User's Guide

Weibull++

VERSION 2021

ReliaSoft

TABLE OF CONTENTS

Getting Started

IN THIS CHAPTER

Introduction to Weibull++

ReliaSoft **Weibull++** by HBM Prenscia is the industry standard in life data analysis (Weibull analysis) for thousands of companies worldwide. Accelerated Life Testing (ALTA) provides an intuitive and user-friendly way to utilize tremendously complex and powerful mathematical models for quantitative accelerated life testing data analysis. Reliability Growth (RGA) is a powerful software tool that allows you to apply reliability growth models to analyze data from both developmental testing and fielded repairable systems. In the development stage, the software allows you to quantify the reliability growth achieved with each successive design prototype and also provides advanced methods for reliability growth projections, planning and management. For systems operating in the field, RGA allows you to calculate optimum overhaul times and other results without the detailed data sets that normally would be required for repairable system analysis.

What's Changed? Starting in Version 2020, ALTA and RGA are now part of Weibull++. ALTA is now referred to as "Accelerated Life Testing" and RGA is now referred to as "Reliability Growth".

This application is part of the suite of ReliaSoft desktop applications, which offers intelligent integration between reliability program activities and tools, while simultaneously facilitating effective information sharing and cooperation between engineering teams of any size. For information about features that are shared by all (or most) of the ReliaSoft desktop applications, see the Databases and Projects, Desktop Application Interfaces and ReliaSoft Common Tools chapters.

Other ReliaSoft software by HBM Prenscia includes: BlockSim, Lambda Predict, XFMEA, RCM++, MPC, SEP web portal and XFRACAS.

What's Changed? The ReliaSoft RBI application has been discontinued as of Version 2021. For information on RBI functionality, please refer to the product help specific to your version of the software, available from https://help.reliasoft.com.

Quick Tour

Quick Tour of Weibull++

Weibull++ offers the most comprehensive toolset available for **reliability life data analysis**, with support for all life data types and all commonly used product lifetime distributions.

Accelerated Life Testing (ALTA) and **Accelerated Life Testing PRO (ALTA PRO)** utilize powerful life-stress relationship models to design and analyze data from **accelerated life tests**.

Reliability Growth (RGA) supports all of the major **reliability growth models** and can analyze data from both developmental testing and fielded repairable systems. It also includes a comprehensive toolset for advanced reliability growth projections, planning and management.

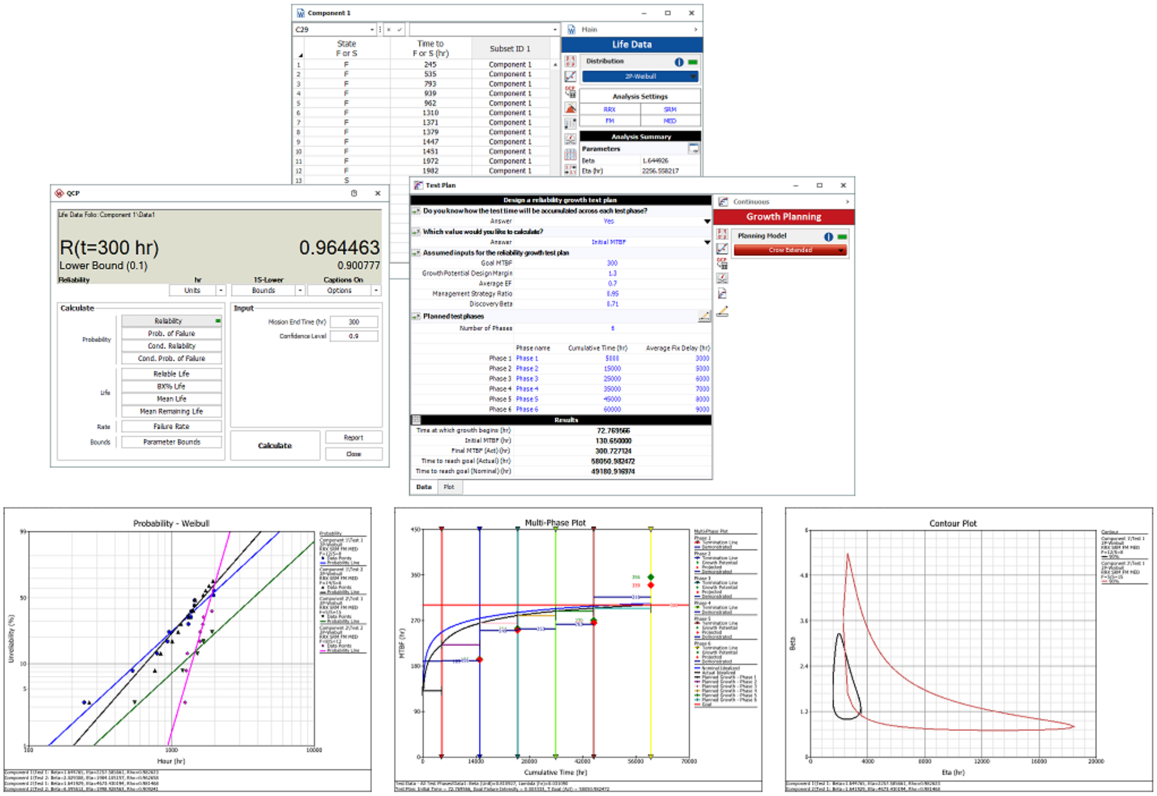
These applications are also packed with tools for related reliability analysis such as degradation data analysis, stress-strength comparison, reliability test design and design of experiments (DOE). In addition, Weibull++ also supports warranty data analysis, non-parametric data analysis and recurrent event data analysis.

For life data and life-stress data analysis, you can use this application to answer a wide variety of questions such as:

- How many units do you need to test? How much test time? At which stress levels?
- Will the product meet the reliability requirement?
- What is the influence of different failure modes? How does accelerated stress affect life?
- Which design (or supplier) will be more cost-effective over time?
- How many units are likely to be returned under warranty?
- Is there an emerging reliability problem in the field? With a particular supplier? In a particular region?
- When will a crack, wear, etc. degrade to the point of failure?
- What is the typical time to repair? How effective are the repairs?

For growth data analysis, you can use this application to answer a wide variety of questions, such as:

- Will the product meet the MTBF goal? How much test time is required?
- How does the product perform across different test phases? At which phase will the product meet the reliability requirement? What is the initial MTBF?
- How do the test data compare to the reliability growth test plan?
- What failure modes should be fixed and when?
- What is the optimum growth maintenance strategy? When is the best time to schedule maintenance/overhauls?



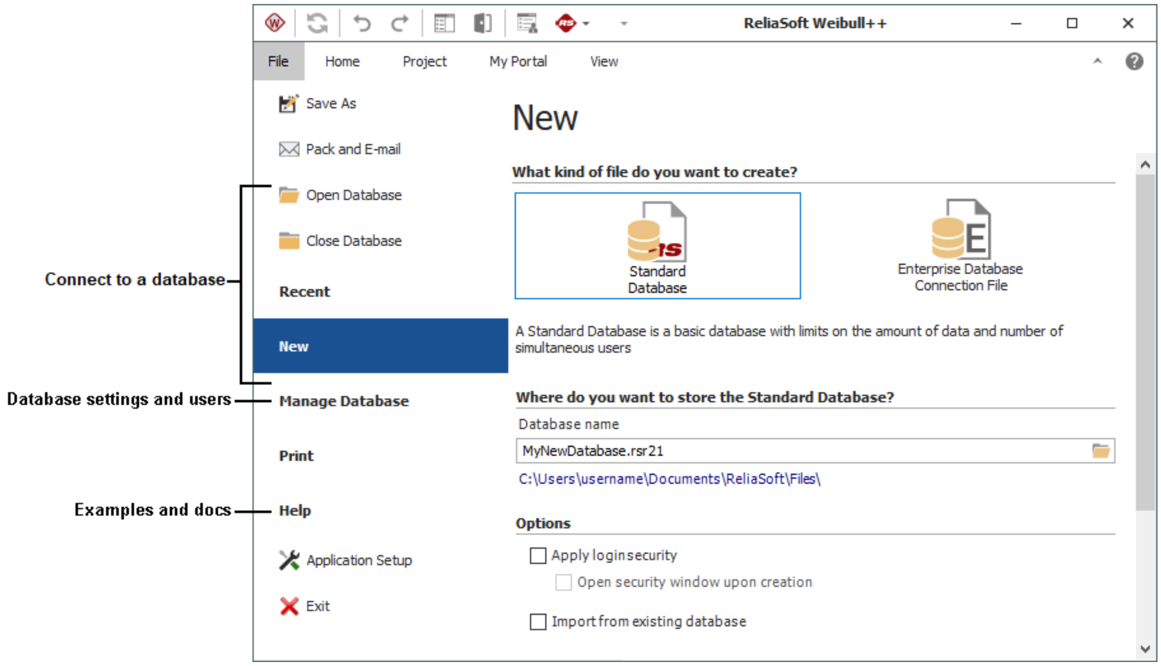
Using Databases and Projects

File Menu

Use commands on the **File** menu (**New**, **Recent** or **Open Database**) to create or open a database). (See Database Types.)

Each database supports simultaneous access by multiple users. If you have admin permissions, use **File > Manage Database** to control user access and manage configurable settings that apply throughout the database. (See Security Options and Backstage View.)

Use **File > Help** to access online help and shipped examples.

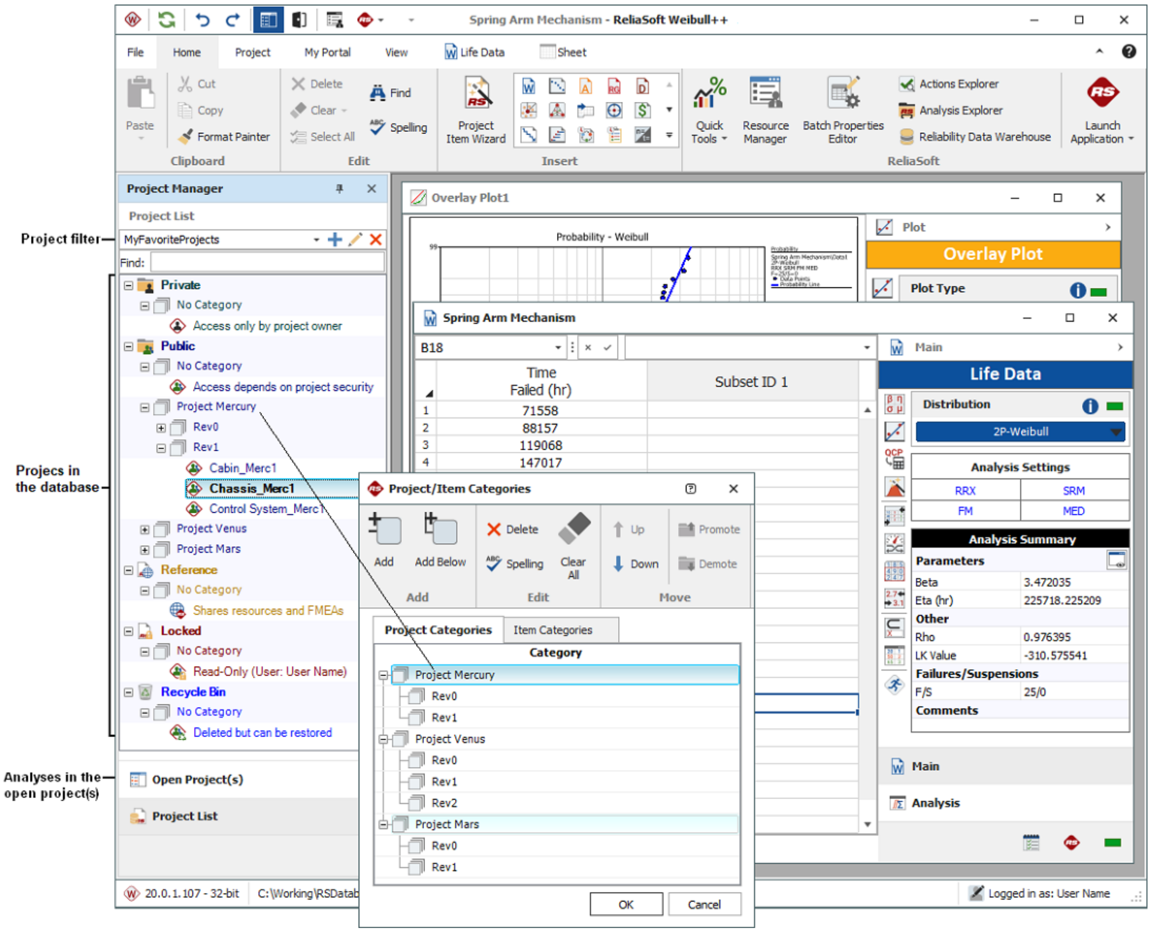


Project List

Each database can contain multiple projects. Use the **project list** to create or open a project. (See Project Manager.)

If you have a lot of projects:

- Use the **project filter** to limit what's displayed.
- Use the **project category** to group projects within the list.



Current Project Explorer

Use the **current project explorer** to create or open analyses and reports. (See Current Project Explorer.)

You can open the same project in any ReliaSoft desktop application. Analyses and reports are visible only in the application(s) that can edit them:

- Life data folios are visible only in Weibull++
- Fault trees are visible only in BlockSim
- FMEAs are visible only in XFMEA/RCM++
- Etc.

Common features such as **project plans**, **resources** and **project attachments** are visible in all ReliaSoft desktop applications.

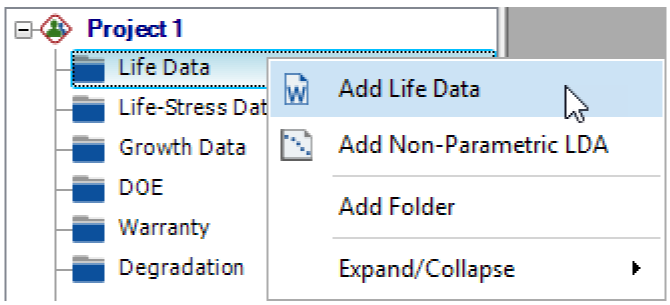
Using Life Data Folios

Weibull++'s life data folio allows you 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 obtain reliability information about a product such as probability of failure, reliability, mean life or failure rate.

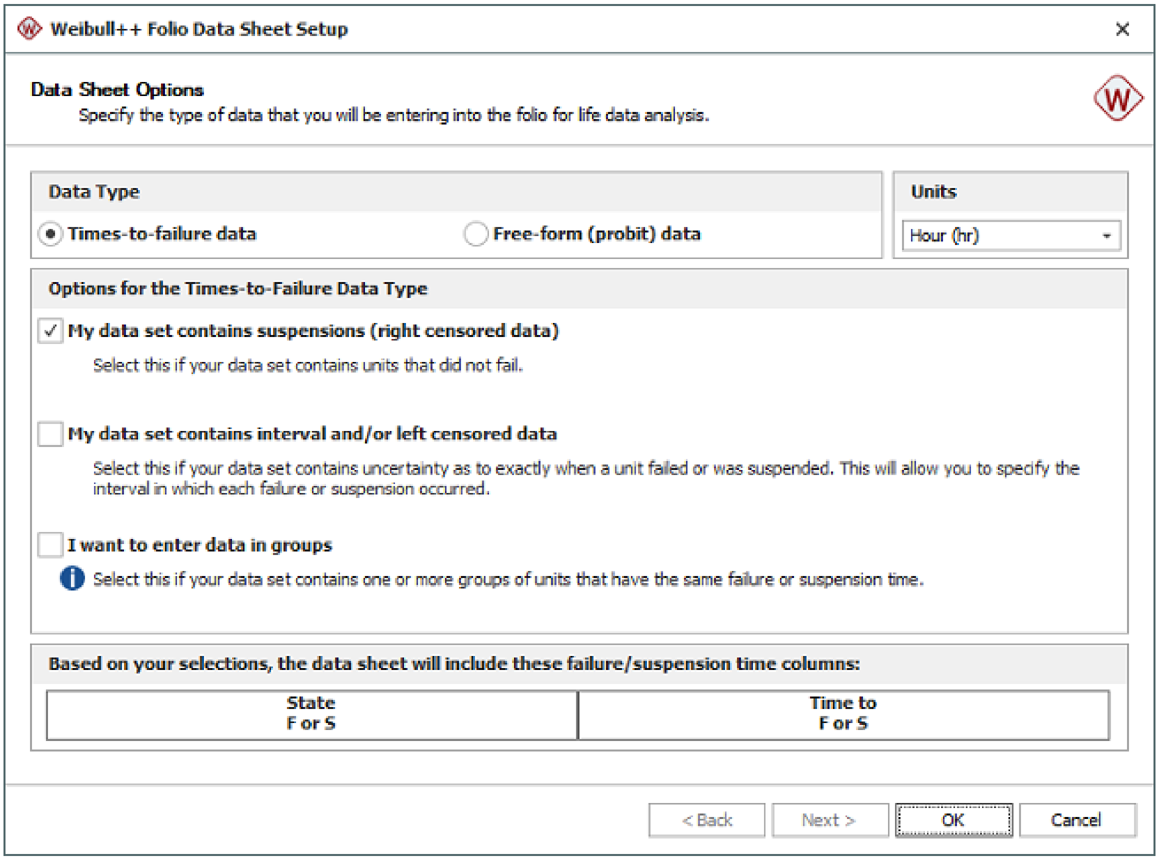
The basic steps for using a life data folio are:

1. Add life data folio

   Right-click the Life Data folder in the current project explorer and choose **Add Life Data**.

