with a probability of 79.4675%., as shown next.



21.4 Stress-Strength Test Example

Steel rods with a diameter of 0.5 +/- 0.015 inches are to be used in an application where the load is purely tensile. Investigations of the application reveal that the load is not constant in magnitude. The following tables show the typical load values and strength data for 50 specimens.

Table I: Typical Load Values (lbs)

Typical Load Values (lbs)			
10455	12372	16559	19703
16961	10595	10898	5814
17279	12849	11661	6795
7821	13017	11263	17934
6821	15426	14656	17703

Table II: Ultimate Tensile Strength Data of 50 specimens (psi)

Ultimate Tensile Strength Data of 50 specimens (psi)				
103779	103633	103779	103633	103799
102906	102616	101162	107848	103488
104796	106831	102470	99563	102906
103197	102325	105232	105813	101017
100872	104651	103924	108430	104651
97383	105087	102325	106540	103197
101162	106399	105377	101744	105337
98110	100872	104796	101598	101744
104651	104360	106831	103799	106104
102906	101453	105087	100145	100726

The objective is to estimate the reliability of the steel rods against fractures.

Note: The data sets in this example use units that are not shipped with the software (inches, pounds, and pounds per square inch). If you have the “Manage other repository settings” permission, you could add these units to the database, define their conversion factors and have them available for use for any analysis performed within the database. *See page 58 in Chapter 2*. For the purpose of this example, it is not necessary for the software to able to convert these units to some other units, so we will use the simpler approach of setting up all the units to be in hours.

Obtain the Stress Distribution

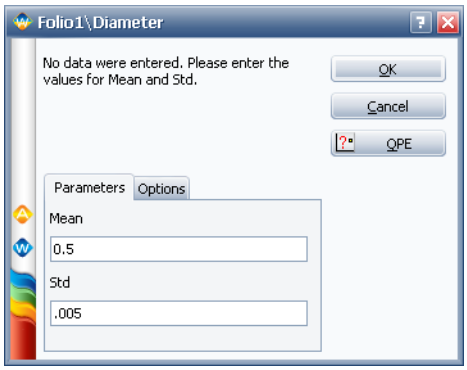
In this example, stress is defined as:

$$stress = \frac{load}{sectional - area} = \frac{F}{\left(\pi d^2\right)/4} \quad \text{(S.1)}$$

where F is the tensile load on the rod and d is the rod's diameter. These variables are distributed; therefore, in order to obtain the stress data set, we will first need to obtain the distributions for these two variables.

1. Let us assume that the diameters of the rods are normally distributed. The parameters of the distribution can then be determined from the given base dimension and tolerance value of the rods. Therefore, the mean of the assumed normal distribution, which is taken to be the base dimension, is 0.5 inches. The standard deviation, which is taken to be a sixth of the total tolerance value, is 0.005 inches.

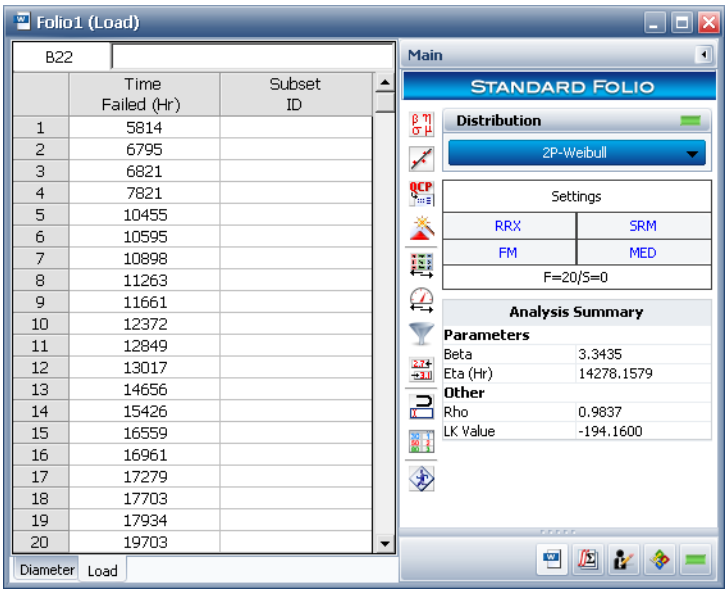
   To use this information, create a new Weibull++ standard folio in the project. Rename the data sheet to "Diameter." On the control panel, choose the **Normal** distribution and the **RRX** parameter estimation method. Click the **Calculate** icon on the Main page of the control panel, then in the input window, enter **0.5** for the mean and **0.005** for the standard deviation, as shown next.



   Click **OK**. The data sheet will be empty but the parameters of the distribution will be visible in the **Analysis Summary** area of the control panel.

2. For the distribution of the tensile load, we can use the data from Table I. Create a new data sheet in the same folio by right clicking the data sheet tab area (the area at the bottom of the window that shows the name of the data sheets) and choosing **Insert Data Sheet** on the shortcut menu. Rename the data sheet to "Load."

3. Enter the data from Table I and calculate the parameters using the **2P-Weibull** distribution and the **RRX** parameter estimation method. The parameters of the distribution for the tensile load *F* are beta = 3.3435 and eta = 14278.1579 lbs, as shown next.

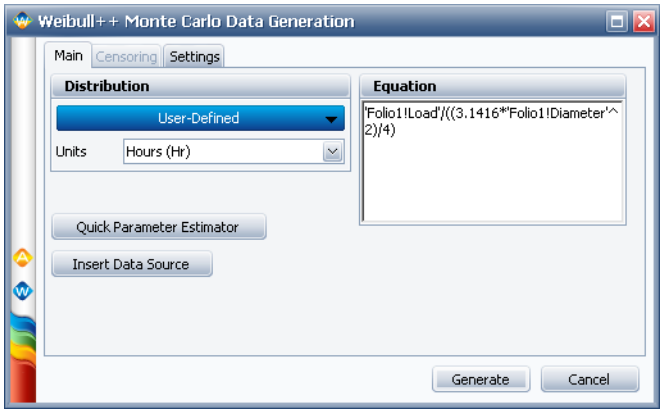


4. Now that the distributions of the variables governing Eqn. (S.1) are available, we can now obtain the stress data set. There are several conventional methods for synthesizing distributions. For this example, we will use a Monte Carlo simulation:

   - Open the Monte Carlo utility by choosing **Home > Tools > Weibull++ Monte Carlo**.



   - In the distribution drop-down list, choose **User-Defined**. Enter Eqn. (S.1) in the Equation field and click the **Insert Data Source** button to insert the distributions of the variables in the equation, as shown next.

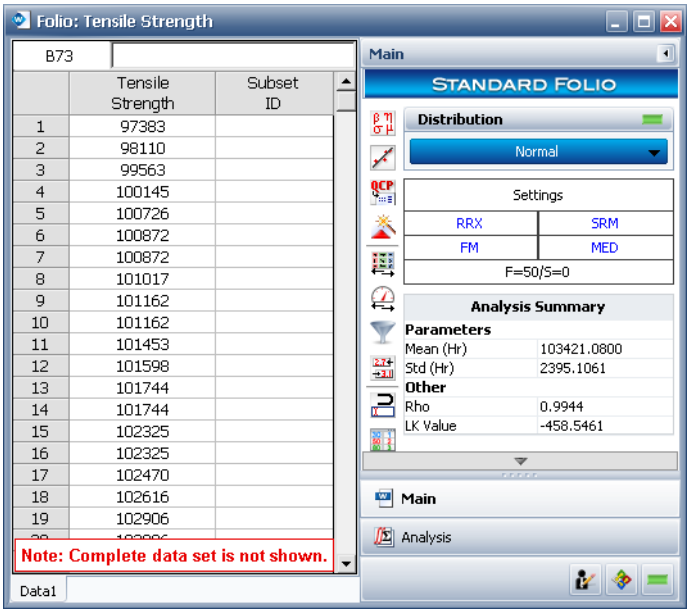


   Click the Settings tab. In the **Data Points** area, enter **1,000** and then click **OK** to generate the stress data set. This creates a new folio in the project. Rename the new folio to "Stress Values."

   - Finally, fit a distribution to the data set in the "Stress Values" folio. For this example, the distribution that best fits the stress data is a generalized gamma distribution with a mu = 11.2127 psi, sigma = 0.2856 psi and lambda = 1.0632 psi. Note that your results may vary because the data were obtained through simulation.

Obtain the Strength Distribution

5. To obtain the strength distribution, use the tensile strength data from Table II. Create a new Weibull++ standard folio and rename it to "Tensile Strength."
6. Enter the data from the table in the data sheet and calculate the parameters using the **Normal** distribution and the **RRX** parameter estimation method, as shown next (showing only the first 19 rows of data).



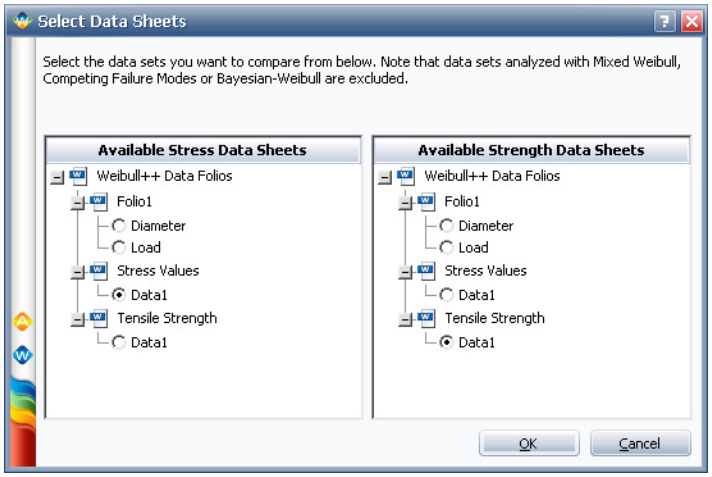
The parameters for the normal distribution of the tensile strength are mean = 103421.0800 psi and standard deviation = 2395.1061psi.

Calculate the Reliability

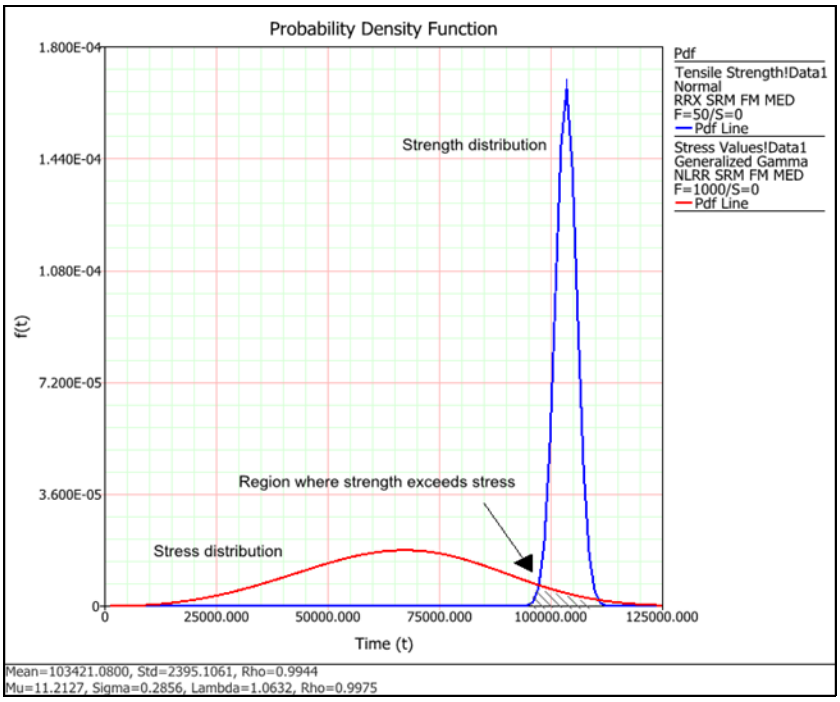
7. Finally, use the stress-strength folio to calculate the reliability of the steel rods. Add the folio to the project by choosing **Insert > Tools > Stress-Strength**.



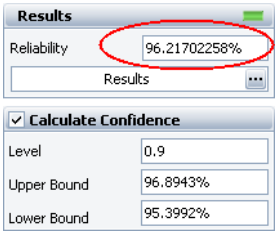
Select the data sheets containing the stress and strength data, as shown next. Click **OK**.



The stress-strength test shows the following plot (with the scaling adjusted and an area of the plot shaded to show the region where the strength exceeds the stress).



The result on the control panel indicates that the reliability of the steel rods against fractures is estimated to be 96.2171%, as shown next.



21.5 Target Reliability Parameter Estimator

The life comparison and stress-strength tools include the Target Reliability Parameter Estimator, which allows you to explore how you could attempt to change one of the parameters in order to meet a specified target reliability. The plot and reliability estimate will be based on the altered distributions, and therefore may be appropriate only within the context of your specific "what-if" analysis.

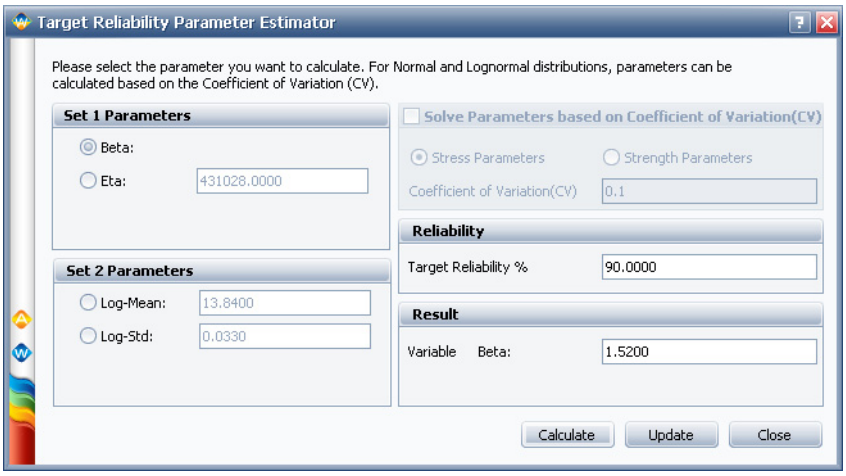
To open the Target Reliability Parameter Estimator window, choose **[Life Comparison/Stress-Strength] > Analysis > Parameter Estimator** or click the icon on the control panel.



You can use the Target Reliability Parameter Estimator with any combination of distributions. The Weibull++ and ALTA versions of the tool are similar, except for one setting:

- The **Solve Parameters based on Coefficient of Variation (CV)** check box is available only in the Weibull++ version of the tool *and* when the distributions are either both normal or both lognormal. Select the parameters to estimate and then enter the CV value. The tool will then estimate the parameters that would result in the target reliability.

The example shown next displays the Weibull++ version of the tool. In this example, the first data set shows the parameters of a Weibull distribution, while the second data set shows the parameters of a lognormal distribution. If we are trying to achieve a target reliability of 90% and select to estimate the beta of the first data set, then the tool will hold the rest of the parameters constant and estimate a beta value that would result in the 90% target reliability. For this example, the result is a beta estimate of 1.5200.



To see how the change in the parameter value affects the plot, click **Update**. The Target Reliability Parameter Estimator window will close and the plot will refresh. The control panel will display the new probability value or reliability estimate, as well as a status that indicates that the parameters have been altered.[4]

4. For a practical example on how you might use the Target Reliability Parameter Estimator tool, see: http://www.ReliaWiki.org/index.php/Stress-Strength_Analysis_in_Design_for_Reliability.

Maintenance Planning 22

In life data analysis, it is assumed that the components being analyzed are non-repairable; that is, they are either discarded or replaced upon failure. When non-repairable components are assembled as part of a system, the system's maintenance activities will involve replacing those worn or failed components. With the Maintenance Planning tool, you can use the information obtained from life data analysis to determine the most cost-effective time to replace an individual component.

The Maintenance Planning tool offers two strategies:

- **Planned replacement** is the practice of replacing components before they fail in order to promote continuous system operation or to avoid dangerous or costly failures. Cost is always a factor with scheduled replacements: replace components too often and you increase the maintenance costs; schedule replacements too far apart and you increase the risk of failures. The goal, then, is to minimize overall costs by finding the optimal time to replace components.
- **Inspections** are used to reveal impending failures, such as corrosion or other forms of degradation. In general, a component is not replaced during inspection unless it is found failed, but if there is evidence that the component is approaching the end of its life, then it may be possible to schedule a replacement at the earliest convenience rather than allowing the failure to occur and possibly cause severe consequences. The goal, then, is to find the optimal time to perform the inspections such that oncoming failures are detected and the component can be scheduled for replacement before any failures occur.

The tool uses the optimum age replacement model to estimate the ideal planned replacement and inspection intervals. The model uses the following assumptions: a) the failure rate of the component increases with time, implying wearout, and b) the cost of replacing the component before failure occurs is significantly less that the cost of replacing the component at the time of failure.

22.1 Generating a Cost vs. Time Plot

Follow the steps outlined below to generate a cost vs. time plot.

1. Open the tool by choosing I**nsert > Tools > Maintenance Planning**.



   Select the data sheet to use for the analysis. Note that the data sheet must have already been analyzed (i.e., the parameters of the distribution have been calculated) in order for you to select it.

2. Click the **Plot Type** drop-down list on the control panel and select which plot to generate. You can choose to create a Planned Replacement plot, an Inspection Replacement plot, or place both on the same plot.

3. In the **Costs** area, enter the required inputs. If you have selected the **Planned Replacement** plot, only the first two inputs are required. For the two other plots, all four inputs are required. These are:
   - **Planned Replacement** is the average cost of replacing the component before failure occurs. This includes downtime costs and other associated costs.
   - **Unplanned Replacement** is the average cost of replacing the component due to failure. This includes downtime costs and other costs associated with the risk, such as lawsuits over the failure of a safety-critical item, loss of goodwill, etc.
   - **Inspection** is the average cost of each inspection procedure.
   - **Failure Detection** is a value between 0 and 1. It indicates the percentage of the component's life that must have elapsed in order for the oncoming failure to be detected. For example, if the component is estimated to fail in 1,000 days and the failure detection threshold value is 0.9, then the oncoming failure can begin to be detected at 900 days.

Note: By default, the planned and unplanned replacement costs are based on the average long-term costs, but there may be cases where the average costs change with time, or when components are replaced with either costlier or less expensive versions. In this situation, you may wish to compute for the optimal time based on the average costs within one replacement cycle (defined as the time when a failure occurs or the time of the scheduled replacement when no failure occurs). To make this change, select the **Plot One Cycle Cost/Time** option on the Settings page of the control panel.

4. In the **Scaling** area, the **X** and **Y** boxes show the minimum and maximum values for the x- and y-axes. To change these values, clear the check box beside the value range. If it is selected, the application will automatically choose appropriate values for the range.
5. To calculate the results and generate the plot, choose **Maintenance Planning > Analysis > Plot** or click the icon on the control panel.



The **Parameters** area of the control panel shows the parameters of the distribution of the data set, while the **Results** area shows the estimated minimal cost and optimal time for the replacement. To see a summary report of the analysis, click anywhere within these two areas. To open the standard folio that contains the data set, click the blue link that displays the name of the folio and data sheet.

22.1.1 Integration with BlockSim

You can use the results from the maintenance plan to create preventive and/or inspection tasks for use in BlockSim simulation diagrams. In this case, the tasks represent the maintenance activities for non-repairable components (i.e., the tasks are configured to 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).

To create a task, choose **Maintenance Planning > Analysis > Create Task**.



This opens a Maintenance Task window that is identical to the interface used in the BlockSim application. The Task Scheduling and Restoration properties are pre-configured based on the analysis results. You can edit these properties and/or define all the other properties of the task, if desired.

When you create a task, the task's name will appear as a link on the control panel, under the **Results** area. You can click the link to view its properties.

The tasks you create can also be accessed by both applications via the Scheduled Tasks page of the Resource Manager by choosing **Project > Synthesis > Resource Manager**.



22.2 Maintenance Planning Control Panel Settings

The Maintenance Planning tool's control panel consists of two pages: Analysis page and Settings page.

Analysis Page

The Analysis page contains the options for generating a cost vs. time plot. The following commands available on this page.



Plot updates the plot to reflect any changes that have been made, and recalculates the estimates for the minimal cost and optimal time.



Plot Setup opens the Plot Setup window, which allows you to customize most aspects of the plot including the titles, line styles and point styles. *See Chapter 15*.



RS Draw launches ReliaSoft Draw, which allows you to annotate your plot and view your plot in greater detail. *See Chapter 16*.



Export Plot Graphic opens the Save As window, which allows you to save the current plot graphic in one of the following formats: *.jpg, *.gif, *.png or *.wmf.



Select Data Sheet opens a window that allows you to select the data sheet to analyze.



Create Task opens the Maintenance Task window, which allows you to create a preventive and/or inspection task based on the analysis results for use in BlockSim simulation diagrams. The name of the last created task will appear as a link on the Main page of the control panel, under the **Results** area. You can click the link to view the task's properties.

Settings Page

The Settings page contains the options for setting how you want the plot to be generated and how you want the optimum time to be computed.

- **Iterations** specifies the maximum number of iterations that the algorithm will perform in order to obtain a solution.
- **Accuracy** specifies the tolerance value that will be used as the convergence limit for the iterations.
- **Plot One Cycle Cost/Time** computes for the optimum time based on the average costs within one replacement cycle, which is defined as the time when a failure occurs or the time of the scheduled replacement when no failure occurs. This calculation may be useful in cases where the average costs change with time, or when components are replaced with either costlier or less expensive versions.
- **Plot Long Term Cost/Cycle** computes for the optimum time based on the average long term costs. This is the default calculation method.

Equation Fit Solver 23

The Equation Fit Solver is a tool that allows you to estimate the parameters of any user-defined non-linear equation. This gives you the flexibility to perform simple parameter estimation on statistical models other than the life distributions available in Weibull++.

WEIBULL++ Only

23.1 Using the Equation Fit Solver

To add an Equation Fit Solver folio to a project, choose **Insert > Tools > Equation Fit Solver** or right-click the **Tools** folder in the current project explorer and choose **Add Equation Fit Solver** on the shortcut menu.



Example

Consider a product that has the following mean life for each stress level:

Table 1: Stress vs. Mean Life

Stress Level Temperature (K)	Mean Life (Hr)
393	6685.111904
408	5730.096004
423	4775.08002

The relationship between the stress level and mean life can be described by the Eyring acceleration model:

$$m = \frac{1}{T} e^{-\left(A - \frac{B}{T}\right)}$$

where:

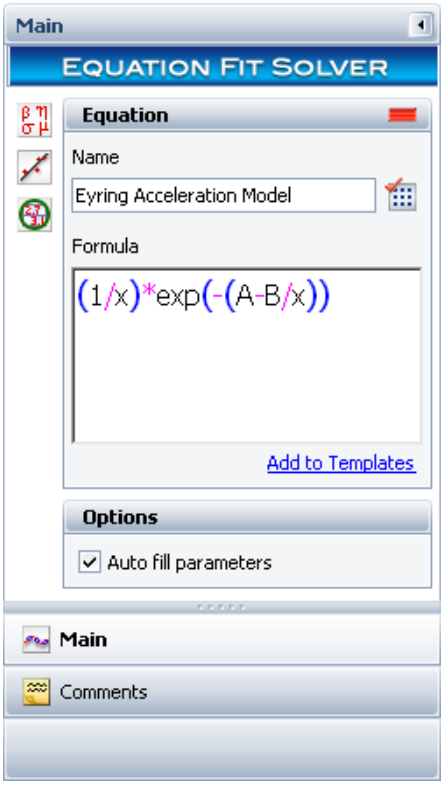
- m is the mean life.
- T is the operating temperature (i.e., use stress level).
- A and B are the parameters of the equation.

In order to estimate the mean life at any given temperature, parameters A and B must be determined. To solve for these parameters, follow the steps outlined below:

1. The first step is to write the Eyring acceleration model in the form $y = F(x)$, where x is the random variable. For this example, let $T = x$ and $m = y$. This results in:

$$y = \frac{1}{x} e^{-\left(A - \frac{B}{x}\right)}$$

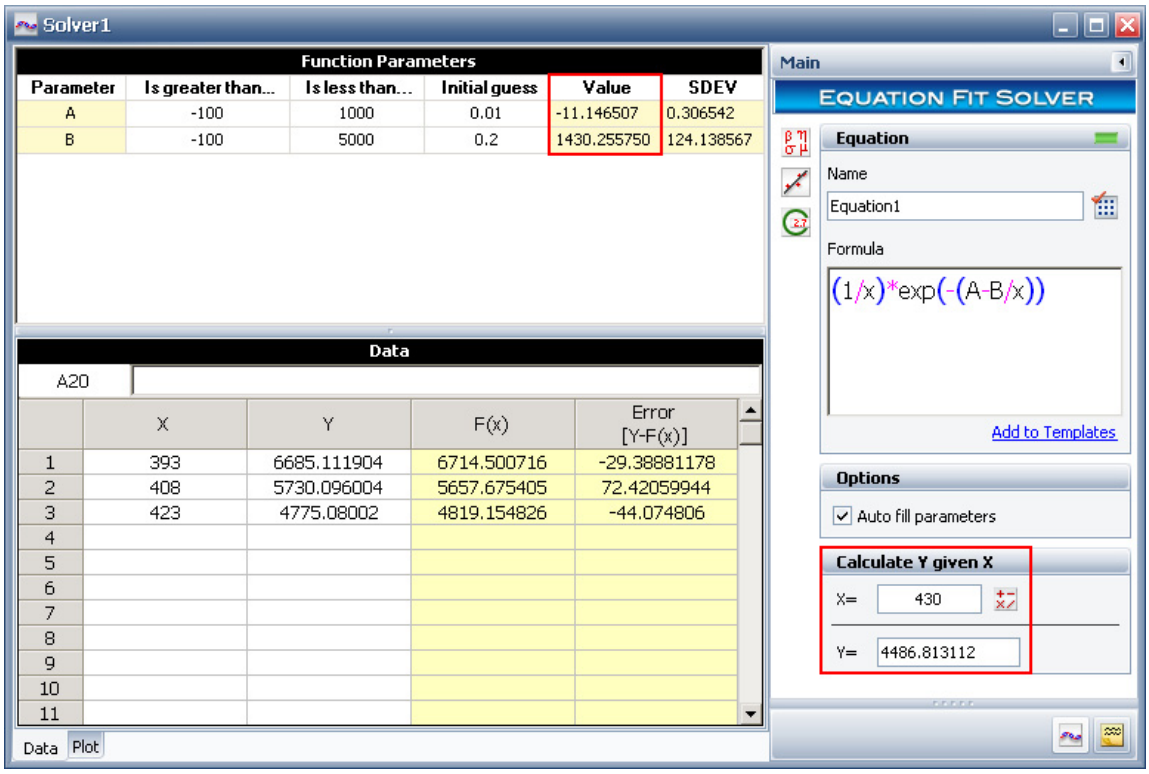
2. Enter the equation in the **Formula** field of the control panel, as shown next.



3. Use the **Function Parameters** area of the folio to provide a starting point for estimating the parameters.
   - In the **Is greater than** and **Is less than** columns, set an estimate for the minimum and maximum values for each parameter in the equation. If the iteration does not converge at the specified range, you will need to edit the range values until a solution can be obtained. For this example, use the following values:
     - A is estimated to be greater than **-100** and less than **1000**.
     - B is estimated to be greater than **-100** and less than **5000**.
   - In the **Initial Guess** column, set a value from which the approximation for the corresponding parameter will start. If the iteration cannot arrive at a solution, you will need to edit the initial guess values until a solution can be obtained. For this example, the initial guess for A is **0.01** and the initial guess for B is **0.2**.

4. In the **Data** area, enter the information from Table 1. Enter the stress level values in the **X** column and enter the mean life values in the **Y** column.
5. Click the **Calculate** icon on the control panel to solve for the parameters of the equation.

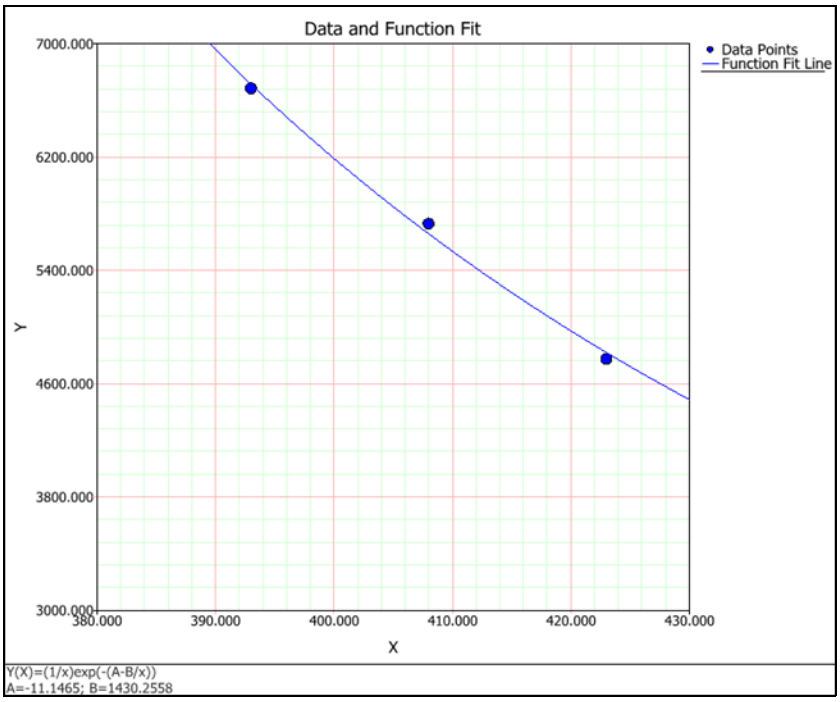
   The parameter values are found to be A = -11.1465 and B = 1430.2558, as shown next. By obtaining the parameters, we can now estimate the mean life (y) for any given stress level (x) by using the **Calculate Y given X** tool. For example, in the picture shown next, the estimated mean life for a temperature of 430 K is 4486.81 hours.



The following columns allow you to evaluate the results of the parameter estimation:

- The **SDEV** column shows the standard deviation of the values of the parameters, which helps you to evaluate the amount of variability in the data set.
- The **F(x)** column shows the calculated mean life (y) at each stress level (x) when the estimated values of the parameters are substituted in the user defined equation.
- The **Error [Y-F(x)]** column shows the calculated distance (the error) of the known value in the Y column to its corresponding value in the F(x) function. This helps you to evaluate the fit of the F(x) function to the data set.

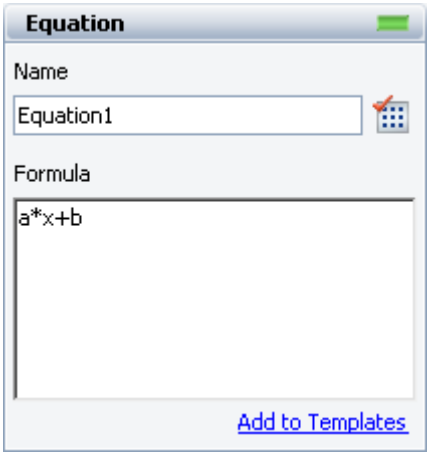
6. To visually assess how well the function fits the known data points of the equation, click the **Plot** icon on the control panel. The Data and Function Fit plot is shown next (with the scaling adjusted).



23.2 Control Panel Settings

The Equation Fit Solver control panel contains the following settings:

- The **Add to Templates** command on the control panel gives you the option to save user-defined equations as templates for future use. To use this feature, enter the equation in the **Formula** field, type a name for the equation and then click **Add to Templates**, as shown in the following example.



- To use a saved template, choose **Equation Fit Solver > Solver Templates > Select from Templates** or click the icon on the control panel.



This opens the Existing Equation Templates window, which shows a list of all the saved templates. The buttons at the lower left side of the window allow you to add, edit or delete templates.

- The **Auto Fill Parameters** check box allows you to automatically copy all the unknown variables in the user-defined equation (except variable x) to the Function Parameters area. If the check box is cleared, you can automatically copy the parameters of the equation to the Function Parameters area by choosing **Equation Fit Solver > Analysis > Initialize Parameters** or by clicking the icon on the control panel.



- The **Calculate Y given X** tool uses the user-defined equation and the fitted parameters to calculate the value of y given the value of x. This tool is available only when you have calculated the parameters. Enter a value for x and click the **Analyze** button to compute the results.



Folio Tools

Additional folio tools are arranged on the left side of the control panel:



Calculate estimates the parameters of the user-defined equation and calculates other results you could use to evaluate how well the function fits the data. This tool is also available by choosing **Equation Fit Solver > Analysis > Calculate**.



Plot creates a plot demonstrating the fit of the function for each known data point that you defined in the Data area. The scaling, setup, exporting and other features of the plot are similar to the options available for other Weibull++ plot sheets. Features that are not applicable to this plot will be hidden or disabled. This tool is also available by choosing **Equation Fit Solver > Analysis > Plot**.



Initialize Parameters automatically copies all the unknown variables in the user-defined equation (except the random variable *x*) to the Function Parameters area. This also resets the minimum, maximum and initial guess values to their default values. This tool is also available by choosing **Equation Fit Solver > Analysis > Initialize Parameters**.

ALTA Test Plan Utility 24

The ALTA Test Plan utility is a planning tool that helps you design a one-stress or two-stress accelerated life test that will reduce the estimation uncertainty of product life under normal operating conditions. It generates a test plan with recommendations for what stress levels should be used in the test and how many units should be tested at each stress level.

This chapter describes how to use the ALTA Test Plan utility, including:

- Generating a test plan - Section 24.1 (p. 515).
- An overview of the available types of test plans - Section 24.2 (p. 516).
- Test Plan Results sheet - Section 24.3 (p. 518).
- The Test Plan control panel - Section 24.4 (p. 518).
- Example of using the Test Plan utility - Section 24.5 (p. 519).

24.1 Generating a Test Plan

ALTA Only

Follow the steps outlined below to generate a recommended test plan.

- Create a new Test Plan folio by choosing **Insert > Tools > ALTA Test Plan**.



- From the **Number of Simultaneous Stresses** drop-down list, choose whether you will be designing a test that applies only one type of stress during the test, or a combination of two stresses applied simultaneously (e.g., temperature and humidity).
- From the **Test Plan Type** drop-down list, choose an appropriate test plan *(see Section 24.2 on page 516)*. The test plan *level* is the number of stress levels that the plan recommends (e.g., a 3 level plan will recommend subjecting some units to a low stress level, some to a mid level and the rest to a high level). Test plans may also differ according to how they allocate the units to stress levels. Your specified test plan will minimize the variance on the value you enter in the **BX Life Estimate Sought** field.
- In the **BX% Life Estimate Sought** field, enter (as a percentage) the BX% life (under normal operating conditions) that you intend to estimate from the test (e.g., for the B10 life, you would enter 10).
- In the **Available Test Time** field, enter the duration of the accelerated life test you are planning. (The duration can be entered in any unit of time, but you must use the same time unit throughout the utility.)

- From the **Unit Allocation** drop-down list, choose **Show Allocations as %** if you do not want to specify the total number of units that will be available for testing. In this case, the test plan will only display the percentage of the available units that should be allocated to each stress level.

  If you want the test plan to display both the percentage and the quantity of test units allocated to each stress level, choose **Show Allocations as Qty** and then enter the total number of units that will be available in the **Number of Units Available** field.

- From the **Lifetime Distribution** drop-down list, select the distribution that describes your product's life behavior. You can select **Weibull** and enter the shape parameter in the **Beta** field, or select **Lognormal** and enter the shape parameter in the **Std** field.

> **Tip:** If your product's underlying life distribution is exponential, select Weibull from the **Life Distribution** drop-down list and enter **1** in the **Beta** field. This is equivalent to using an exponential distribution.

- Next, in the **Stress 1** area, choose the appropriate **Life-Stress Relationship** from the drop-down list. Then enter the stress level at normal use conditions in the **Use Stress Value** field. In the **Maximum Stress Value** field, enter the maximum stress level that could be applied during the test without introducing failure modes that would not occur under normal use conditions. Repeat these steps for **Stress 2** if you selected to design a two-stress test.
- In order to provide reasonable recommendations for the stress levels to be used in the test, the software must estimate a life-stress relationship for your product. The utility will estimate a life-stress relationship using the values you enter in the **Probabilities of Failure (%)** area. Each probability of failure is for the time you entered in the **Available Test Time** field and at the stress level described.
  - For example, if you specified a test time of 100 and one of the fields in this area is **P(Time, Use Stress 1, Maximum Stress 2)**, then you should enter the estimated probability of failure for the product at time = 100, where the product experiences Stress 1 at the specified use stress value and Stress 2 at the specified maximum stress value.
- Click **Calculate** to generate the recommended test plan. The Test Plan Results sheet will appear (*see Section 24.3 on page 518 for information on how to read the results and evaluate the test plan*). You can further evaluate the recommended plan by using the utility's control panel.

24.2 Test Plan Types

The ALTA Test Plan utility allows you to choose a test plan type that will best fit your test goals and resource constraints. Which test plans are available will depend on whether you chose to design a one-stress or two-stress test in the **Number of Simultaneous Stresses** drop-down list. Each test plan will attempt to minimize the variance on the BX% life you specified in the **BX% Life Estimate Sought** field.

If you selected to design a single-stress test, the following test plans will be available:

- **2 Level Statistically Optimum Plan**
  Recommends two stress levels: a high stress level, which is the maximum allowable stress value that you specified during setup, and a low stress level. The low stress level and the proportion of units allocated to each stress level are calculated in order to minimize the variance on the BX% life.
- **3 Level Best Standard Plan**
  Recommends three stress levels: a high stress level, which is the maximum allowable stress value that you specified during setup, a low stress level and a mid stress level. The low and mid levels are calculated in order to minimize the variance on the BX% life. The test units are allocated equally to each stress level.
- **3 Level Best Compromise Plan**
  Recommends three stress levels using the same approach described above for the 3 Level Best Standard Plan. You define the proportion of test units to allocate to the mid stress level during setup in the **Proportion of Units at Mid Stress** field. The other proportions are calculated in order to minimize the variance on the BX% life.

- **3 Level Best Equal Expected Number Failing Plan**
  Recommends three stress levels using the same approach described above for the 3 Level Best Standard Plan. The proportion of units allocated to each stress level is calculated such that the number of units expected to fail at each stress level is equal.
- **3 Level 4:2:1 Allocation Plan**
  Recommends three stress levels using the same approach described above for the 3 Level Best Standard Plan. The proportion of test units tested at the high, mid and low stress levels will be calculated to be as close as 4:2:1 as possible (e.g., if you specified that 7 units are available in the **Total Number of Units** field, then this plan will recommend testing 4 units at the high level, 2 at the mid level and 1 at the low level).
  - When you select this plan, you must also enter a value in the **Low Stress Adjustment Factor** field. The reduction factor must be a decimal between 0 and 1. The lower the value, the closer the recommended low stress level will be to the product's specified use stress level. Thus, if the reduction factor is 0, then the low stress level will be equal to the use stress level. If the reduction factor is 1, then the low stress level will equal the stress value that minimizes the variance on the BX% life.

If you selected to design a two-stress test, the following test plans are available:

- **3 Level Optimum Plan**
  Recommends three stress levels, each consisting of a combination of stress values for the two stress types. The combinations are structured such that:
  - The 1st combination has both Stress 1 and Stress 2 at the maximum allowable stress values that you specified during setup.
  - The 2nd combination will have Stress 1 at a calculated mid stress level and Stress 2 at the use stress level.
  - The 3rd combination will have Stress 1 at the use stress level and Stress 2 at a calculated mid stress level.

  The mid stress level is calculated such that the units tested at the 2nd and 3rd stress combinations will have the same probability of failure at the end of the test. The proportion of units allocated to each stress level is computed in order to minimize the variance on the BX% life.
- **5 Level Best Compromise Plan**
  Recommends five stress levels, each consisting of a combination of stress values for the two stress types. The combinations are structured such that:
  - The 1st combination will have both Stress 1 and Stress 2 at the maximum stress values that you specified during setup.
  - The 2nd combination will have Stress 1 at a calculated mid stress value A and Stress 2 at the use stress level.
  - The 3rd combination will have Stress 1 at the use stress level and Stress 2 at a calculated mid stress value B.
  - The 4th combination will have Stress 1 at the specified maximum stress value and Stress 2 at a calculated mid stress value C.
  - The 5th combination will have Stress 1 at a calculated mid stress value D and Stress 2 at the specified maximum stress value.

  The mid stress values A and B are calculated such that the units tested at the 2nd and 3rd stress combinations will have the same probability of failure at the end of the test. The mid stress values C and D are calculated such that the units tested at the 4th and 5th stress combinations will have the same probability of failure at the end of the test. The proportion of units allocated to each stress level is computed in order to minimize the variance on the BX% life.

24.3 Test Plan Results Sheet

The Test Plan Results sheet displays a summary of your inputs, the utility's recommended test plan and the time at which unreliability equals the value you entered in **BX% Life Estimate Sought** field (i.e., Tp). If you entered a number of available test units, it also displays the standard deviation of Tp.

The results sheet is divided into the following areas:

- The **Test Plan Inputs** area displays a summary of all the inputs that were used to generate the test plan. This includes information for each specified stress type and probabilities of failure at various stress levels.
- The **Recommended Test Plan** area displays the recommended stress levels to be used in the test and the recommended allocation of units to each stress level. Each row describes a stress level (which, for two-stress tests, will be a combination of stress values) and the number of units to be tested at that stress level. For example, consider the following plan.

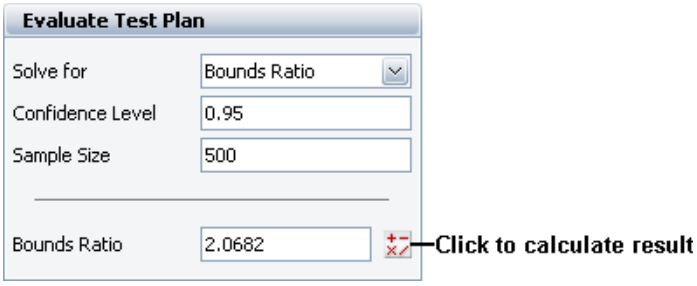
21	Recommended Test Plan					
22	**Stress Level**	**Stress 1**	**Stress 2**	**Unit Allocation (%)**	**Unit Allocation (Qty)**	**Probability of Failure**
23						
24	First Stress Level	400	25	25.2	12.6	0.8772
25	Second Stress Level	285.4838	10	27.65	13.8237	0.2127
26	Third Stress Level	275	10.7376	47.15	23.5763	0.2127
27						
28	**BX% Life Estimate**					
29	Time at Which Unreliability (Tp)=5%	55.538				
30	Standard Deviation of Tp	10.3751				

The row labeled “First Stress Level” recommends that 25.2% of the available units be tested at a level of 400 for Stress 1 and 25 for Stress 2. The **Unit Allocation (Qty)** column is visible when the **Show Allocations as Qty** option was chosen from the **Unit Allocation** drop-down list on the Test Plan Inputs sheet. In this example, it shows that 25.2% of the number of available units is 12.6.

  - **Time at Which Unreliability (Tp) = 5%** is the time at which unreliability equals the value you entered for **BX Life Estimate Sought**. This value is calculated based on the life distribution and probabilities of failure you entered in the Test Plan Inputs sheet.
  - **Standard Deviation of Tp** is calculated only if you choose **Show Allocations as Qty** on the Inputs sheet. The smaller the standard deviation of Tp, the less variation in Tp you can expect in repeated samplings. Thus, smaller values indicate a more robust test plan.

24.4 Test Plan Control Panel

The Main page of the Test Plan's control panel gives you options to further evaluate the recommended test plan by solving for any one of the criteria in the **Solve for** drop-down list.



- The **Confidence Level** is the confidence level (as a decimal) of the 2-sided bounds on the BX% life estimated from data obtained using the recommended plan. The calculated confidence level can be increased by increasing the bounds ratio or sample size.

- The **Bounds Ratio** is the ratio of the upper confidence bound to the lower confidence bound at the specified confidence level. It indicates the relative size of the variance on the BX% life estimate. The narrower the confidence bounds, the closer the ratio is to 1, indicating less uncertainty. The calculated bounds ratio can be decreased by decreasing the confidence level or increasing the sample size.
- The **Sample Size** is the number of units that are used in the test. The sample size can be decreased by decreasing the confidence level or increasing the bounds ratio.

To solve for one of these three criteria, you must provide values for the other two. By using the Main page of control panel, you can determine whether your goals with respect to these criteria are realistic (given factors such as sample size constraints), and you can explore various possible trade-offs between the three criteria.

Tip: ALTA also includes a simulation tool called SimuMatic that you can use to visualize the confidence bounds associated with the bounds ratio presented in the test plans utility. The Test Plan utility uses an analytical solution that is based on the assumption of a large sample size; SimuMatic does not make any assumptions about the sample size and thus may be preferred in the case of small sample sizes. *The SimuMatic utility is discussed in Chapter 17*.