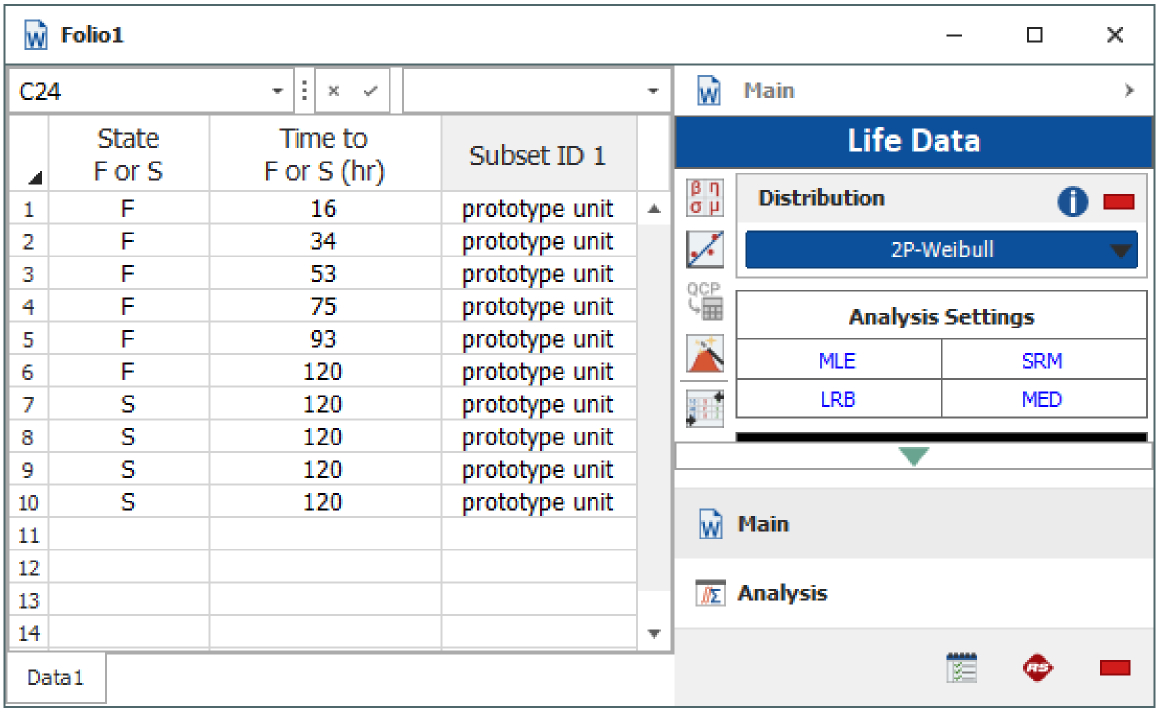


   In the setup window, select the desired data type and the units for time/usage values (hours, cycles, etc.). (See Weibull++ Data Types.)



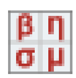
2. Enter data

Enter the data and use the control panel to specify the life distribution and analysis settings. (See Life-Data Folio Control Panel for descriptions of all available settings.)

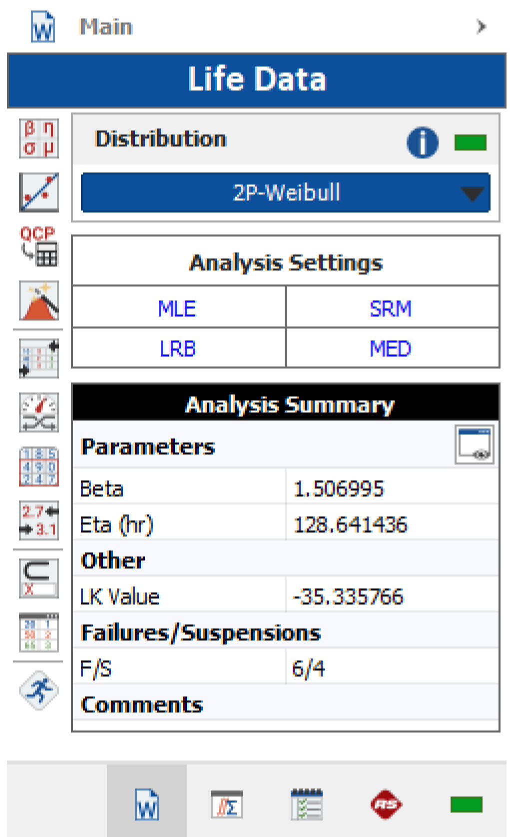


3. Analyze data

Click the **Calculate** icon on the control panel.

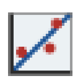


The Analysis Summary area displays the parameters of the distribution. Click the **Detailed Summary** icon  to view more information.

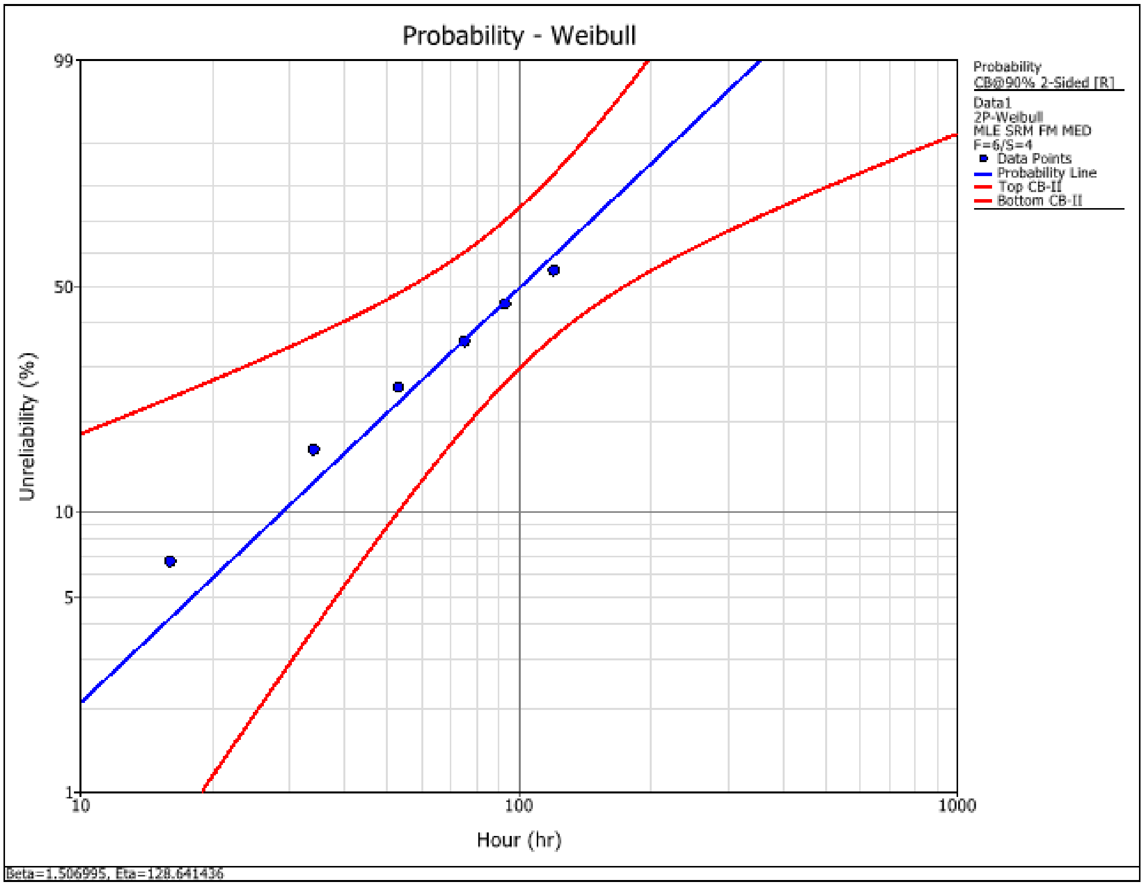


4. View plots

Click the **Plot** icon on the control panel to open the plot sheet. (See Life Data Analysis Plots for available plot types.)



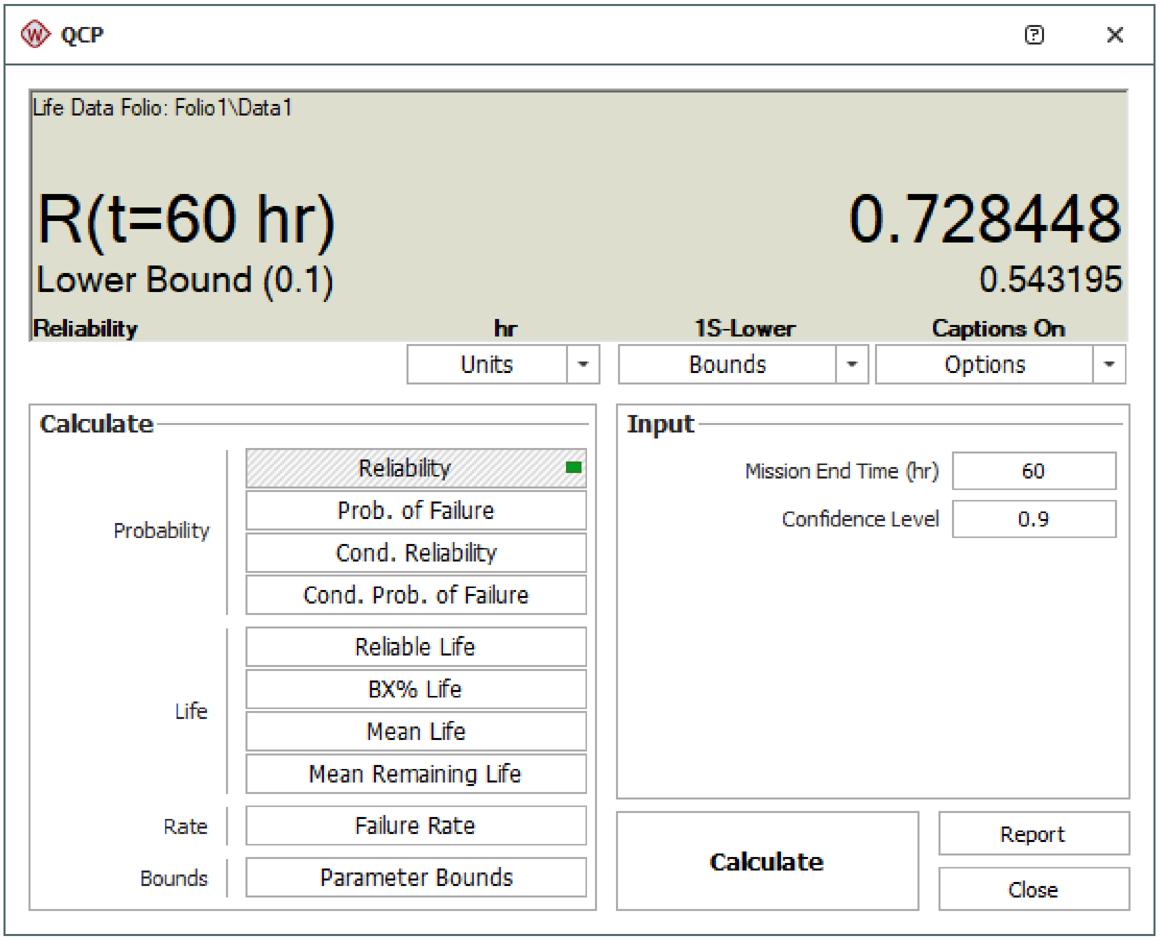
To show the confidence bounds, click the **Confidence Bounds** link on the plot's control panel, and then set the desired type of bounds and confidence level.



5. Calculate metrics

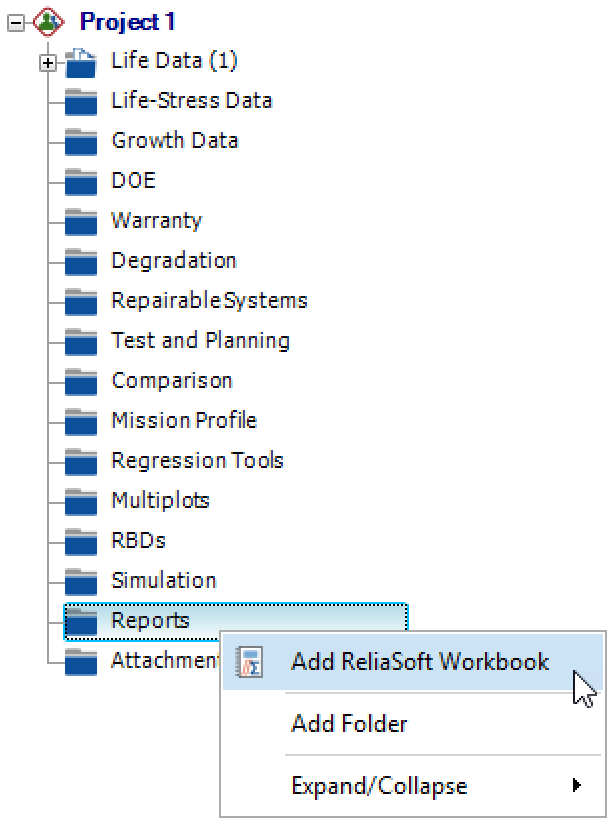
Click the **QCP** icon on the control panel to open the Quick Calculation Pad. (See QCP Calculations for Life Data Analysis for available calculations.)

The following example shows the estimates for the reliability and the lower one-sided 90% confidence bound.