To solve for one of these criteria, follow the steps outlined below:

- From the **Solve for** drop-down list in the **Input** area of the control panel, choose the value you wish to solve for. The two input fields immediately below the drop-down list will change depending on what you choose. For example, if you choose **Sample Size**, the two input fields will be **Confidence Level** and **Bounds Ratio**.
- Enter the appropriate values in the two input fields in the **Input** area.
- Click the **Calculate Result** icon inside the **Result** area of the control panel to solve for your selected value.

The control panel also includes a Comments page that allows you to enter notes or other text that will be saved with the folio.

24.5 Test Plan Utility Example

This section presents an example of how to use the ALTA Test Plan utility to generate a test plan. The test plan will recommend the stress levels that should be used in a test and the number of units that should be tested at each stress level.

A reliability group in a semiconductor company is planning an accelerated life test for an electronic device. 150 test units will be employed for the test, and the test is planned to last for 600 hours. Temperature and voltage have been determined to be the main factors affecting the reliability of the device, and three chambers are available for testing the units. The goal of the test is to estimate the B10 life (i.e., the time at which unreliability = 10%) of the device under the normal use conditions of 300 K for temperature and 4 V for voltage.

The engineers have determined that the maximum stress values that could be applied during the accelerated life test are temperature = 360 K and maximum voltage = 10 V. Stresses beyond this level would introduce failure modes that do not occur under normal conditions. The product follows a Weibull distribution with beta = 3. Based on their knowledge of the life-stress relationship, the engineers estimate the following probabilities of failure after the product is tested for 600 hours at the different combinations of use level and maximum level stress values:

- 2% at 300 K and 4 V.
- 40% at 360 K and 4 V.
- 90% at 300 K and 10 V.

Generating a Test Plan

Follow the steps outlined below to generate a test plan:

- Open the Test Plan utility by choosing **Insert > Tools > ALTA Test Plan**.
- Select to design a two-stress test by choosing **2** from the **Number of Simultaneous Stresses** drop-down list.
- Because only three testing chambers are available, three different stress levels will be used for each of the two stress types. Choose **3 Level Optimum Plan** from the **Test Plan Type** drop-down list.
- Enter **10** in the **BX% Life Estimate Sought** field to specify that the test will be used to estimate the B10 life.
- Enter **600** in the **Available Test Time** field.
- From the **Unit Allocation** drop-down list, choose **Show Allocations as Qty**. Then enter **150** in the **Number of Units Available** field.
- Choose **Weibull** from the **Lifetime Distribution** drop-down list. Then enter **3** in the **Beta** field.
- In the **Stress 1** area, enter the information for the first stress type (i.e., temperature).
  - Choose **Arrhenius** from the **Life-Stress Relationship** drop-down list. This is the life-stress relationship that is typically used for temperature.
  - Enter **300** in the **Use Stress Value** field.
  - Enter **360** in the **Maximum Stress Value** field.
- In the **Stress 2** area, enter the information for the second stress type (voltage).
  - Choose **Power** from the **Life-Stress Relationship** drop-down list. This is the life-stress relationship that is typically used for voltage.
  - Enter **4** in the **Use Stress Value** field.
  - Enter **10** in the **Maximum Stress Value** field.
- Enter the estimated probabilities of failure at the end of the test, as shown next.

Probabilities of Failure (%) at Time=600	
P(Time, Use Stress 1, Use Stress 2)	2
P(Time, Maximum Stress 1, Use Stress 2)	40
P(Time, Use Stress 1, Maximum Stress 2)	90

  Each probability of failure is a function to time and two stress levels. For example, the value entered in the **P(Time, Use Stress 1, Maximum Stress 2)** field is the probability of failure at 600 hours (specified in the **Available Test Time** field), assuming the product experiences the specified use stress level for the first stress (300 K) and the specified maximum stress level for the second stress (10 V). The stress values were entered in the **Stress 1** and **Stress 2** areas.
- Click the **Generate Test Plan** button. The Test Plan Results sheet will appear.

- The recommended stress levels and unit allocations are displayed under the **Recommended Test Plan** heading near the bottom of the results sheet. This area of the sheet is shown below.

26	Recommended Test Plan					
27	**Stress Level**	**Stress 1**	**Stress 2**	**Unit Allocation (%)**	**Unit Allocation (Qty)**	**Probability of Failure**
28						
29	First Stress Level	360	10	19.4	29.1	1
30	Second Stress Level	357.0955	4	32.6825	49.0238	0.3611
31	Third Stress Level	300	7.2852	47.9175	71.8762	0.3611
32						
33	**BX% Life Estimate**					
34	Time at Which Unreliability (Tp)=10%	1040.4967				
35	Standard Deviation of Tp	118.3922				

  For example, according to this recommendation, the first stress level of the test should be combination of 360 K and 10 V, and about 29 units (or 19.4% of the total number of available units) should be tested at that level.

Evaluating the Plan

The reliability group may also wish to evaluate the test plan in relationship to other test goals. For example, the company may desire a test that will estimate the B10 life with 90% confidence and a bounds ratio of 2 or less. The control panel can help you decide whether this can be achieved with the given test plan and with the specified sample size.

To use the control panel, follow the steps outlined below.

- From the **Solve for** drop-down list in the **Input** area of the control panel, select **Bounds Ratio**.
- In the **Confidence Level** input field, enter **0.90**.
- In the **Sample Size** input field, enter **150**, which is the total number of units that will be tested, according to the current test plan.
- Click the **Calculate Result** icon inside the **Result** area of the control panel to solve for the bounds ratio.

In this example, the bounds ratio is calculated to be 1.3575, which is better than the company's desired ratio of 2. As a result, the engineers may consider using a smaller sample size in order to plan a more efficient test. For example, if they entered **40** in the **Sample Size** field and then clicked **Calculate Result** again, the calculated bounds ratio would be 1.8073, which suggests that 40 units may be sufficient for the purposes of the test.

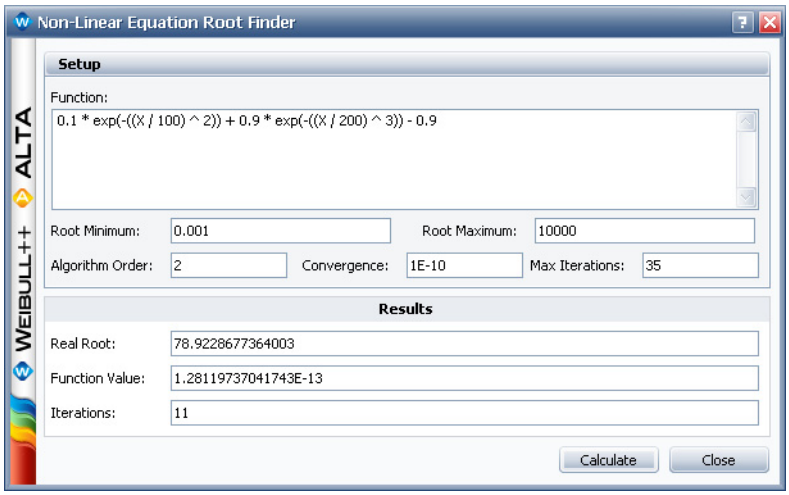
Non-Linear Equation Root Finder 25

Many engineering math problems result in a need to solve non-linear equations. Solving these equations can be complex, tedious and in some situations, no solution may exist at all. The Non-Linear Equation Root Finder is a utility that allows you to quickly solve for the root of any user-defined non-linear equation. This helps you eliminate some of the guesswork of solving for the value of the unknown variable that brings the function as close to zero as possible.

You can access the utility by choosing **Home > Tools > Non-Linear Equation Root Finder**.



The following is an example of a calculation for the root of an equation.



To use the utility, follow the steps below:

- Enter the user-defined equation in the **Function** box. Use the variable x for the unknown variable.
- In the **Root Minimum** and **Root Maximum** fields, enter the initial guess estimates of the root.
- In the **Algorithm Order** field, enter the variable order which is to be improved in every iteration. Note that the algorithm order must be greater than or equal to 2.
- In the **Convergence** field, set a tolerance value at which the root of the equation should converge (i.e., $f(x) = 0$).
- In the **Max Iterations** field, set the maximum number of iterations the algorithm will perform in order to solve for the root of the equation.

Click **Calculate** to solve for the root of the equation. You may need to edit the inputs in order to obtain the best possible solution for the equation. The **Results** area will return the following:

- **Real Root** returns the root of the equation. If the root is not within the initial specified range, you will need to edit the values in the Root Minimum and Root Maximum fields until the root is within range of those values.
- **Function Value** shows the value of the user-defined equation at the root value. You may need to edit the tolerance value in the Convergence field in order to obtain a solution that is as close as possible to zero.
- **Iterations** returns the number of iterations that were performed in order to obtain the solution.

Quick Statistical Reference (QSR) 26

The Quick Statistical Reference (QSR) tool allows you to quickly and easily calculate many common statistical values (e.g., median ranks and chi-squared values) and interpolate (or extrapolate) values using the polynomial interpolation function.

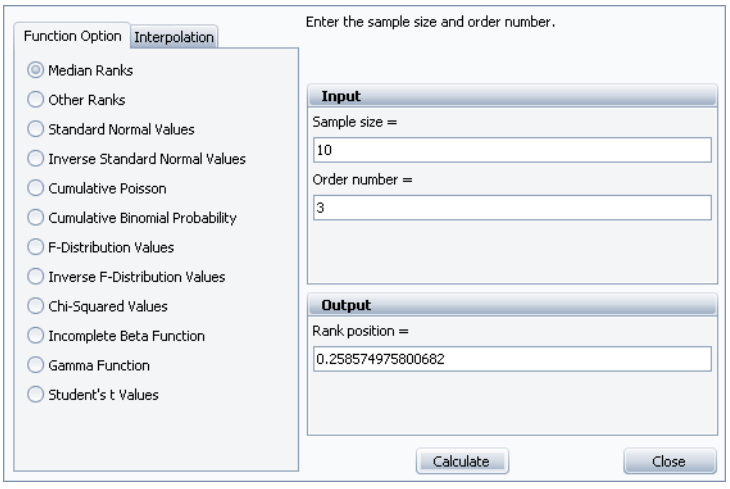
To open the QSR, choose **Home > Tools > Quick Statistical Reference**.



The Function Option page of the QSR is for calculating statistical values. The Interpolation page is for interpolating (or extrapolating) values using the polynomial interpolation function. Both pages are described in the sections below.

26.1 Calculating Statistical Values

To calculate a statistical value, select the appropriate function option on the left side of the Function Option page. Then enter the required inputs and click **Calculate**.



The following options are available.

26.1.1 Median Ranks

The Median Ranks option returns the probability of failure based on the sample size and order number of the failure. The probability estimates are at a 50% confidence level.

The median rank is obtained by solving the following equation for *Z*:

$$0.5 = \sum_{k=j}^{N} \binom{N}{k} Z^k (1-Z)^{N-k}$$

where:

- *N* is the sample size.
- *j* is the order number.
- *Z* is the median rank.

26.1.2 Other Ranks

The Other Ranks option returns the probability of failure based on the sample size and order number of the failure. The probability estimates are at a confidence level percentage point that is specified by the user.
The rank is obtained by solving the following equation for *Z*:

$$P = \sum_{k=j}^{N} \binom{N}{k} Z^k (1-Z)^{N-k}$$

where:

- *N* is the sample size.
- *j* is the order number.
- *P* is the confidence level.
- *Z* is the rank.

26.1.3 Standard Normal Values

The Standard Normal Tables option returns the probability of observing a value less than or equal to *x* on the standard normal curve, given a value for *x*. To find the value of x given the probability, use the Inverse Standard Normal Values option.

The probability is obtained by solving the following equation for *Z*(*x*):

$$1 - Z(x) = \int_{x}^{\infty} \frac{1}{\sqrt{2\pi}} e^{-\frac{t^2}{2}} dt$$

where *Z*(*x*) is the probability of observing a value less than or equal to *x*.

26.1.4 Inverse Standard Normal Values

The Inverse Standard Normal Tables option returns a value for *x* on the standard normal curve, given the probability of observing a value less than or equal to *x*. To find the probability given *x*, use the Standard Normal Tables option.

The output is obtained by solving the following equation for *x*:

$$1 - Z(x) = \int_{x}^{\infty} \frac{1}{\sqrt{2\pi}} e^{-\frac{t^2}{2}} dt$$

where *Z*(*x*) is the probability of observing a value less than or equal to *x*.

26.1.5 Cumulative Poisson

The Cumulative Poisson option returns the probability of an event occurring n times during a specified interval. The required inputs are *n* and the average rate of occurrence for the event, λ, where $\lambda > 0$.

The probability is obtained by solving for $P(n, \lambda)$ in the following equation:

$$P(n, \lambda) = e^{-\lambda} \sum_{i=0}^{n} \frac{\lambda^i}{i!}$$

where:

- *n* is the maximum number of occurrences (and the upper limit of the summation).
- λ is the average rate of occurrence for the event.

26.1.6 Cumulative Binomial Probability

The Cumulative Binomial Probability option returns the probability of an event occurring *k* or more times in *N* trials. The required inputs are *k*, *N* and the probability (entered as a decimal number) of the event occurring per trial.

The probability is obtained by solving for *P* in the following equation:

$$P = \sum_{i=k}^{N} \binom{N}{i} p^i (1-p)^{N-i}$$

where:

- *P* is the probability of the event occurring *k* or more times in *N* trials.
- *p* is the probability of the event occurring per trial.
- *N* is the minimum number of trials (and the end of the summation).
- *k* is the minimum number of occurrences (and the starting point of the summation).

26.1.7 F-Distribution Values

The F-Distribution option returns $Q(F|n_1, n_2)$, the significance level at which we can reject the hypothesis that one sample has a smaller variance than another. The three inputs required are the degrees of freedom for both samples and the ratio of the observed dispersion of the first sample to that of the second. To find the ratio of the observed dispersion, use the Inverse F-Distribution Values option.

The output is obtained by solving for $Q(F|n_1, n_2)$ in the following equation:

$$Q(F|n_1, n_2) = 1 - \frac{n_1^{\frac{1}{2}n_1} n_2^{\frac{1}{2}n_2}}{B\left(\frac{1}{2}n_1, \frac{1}{2}n_2\right)} \int_0^F t^{\frac{1}{2}(n_1-2)} (n_2 + n_1 t)^{-\frac{1}{2}(n_1+n_2)} dt$$

where:

- n_1 is the degrees of freedom for the first sample.
- n_2 is the degrees of freedom for the second sample.
- *F* is the ratio of the observed dispersion of the first sample to that of the second.
- *B* is the beta function.

26.1.8 Inverse F-Distribution Values

The Inverse F-Distribution option returns *F*, the ratio of the observed dispersion of one sample to that of another. The three inputs required are the degrees of freedom for both samples and the significance level at which we can reject the hypothesis that the first sample has a smaller variance than the second. To find the significance level at which we can reject the hypothesis, use the F-Distribution Values option.

The output is obtained by solving for *F* in the following equation:

$$Q(F|n_1, n_2) = 1 - \frac{n_1^{\frac{1}{2}n_1} \, n_2^{\frac{1}{2}n_2}}{B\left(\frac{1}{2}n_1, \frac{1}{2}n_2\right)} \int_0^F t^{\frac{1}{2}(n_1-2)} (n_2 + n_1 t)^{-\frac{1}{2}(n_1+n_2)} dt$$

where:

- n_1 is the degrees of freedom for the first sample.
- n_2 is the degrees of freedom for the second sample.
- $Q(F|n_1, n_2)$ is the significance level at which we can reject the hypothesis that the first sample has a smaller variance than the second.
- B is the beta function.

26.1.9 Chi-Squared Values

The Chi-Squared Values option returns the chi-squared value at the (1-*d*) percentile. The required inputs are the area to the right of the critical value and the number of degrees of freedom.

The output is obtained by solving for $x^2_{d;v}$ in the following equation:

$$P(X^2 > x^2_{d;v}) = \left[2^{\frac{v}{2}}\Gamma\left(\frac{v}{2}\right)\right]^{-1} \int_{x^2_{d;v}}^{\infty} e^{-\frac{t}{2}} t^{\frac{v}{2}-1} dt$$

where:

- d is the area to the right of the critical value.
- v is the number of degrees of freedom.
- X^2 is a chi-squared random variable with v degrees of freedom.

26.1.10 Incomplete Beta Function

The Incomplete Beta Function option returns the value of $I_x(a,b)$. The required inputs are the values of x, a and b.

The output is obtained by solving for $I_x(a,b)$ in the following equation:

$$I_x(a, b) = \frac{1}{B(a, b)} \int_0^x t^{(a-1)} (1-t)^{(b-1)} dt$$

where $0 < x < 1$.

26.1.11 Gamma Function

The Gamma Function option returns the value of $\Gamma(n)$. The only required input is n.

The output is obtained by solving for $\Gamma(n)$ in the following equation:

$$\Gamma(n) = \int_0^\infty t^{x-1} e^{-t} dt$$

where $n > 0$.

26.1.12 Student's t Values

The Student's t Values option returns the t-value of the Student's t-distribution, given the probability of observing a value equal or less than the t-value and the number of degrees of freedom.

The output is obtains by solving for t in the following equation:

$$a = P(t > t_{a;v}) = \int_{t_{a;v}}^\infty f(t) dt$$

where:

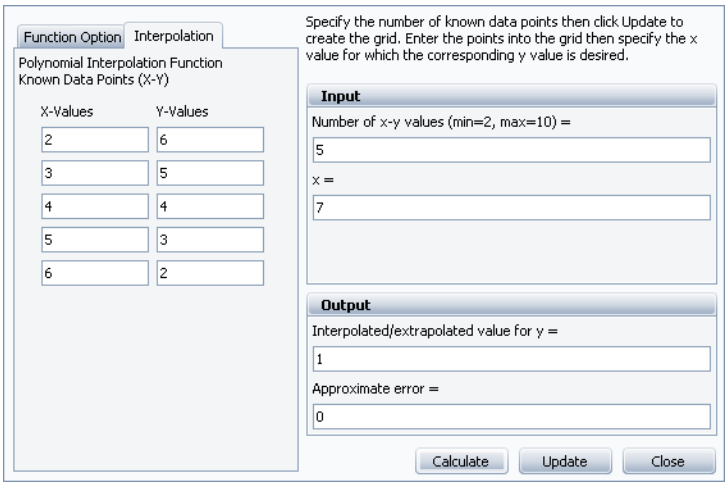
- a is the probability of observing a value equal to or less than t.
- v is the degrees of freedom.

26.2 Interpolating a Value from a Data Set

The Interpolation page uses the polynomial interpolation function to interpolate (or extrapolate) values given a set of known data points that you provide.

The QSR attempts to fit a polynomial to the given data points. The polynomial is of the (i - 1) order, where i is the number of data points. You must enter at least 2 data points, and you cannot enter more than 10.

Note that if the given data points are far from the point of interest, the resulting polynomial can oscillate between values, thus yielding erroneous results. Moreover, interpolation based on only data points, with no background information on the actual function, can also yield erroneous results. For these reasons, an approximate error value is returned with each interpolation.



Follow the steps outlined below to obtain values with the QSR:

- In the **Input** area of the Interpolation page, specify the number of data points you wish to enter in the **Number of x-y Values** field. Then click the **Update** button to create a table on the left of the page where you can enter the known data points that you wish to interpolate from.
- In the table on the left side of the page, enter the x- and y-values data points that you wish to interpolate from.
- In the **Input** area, enter the x-value for which the corresponding y-value will be obtained.
- Click the **Calculate** button to obtain the corresponding y-value of the x-value you entered in the **Input** area. The y-value and an approximate error value are displayed in the **Output** area.

Analysis Workbooks 27

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 27.1 (p. 534)
- Managing the associated data sources - Section 27.2 (p. 534)
- Creating and using saved templates - Section 27.3 (p. 536)
- Using the analysis workbook's control panel - Section 27.4 (p. 537)
- Using the Function Wizard to insert data/results - Section 27.5 (p. 539)
- Using the Chart Wizard and Chart Designer to create custom plots/charts - Section 27.6 (p. 546)
- Tools for working with spreadsheets - Section 27.7 (p. 547)
- An analysis workbook example - Section 27.8 (p. 557)

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.

27.1 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:

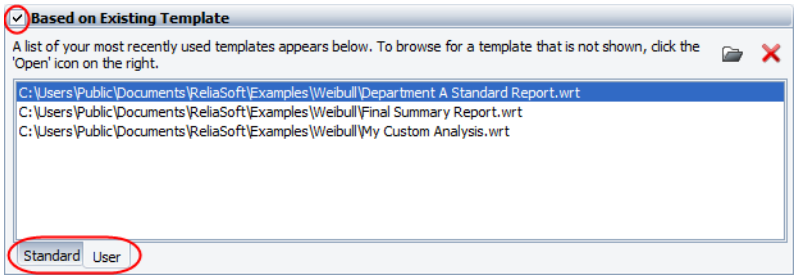
Click **Select** and choose one of the available analyses (i.e., a data sheet in Weibull++/ALTA and RGA, or a diagram in BlockSim). When you return to the wizard, click **OK** to create the report.

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 27.3 on page 536).

27.2 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.

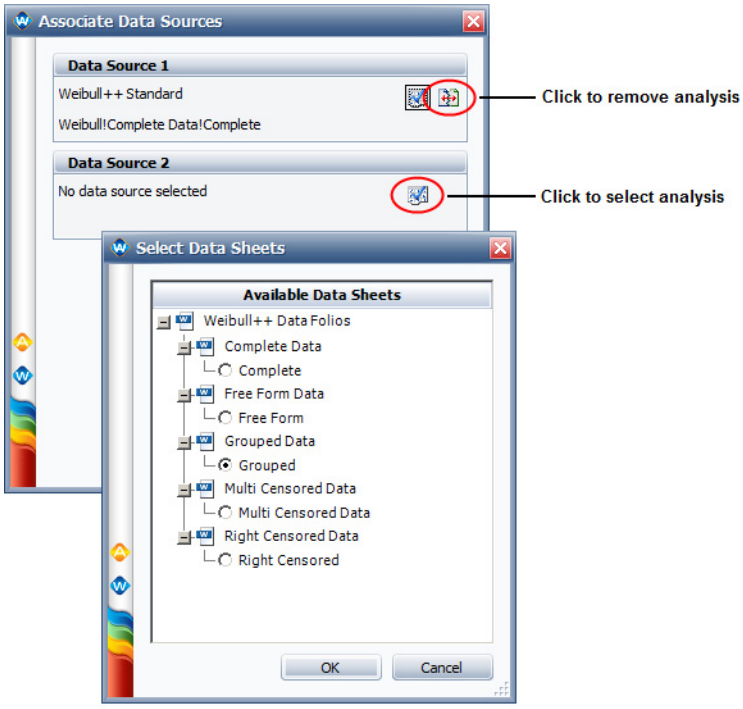
Associating Data Sources

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).`

27.3 Saved Workbook/Report Templates

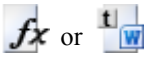
In both analysis workbooks and Word reports, saved templates make it easy to reuse the same report multiple times — in different analysis projects and with different data sets.

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.

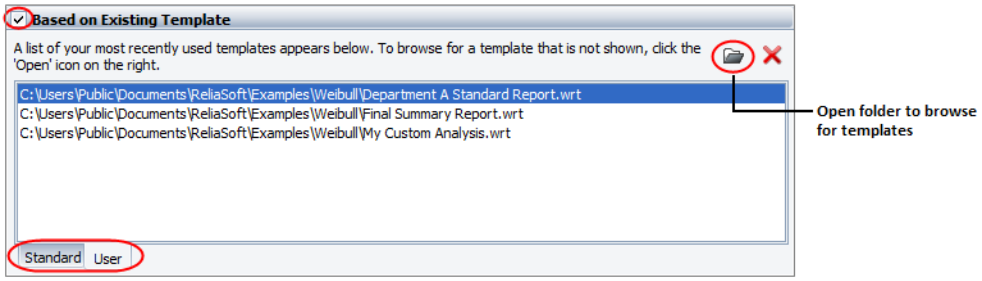
Using a Template to Create a Report

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.

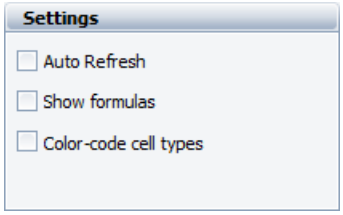
Note that the software automatically appends the appropriate file extension, and each template works only in the application that was used to create it.

Application	File Extension
Weibull++/ALTA	.wrt (workbooks) .rtt (Word reports)
RGA	.rgart (workbooks) .rtt (Word reports)
BlockSim	.brt (workbooks) .rtt (Word reports)

27.4 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.)

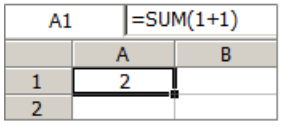
Settings



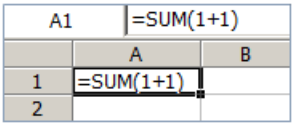
- **Auto recalculate** automatically recalculates formulas when relevant values change in any of the referenced recalculated data sources. This option is also available from the Calculation Options window (**Sheet > Format and View > More Settings > Calculation Options**).

  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

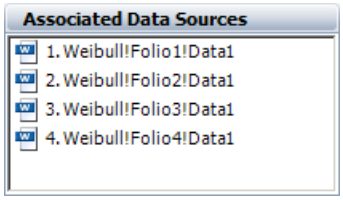


Formula displayed in A1

- **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.



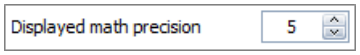
- To open a folio or diagram, click the name.
- To change one or more of the data sources, click the **Associate Data Sources** icon.



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 550*) 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 27.7.3 on page 548*), 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.

27.5 Function Wizard

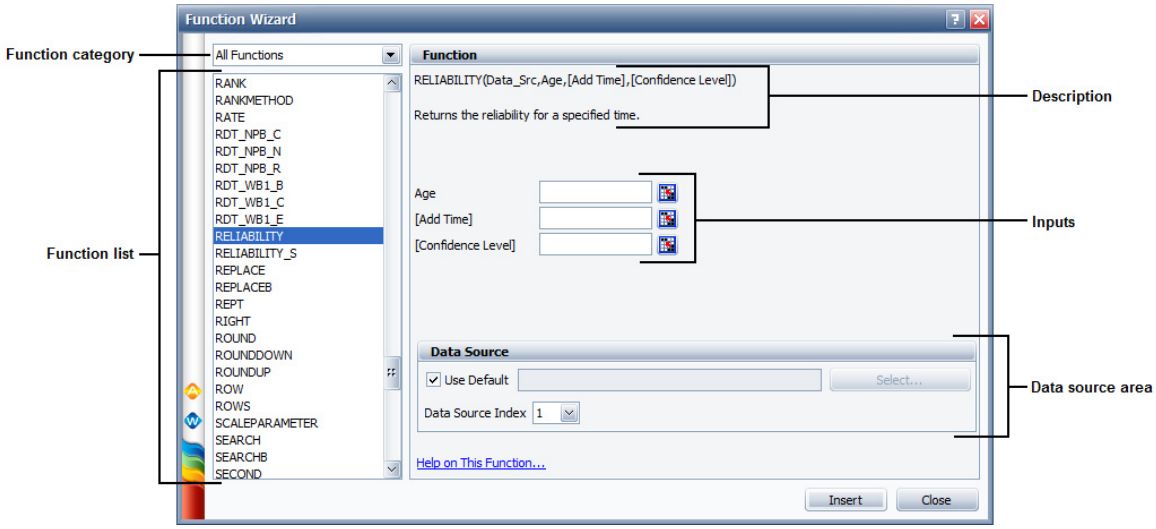
27.5.1 Using the Function Wizard in Any of the Reporting Tools

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/weibull_alta9/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.

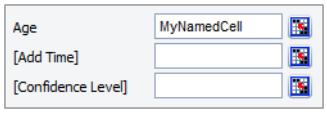
```
[RELIABILITY(Source Number)(Time)]
```

- 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.)

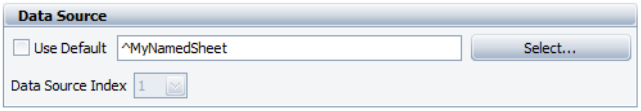
  The **Insert Workbook Reference** icon provides a quick way to insert a reference to the cell that is currently selected in the sheet. Note that you can move the cursor in the sheet while the Function Wizard is open. Pressing **CTRL** while clicking this icon inserts an absolute reference to the currently 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 27.5.2 on page 542).

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."

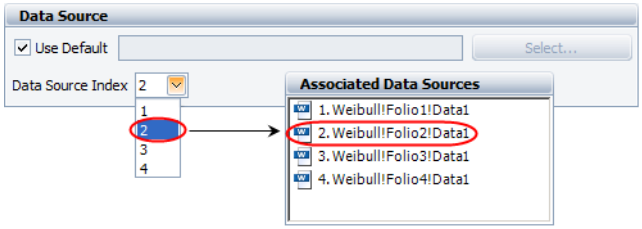
  To do this in the analysis workbook's function wizard, clear the Use Default check box and use the Select button to choose the data source, as shown next. (In the general spreadsheet's function wizard, you will always need to choose a specific data source.)



- 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.

27.5.2 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/weibull_alta9/spreadsheet_functions.htm).

Working with Date Functions

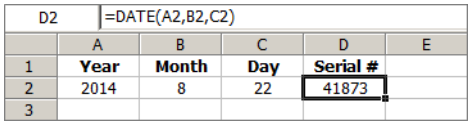
When using one of the spreadsheet date functions (`DAY`, `DAYS360`, `MONTH`, `WEEKDAY` and `YEAR`) to enter a date, you can use one of the following accepted text formats:

- 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.

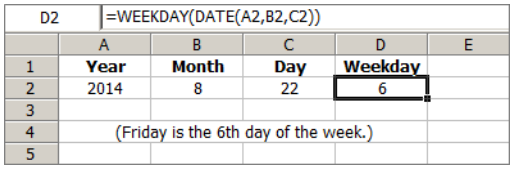
- You can obtain a date's serial number using either of the following two functions. This may be helpful in cases where you want to filter, sort or use the date(s) in calculations.
  - The `DATE` function uses the inputs of other cells to obtain the serial number. For example, if you have dates specified 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))`



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					

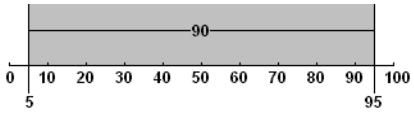
  - The `TIMEVALUE` function requires you to enter the time as text in one of the accepted text formats. For example, `=TIMEVALUE("4:48:10 PM")` returns 0.70011574.

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.

27.5.3 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.

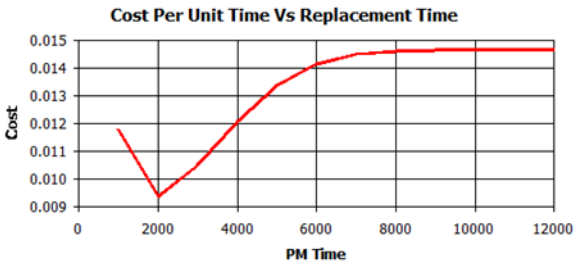
For more information about confidence bounds, refer to:
http://www.ReliaWiki.org/index.php/Confidence_Bounds.

27.6 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	Cost for Planned Replacement:	$10
Increment By:	1000	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.

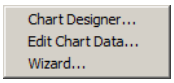


2. In the worksheet, click inside the worksheet and drag to select the area where you want to insert the chart.
3. In the Gallery window, select a chart type. Click **Next** and follow the wizard to add chart and axis titles.
4. Click **Finish** to insert the report.

See the analysis workbook example (Section 27.8 on page 557) 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.

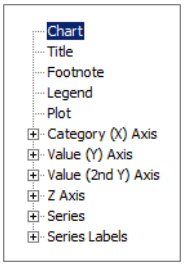


To Use the Chart Wizard:

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.

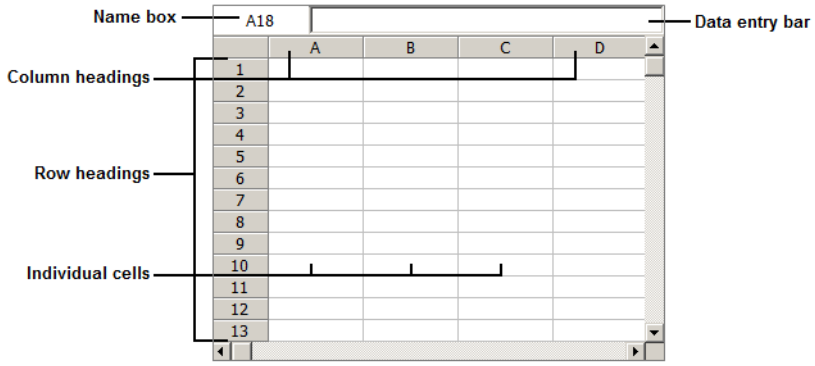


3. Click **OK** when done.

27.7 Tools for Working with Spreadsheets

27.7.1 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.

27.7.2 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.

27.7.3 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

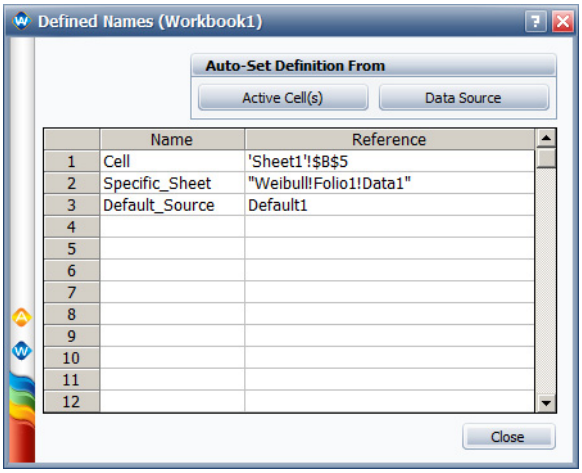
To define a name, do the following:

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++.

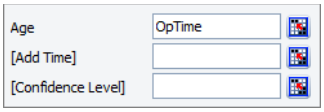
To reference a...	Do the following...
Cell	Identify the location by the sheet name and the absolute cell reference (e.g., Sheet1!$B$5). You can enter this manually or select the cell in the spreadsheet and then click **Active Cell(s)**.
Specific data source	Identify the location by the folio type, folio 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.



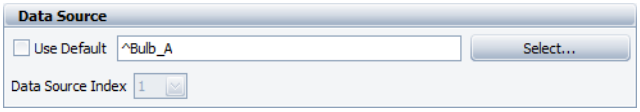
4. Click **Close** to save the changes and return to the spreadsheet.

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++.



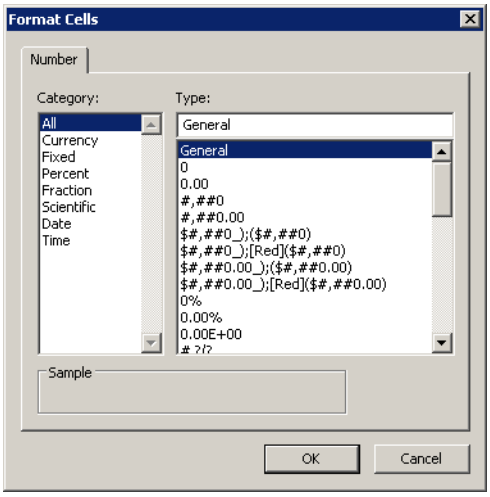
Note that quotation marks around the name in the function expression (e.g., `=RELIABILITY("Bulb_A,"$B$5)`) will result in a "Sheet not found" error.

27.7.4 Format Cells - Number Window

You can set the number format within the selected cells in the current spreadsheet by choosing **Sheet Options > Format and View > Custom Number**.



The Format Cells - Number window will appear.



After you have made your selection, click **OK**.

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**).

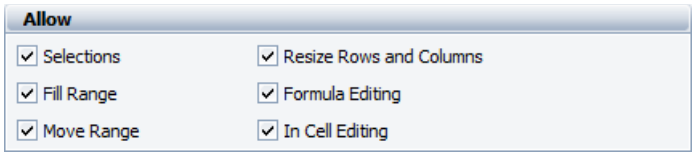
27.7.5 Sheet Options Window

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.

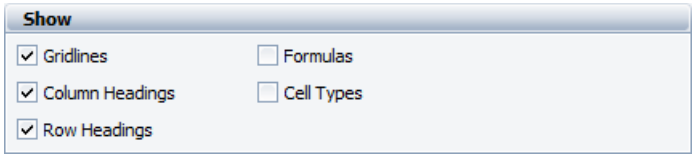


To restrict access:

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

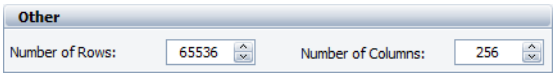
If you clear...	Results
Gridlines	Hides the gridlines on the spreadsheet.
Column and/or **Row Headings**	Hides the column and/or row headings

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.

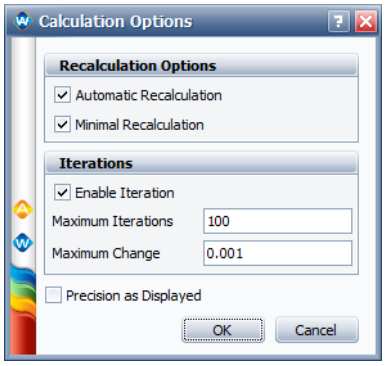


27.7.6 Calculation Options

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.



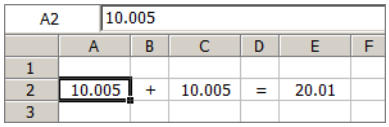
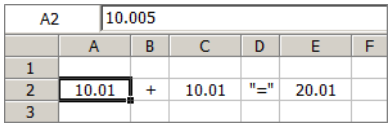
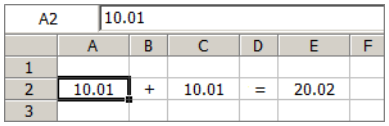
After you change any of these options, click **OK** to save the changes and then choose **Sheet > Format and View > More Settings > Recalculate Formulas** to recalculate the formulas within the spreadsheet based on the changes that have been made.

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.

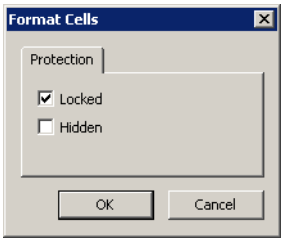
The behavior will differ based on the combination of options you select. For example, consider a simple scenario in which values entered directly into two cells are being added together:

If the cell uses the default number format (General)…	**If the cell uses a custom number format (e.g., 0.00)…**
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). 	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). 

27.7.7 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.

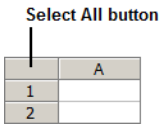


To lock/hide and protect cells, do the following:

1. If cell protection is already enabled, toggle it off temporarily by choosing **Sheet > Format and View > More Settings > Enable Protection**.



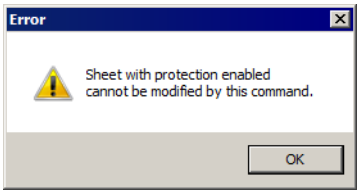
2. Select the cell(s) you want to configure. Note that if you want to select the whole sheet, click the **Select All** button.



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**.

27.7.8 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**.

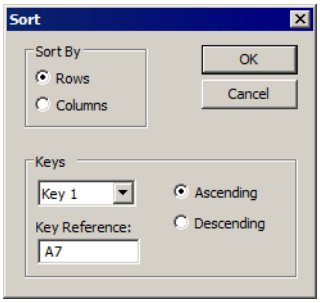


Sort Ascending or Descending

Choose **Sort Ascending** or **Sort Descending**. The data in the selected cells will automatically be sorted in the appropriate order.

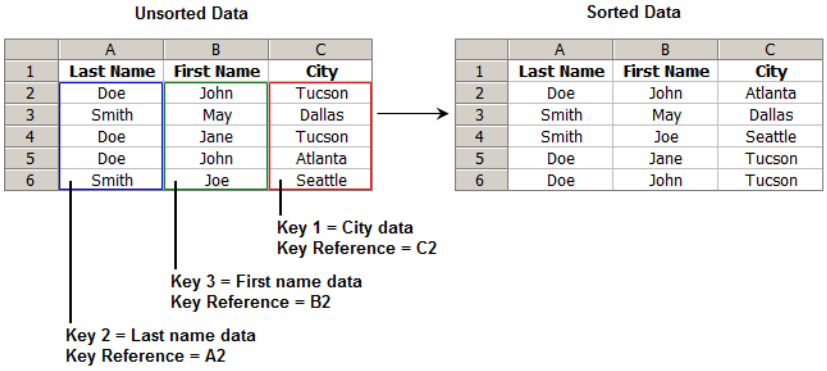
Sort with Multiple Criteria

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.



27.7.9 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.



27.8 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.

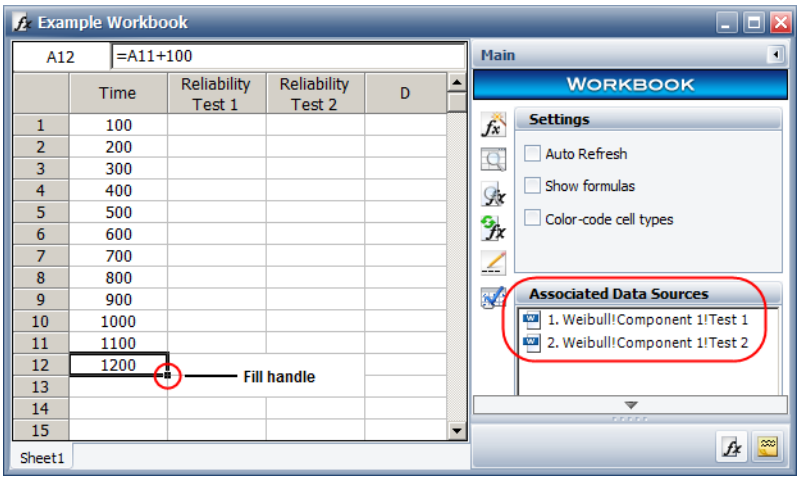


2. Click the Associate Data Source icon and assign the second data source, the "Test 2" data sheet for Component 1.

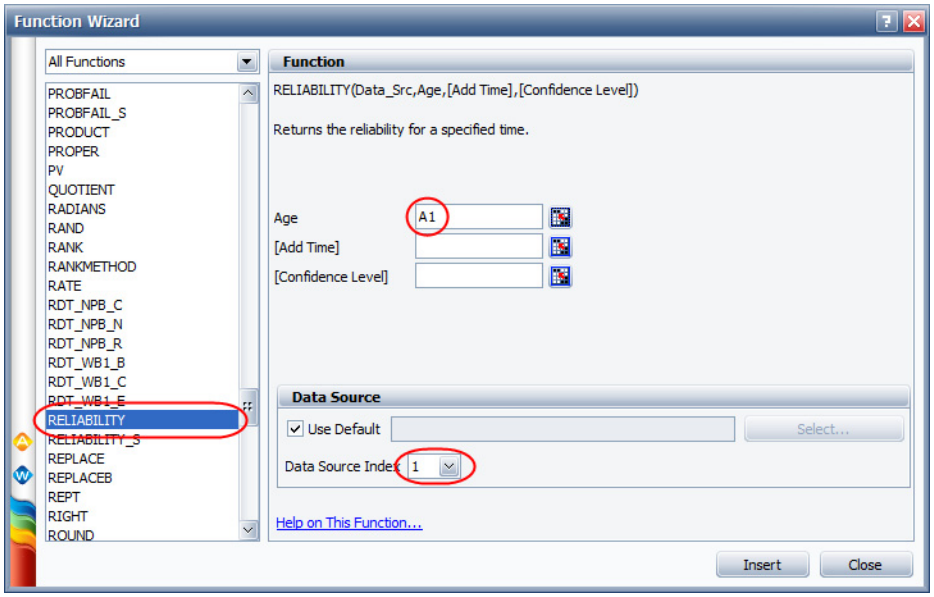


3. Update the column headings in the report: **Time** in column A, **Reliability Test 1** in column B and **Reliability Test 2** in column C.