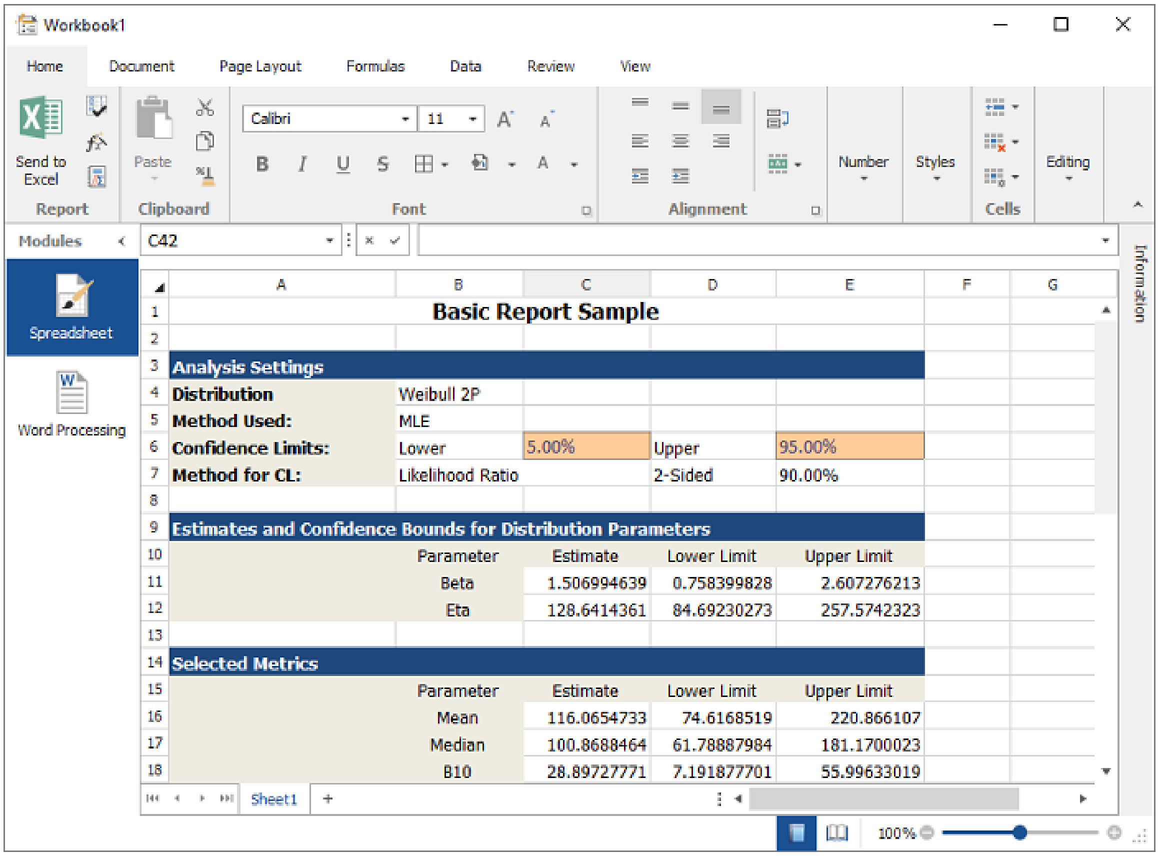


6. Create report

Right-click the Reports folder in the current project explorer and choose **Add ReliaSoft Workbook**.



In the ReliaSoft Workbook wizard, select the folio as the default data source. If desired, you can also select a standard or user-defined report template (e.g., the "Weibull Summary Style 1 Template" is shown below). (See ReliaSoft Workbooks.)



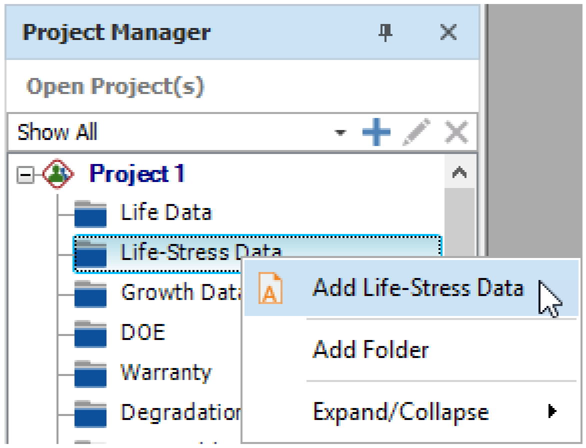
Using Life-Stress Data Folios

Weibull++'s life-stress data folio allows you to perform quantitative accelerated life testing data analysis. In this analysis, you extrapolate 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, the analysis allows you to obtain reliability information about a product (e.g., mean life, probability of failure at a specific time, etc.) in a shorter time.

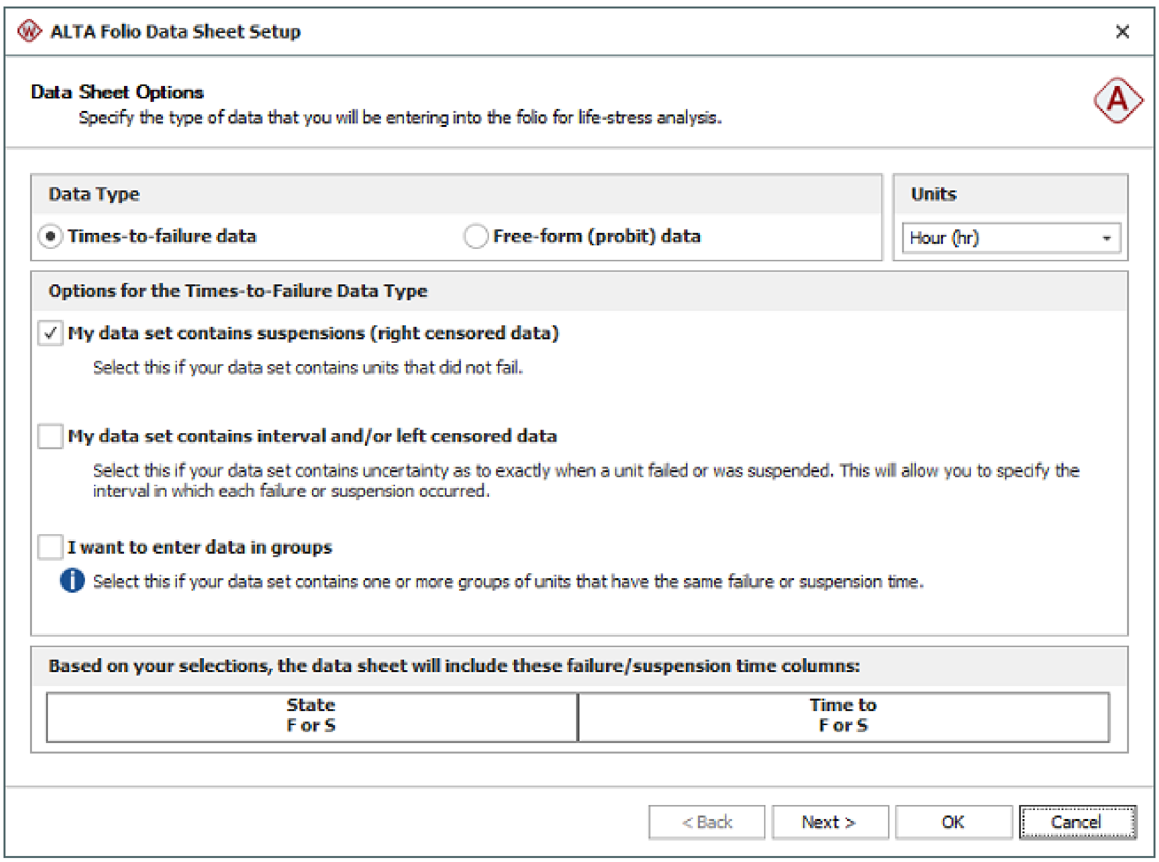
The basic steps for using a life-stress data folio are:

1. Add life-stress data folio

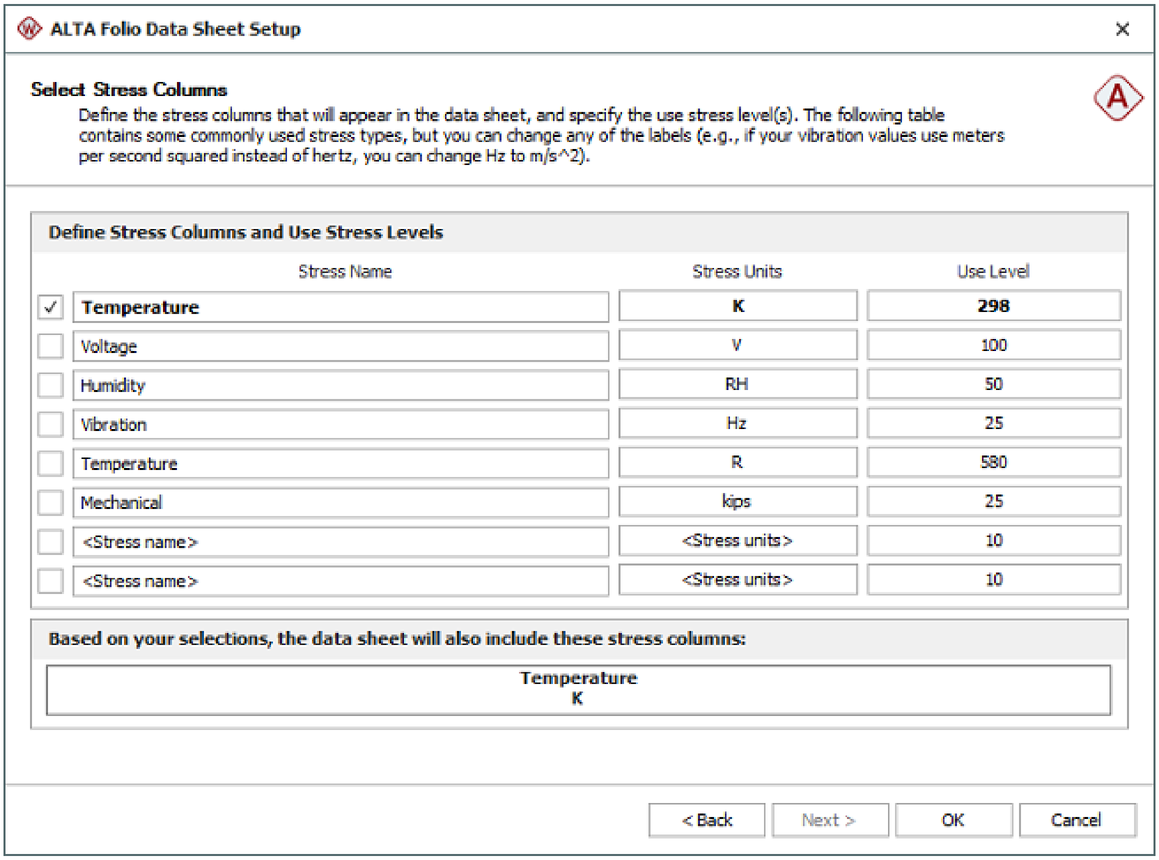
   Right-click the Life-Stress Data folder in the current project explorer and choose **Add Life-Stress Data**.



In the setup window, select the desired censoring type and units for time/usage values (hours, cycles, etc.). (See Accelerated Life Testing Life-Stress Data Folio Setup.)

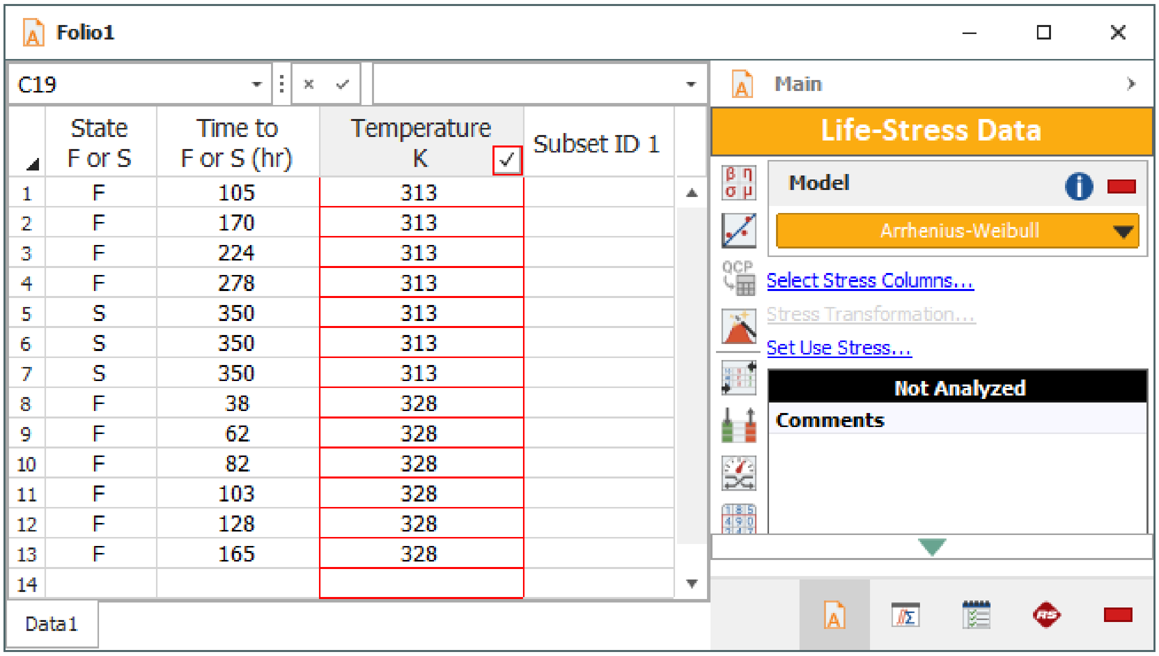


On the next page, define the stress type(s) to use in the analysis and the use stress level.



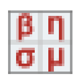
2. Enter data

Enter the data and use the control panel to specify the life-stress model. (See Life-Stress Data Folio Control Panel for available models.)

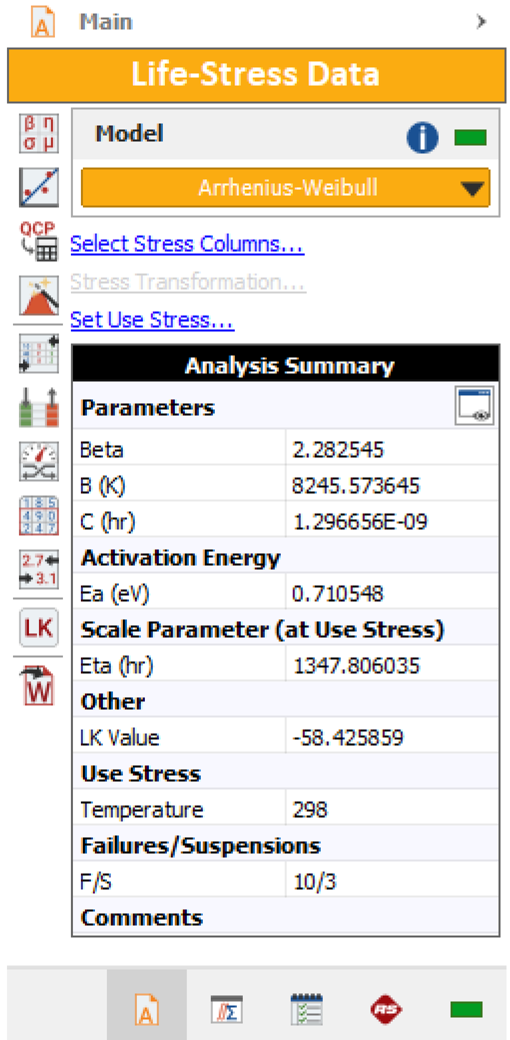


3. Analyze data

Click the **Calculate** icon on the control panel.

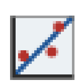


The Analysis Summary area displays the parameters of the distribution. Click the **Detailed Summary** icon  to view more information.

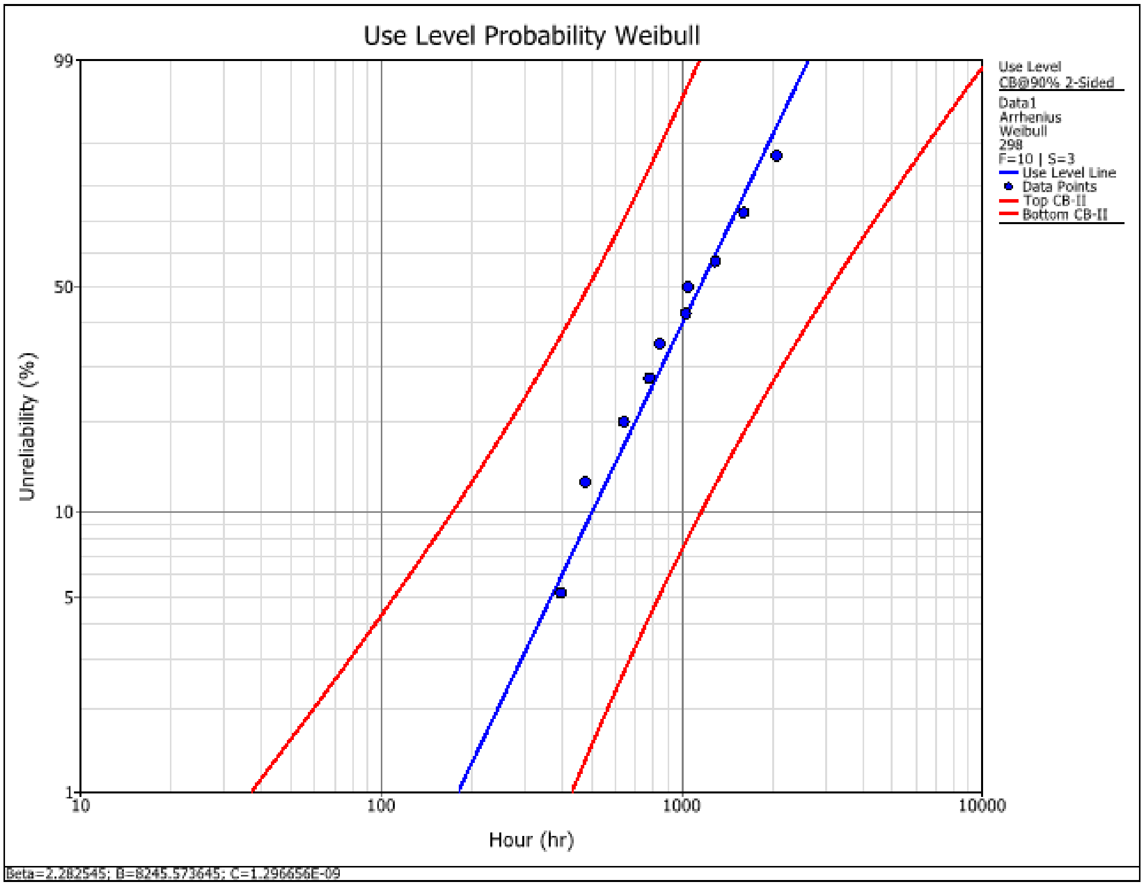


4. View plots

Click the **Plot** icon on the control panel to open the plot sheet. (See Accelerated Life Testing Plots for available plot types.)