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				

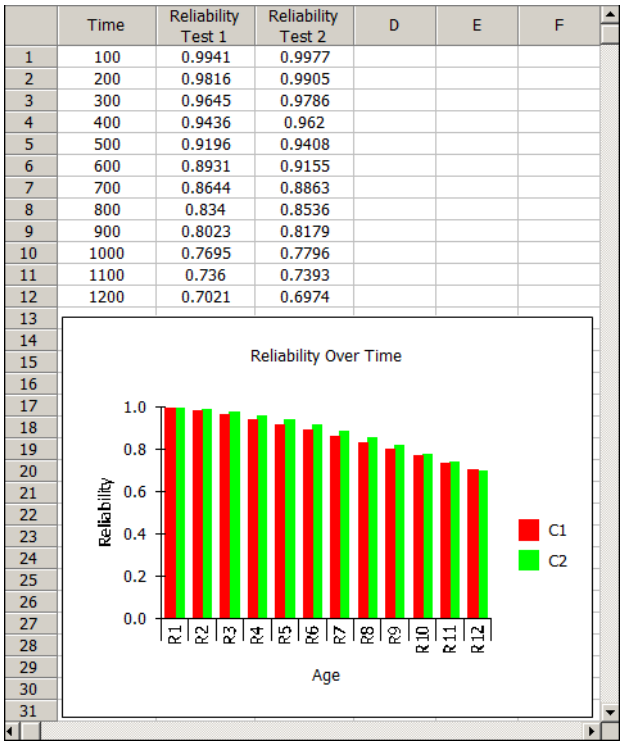
By looking at the data in the table you can see which design has the better reliability at a given time.

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:



Word Report Templates 28

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 27 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 28.1 (p. 562)
- Using the plot wizard to insert plots - Section 28.2 (p. 563)
- Tools for working with report templates - Section 28.3 (p. 564)
- Word report template Ribbon - Section 28.4 (p. 571)
- A Word report example - Section 28.5 (p. 577)

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.

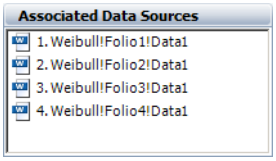
In Weibull++/ALTA and RGA, you also have the option to insert general spreadsheets into any data folio. These flexible spreadsheets provide functionality that is very similar to analysis workbooks.

28.1 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.



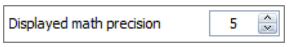
- To open a folio or diagram, click the name.
- To change one or more of the data sources, click the **Associate Data Sources** icon.



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.

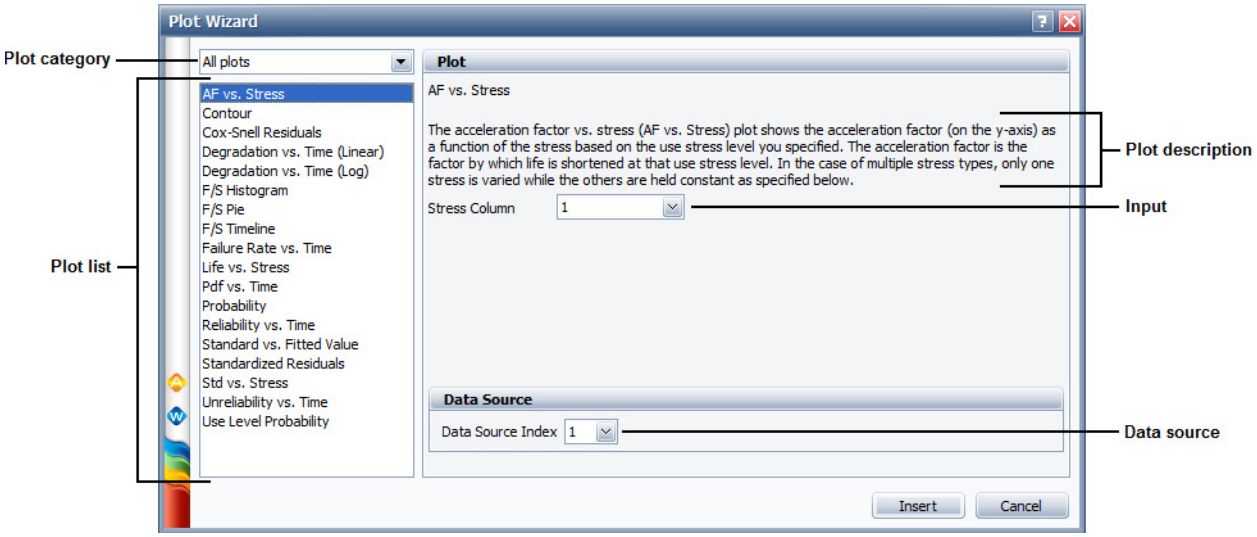
Create Report generates the report with calculated results and plots in Microsoft Word.

28.2 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:

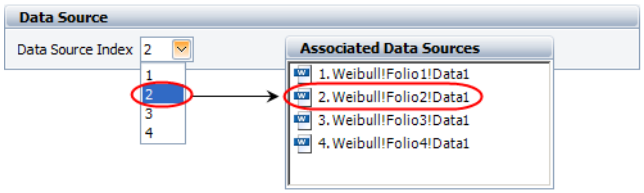
1. Select the plot type (and enter the inputs, if applicable)
2. Select the data source
3. Insert the plot holder

Select the Plot Type

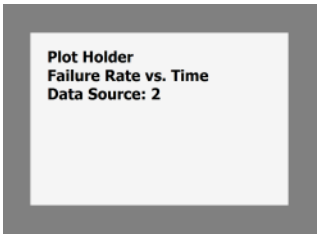
First, select one of the available plot types from the panel on the left side. The right side of the wizard displays some information about the plot that is currently selected. If applicable, this area also allows you to make relevant inputs (e.g., for ALTA plots that require you to select a specific 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

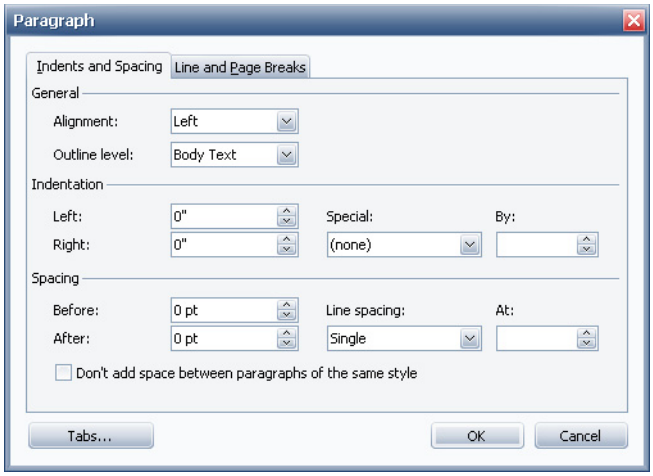
When the plot is fully defined, click **Insert** to place it into the report template at the current cursor location. You can move and/or resize the plot holder after it has been inserted. Note that the plot holders inserted in a Word report template will not return any results until the report is generated (**Home > Report > Create Report**).

28.3 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.

28.3.1 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:

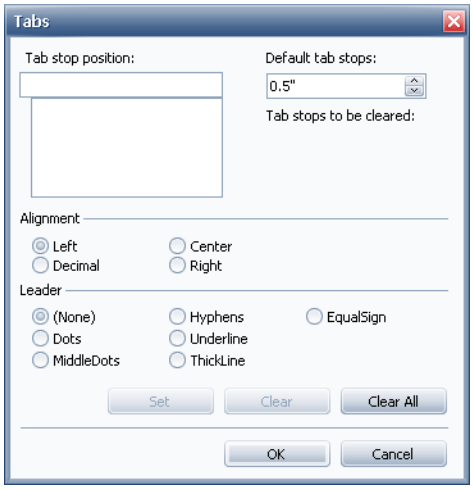
On the **Indents and Spacing** tab:

- 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:

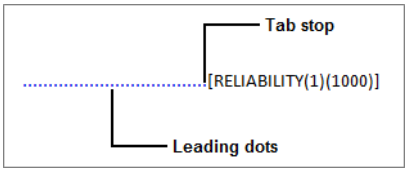
- Select **Keep lines together** to prevent the paragraph lines from being on different pages. The option can be applied to selected paragraph(s) or the entire document.
- Select **Page break before** to force a page break before the paragraph.

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.

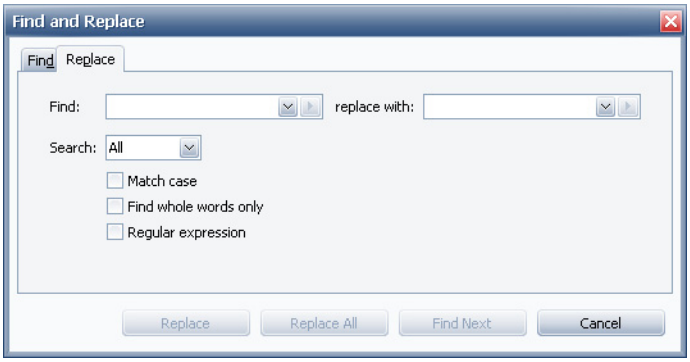
- Under **Alignment**, click the type of tab stop that you want.
- If you want to add dots with your tab stop, or to add another type of leader, click the option that you want under Leader.



- Click **Set** to create the tab stop. You can add additional tab stops, or click **OK** to add the tab stops to the paragraph.

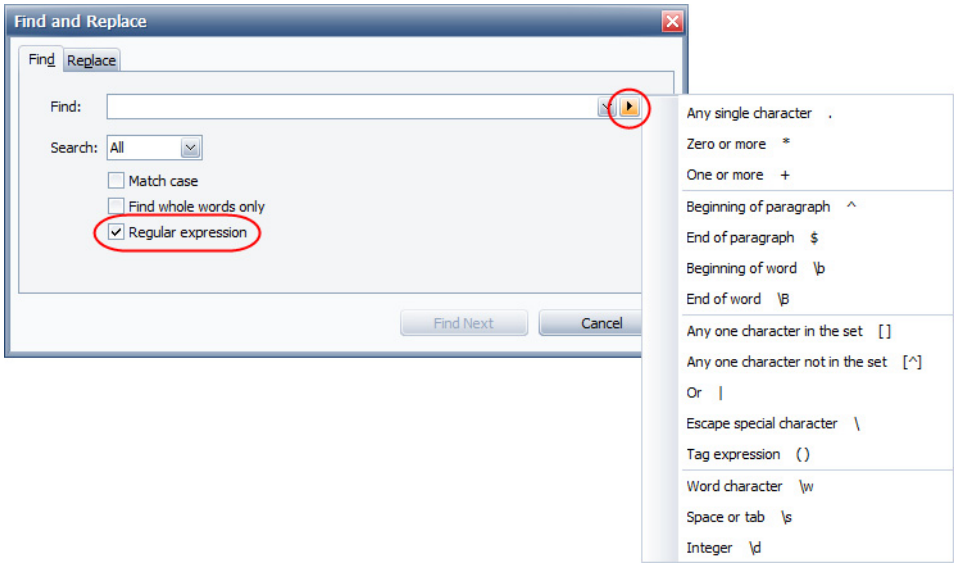
28.3.2 Find and Replace Text or Special Characters

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.

28.3.3 Inserting Pictures and Symbols

Insert a Picture

- Place the cursor into the desired position and choose **Insert > Items > Picture**.



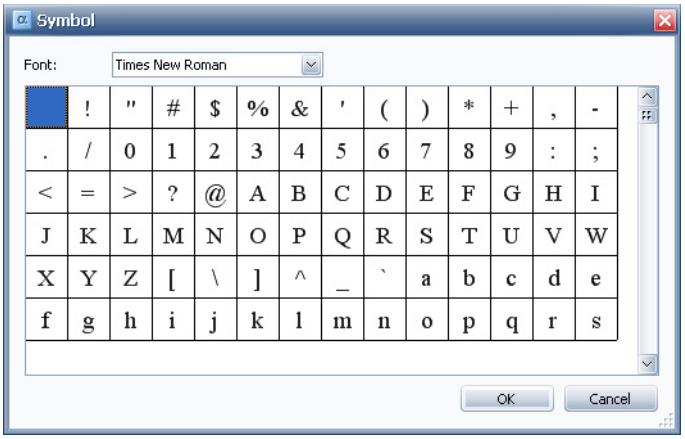
- Select a picture and click **Open**. Resize the picture, if desired.

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.



28.3.4 Inserting Bookmarks and Hyperlinks

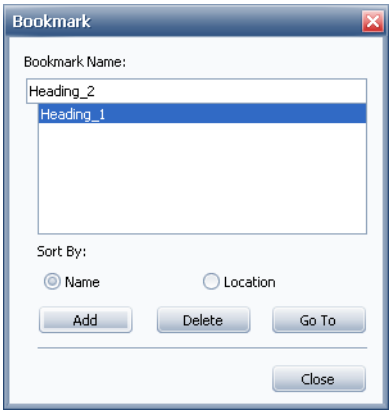
A bookmark identifies a location or a selection of text that you name and identify for future reference. For example, you might use a bookmark to identify text that you want to revise at a later time. Whereas, a hyperlink can be text or graphics that you click to go to a specific location in a document, 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").

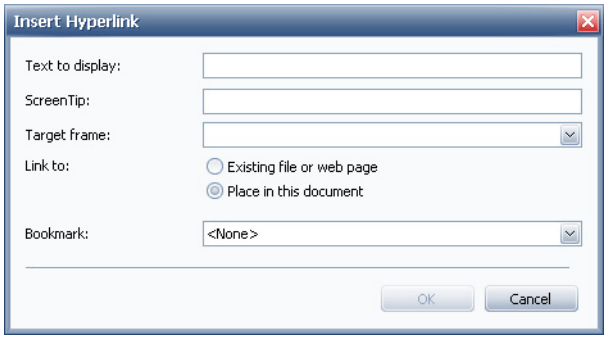


Insert a Hyperlink

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 holds the mouse 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.

28.3.5 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.

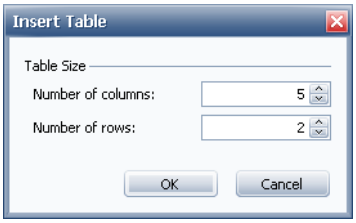
Note that these user-created tables are different from the table functions that return results in table form (e.g., functions with parameter bounds in Weibull++ or component failure in BlockSim). Table functions are not used in RGA.

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**.

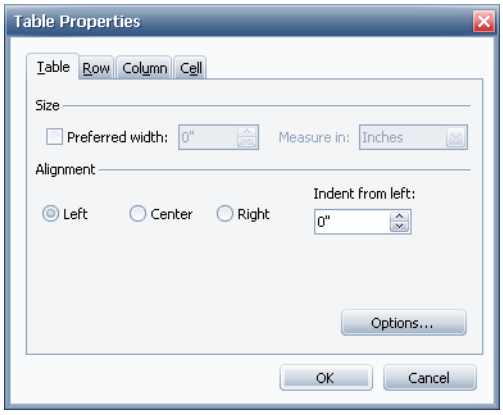


Customize the Table Properties

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:

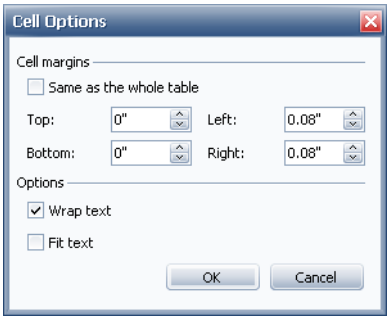
- To fix the height of a row, select the **Specify height** check box, then enter a value. The row height options are: **At Least** (the minimum height the cell can be) or **Exactly** (will not allow the row expand to accommodate the content).
- To allow the row to split across a page break, select the **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.



28.4 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.

28.4.1 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**.

28.4.2 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**.

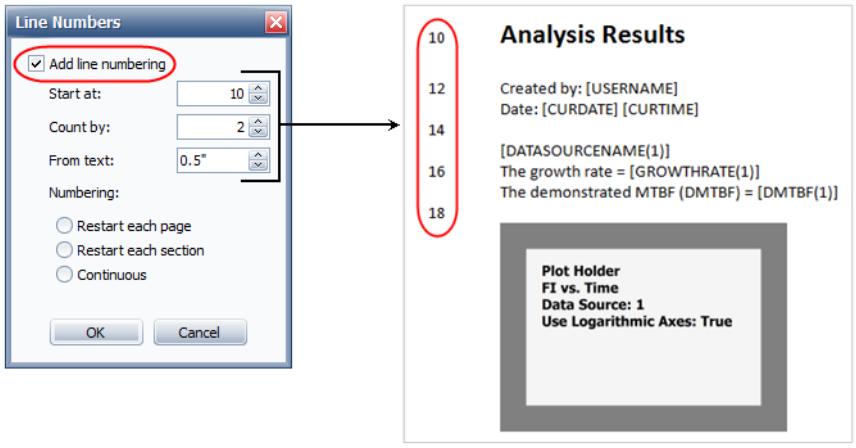
  Remove line numbers:

  - 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.

28.4.3 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**.

28.4.4 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.

28.4.5 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 mouse 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.

28.5 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.

Reliability Results for Bulb Testing

Created by:
Date:

[DATASOURCENAME(1)]	[DATASOURCENAME(2)]
The reliability at 400 hours =	The reliability at 400 hours =
The B10 life =	The B10 life =

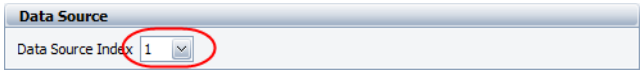
5. Insert the user name and date functions.
   - Place the cursor after the "Created by:" label and then click the **Function Wizard** icon on the template's control panel.



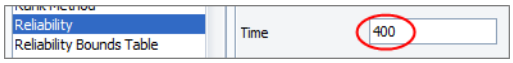
   - Select `User Name` from the function list, then click **Insert**.
   - Repeat the process to insert the Current Date function after the "Date:" label. The inserted functions are shown next.

Created by: [USERNAME]
Date: [CURDATE]

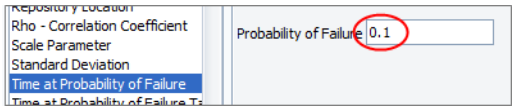
6. Use the Function Wizard to insert the data source name, reliability and B10 functions for the in-house data. For each function, select **1** as the data source index.



   - **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.



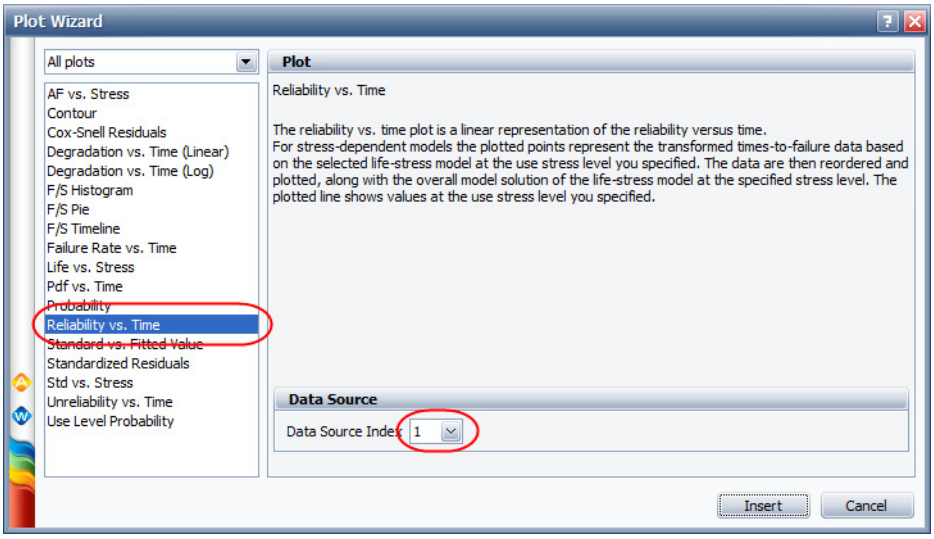
   The inserted functions are shown next.

[DATASOURCENAME(1)]
The reliability at 400 hours = [RELIABILITY(1)(400)]
The B10 life = [TIMEATPF(1)(0.1)] hours

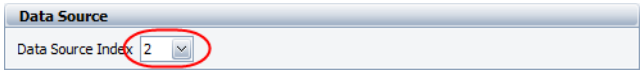
7. Insert the reliability plot for the in-house data (data source 1).
   - Place the cursor where you want to insert the plot and then click the Plot Wizard icon on the template's control panel.



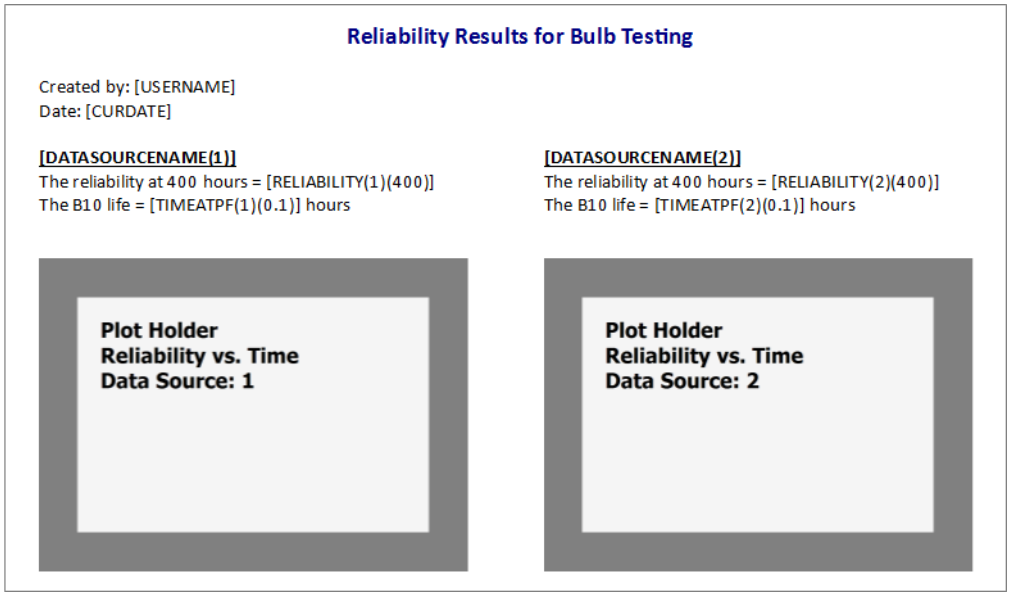
- Select **Reliability vs. Time** from the plot list, select **1** as the data source index and click **Insert**.



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:

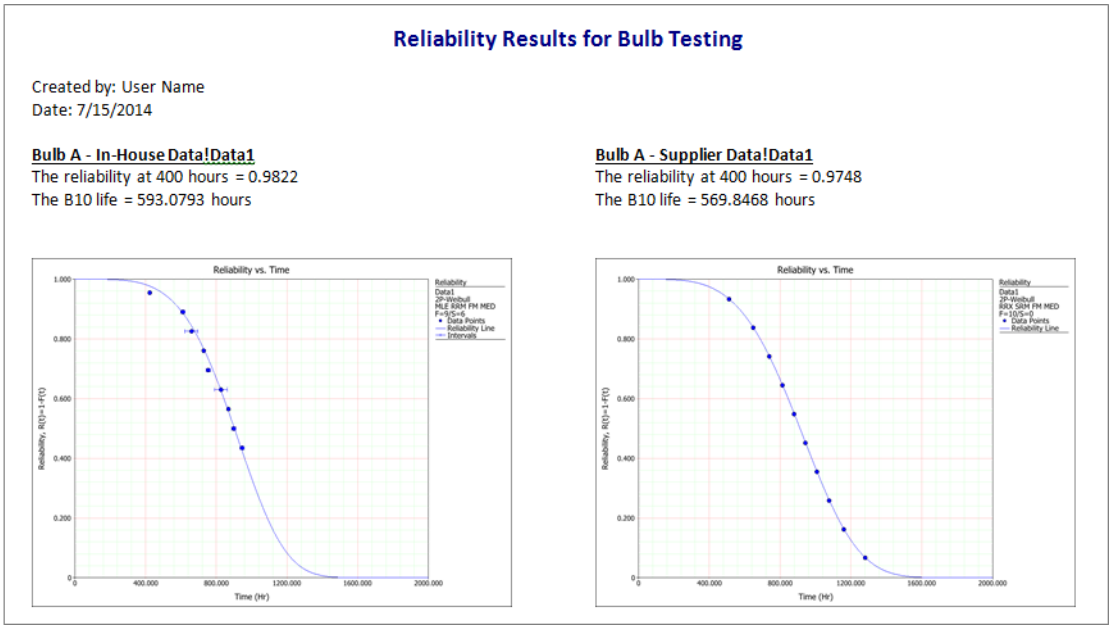


Generating the Report

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

Printing 29

This chapter describes the various printing options that are available in Weibull++ and ALTA, including:

- Using the Page Setup window to make selections on how the pages will be printed - Section 29.1 (p. 581)
- Previewing the pages that will be printed - Section 29.2 (p. 585)
- Using the Print Window to specify the printer to send the pages to - Section 29.3 (p. 587)

29.1 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**.

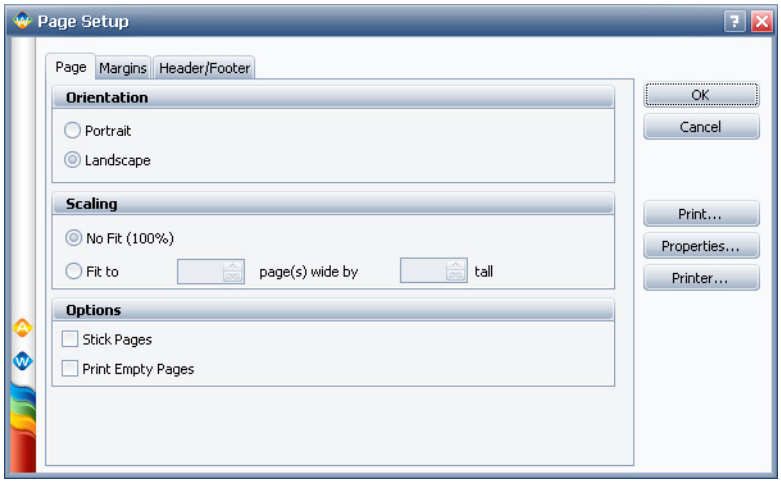


The Page Setup window is made up of three tabs. The appearance of this window will vary depending on the type of sheet that currently has focus (e.g., diagram sheet, spreadsheet, plot/overlay plot, etc.). Each of the tabs in the Page Setup window and its components are presented next.

The **Properties** button is always available. It opens the Windows Page Setup window, which allows you to set printing properties for the selected printer.

29.1.1 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 data sheets and general spreadsheets 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 diagrams the following options apply:

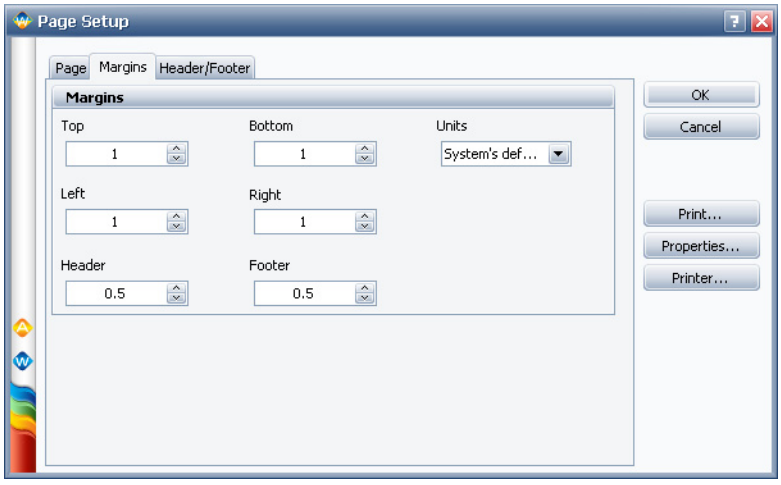
- **Scaling**
  - **No Fit (100%)** prints the item at its actual size.
  - **Fit to __ page(s) wide by __ tall** allows you to specify the number of pages you want the sheet printed on.
- **Options** allows you to specify the details of the printing process.
  - **Stick Pages** prints the pages without margins.
  - **Print Empty Pages** prints any blank pages in the item. If not selected, blank pages will not be printed.

For 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.

29.1.2 Margins Tab

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. You can also center the printed page horizontally, vertically or both, except when working with plot sheets and diagrams.

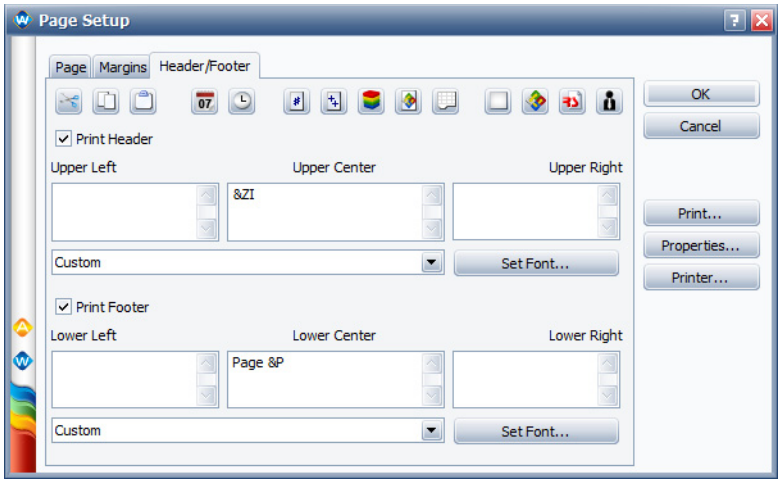


The **Units** drop-down field allows you to specify the units in which the margins will be printed:

- **System's defaults** uses the units used by your computer as the measurement unit by which the margins will be printed. *See page 13 in Chapter 1*.
- **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.

29.1.3 Header/Footer Tab

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.
- The **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.
- **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.



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 inserts the sheet name.



Item Name inserts the name of the active sheet, or the folio with which the active plot sheet is associated.



Application Name inserts the program name (i.e., Weibull++ or ALTA).



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.

29.1.4 Header/Footer Format Codes

The format codes that can be used for printing headers and footers are presented next.

Format Code	Description
&L	Left aligns the item.
&C	Center aligns the item.
&R	Right aligns the item.
&A	Prints the sheet name.
&D	Prints the current date.
&T	Prints the current time.
&P	Prints the page number.
&P+number (e.g., &P+2)	Prints the page number plus the specified number.
&P-number (e.g., &P-2)	Prints the page number minus the specified number.
&&	Prints an ampersand.
&N	Prints the total number of pages.
&ZA	Prints the name of the application (i.e., Weibull++ or ALTA).
&ZC	Prints the company name.
&ZU	Prints the user name.
&ZP	Prints the project name.
&ZF	Prints the project file name and location.
&ZI	Prints the name of the item (Folio, Plot, Diagram, etc.).

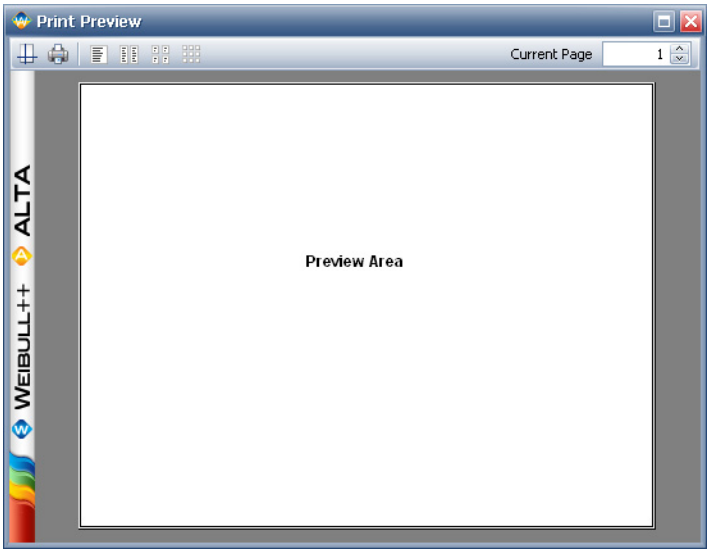
By default, text will be centered in the headers/footers unless otherwise specified. Please note that the alignment codes (e.g., &L, &C, &R) restart each format code section. For example, in the format code &L&P &R&ZI, the page number will be left aligned and the sheet name will be right aligned.

29.2 Print Preview

Print preview is available for data sheets in all folio types and in the non-linear equation fit solver, the Results window, SimuMatic, general spreadsheets, side-by-side plots, 3D plots and reports. It can be accessed by choosing **Home > Print > Print Preview** or by clicking **Preview** in the Print window.



The Print Preview window appears, as shown next.



Depending on your screen resolution and screen size, the print preview of the spreadsheet 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.

Page Setup allows you to specify printing options. *See Section 29.1 on page 581*.

Print sends the current document to the printer.

Single Page Layout displays one page in the Preview area.

Double Page Layout displays two pages in the Preview area.

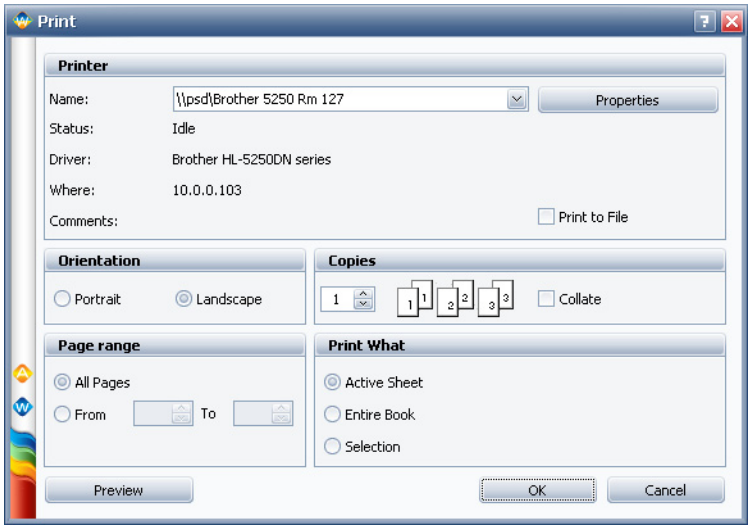
Four Page Layout displays four pages in the Preview area.

Nine Page Layout displays nine pages in the Preview area.

Depending on your screen resolution and screen size, the print preview of data sheets, general spreadsheets or reports 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, as well as any objects. Please note that objects such as charts on a general spreadsheet will not be shown in full detail on the preview, but they will print correctly. 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).

29.3 Print Window

The Print window allows you to specify the printer to send the data sheets for all types of folios, general spreadsheets, reports, other tools (i.e., non-linear equation fit solver and SimuMatic), plot sheets and diagrams, as well as printing options for the sheet.



- **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 data sheet or a spreadsheet 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 spreadsheet.
  - **Selection** prints only the selected cell range in the currently active sheet in the spreadsheet. 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 data sheet or spreadsheet's window, not the Print window. *See Section 29.2 on page 585*.

Setup Windows 30

Synthesis applications provide two setup windows that allow you to customize workspace preferences to fit the way you work:

- The **Synthesis Setup** allows you to 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** allows you to 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 36 in Chapter 2*) and other relevant topics.

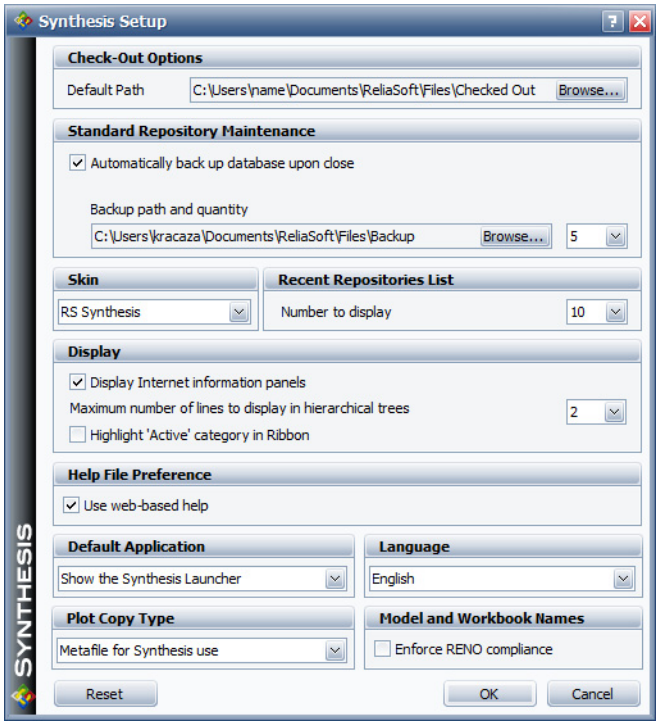
30.1 Synthesis Setup

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 73 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. Some Synthesis applications also offer diagram skins, which control the default appearance of new diagrams. *See page 468 in Chapter 18*.

- 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 62 in Chapter 2*.

30.2 Application Setup

The Application Setup window contains settings that apply specifically to Weibull++ and ALTA. 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.

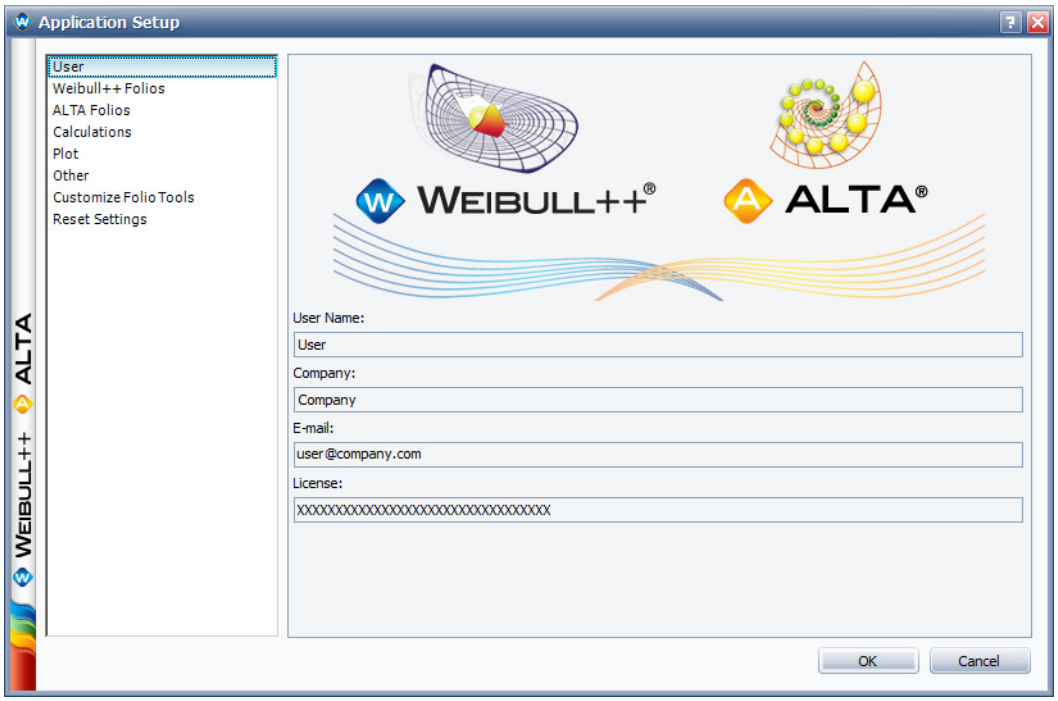
To open the Application Setup window, choose **File > Application Setup**.



What's Changed? In previous versions of the software, the Application Setup was known as the User Setup.

30.2.1 Application Setup User Page

In Synthesis 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:

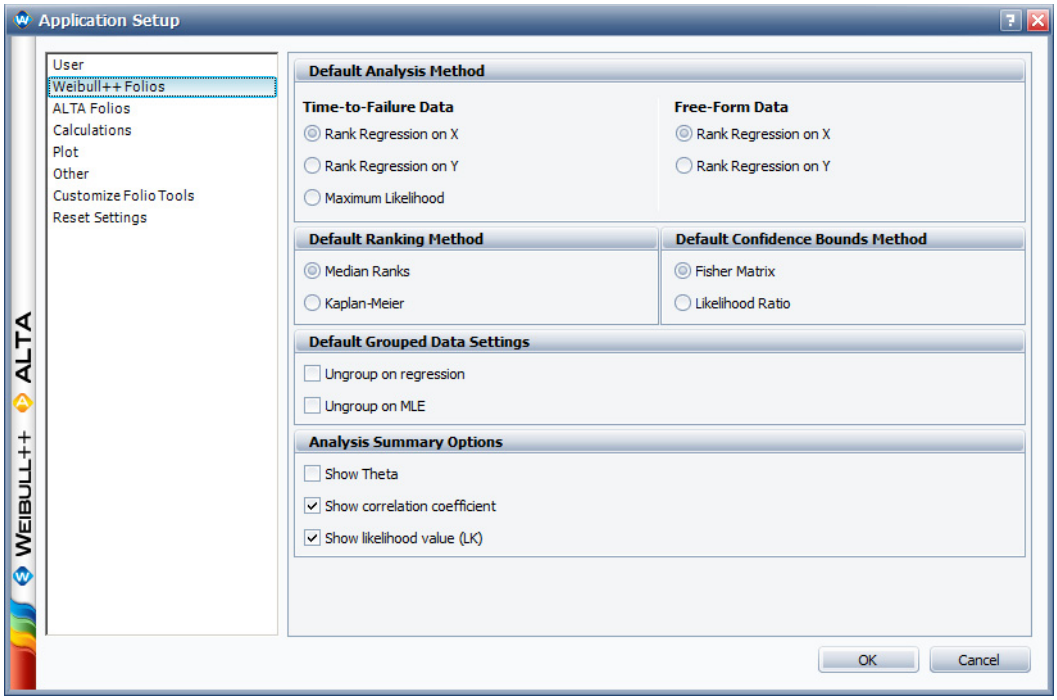
- In plots and in some report output
- For notification e-mails, if enabled for the database

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.

30.2.2 Application Setup Weibull++ Folios Page

WEIBULL++ Only

The Weibull++ Folios page of Weibull++/ALTA's Application Setup allows you to set the options that apply only to Weibull++ folios. Options that apply to both Weibull++ and ALTA appear on other pages.



The page contains the following settings:

- Use the **Default Analysis Method** area to set the default analysis method for data sheets. Note that the options are different for time-to-failure data sheets and free-form data sheets as free-form data sheets do not support the MLE method.
- Use the **Default Ranking Method** area to select the default ranking method for rank regression analysis.
- Use the **Default Confidence Bounds Method** area to select the default method used to calculate the confidence bounds.
- Use the settings in the **Default Grouped Data Settings** area to set how Weibull++ assigns an individual median rank value to each data point (observation) within each group. When you select an option and plot the data, the points in each group will appear vertically aligned. If you do not select an option then the observations in each group will be represented by a single point at the highest median rank value within each group.
  - Select **Ungroup on regression** to fit the regression line to all data points in each group. If not selected, only the points with the highest median rank value within each group will be considered when fitting a line to the points.
  - Select **Ungroup on MLE** to plot all of the data points in each group on the MLE probability plot. If not selected, only the points with the highest median rank value within each group will be plotted. The line is independent of whether data points are treated as grouped or ungrouped.

- **Analysis Summary Options**
  - To display the theta parameter in the control panel analysis summary area when using the Weibull distribution, select the **Show Theta** option. Theta is defined as the sum of the scale and location parameters (Eta and Gamma):

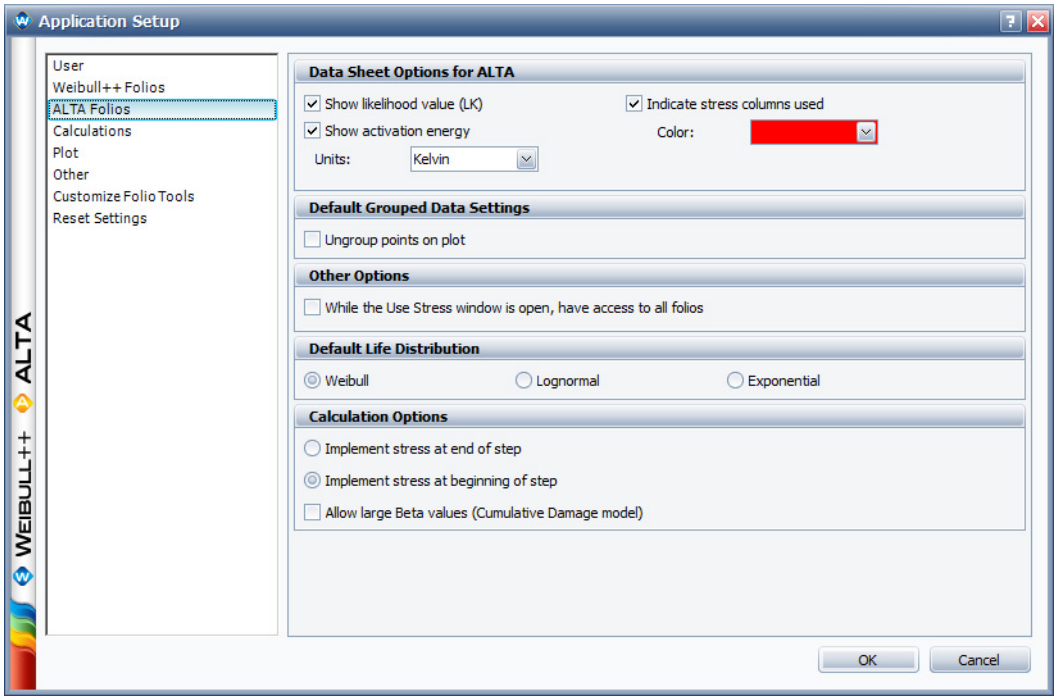
$$\theta = \eta + \gamma$$

  - To display the correlation coefficient in the control panel analysis summary area, select the **Show correlation coefficient** option. Note that the correlation coefficient is only applicable when using rank regression as the analysis method.
  - To display the likelihood function value in the control panel analysis summary area, select the **Show likelihood value (LK)** option.