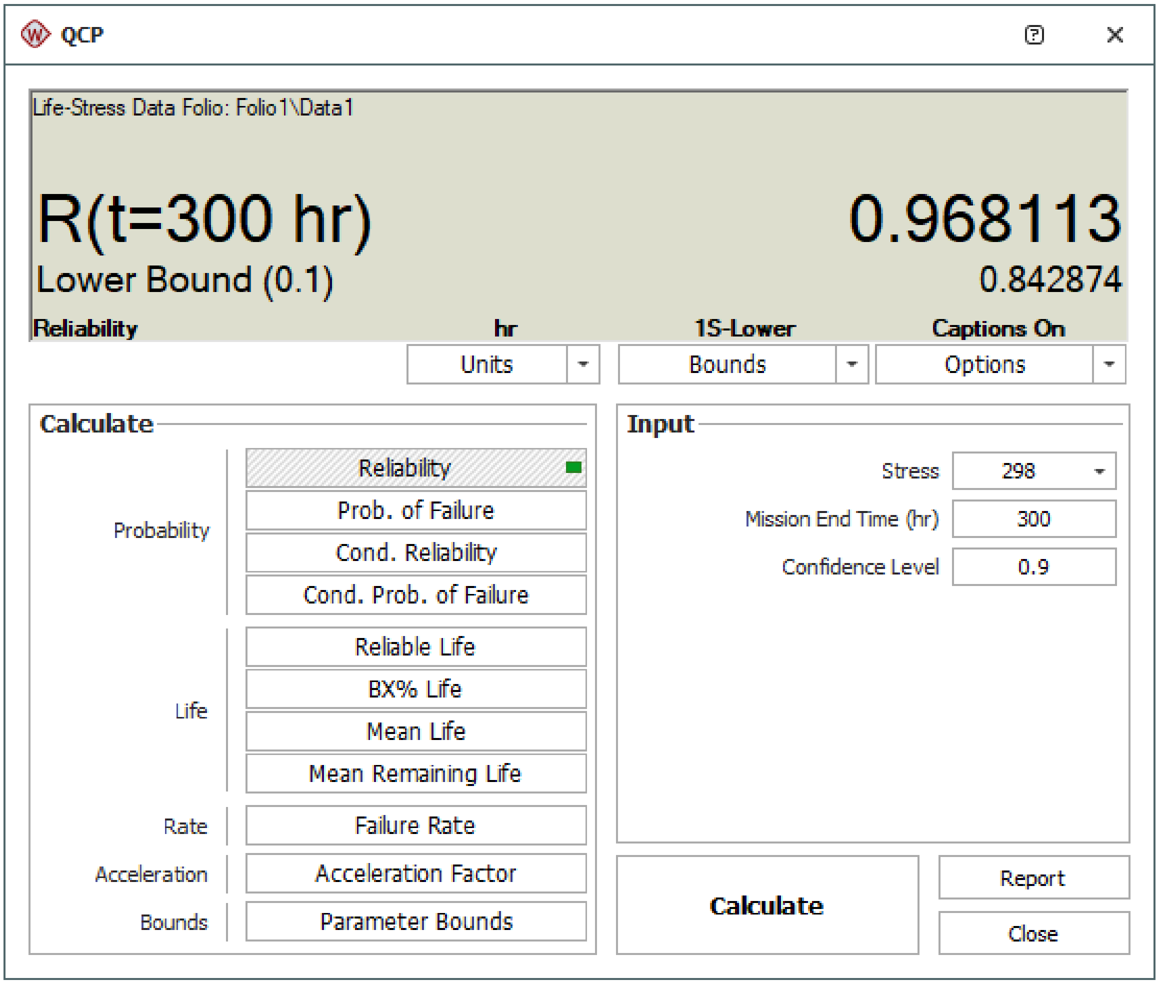
To show the confidence bounds, click the **Confidence Bounds** link on the plot's control panel, and then set the desired type of bounds and confidence level.



5. Calculate metrics

Click the **QCP** icon on the control panel to open the Quick Calculation Pad. (See QCP Calculations for Accelerated Life Testing Analysis for available calculations.)

The following example shows the estimates for reliability and the lower one-sided 90% confidence bound.



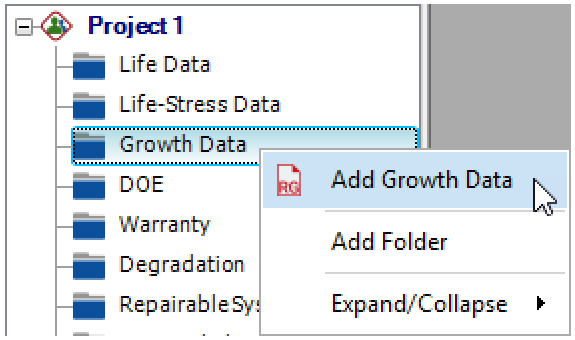
Using Growth Data Folios

Weibull++'s growth data folio supports traditional reliability growth analysis, repairable systems analysis and reliability growth projections. The data type you select determines the type of analysis you can perform.

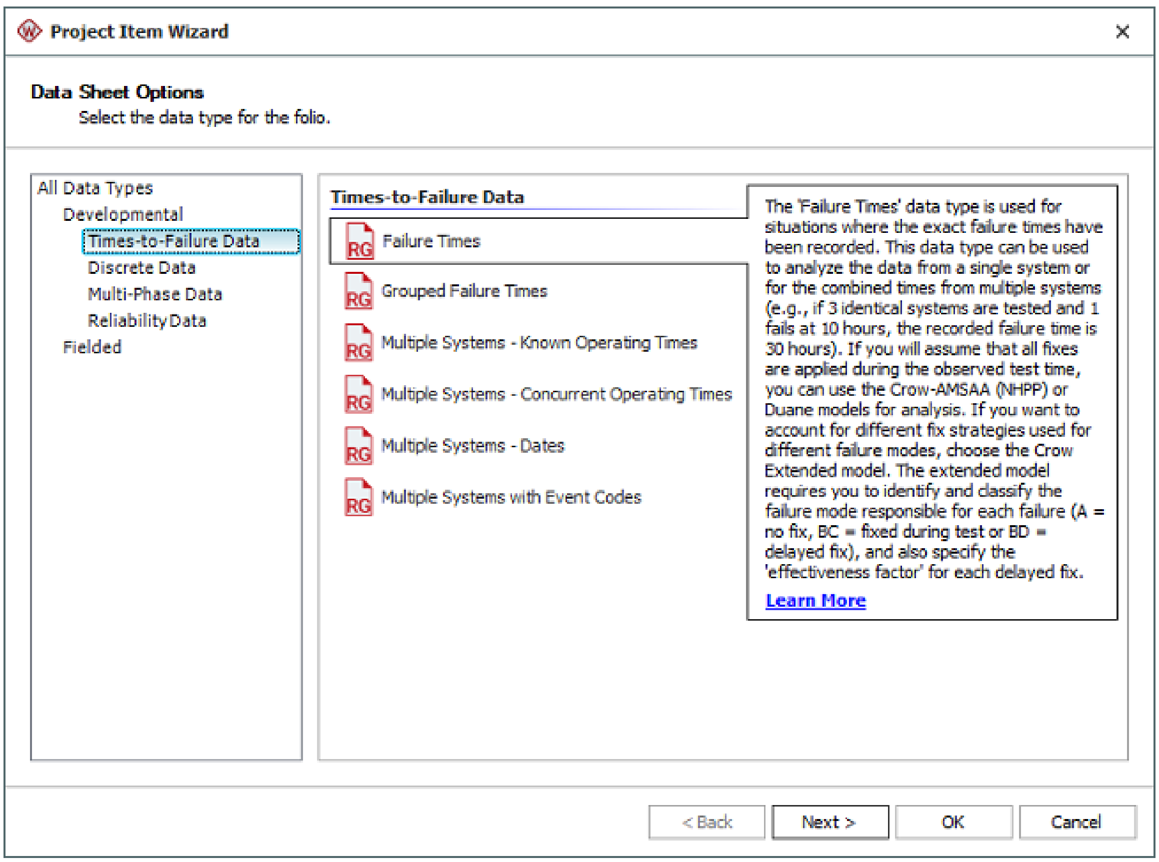
The basic steps for using a growth data folio are:

1. Add growth data folio

   Right-click the Growth Data folder in the current project explorer and choose **Add Growth Data**.

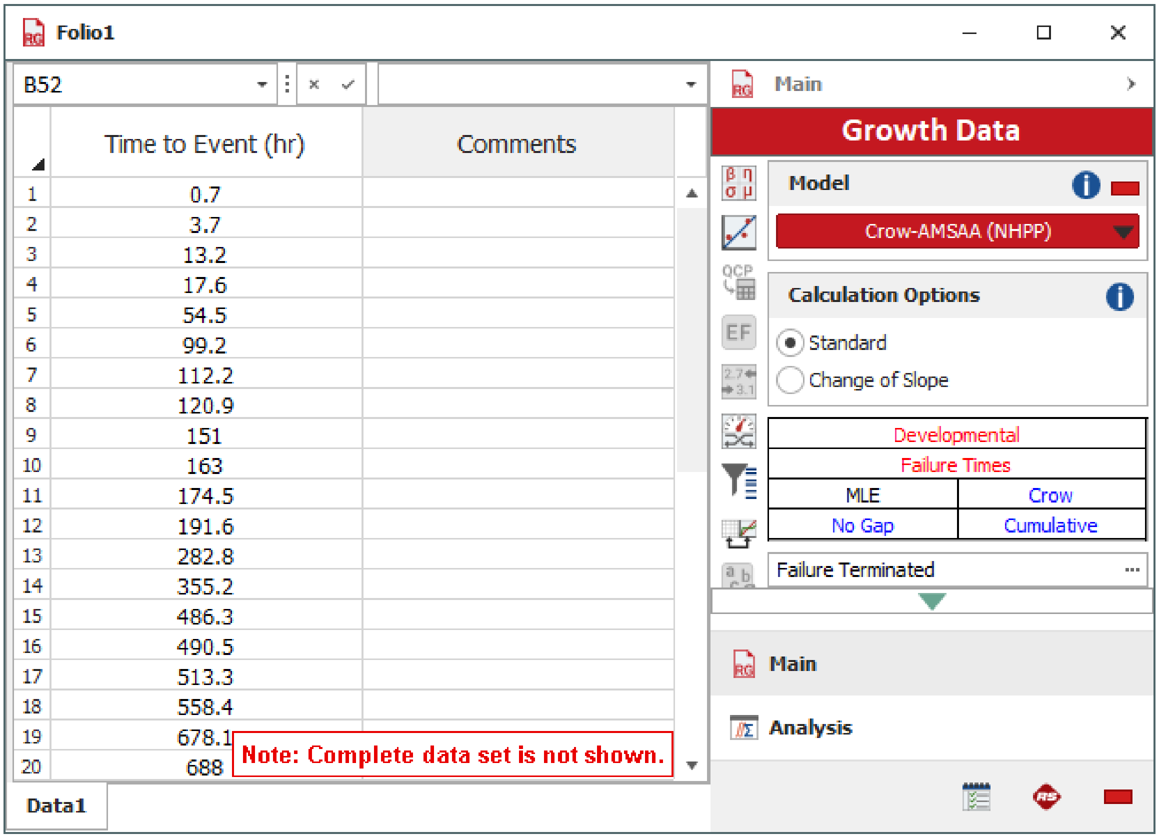


In the setup window, select the desired data type. (See Reliability Growth Data Types.) Then click **Next** to set the units for values (hours, cycles, etc.) and other preferences. (See Creating a Growth Data Folio.)



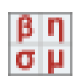
2. Enter data

Enter the data and use the control panel to specify the growth model and analysis settings. (See Growth Data Folio Control Panel for descriptions of all available settings.)

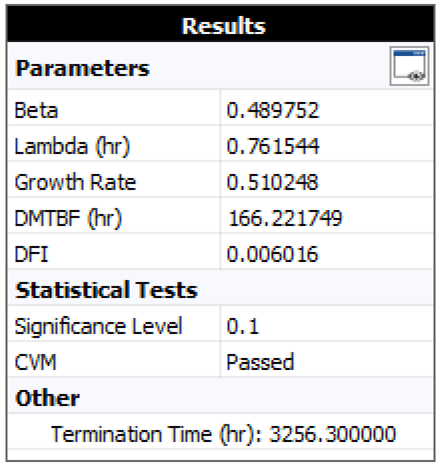


3. Analyze data

Click the **Calculate** icon on the control panel.



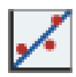
The Analysis Summary area displays the parameters of the growth model. Click the **Detailed Summary** icon to view more information.



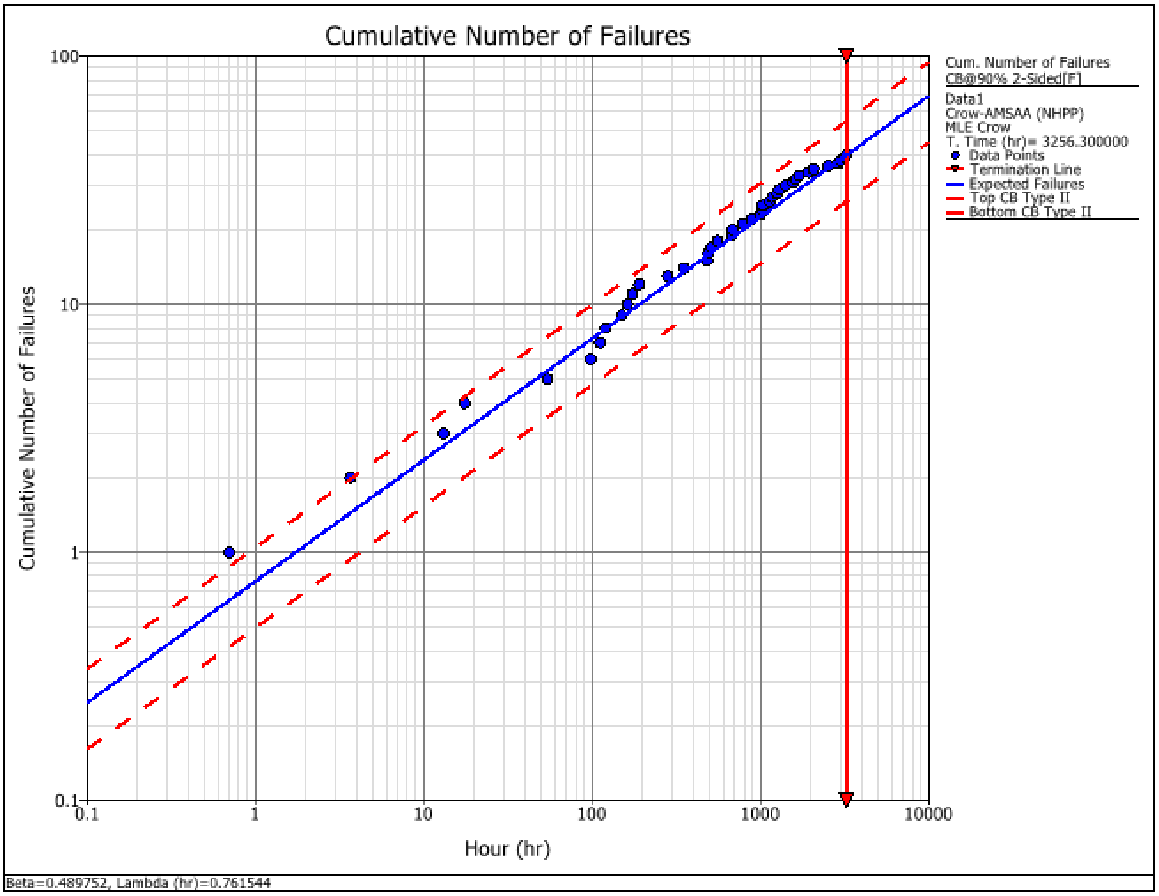
Results	
Parameters	
Beta	0.489752
Lambda (hr)	0.761544
Growth Rate	0.510248
DMTBF (hr)	166.221749
DFI	0.006016
Statistical Tests	
Significance Level	0.1
CVM	Passed
Other	
Termination Time (hr): 3256.300000	

4. View plots

Click the **Plot** icon on the control panel to open the plot sheet. (See QCP Calculations and Plots for Traditional RGA.)