30.2.3 Application Setup ALTA Folios Page

ALTA Only

The ALTA Folios page of Weibull/ALTA's Application Setup allows you to set the options that apply only to ALTA folios. Options that apply to both Weibull++ and ALTA appear on other pages.



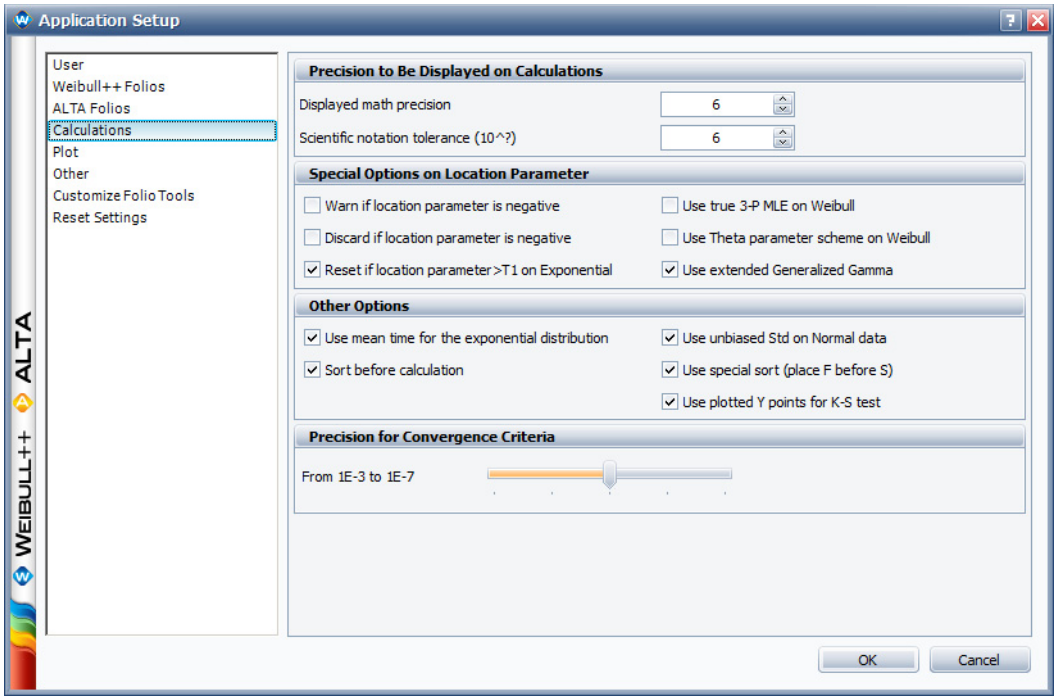
The page contains the following settings:

- **Data Sheet Options for ALTA**
  - To display the likelihood function value in the control panel **Analysis Summary** area, select the **Show likelihood value (LK)** option.
  - To display the activation energy (Ea) in the control panel **Analysis Summary** area, select the **Show activation energy** option. Note that this option does not apply to the inverse power law, proportional hazards or general log-linear life-stress models.
    - In the **Units** drop-down list choose either Kelvin or Rankine for the time units in which to display the activation energy.
  - To display the selected stress columns in the data sheet with a colored border, select the **Indicate stress columns used** option, then click the **Color** box to open the palette and select a border color.
- **Default Grouped Data Settings**
  - If you select **Ungroup points on plot**, ALTA assigns an individual median rank value to each data point (observation) within a group. When plotted, the points in each group will appear vertically aligned. If not selected, the observations in each group will be represented by a single point at the highest median rank value within the group. The line is independent of whether data points are treated as grouped or ungrouped.
- **Other Options**
  - To open other ALTA standard folios while the Use Stress Level window remains open, select the **While the Use Stress window is open, have access to all folios** option. The Use Stress Level window will be linked to the currently active data sheet. If not selected, you must close the Use Stress Level window in order to return to the folios.
- In the **Default Life Distribution** area select whether the Weibull, Lognormal or Exponential distribution will be selected by default for ALTA standard folios. If lognormal is selected as the default and you select a model for which the lognormal distribution is not available (i.e., Proportional Hazards or Cumulative Damage in ALTA 9 PRO), the life distribution will automatically reset to Weibull.
- For time-dependent stress profiles, in the **Calculations Options** area select the option that specifies when ALTA will apply the stress to a time-to-failure data point that occurs at the both the End time of one segment and the Start time of another segment:
  - Select **Implement stress at end of step** to apply the stress from the later segment.
  - Select **Implement stress at beginning of step** to apply the stress from the earlier segment.

  For example, suppose that the selected stress profile applies a stress of 300K to failure times between 0 and 100 hours and a stress of 350K to failure times between 100 and 200 hours. If the observed failure time is 100 hours, you must determine whether to apply the 300K or 350K stress to that data point. These settings are available only in ALTA PRO and apply only when Cumulative Damage is selected as the life-stress model.
- Select **Allow large Beta values (Cumulative Damage model)** to display beta values up to 250 when using the cumulative damage model. If not selected, the default maximum beta value is 100. This option is available only in ALTA PRO and applies only when Cumulative Damage is selected as the life-stress model.

30.2.4 Application Setup Calculations Page

The Calculations page of Weibull/ALTA's Application Setup allows you to set the numerical precision used in the calculated results and plots, and whether to sort a data sheet before or after calculations.



The page contains the following settings:

- **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.
- **Special Options on Location Parameter** WEIBULL++ **Only**

  Use this area to set the options regarding the location parameter (Gamma).
  - If you want to be notified if the location parameter contains a negative value, select **Warn if location parameter is negative**.
  - If you select **Discard if location parameter is negative** the software behaves as follows:
    - If you are using RRX or RRY to calculate the parameters:
      - For the 2-parameter exponential distribution, the negative location parameter is discarded and the data set is recalculated using the 1-parameter exponential distribution. The location parameter is displayed as zero.
      - For the 3-parameter Weibull distribution, the negative location parameter is discarded and the data set is recalculated using the 2-parameter Weibull distribution. The location parameter is displayed as zero.

- If you are using MLE to calculate the parameters:
  - For the 2-parameter exponential distribution, the negative location parameter is discarded and the data set is recalculated using the 2-parameter exponential distribution with the location parameter constrained between zero and the first failure time.
  - If you have not selected the **Use true 3-P MLE on Weibull** option on this page of the Application Setup, the negative location parameter is discarded and the data set is recalculated using the 2-parameter Weibull distribution. The location parameter is displayed as zero.
  - If you have selected the **Use true 3-P MLE on Weibull** option, the negative location parameter is discarded and the data set is recalculated using the 3-parameter Weibull distribution with the location parameter constrained between zero and the first time-to-failure.

- Select **Reset if location parameter > T1 on Exponential** to reset the value of the location parameter equal to the first time-to-failure if the location parameter is greater than the first failure time. The parameters will then be recalculated based on the new value of the location parameter. This option pertains to the exponential distribution for both rank regression on X (RRX) and rank regression on Y (RRY).
- Select **Use true 3-P MLE on Weibull** to use a true 3-parameter MLE solution. The 3-parameter solution is inherently problematic for some Beta values and can fail to converge for some data sets. When not selected, Weibull++ uses non-linear regression to compute Gamma, and then computes the MLE of Beta and Eta using a transformed $T' = (T - \gamma)$.
- Select **Use Theta parameter scheme on Weibull** to replace the Weibull scale parameter Eta in all results with Theta, where Theta = Eta + Gamma.
- Select **Use extended Generalized Gamma** to allow the Lambda parameter of the generalized gamma distribution to take negative values.

- **Other Options**
  - 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.
  - Select **Sort before calculation** to sort the times in ascending order before calculating the data sheet. Note that this will not affect the results of the calculation.
  - **Use unbiased Std on Normal data** WEIBULL++ Only

    Use this check box to correct the MLE standard deviation (sigma) value for biasedness. Note that this setting applies only to the normal distribution with complete data.
  - **Use special sort (place F before S)** WEIBULL++ Only

    The software always sorts entries based on failure times. In special cases where two or more failure times are identical to suspension times, the sort algorithm does not distinguish between them and sorts them based on the way they were entered. For example, if given a suspension time of 100 hours and a failure time of 100 hours, the results will vary depending on the order in which the data were entered and whether you are using rank regression. When the **Use special sort (place F before S)** option is selected and two identical times are encountered, the software will always put failures before suspensions. This involves a double sort routine and might be slightly slower than the standard sort.
  - **Use plotted Y points for K-S test** WEIBULL++ Only

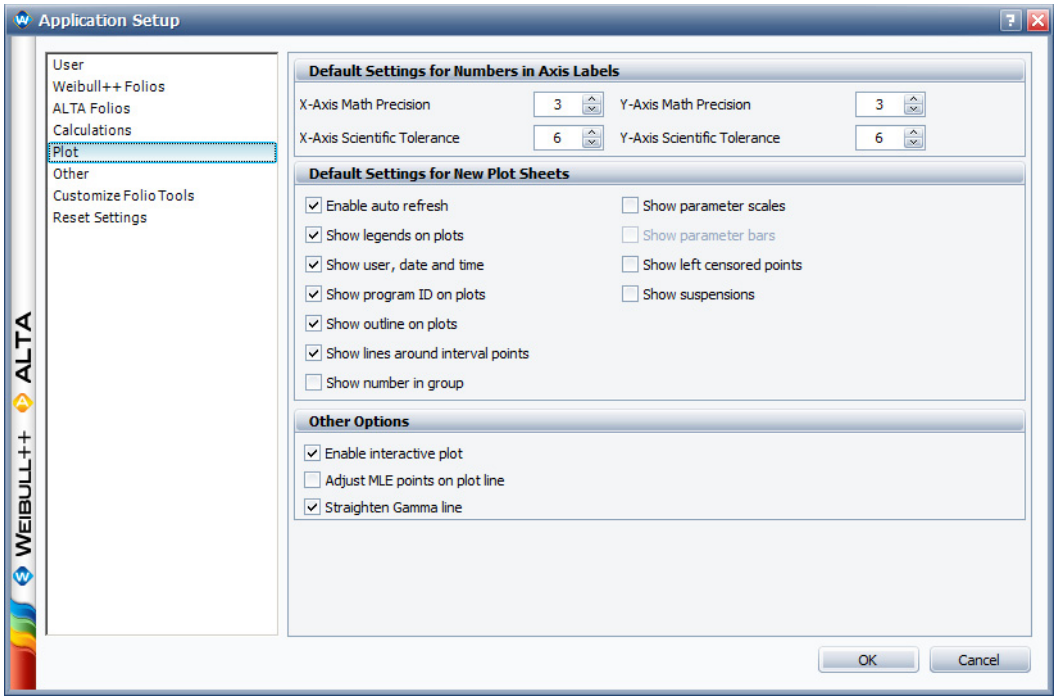
    Select this check box to use the plotted points (using median ranks or Kaplan-Meier) to calculate the difference between observed and model estimated probability for the Kolmogorov-Smirnov test, which is used by the Goodness of Fit Results window and the Distribution Wizard. If not selected, the non-parametric

(empirical) values of the probability of failure are used as the observed value for the Kolmogorov-Smirnov test.

- 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).

30.2.5 Application Setup Plot Page

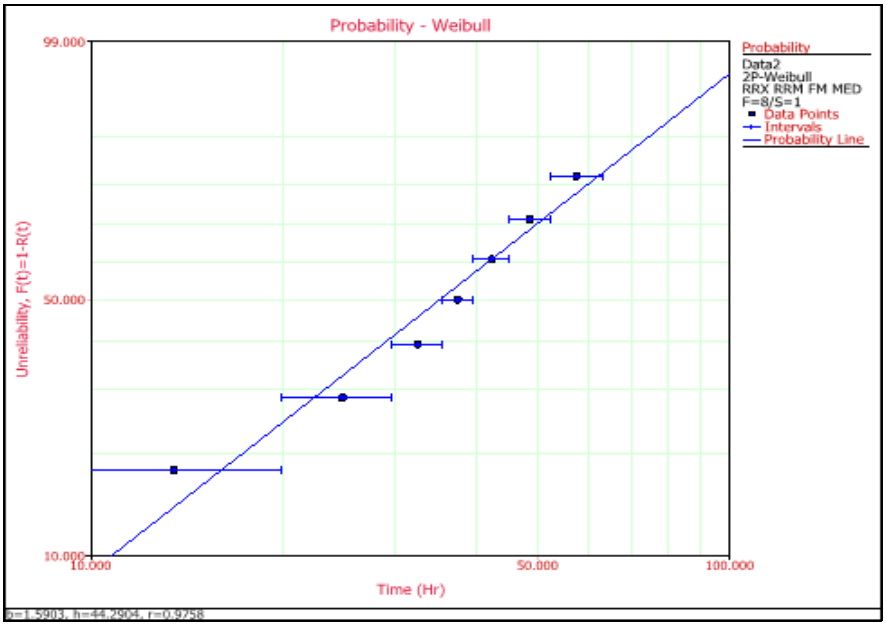
The Plot page of Weibull/ALTA's Application Setup allows you to set the default display preferences for each new plot sheet that you create from this computer.



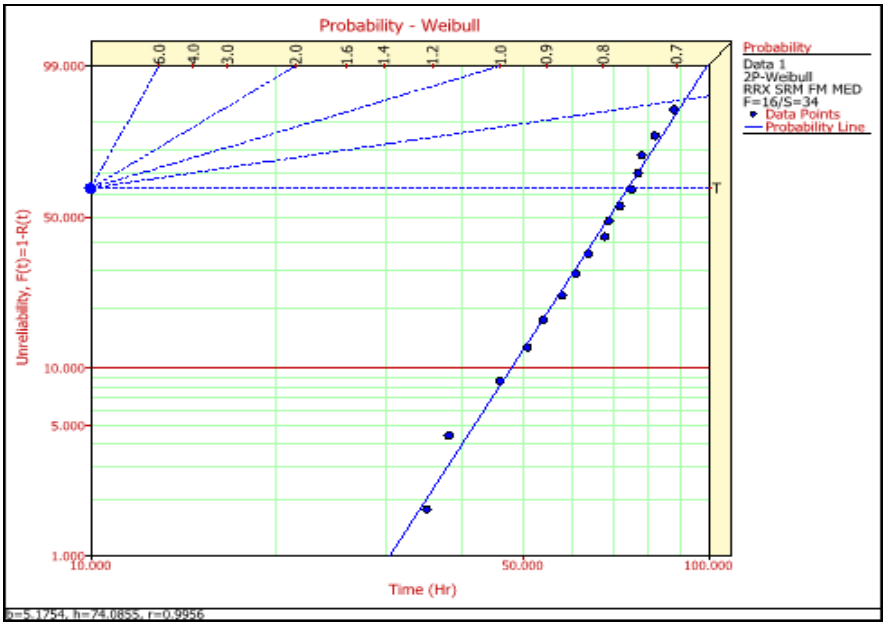
The page contains the following settings:

- 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 your user account in the current database (**My Portal > Users > My Profile**) and is displayed at the bottom right corner of the plot.
  - Select **Show program ID on plots** to display the software name and URL in the Header Title section of the plot (e.g., Weibull - www.ReliaSoft.com).

- Select **Show outline on plots** to display the canvas border on the plot.
- Select **Show lines around interval points** to show lines around the points for interval censored data. The lines represent the starting and ending time for each interval. For example:



- Select **Show number in group** to show the number of data elements represented by each point on the plot when plotting grouped data.
- Select **Show parameter scales** to show the parameter scales on a probability plot. The probability scales allow you to read the parameters directly from the plot.
- Select **Show parameter bars** to place a box around the parameter scales on a probability plot. (To see the parameter scales and the box, open a probability plot, choose **Plot > Plot Actions > Show/Hide Plot Items** and then select both **Scales** and **Bars**.) For example:

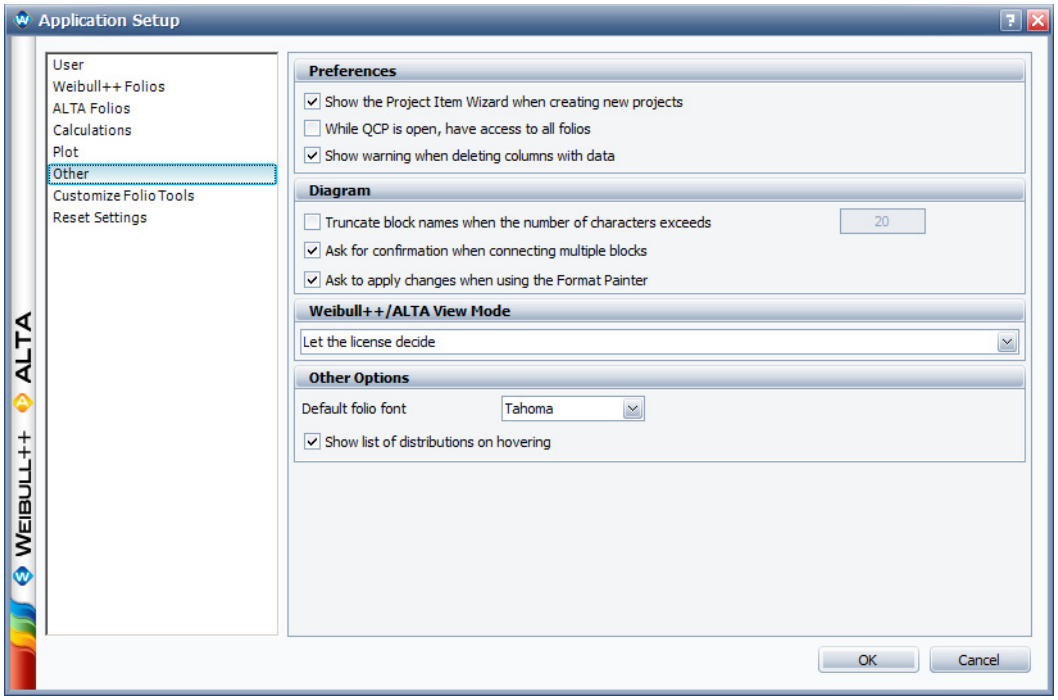


- Select **Show left censored points** to display the left censored points as triangles on the bottom of the plot when plotting interval or left censored data.
- Select **Show suspensions** to display the suspension (right-censored) data points as triangles on the bottom of the plot when plotting suspended data.

- **Other Options**
  - Select **Enable interactive plot** to highlight each point, line, slice or bar on the plot when you move the cursor over it. Point coordinates/values will be displayed on highlighted objects.
  - **Adjust MLE points on plot line**. For MLE plots, it adjusts the y-axis positions of the plotted points to fit the MLE solution line (instead of plotting the points from rank regression).
  - Select **Straighten Gamma line** to straighten the parameter line on the probability plot for data sheets that use the gamma or generalized gamma distributions. Note that if you change this setting, you must refresh an affected plot to see the change.

30.2.6 Application Setup Other Page

The Other page of Weibull/ALTA's Application Setup allows you to set some other preferences that are applicable only to Weibull and ALTA.



The page contains the following settings:

- **Preferences**
  - Select **Show the Project Item Wizard when creating new projects** to display a wizard that helps you select the first new folio to create in a project. If not selected, the new project will be blank.
  - 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 data sheet. If not selected, you must close the QCP in order to return to the folios. 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 deleting columns with data** to display a confirmation message when using the Add or Remove Columns window to delete a column that has data.
- **Diagram**

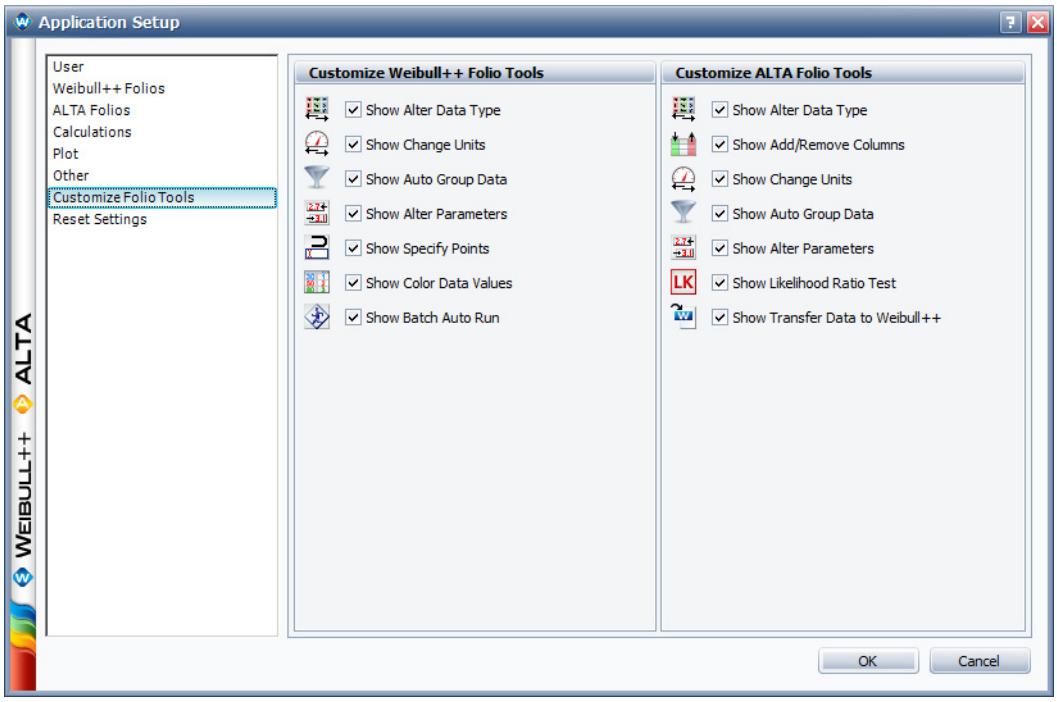
  Note: These setting apply only to the diagrams that are created using the Diagrams feature in Weibull++ or ALTA (*see Chapter 18*).