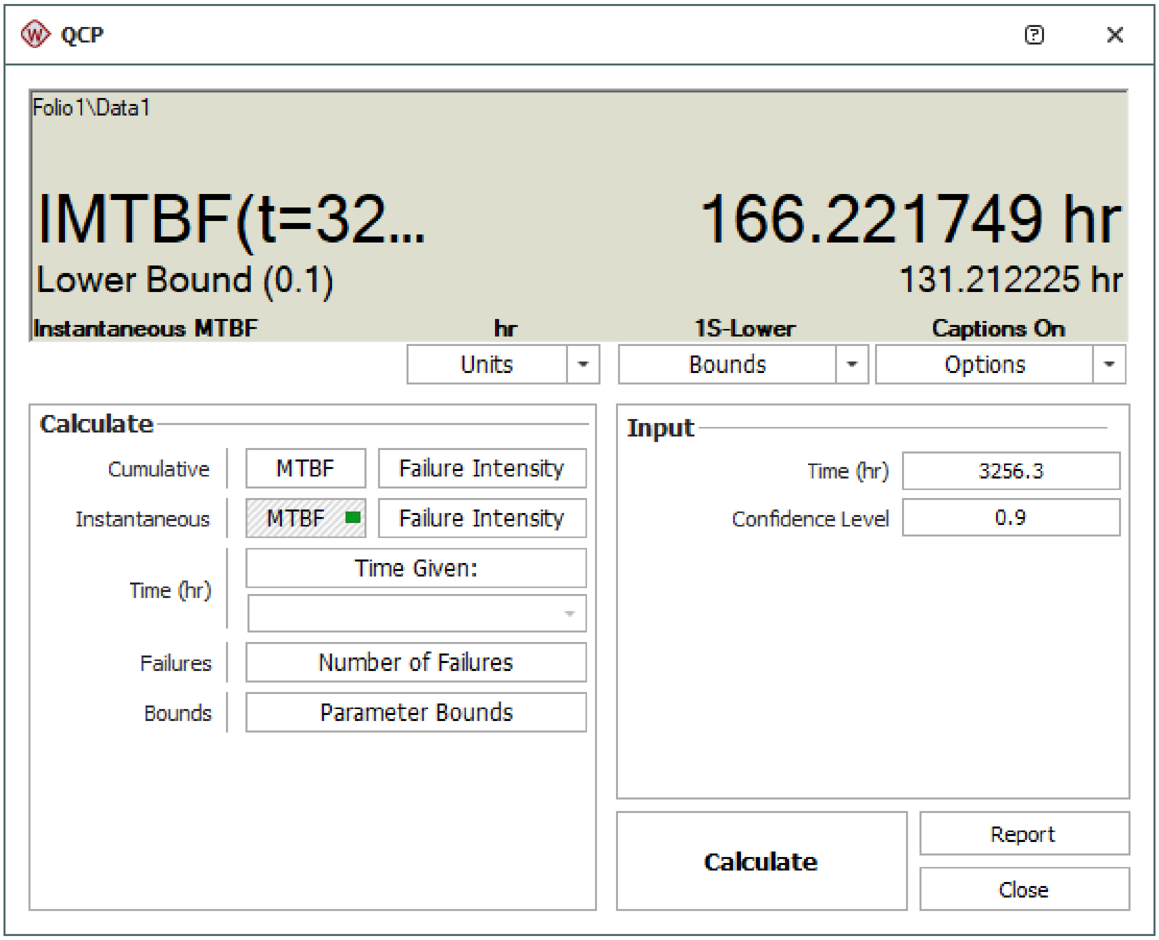
To show the confidence bounds, right-click the plot, choose **Show Confidence Bounds**, and then set the desired type of bounds and confidence level.



5. Calculate metrics

Click the **QCP** icon on the control panel to open the Quick Calculation Pad. (See QCP Calculations and Plots for Traditional RGA.)

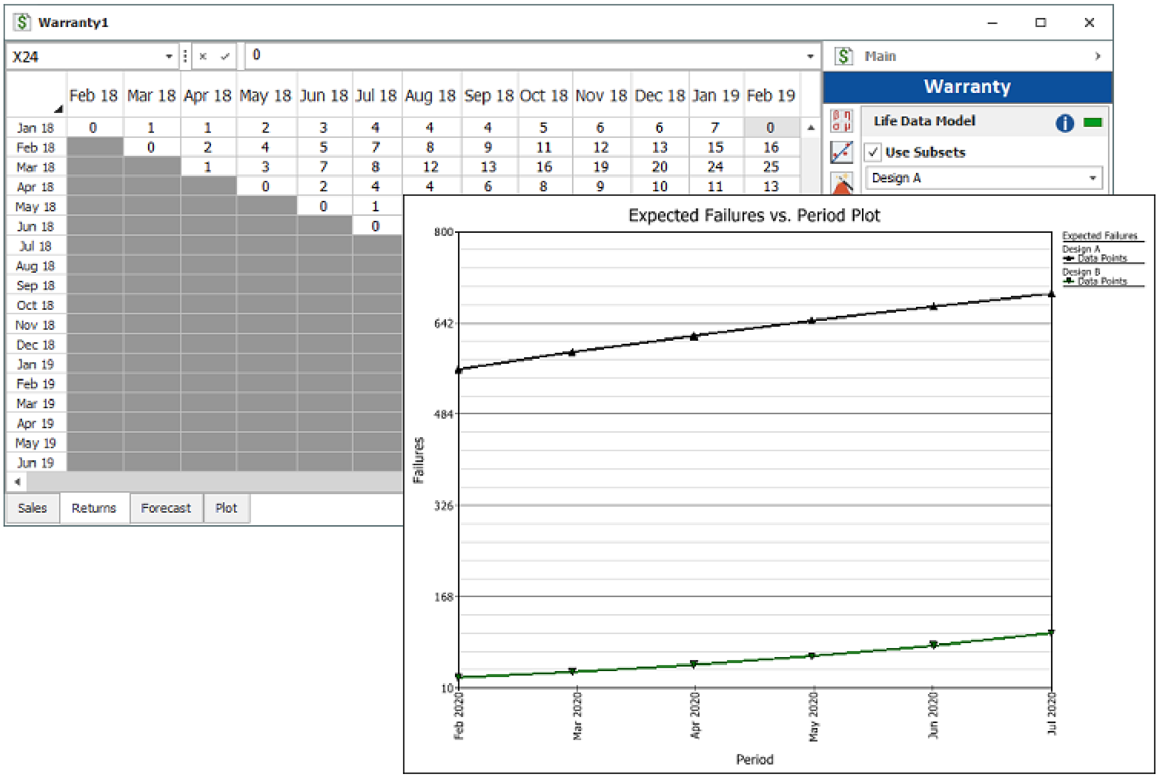
The following example shows the instantaneous MTBF with the lower one-sided 90% confidence bound.



More Analyses and Tools

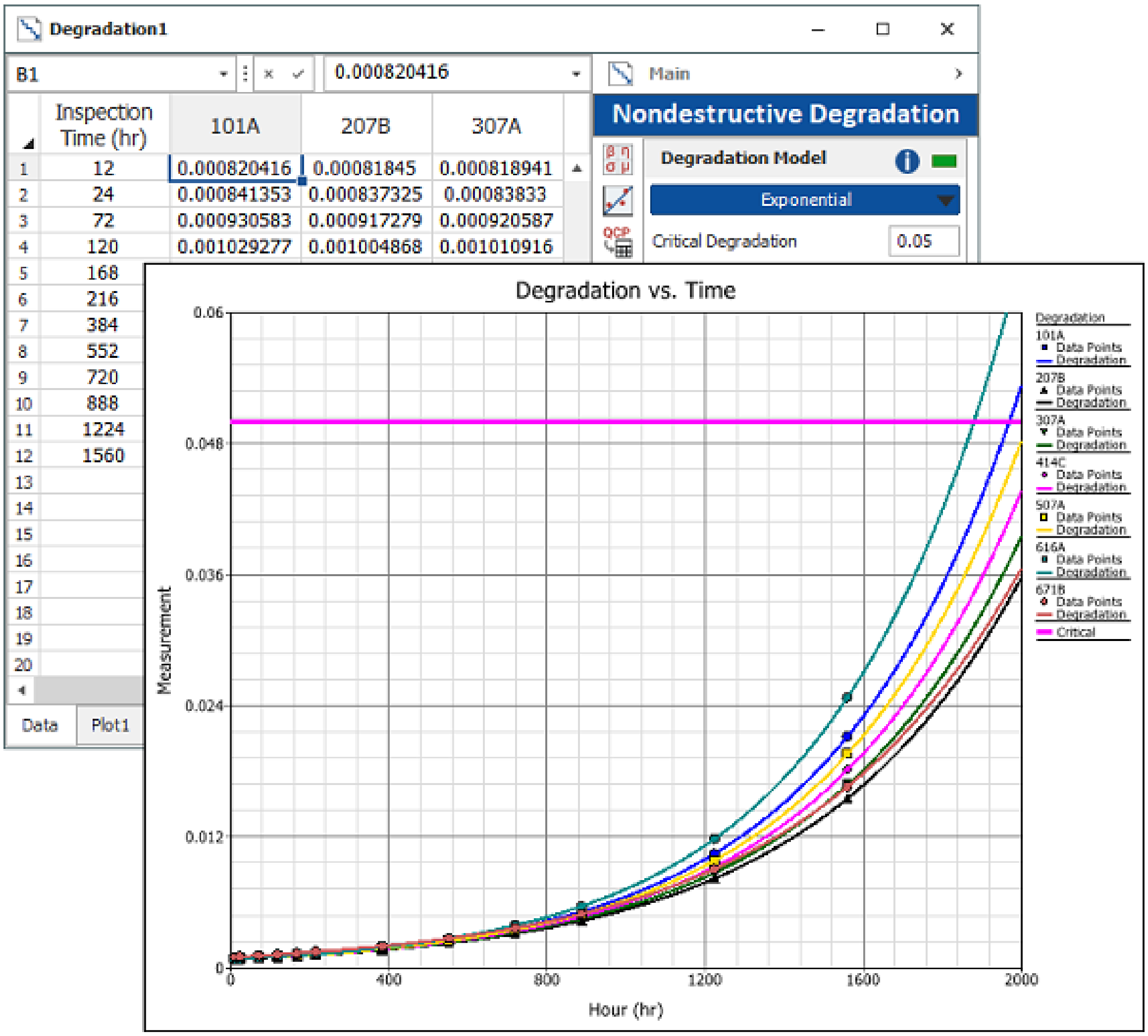
Warranty Data Analysis

For life data analysis, the warranty folio offers a choice of 4 data entry formats that use sales and returns data to predict future returns under warranty: Nevada, Times-to-Failure, Dates of Failure and Usage.


Warranty1
Main
Warranty
Life Data Model
Use Subsets
Design A
Sales
Returns
Forecast
Plot
Expected Failures vs. Period Plot
Failures
Period

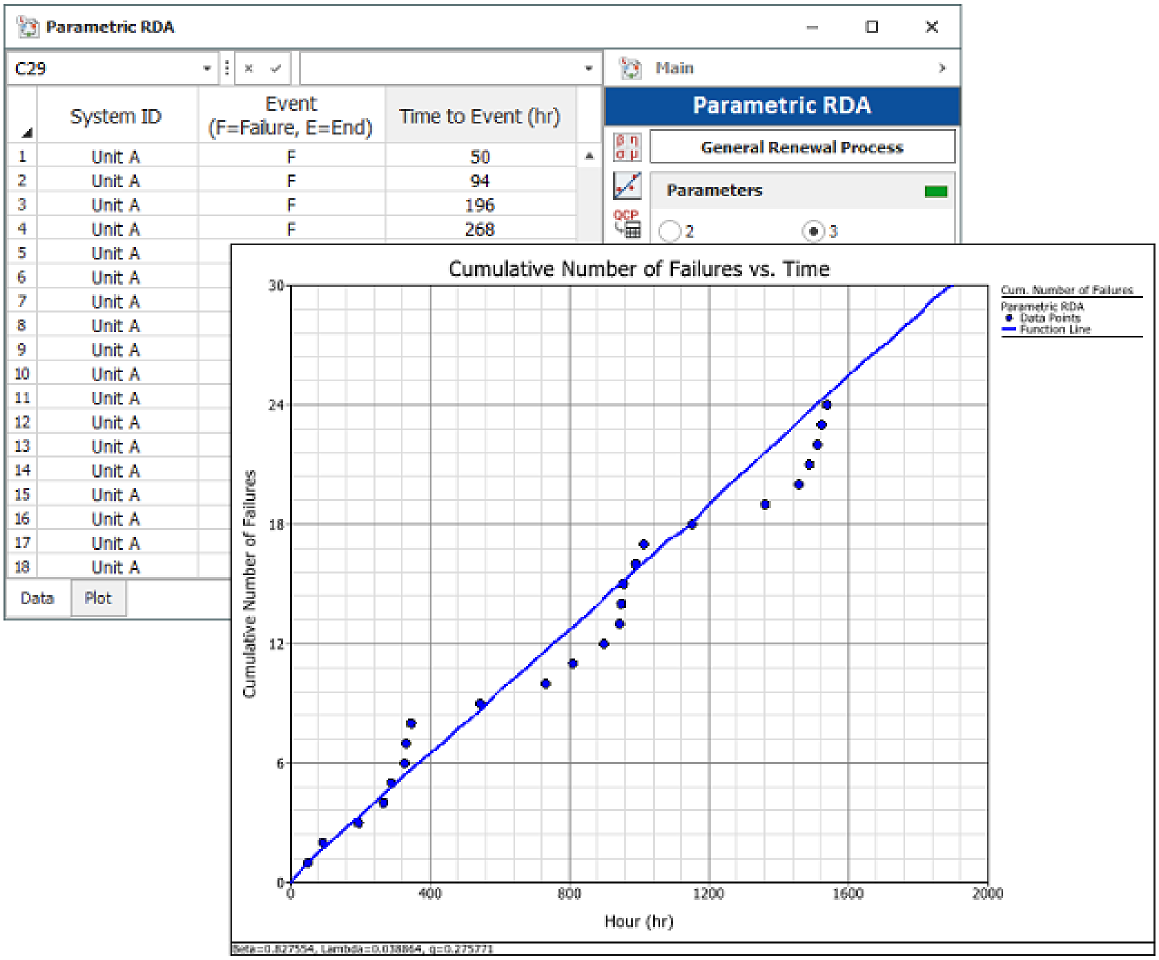
Degradation

For life data and life-stress analyses, the degradation folio extrapolates expected failure times based on how a performance measure (e.g., crack propagation, tread depth, vibration) has degraded over time.



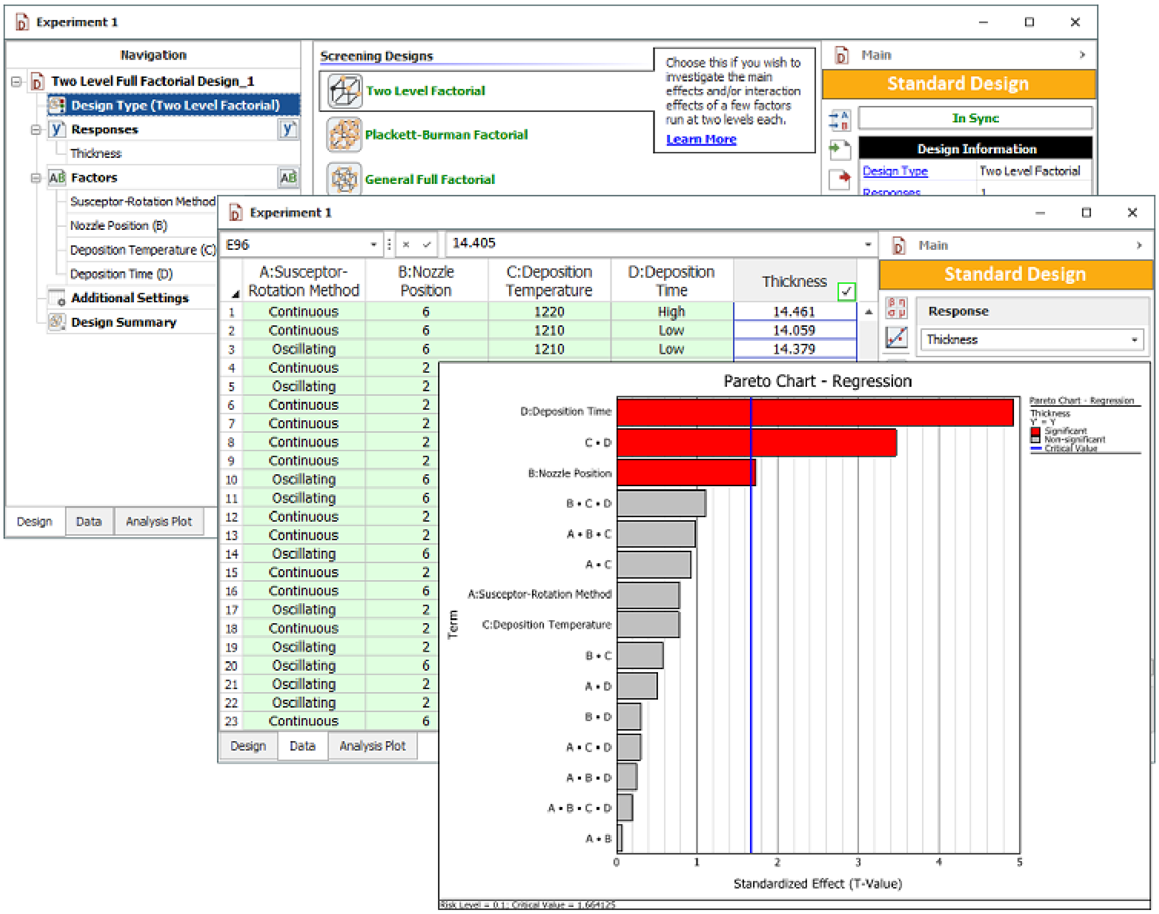
Recurrent Event Data Analysis

For life data analysis, the non-parametric RDA folio uses the Mean Cumulative Function to plot the average number of recurring failures over a given period of time. The parametric RDA folio uses the General Renewal Process (GRP) model, which considers the effectiveness of repairs, to model the cumulative number of failures over time.



Design of Experiments

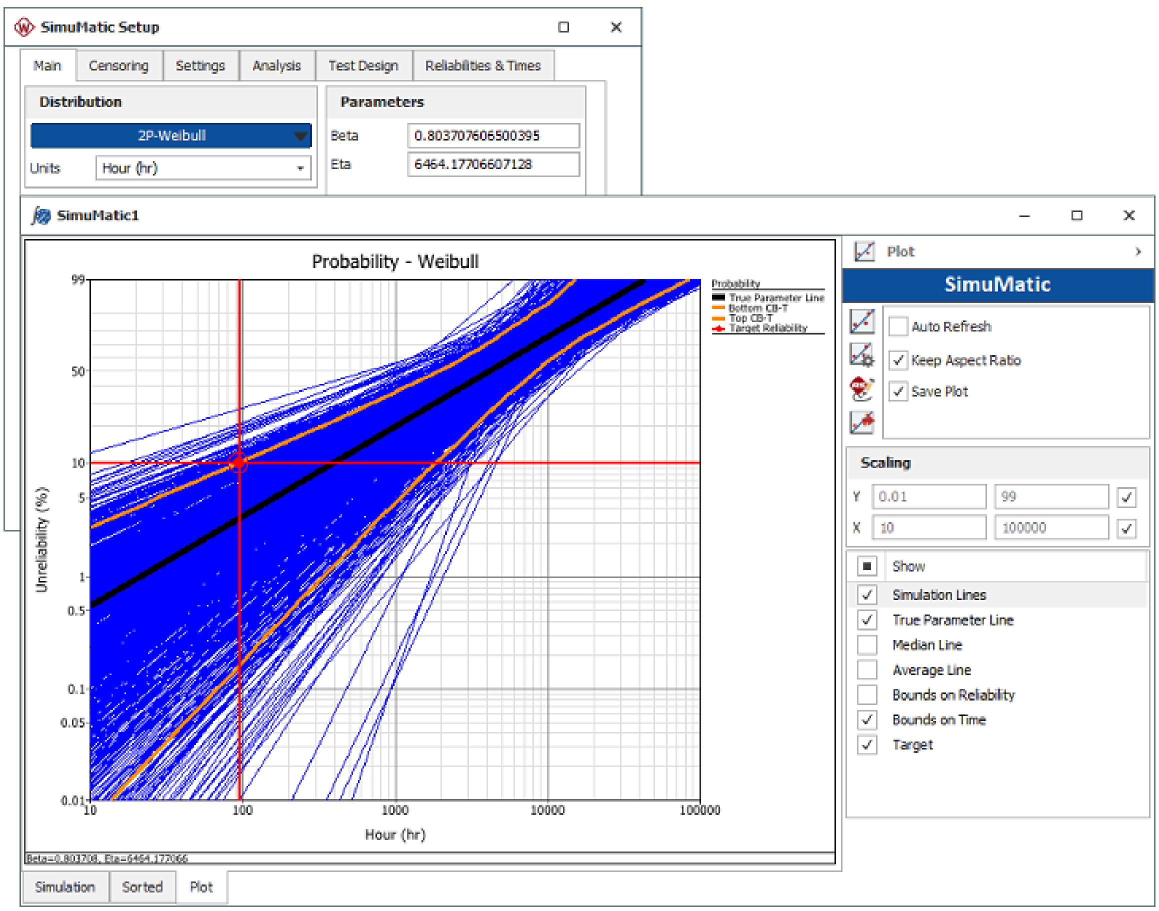
A set of design folios provide techniques for DOE analyses such as identifying key factors that affect performance (standard design), minimizing a system's sensitivity to noise (robust design) and determining the best proportion of components in a mixture (mixture design).



Monte Carlo Data Generation and SimuMatic

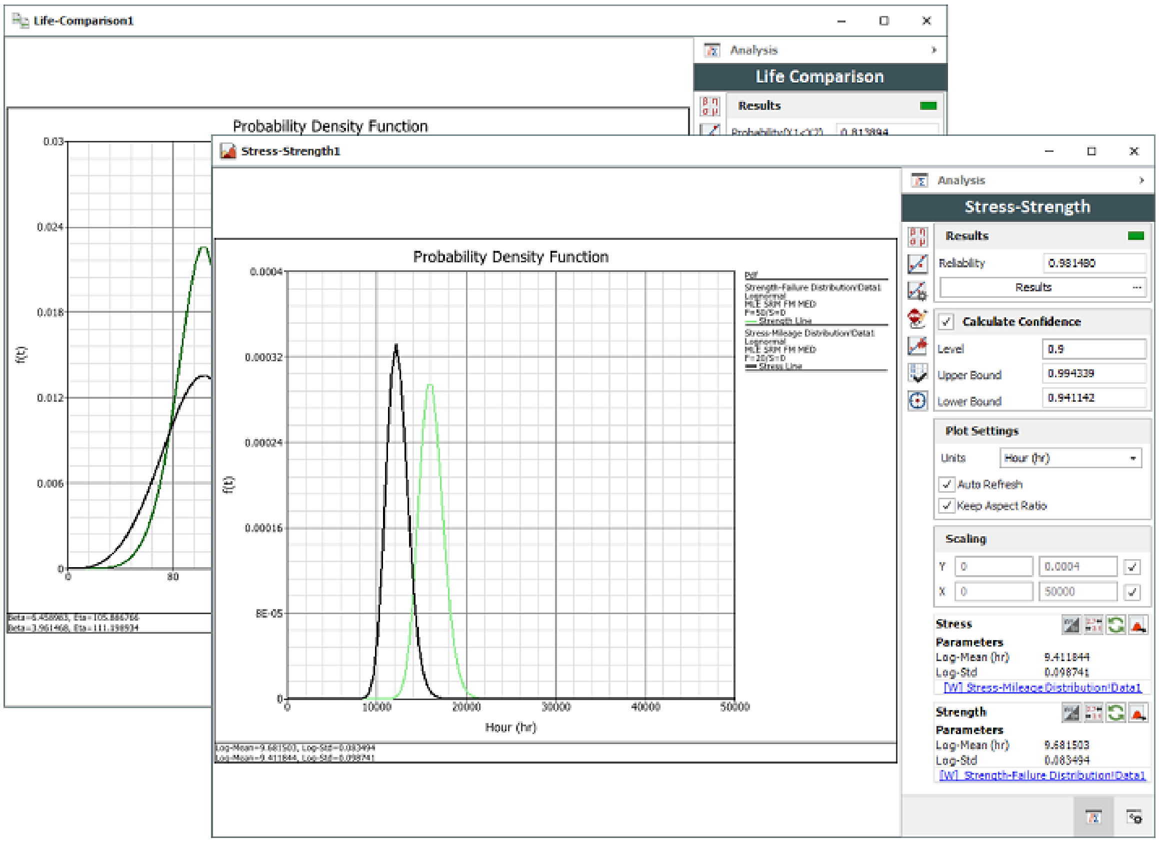
For life data and life-stress data analyses, the Monte Carlo utility generates a data set based on a specified probability distribution. SimuMatic performs analyses on a large number of simulated data sets, based on a specified probability distribution. For growth data analysis, the Monte Carlo utility generates a data set based on the Crow-AMSAA model, with specified beta and lambda parameters while SimuMatic performs analyses on a large number of simulated data sets, based on the Crow-AMSAA model.

The life data versions look like this:



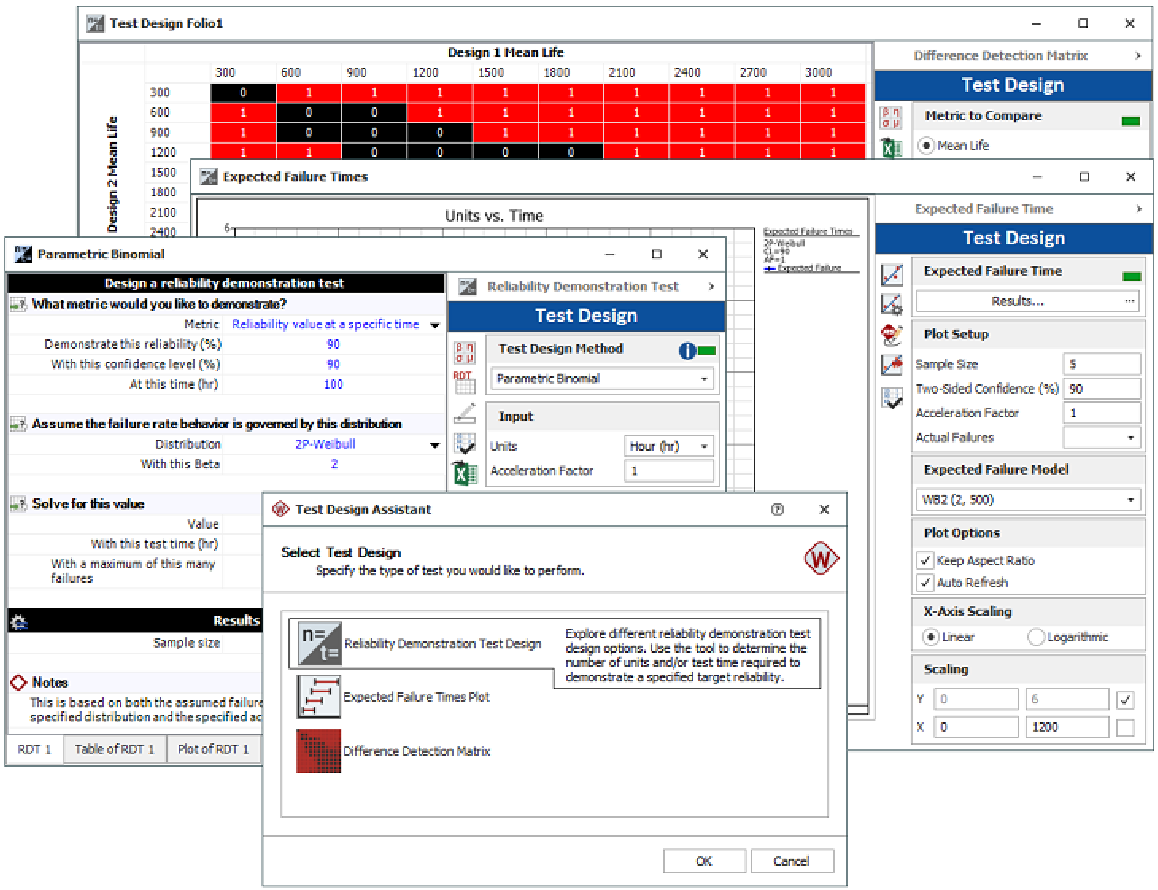
Comparison Folios

The life comparison folio determines the probability that the life of one population will be greater or less than that of the second. The stress-strength analysis folio determines the probability of failure based on the probability of stress exceeding strength.



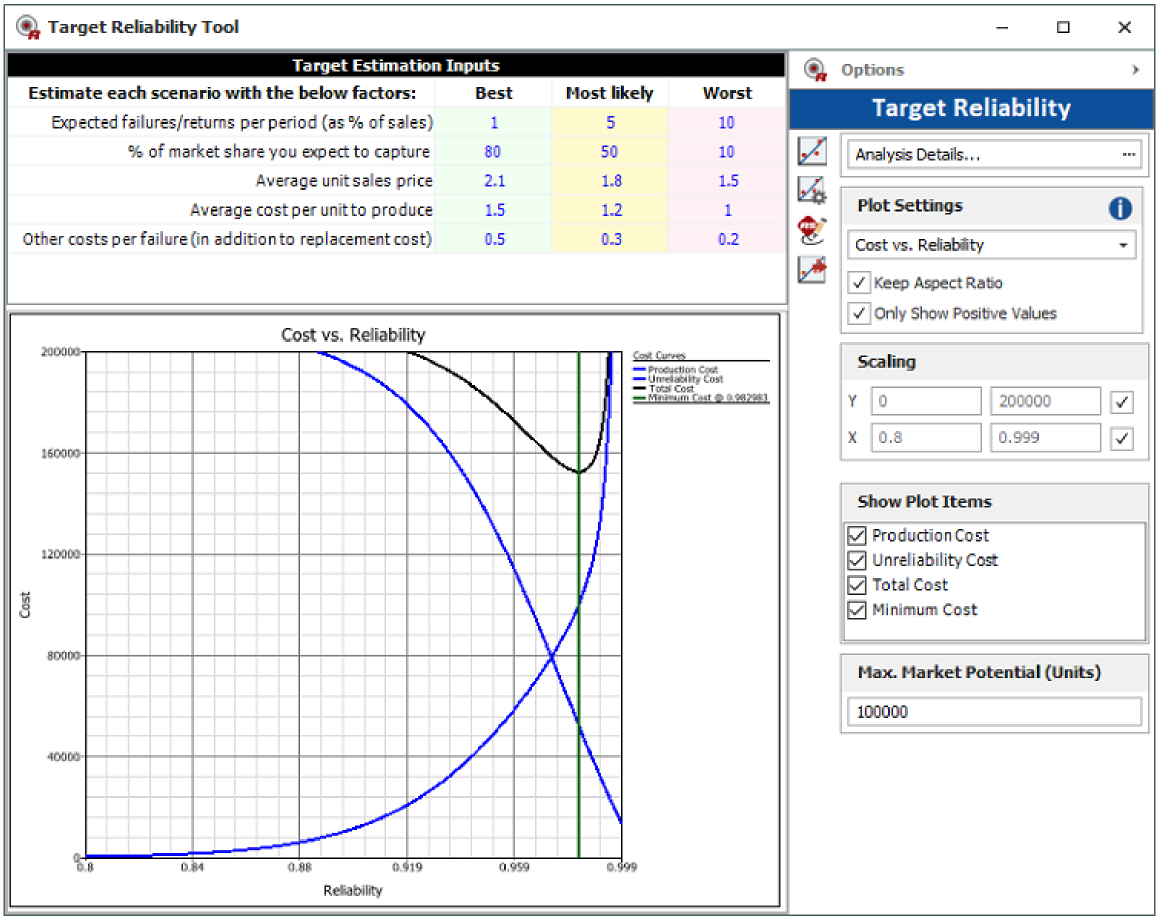
Test Design

For life data analysis, Weibull++ offers a choice of tools to assist with reliability test design: Reliability Demonstration Test (Parametric Binomial, Non-Parametric Binomial, Exponential Chi-Squared or Non-Parametric Bayesian), Expected Failure Times Plot and Difference Detection Matrix. For growth data analysis, the Test Design folio uses the non-homogeneous Poisson process (NHPP) to design a demonstration test for repairable systems by solving for either the test time required per system or the number of systems that must be tested.