  - Use the **Truncate block names when the number of characters exceeds** area to set the maximum number of characters displayed for block names.
  - 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.
  - 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.
- Use the **Weibull++/ALTA View Mode** drop-down list to select which product view mode to use when opening a database.
- **Other Options**
  - Use the **Default folio font** drop-down list to select the default font for data sheets in new folios.
  - Select **Show list of distributions on hovering** to display the distribution/model drop-down list in control panels when you hover over the bar. If not selected, you must click the bar in order to display the list.

30.2.7 Application Setup Customize Folio Tools Page

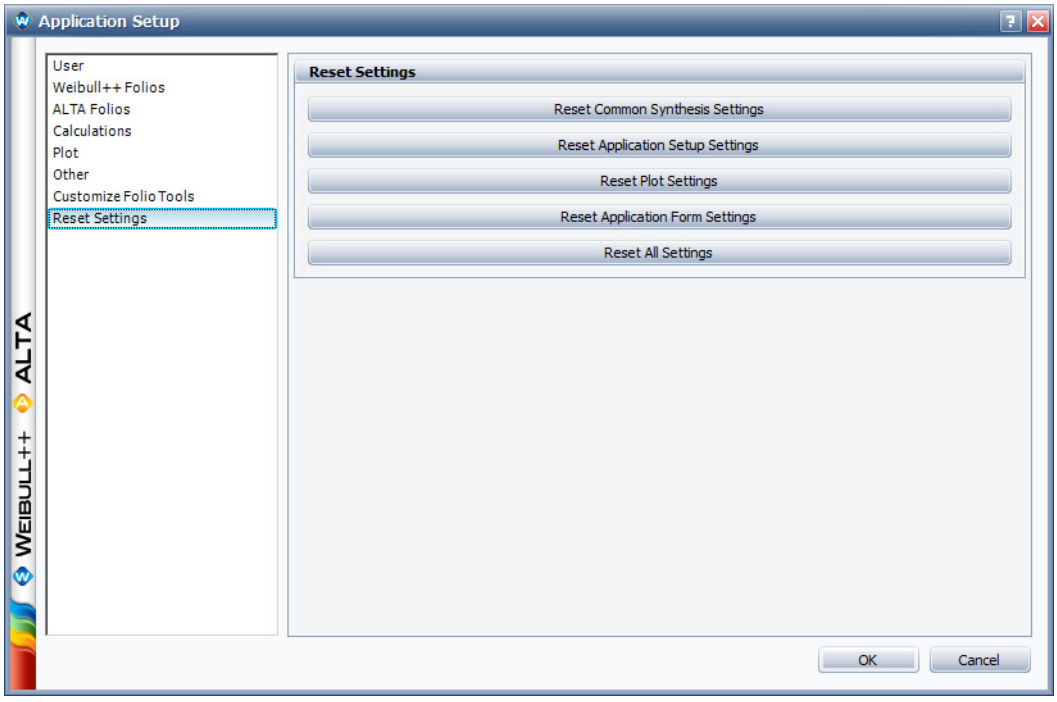
The Customize Folio Tools page of Weibull/ALTA's Application Setup allows you to specify which tools you want available on the standard folio control panels along with the standard tools that are always displayed (Calculate, Plot, QCP and Distribution Wizard).

In the **Customize Weibull++ Folio Tools** and **Customize ALTA Folio Tools** areas select or clear the options that you want displayed in the control panel. Note that these options will still be accessible from the applicable Ribbons.



30.2.8 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 applications on this computer; others apply only to the application that you are currently using.



The page contains the following settings:

- **Reset Common Synthesis Settings** resets only the settings that are shared by all Synthesis 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.

Minimum Data Requirements

A

This appendix describes the minimum data requirements for performing calculations in Weibull++/ALTA.

For Weibull++, the minimum data requirements are given for:

- Standard folios - Section 31.1.1 (p. 605).
- Degradation analysis folios - Section 31.1.2 (p. 610).
- Non-parametric life data analysis (LDA) folios - Section 31.1.3 (p. 610).
- Warranty analysis folios - Section 31.1.4 (p. 610).
- Event log folios - Section 31.1.5 (p. 611).
- Parametric recurrent event data analysis (RDA) folios - Section 31.1.6 (p. 611).
- Non-parametric recurrent event data analysis (RDA) folios - Section 31.1.7 (p. 612).

For ALTA, the minimum data requirements are given for:

- Standard folios - Section 31.2.1 (p. 612).
- Degradation analysis folios - Section 31.2.2 (p. 613).

A.1 Requirements for Weibull++

WEIBULL++ Only

The following sections describe the minimum data requirements for using different folios in Weibull++.

A.1.1 Standard Folios

Depending on the data type and the number of failures and/or suspensions present within your data, Weibull++ can perform all or some of the available analyses and calculations in a standard folio. Note that data sets with 2 or more distinct failure times, with or without suspensions, can be calculated with any of the available distributions, except for the mixed Weibull distribution.

For the mixed Weibull distribution, the following requirements apply:

- At least 5 unique failure times (i.e., data points with different time values) are required for the 2-subpopulation analysis.
- At least 8 unique failure times are required for the 3-subpopulation analysis.
- At least 11 unique failure times are required for the 4-subpopulation analysis.

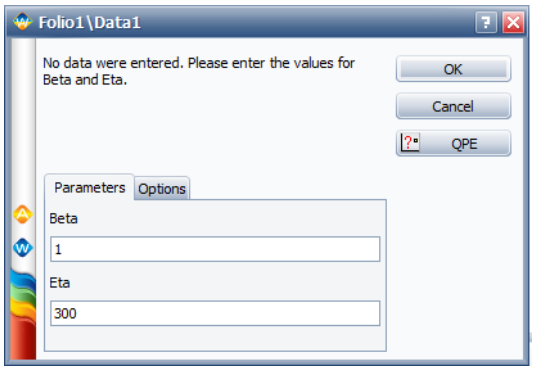
In addition, Weibull++ allows you to work with data sets that have fewer than 2 distinct failures. However, the analysis methods that can be performed will vary depending on the data and particular case.

There are a total of five possible cases as to how data can be entered. For each scenario, a description of the available analysis options is provided next.

A.1.1.1 Case 1: No Data

Data Types	ALL
Methods of Analysis	NONE (parameters supplied by user)
Confidence Bounds Calculations?	NO
Distributions	All except 1-parameter Weibull, Weibull-Bayesian and CFM-Weibull
Competing Failure Modes?	NO

When no data is entered and you click the **Calculate** icon, Weibull++ will prompt you to enter the values of the parameters for the selected model, as shown next for the 2-parameter Weibull. The appearance of the Parameter Input window will vary depending on the distribution selected.

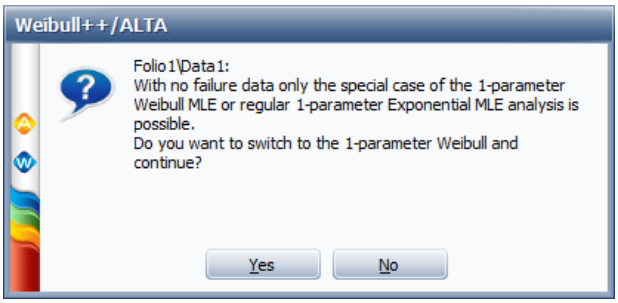


Specify values for the parameters based on the distribution selected and click **OK**. All options and calculations supported by Weibull++ will be available except calculations involving confidence bounds.

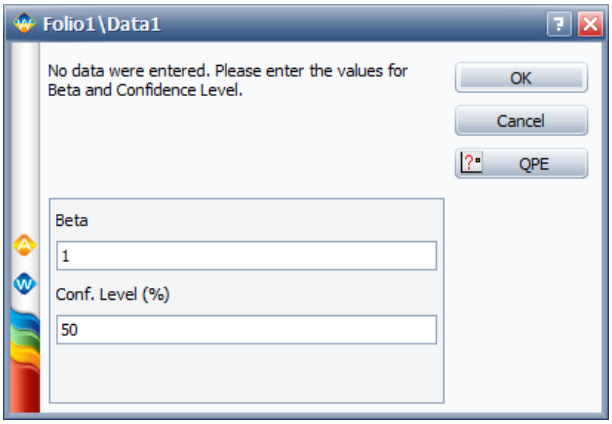
A.1.1.2 Case 2: No Failures and 1+ Suspensions

Data Types	All except free-form
Method of Analysis	MLE
Confidence Bounds Calculations?	NO
Distributions	1-parameter Weibull and 1-parameter exponential
Competing Failure Modes?	NO

Since only suspension data has been entered, Weibull++ will only allow you to perform calculations involving the 1-parameter Weibull distribution with MLE or the 1-parameter exponential distribution with MLE.



To use the 1-parameter Weibull distribution, you will be prompted to enter the shape parameter, beta, as well as the lower one-sided confidence for the scale parameter, eta.



Once you have supplied the parameters, all options and calculations are supported except for calculations involving confidence bounds. The calculated scale parameter, eta, will be the lower one-sided value at the confidence level specified by you.

If using the 1-parameter exponential distribution, the results will be at the 50% confidence level. To use the exponential distribution at a higher confidence level, select the Weibull distribution, specify a beta equal to 1, and enter the desired confidence level.

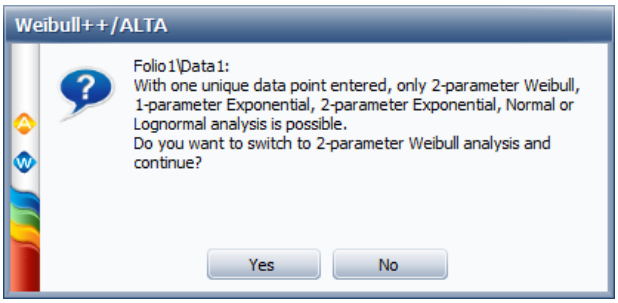
A.1.1.3 Case 3: 1 Failure and No Suspensions

Free-Form Data

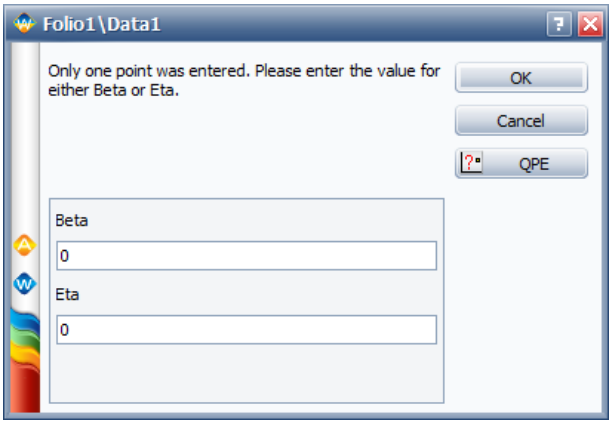
Data Types	Free-form only
Method of Analysis	Rank Regression
Confidence Bounds Calculations?	NO
Distributions	2-parameter Weibull, 1-parameter exponential, 2-parameter exponential, normal and lognormal
Competing Failure Modes?	NO

Only rank regression is possible when using free-form data. MLE will be disabled.

With only one unique data point, only the 2-parameter Weibull, 1-parameter exponential, 2-parameter exponential, normal and lognormal analyses are possible. If you select a different distribution, a message box will appear notifying you that you need to switch analysis types, as shown next.



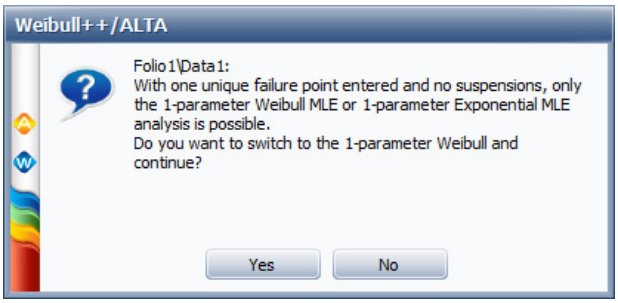
If you select to use a 2-parameter distribution, you will then be prompted to enter a value for one of the parameters. Remember that when using free-form data, you are supplying both the X and Y position for the entered data point. Given this point and a value for one of the parameters, the application will then solve for the other parameter. The parameter that you would like to solve for should be left as zero as shown below.



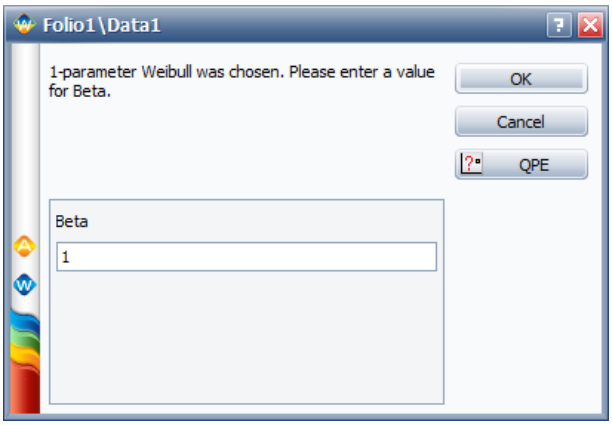
Other Data Types

Data Types	All except free-form data
Method of Analysis:	MLE
Confidence Bounds Calculations?	NO
Distributions:	1-parameter Weibull and 1-parameter exponential
Competing Failure Modes?	NO

With only one unique failure time, the application will notify you that only the 1-parameter Weibull and 1-parameter exponential distributions with MLE are available.



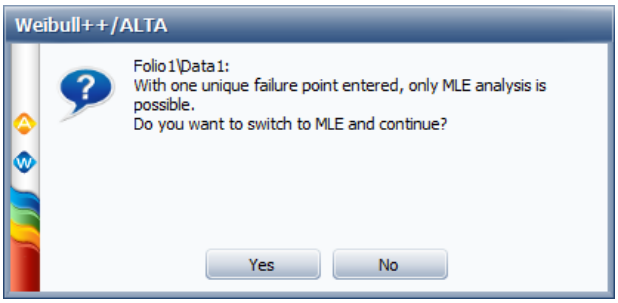
If you select the 1-parameter Weibull, you will then be prompted to enter the value of the known parameter.



A.1.1.4 Case 4: 1 Failure and 1+ Suspensions

Data Types	All data types with suspensions
Method of Analysis:	MLE
Confidence Bounds Calculations?	YES
Distributions:	All except mixed Weibull, 3-parameter Weibull, 2-parameter exponential and generalized gamma. However, depending on the number of suspensions and their location, a solution may be impossible.
Competing Failure Modes?	NO

If you enter only one unique failure and at least one suspension with any analysis method other than MLE, the application will notify you that only MLE is possible.



A.1.1.5 Case 5: 2+ Failures

Data Types	ALL
Method of Analysis:	ALL
Confidence Bounds Calculations?	YES
Distributions:	All except mixed Weibull
Competing Failure Modes?	YES

Note that unless sufficient data are present, some distributions of 3 or more parameters may not be available, or the application may be unable to converge for a solution. In general, when using distributions with 3 or more parameters, you should have at least 5 distinct failure points per parameter.

A.1.2 Degradation Analysis Folios

The minimum data requirements for degradation analysis folios are as follows:

- The unit ID must be entered for all data points.
- For all models except Gompertz, there must be at least 2 data points for each unique unit ID. For the Gompertz model, there must be at least 3 data points for each unique unit ID.
- The data must produce enough extrapolated failure/suspension times to perform life data analysis with the selected life distribution. See the requirements for standard folios.

A.1.3 Non-Parametric Life Data Analysis (LDA) Folios

For non-parametric LDA folios, there must be at least 1 data point entered.

A.1.4 Warranty Analysis Folios

Weibull++ provides four data formats for warranty analysis. The requirements for each format are described next.

Nevada Chart Format

The minimum data requirements for warranty analysis folios in the Nevada chart format are as follows:

- The returns quantity cannot exceed the sales quantity.
- At least one data point must be entered.

- With only Sales data entered (i.e., a data set with suspensions only), only the 1-parameter Weibull or 1-parameter exponential distributions with maximum likelihood estimation (MLE) will be available. To use the 1-parameter Weibull, you will be prompted to enter the beta value and the lower one-sided confidence bound. Confidence bounds will not be available in the QCP or plots. This is similar to Case 2 for standard folios.
- If the Returns data contains failure(s) for only 1 time period, only MLE analysis is possible. In addition, the 3-parameter Weibull, mixed Weibull and generalized gamma distributions will not be available. This is similar to Case 4 for standard folios.
- If the Returns data contains failure(s) for at least 2 time periods, all distributions except mixed Weibull will be available. This is similar to Case 5 for standard folios.

Times-to-Failure Format

The minimum data requirements for warranty analysis folios in the times-to-failure format are the same as those described previously for standard folios, except you cannot calculate with no data entered.

Dates of Failure Format

The minimum data requirements for warranty analysis folios in the dates of failure format are as follows:

- The "End of Observation Period" (entered on the Main page of the control panel) is required.
- The "Date In-Service" is required for all Sales and Returns data points. Each date entered in the Returns sheet must be identical to a date that has been entered in the Sales sheet.
- At least one data point must be entered.
- With only Sales data entered (i.e. a data set with suspensions only), only the 1-parameter Weibull or 1-parameter exponential distributions with maximum likelihood estimation (MLE) will be available. To use the 1-parameter Weibull, you will be prompted to enter the beta value and the lower one-sided confidence bound. Confidence bounds will not be available in the QCP or plots. This is similar to Case 2 for standard folios.
- If the Returns data contains only one unique failure point, only MLE analysis is possible. In addition, the 3-parameter Weibull, mixed Weibull and generalized gamma distributions will not be available. This is similar to Case 4 for standard folios.
- If the Returns data contains failure(s) for at least two time periods, all distributions except mixed Weibull will be available. This is similar to Case 5 for standard folios.

Usage Format

The minimum data requirements for warranty analysis folios in the usage format are the same as those described above for warranty folios in the dates of failure format.

A.1.5 Event Log Folios

The minimum data requirements for event log folios are as follows:

- The system start date must be earlier than or the same as the earliest event date.
- At least one event must be entered.

A.1.6 Parametric Recurrent Event Data Analysis (RDA) Folios

The minimum data requirements for parametric recurrent RDA folios are as follows:

- The System ID must be entered for all data points.
- There must be at least 3 unique Failure (F) data points entered.

A.1.7 Non-Parametric Recurrent Event Data Analysis (RDA) Folios

- The minimum data requirements for non-parametric RDA folios are as follows:
- The System ID must be entered for all data points.
- If confidence bounds are not required, there must be at least 1 Failure (F) data point for each unique System ID.
- If confidence bounds are required, there must be at least 2 unique System IDs with at least 1 Failure (F) data point each.

A.2 Requirements for ALTA

ALTA Only

The following sections describe the minimum data requirements for standard folios and degradation analysis folios in ALTA.

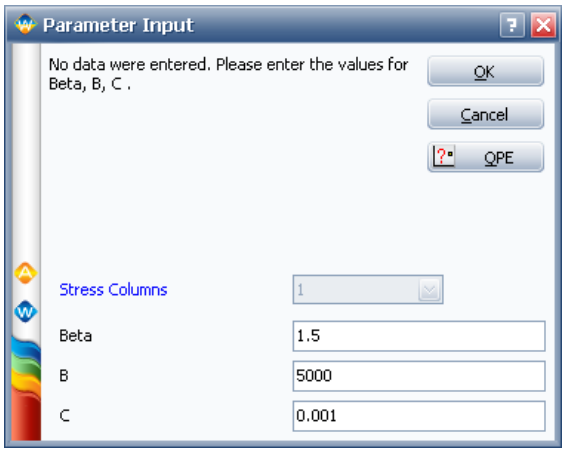
A.2.1 Standard Folios

The minimum data requirements for ALTA standard folios are broken down by model in the table below. Note the following:

- "Number of failures" assumes unique (i.e., different) failure times.
- "Number of stress levels" assumes unique stress levels.
- "Stresses" refers to the number of stress columns used in the analysis (this may not necessarily equal the number of stress columns in the data sheet).
- If the data set uses the minimum number of stress levels, then each stress level must have at least one failure.
- If you provide the value of the shape parameter (β for the Weibull distribution or σ for the lognormal distribution), less data will be required for calculation, as shown in the **Give β or σ and Solve** column of the minimum data requirements table below. If the data set you enter meets only these minimum requirements, when you click **Calculate**, you will be prompted to provide the value of the shape parameter in a window. Note that confidence bounds calculations are not possible in these situations.

Life-Stress Model	Give β or σ and Solve	Exponential	Weibull/Lognormal
Arrhenius/Eyring/Inverse Power Law	Failures = 2 Stress Levels = 2 *Note:* If you provide 3 or more failures at only 1 stress level, you can provide the acceleration factor to calculate.	Failures ≥ 2 Stress Levels ≥ 2	Failures ≥ 3 Stress Levels ≥ 2
Temperature-Humidity/ Temperature-Nonthermal	Failures = 3 Stress Levels = 3	Failures ≥ 3 Stress Levels ≥ 3	Failures ≥ 4 Stress Levels ≥ 3
Generalized Eyring	Failures = 4 Stress Levels = 4	Failures ≥ 4 Stress Levels ≥ 4	Failures ≥ 5 Stress Levels ≥ 4
Proportional Hazards/ General Log-Linear	Failures = stresses + 1 Stress Levels = stresses + 1	Failures ≥ stresses + 1 Stress Levels ≥ stresses + 1	Failures ≥ stresses + 2 Stress Levels ≥ stresses + 1 *Note:* The lognormal distribution is not available for proportional hazards.
Cumulative Damage	Failures = stresses + 1 Stress Levels = stresses	Failures ≥ stresses + 1 Stress Levels ≥ stresses	Failures ≥ stresses + 2 Stress Levels ≥ stresses *Note:* Only one stress column can be used with the lognormal distribution.

If you enter no data at all and click the **Calculate** icon, ALTA will prompt you to enter the values of the parameters for the selected model, as shown next. The appearance of the Parameter Input window will vary depending on the life-stress model and distribution selected.



Specify values for the parameters based on the selected life-stress model and distribution and click **OK**. All options and calculations supported by ALTA will be available except calculations involving confidence bounds and plots that require individual stress levels.

A.2.2 Degradation Analysis Folios

The minimum data requirements for degradation analysis folios are as follows:

- The Unit ID must be entered for all data points.
- For all models except Gompertz, there must be at least two data points for each unique Unit ID. For the Gompertz model, there must be at least three data points for each unique Unit ID.
- The data must produce enough extrapolated failure/suspension times to perform accelerated life testing data analysis with the selected ALTA model. See the requirements for standard folios.

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