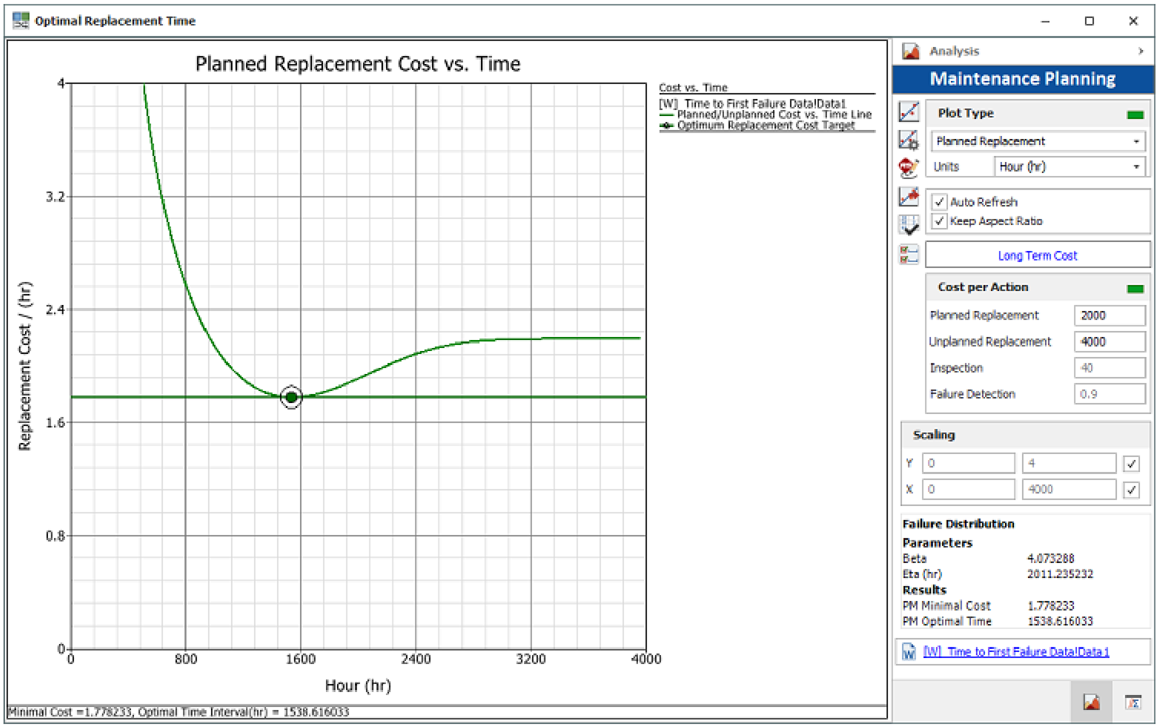
Target Reliability Planner

For life data analysis, the Target Reliability folio finds the optimum cost and reliability of a product, given estimates for various factors involved in the product's production and sale.



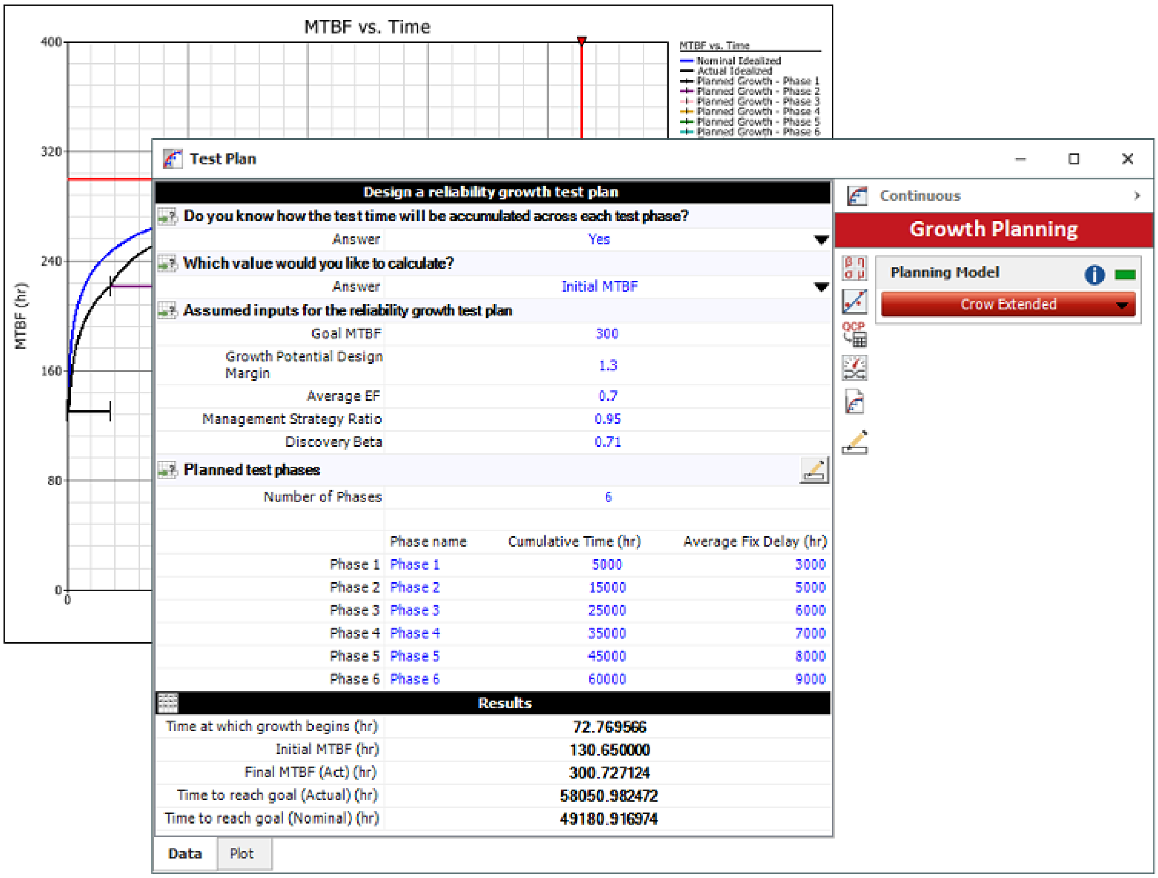
Maintenance Planning

For life data analysis, the Maintenance Planning folio uses the optimum replacement age model to determine the most cost-effective time to replace aged components.



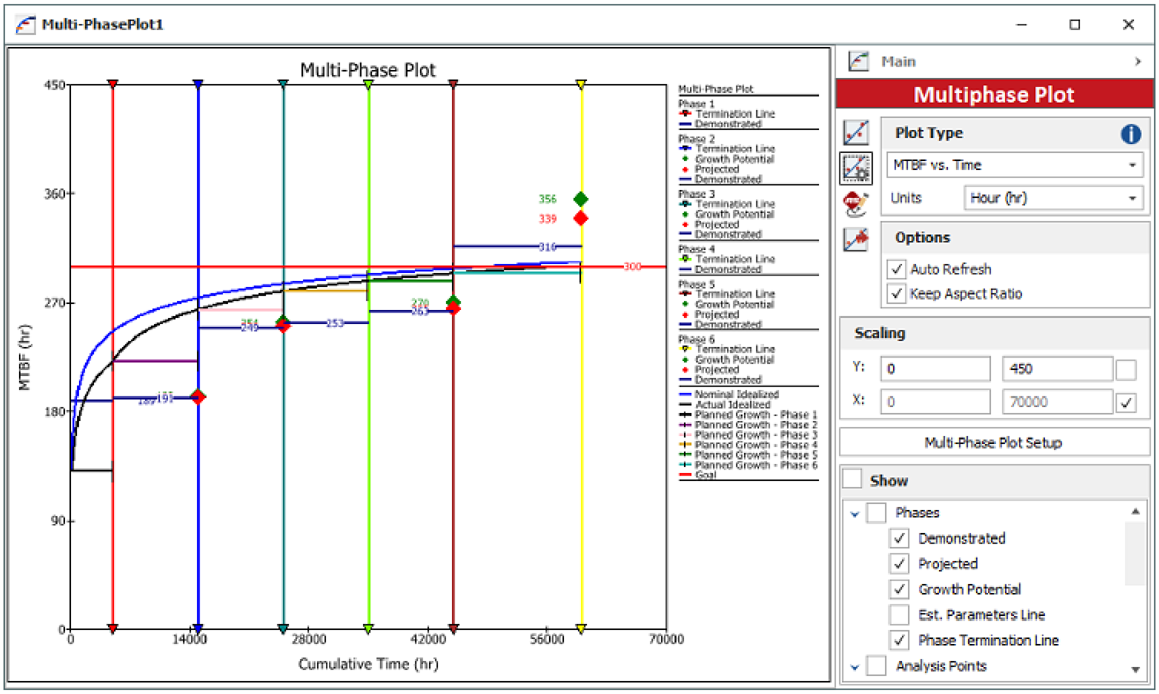
Growth Planning

For growth data analysis, the growth planning folio helps you to create a multi-phase reliability growth testing program that is designed to achieve a specific MTBF or reliability goal.



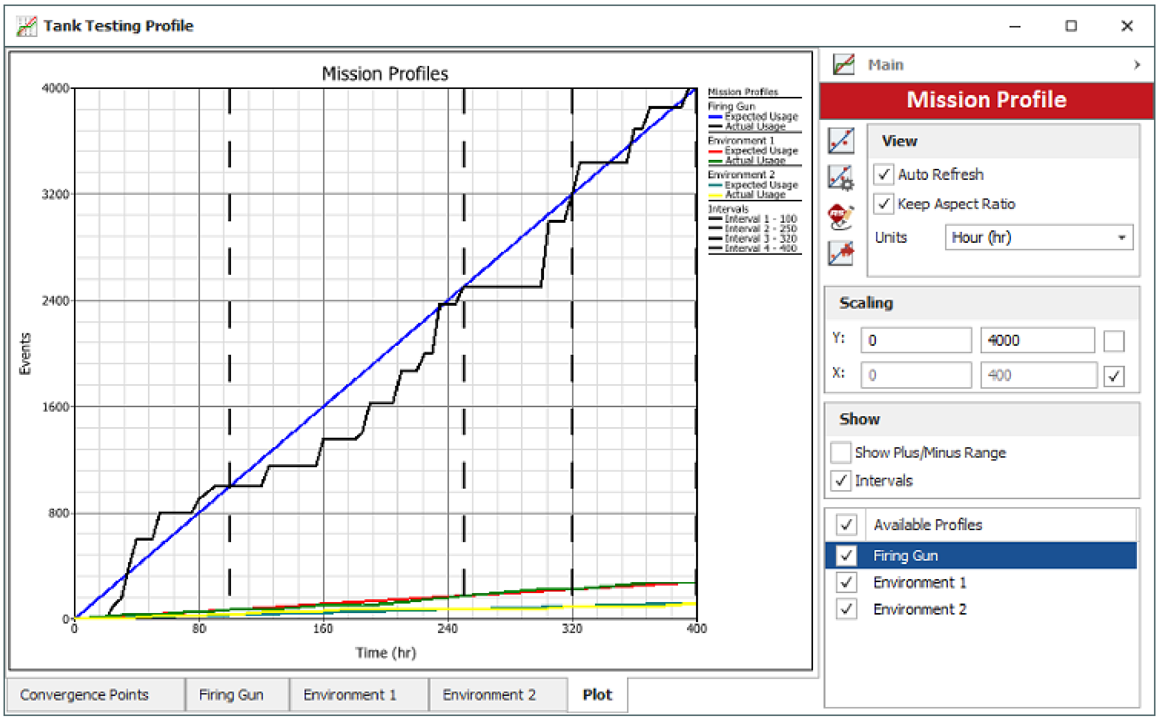
Multi-Phase Plot

For growth data analysis, the multi-phase plot displays actual test results (e.g., demonstrated, projected and growth potential MTBF or failure intensity) across multiple analysis points and/or test phases. It can be linked to a growth planning folio to compare actual test results against the test plan.



Mission Profiles

For growth data analysis, mission profiles help to create a balanced operational test plan and track the actual testing against the plan to make sure the data will be suitable for reliability growth analysis.



Technical Support

ReliaSoft reliability software products are renowned for their ease of use and unparalleled after sale support. For users with an active maintenance agreement, we provide technical support for software-related issues via a network of regional offices and partners/distributors throughout the world.

You can request assistance directly from within the software by choosing **File > Help > E-mail Support**. This 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.

To get contact details for technical support worldwide, choose **File > Help > Contact Technical Support** or visit https://support.hbmprenscia.com.

When Requesting Support

When you contact us 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. 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.
- Your product license key.
  - To determine the license for the application on your computer, you can choose **File > Application Setup** and click any of the main headings in the Application Setup window.
- The operating system (e.g., Windows 10), 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: 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 Reliability Consulting Services. HBM Prenscia 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 https://www.reliasoft.com/services/training-courses.

Reliability Consulting

If your organization does not have sufficient time, expertise or objectivity in-house to accomplish specific reliability goals, turning to our 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, Reliability 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 global resources, expertise and contacts**.
- Our **team-based approach to consulting**, combined with ReliaSoft 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 https://www.reliasoft.com/services/reliability-maintainability-engineering-services for more information.

Install and License

System Requirements

In addition to the information below, we provide an automated tool that determines whether your computer and Internet connection meet the requirements for running and activating ReliaSoft desktop applications. It will also determine whether you could install the 64-bit version of these applications, if desired. [Download Requirements and Compatibility Test Tool...]

Operating Systems

64-bit versions are supported, except where noted. [See our 32-bit vs. 64-bit FAQ...]

- Microsoft Windows® 10.
- Microsoft Windows Server 2008 R2 SP1 (64-bit version) or newer. (Requires a license that specifically allows server installations/deployments.)

Note: Upgrading your operating system after your ReliaSoft software has already been installed can cause issues with the activation. To prevent this, it is recommended to uninstall ReliaSoft 2021 prior to upgrading your operating system and then reinstall it after the upgrade. For other options, and for information about how to handle the "activated license is corrupt" message that may appear if you upgrade your operating system with live activations, please refer to https://www.reliasoft.com/windows10-reactivation.

Minimum Hardware Requirements

- RAM: 2 GB or more
- Hard disk space: 1 GB (minimum) free space available
- Screen resolution: 1024 x 768 or higher

Additional Software Requirements

- .NET Framework 4.6.1
- Microsoft Office (Excel®, Word and Outlook®) 2010 or higher (non-subscription-based) for automated exports, report generation and e-mail/calendar integration.

Enterprise Database Platforms

32- and 64-bit versions are supported.

- SQL Server® 2012 - 2017
- SQL Server Express Edition 2012 - 2017
- Oracle® 10g - 12c

Although the enterprise database platform (Microsoft SQL Server or Oracle) could be installed on the same computer where the ReliaSoft 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 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, we recommend a minimum CPU speed of 2 GHz and 4 GB of RAM.

Installation and Licensing

There are two alternative ways that ReliaSoft desktop applications may be licensed on your computer:

- If you received a license file (*.lic) and/or a license settings file (*.prnrsl), see Locally Hosted Licensing
- If you received only a license key (a 32-digit alphanumeric code), see ReliaSoft-Hosted Licensing

One or both methods can be used on the same computer. For example, if you have your own single-user license for BlockSim and will share your department's new multi-user (floating) license for Weibull++, you will receive a license key for BlockSim and a license file for Weibull++.

If you will use both types of licensing on the same computer, use the installation instructions for locally hosted licenses. When you launch any application that isn't included in your locally hosted license or that you have specified as using ReliaSoft-hosted licensing via the License Settings Tool (see https://www.reliasoft.com/license-settings-tool), the ReliaSoft License Manager will allow you to perform the activation of your ReliaSoft-hosted license(s) using a product license key.

Requirements for All Installations

- Before running the installation program, make sure your computer meets the system requirements for ReliaSoft desktop applications.
- 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.

Locally Hosted Licensing

If you received a license file (*.lic) or license settings file (*.prnrsl), you will be using locally hosted licensing. This allows you to manage available license seats or CRS units in-house without the requirement to connect to the ReliaSoft external license server.

The first step is for a license administrator to configure your organization's license server and provide each individual user with the path to the license server/host (e.g., 6220@servername) and, in some cases, a license settings file (*.prnrsl) that specifies which ReliaSoft applications on your computer will use locally hosted licensing. (For detailed instructions, see https://www.reliasoft.com/locally-hosted-licensing.)

Once the license server has been set up, perform the following steps for each client installation. If you have previously connected the client computer to the appropriate license server, you can start at step 4.

1. Open the ReliaSoft ePack and decompress the *.zip archive in the Locally Hosted Licensing Utility folder
2. Install the HBM Prenscia Licensing Administration utility by double-clicking licinst.exe and following the prompts. The utility will start as soon as installation is complete.
3. On the Client tab of the license admin utility, click **Set Client License Path** and then click **Add Network Host** or enter the license path directly in the window (use the network host format; e.g., port@serverhostname).

   The client computer will check for available license seats in the specified order (i.e., it will go to the top server listed and if no license seat is available there, it will check the next server down, and so on).

   Note that you can delete a license location by highlighting the line and deleting it.
4. Install the ReliaSoft desktop applications by double-clicking the installation program from the ePack (e.g., ReliaSoft2021.exe) and following the prompts.

The first time you launch an application, you may be asked to provide your name and basic contact information. This is required to be eligible for technical support.

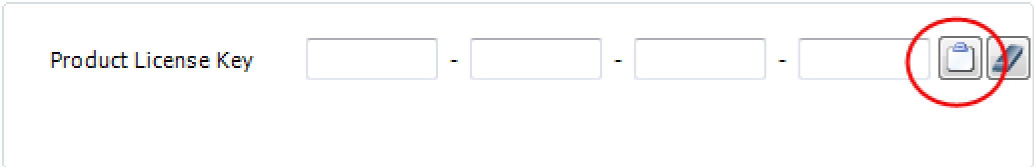
Note: Applications using locally hosted licensing must have an active connection to the license server unless you have borrowed a license. If you see a message that indicates that a license could not be obtained, this could be for a variety of reasons (e.g., you are not connected to the license server, the license server is down or there are no seats or CRS units available for the license). In addition, if you lose connection to the license server while you are using an application, you will see a message warning you to save your work before the application closes.

ReliaSoft-Hosted Licensing

If you are requesting a demo or have already received a license key (a 32-digit alphanumeric code), you will be using ReliaSoft-hosted licensing.

If your computer has an active Internet connection and your firewall allows the software to access the ReliaSoft secure external license server at https://validate.reliasoft.org, only a few simple steps are required for activation:

1. Install the ReliaSoft desktop applications by double-clicking the installation program from the ePack (e.g., ReliaSoft2021.exe) and following the prompts.
2. 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 request a free demo license.
3. 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. This must be your company e-mail address — not gmail, hotmail, etc.
    - **Name and Contact Info**: If this is the first time that you have registered a ReliaSoft desktop application on this computer with this e-mail address, the next page will request your name and basic contact information.
    - **Demo License**: If you need a demo license, most requests will be addressed immediately during normal business hours; for some locations, please allow up to 2 business days.
4. On the next page, enter the product license key provided to you. *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.*



- **Confirmation Keys**: If your license type requires a confirmation key, the next page requires you to copy/paste a key that you will receive 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 ReliaSoft desktop 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.

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

If the computer does not have an active Internet connection, or if you encounter issues with the firewall, there are other ways to complete the activation. Please visit https://www.reliasoft.com/reliasoft-hosted-licensing for details or consult the "ReliaSoft Installation and Licensing" document included with the software. And, of course, you can always contact us for assistance (https://support.hbmprenscia.com).

Uninstalling ReliaSoft Applications

If you wish to stop using all of the ReliaSoft desktop applications on this computer, you can perform the following steps:

1. Make sure you do not have any ReliaSoft desktop applications currently open.
2. For locally hosted licenses prior to Version 2020, if you are uninstalling permanently (i.e., if you are not doing this as part of an uninstall/reinstall), open the Import/Export License Settings utility by choosing **ReliaSoft [Version] > ReliaSoft [Version] License Settings Import-Export** (Windows 10) in the Windows Start menu. Follow the on-screen instructions to delete the license settings file.
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 [Version]** and follow the prompts.
5. If you have installed the HBM Prenscia Licensing Administration tool, you can follow similar steps to uninstall it.

Tip: If you are using ReliaSoft-hosted licensing (i.e., if you received a license key and activated specific application(s) on your computer), you also have the option to deactivate one or more applications without performing a complete uninstall. For instructions, see https://www.reliasoft.com/deactivating-applications.

License Manager

The License Manager allows you to view and manage license-related details for the ReliaSoft desktop applications that are currently activated on your computer. You can use this interface to:

- View and, for ReliaSoft-hosted licenses, edit the **contact information** that is on file with us or **change the password** associated with your ReliaSoft ID. [Learn more…]
- For ReliaSoft-hosted licenses, **deactivate the current application** (if you don't plan to continue to use it on this computer or for this user). [Learn more…]
- For ReliaSoft-hosted floating licenses, **check out or check in a license**. [Learn more...]

To open the License Manager, choose **File > Help** and click the **License Manager** link in the Licensing section.

You may 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**.)

Contact Information and Password

The **Contact Information** page displays the name, phone number and other details that are on file with ReliaSoft. For locally hosted licenses, this information is read-only and can only be edited by uninstalling and reinstalling the software. For ReliaSoft-hosted licenses:

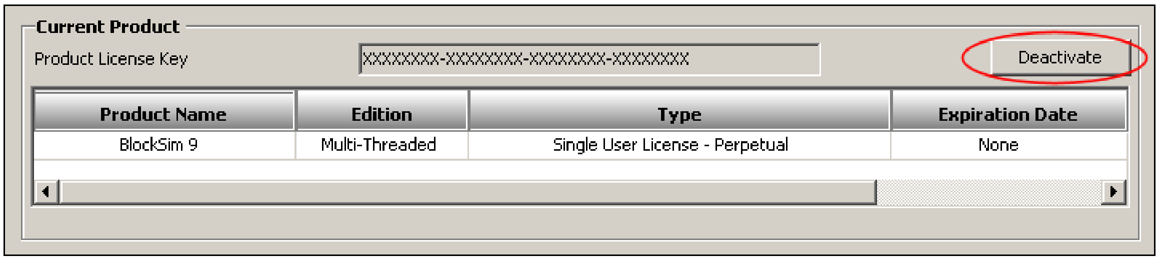
- If you need to download the latest information from the license server, click **Synchronize**.
- If you want to change your password or any of the current contact information, type the new information into the fields on this page and then click **Update Information on Server**.

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

For ReliaSoft-hosted licenses, if you don't plan to continue to use the current ReliaSoft 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.

Borrowing Licenses

For certain license types, your computer must be able to connect to a license server to determine whether shared license seats (or, for token-based licenses, the required number of CRS units) are available when you attempt to launch a ReliaSoft application.

If you need to use the application(s) when your computer is not connected to the license server, you can either "borrow" or "check out" the licenses (depending on the license type). This reduces the number of seats (or CRS units) available to other users until the licenses are returned. (For ReliaSoft-hosted licenses, you can also use this feature to make sure a license seat will be available when you need it.)

Borrowing Locally Hosted Licenses

If you have a locally hosted license, you can use the ReliaSoft License Borrowing utility to borrow licenses for selected product(s) for a specified duration.

IMPORTANT: If at least one ReliaSoft product with a locally hosted license is borrowed, you will only be able to start the product(s) that are borrowed. All desired products must be borrowed or returned at the same time. Note that in versions prior to 18.0.3, you must also be disconnected from the license server to use the borrowed product(s).

If any of the ReliaSoft applications on your computer use a ReliaSoft-hosted license (i.e., a license key rather than a license settings file), they will be unaffected by locally hosted license borrowing. For those products, see Checking Out ReliaSoft-Hosted Licenses below.

BORROW LICENSE(S)

Before attempting to borrow licenses, first make sure that your computer is connected to the license server and shut down all ReliaSoft desktop applications.

1. To open the ReliaSoft License Borrowing utility, navigate to the **ReliaSoft 2021 > Additional Tools** folder in the Windows Start menu and click **ReliaSoft 2021 License Borrowing**.
2. The window shows all of the ReliaSoft desktop applications on your computer that use locally hosted licensing. Specify the borrow duration and select which product(s) you want to borrow. (Remember that all desired products must be borrowed or returned at the same time.)

   Starting in Version 2020, this list also includes any licensed features (e.g., Accelerated Life testing [ALTA] in Weibull++, Process Flow Simulation in BlockSim, etc.). When you select a feature, the parent application is automatically selected; clearing the parent application will also clear the feature.
3. Click **Borrow Selected Licenses**.
4. Close the window.

You will only be able to use the product(s) that are borrowed. Additionally, for versions prior to 18.0.3, you can use the borrowed products only when the computer is not connected to the license server. If you need to change which products are borrowed, you can reconnect to the license server (if necessary) and repeat the same steps again (this will restart the borrow duration for all selected products). If you need to be able to use other (unborrowed) applications, you can return the licenses as described below.

What's Changed? Starting in version 18.0.3, you can use the products you have borrowed while you are connected to the license server.

RETURN LICENSE(S)

Before attempting to return licenses, shut down all ReliaSoft desktop applications and make sure your computer is connected to the license server.

1. Open the ReliaSoft License Borrowing utility.
2. Click **Return All Licenses**.
3. Close the window.

Note: Failure to shut down all ReliaSoft desktop applications prior to borrowing or returning licenses may result in unexpected behavior, such as inability to borrow, applications quitting unexpectedly or double use of license credits.

You can borrow licenses only while your license is valid (e.g., if the annual expiration date of your locally hosted license is 2 days from now, you cannot borrow a license for a duration of 3 days). In addition, you cannot borrow licenses during the 24 hours prior to annual license expiration.

Checking Out ReliaSoft-Hosted Licenses

If you have a ReliaSoft-hosted floating license, you can use the ReliaSoft License Manager to reserve a license.

1. Open the ReliaSoft application.
2. Open the License Manager by choosing **File > Help** and clicking the **License Manager** link in the Licensing section.
3. In the current product area on the **Products** page, click **Check Out** or **Check In**.

Databases and Projects

IN THIS CHAPTER

The ReliaSoft desktop applications use 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.

Your organization (or team) will need to choose the most appropriate database type and security, implement shared settings that will effectively facilitate project management, and establish adequate backups and database maintenance to protect against data loss. In addition, individual users must be aware that changes are saved automatically as they work, and become familiar with the features that facilitate simultaneous access by multiple users.

This chapter discusses some of the basic requirements for using ReliaSoft applications and projects. For information about other features, see ReliaSoft Common Tools.

What's Changed? In versions prior to Version 2020, "databases" were known as "repositories."

ReliaSoft Databases

Database Types

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

 **Standard databases** (*.rsr21) are basic database files for single- or multi-user environments. Implementation of a database server is not required; however, there are limits on the amount of data and the number of simultaneous users.

 **Enterprise databases** require implementation and support of Oracle or Microsoft SQL Server, but they are a more robust platform that can store much more analysis information in the same database and support access by many more simultaneous users. ReliaSoft applications are compatible with Microsoft SQL Server 2008 or later and Oracle 10g or later (including the free Express editions of all of these). They can be accessed via an enterprise database connection file with the extension *.rserp.

What's Changed? Starting in Version 2019, Synthesis files (*.rsf) can no longer be created but you can still open Synthesis files created in prior versions. Such files will be converted to standard databases upon opening.

Standard Databases

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

Creating a New Standard Database

To create a new standard database, choose **File > New > Standard Database**.

In the **Database name** field, specify the filename for the new *.rsr21 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:

- **Apply login security** configures the new database to be login secure. You can then select the **Open security window upon creation** check box if you want to add user accounts as part of the database creation process. Note that you cannot automatically remove security from a database once it has been enabled. However, you can create a new non-secure database and use the **Import from existing database** check box to automatically import all of the data from the secure database to the non-secure one.

- **Import from existing database** allows you to specify an existing database from which to import data when the new database is created. You can select to import projects, database settings, security-related information, restore points and more.

Opening an Existing Standard Database

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

Otherwise, choose **File > Open Database** and then browse for the *.rsr21 file.

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

- **No access to the database**. If you have not been given access to the database, you will see a message stating that your account in the database is not active or not assigned to at least one security group. You will need assistance from someone who can create and update user accounts (see Managing User Accounts).

- **Windows authentication failed**. If your Windows login (domain and username) is different from what was specified for your user account, you will see a message to connect using alternative credentials. You will need assistance from someone who can enable the use of alternative credentials for your user account.

Upgrading a Standard Repository from a Previous Version

If you have a standard repository that was created in a previous version of the software, you can simply open it like any other standard database (**File > Open Database**). An upgraded copy will be created, with "_V21" appended to the filename. For example, if you open "MyDatabase.rsr10," the Version 2021 database that is created will be called "MyDatabase_V21.rsr21."

It is important to remember that upgrading a standard repository will not automatically upgrade any library or template files that you may have been using in conjunction with that database, as these file types are stored separately. Please refer to the documentation on those files for information on upgrading them.

IMPORTANT: For best results, the standard repository should NOT be open in the prior version. This ensures that all information in the repository will be converted to the new database.

Enterprise Databases

Unlike standard databases, enterprise databases require an established database server with Oracle or Microsoft SQL Server. This allows you to store more analysis information in the same database and support access by many more simultaneous users. ReliaSoft applications are compatible with Microsoft SQL Server 2008 or later and Oracle 10g or later (including the free Express editions of all of these).

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

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

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

The following topics will be of interest to database administrators:

- Creating a New Enterprise Database
- SQL Server Logins or Impersonation
- Upgrading an Enterprise Database from a Previous Version
- ReliaSoft Admin Tool

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

- Creating an Enterprise Database Connection File (*.rserp)
- Monitoring Connection Speed

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 databases on the server, you can create a new enterprise database by choosing **File > Manage Database > New Enterprise Database**.

Tip: If you need to be able to create an enterprise database without using a ReliaSoft desktop application (i.e., without taking up one of the available license seats), you can access this same feature from the ReliaSoft Admin tool.

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.
- Options:
  - **Encrypt communication** secures the connection information between the ReliaSoft application and the enterprise database.
  - **Trust server certificate**. Select this option if the server has a self-signed certificate.

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

- **Port**, **Host** and **Service** identifiers for the Oracle server where the new database will be created.
- The **Schema** of the new Enterprise database.
- The **Password** for the new enterprise database schema. If you are including special characters in the password, you must include quotes around each special character when entering the password.
- The administrative username and password for the Oracle server (entered in the **Admin Information** area).

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

Click **OK** to create the database. The database will not open automatically; you must connect to it. (See Connecting to an Existing Enterprise Database.)

Connecting to an Existing Enterprise Database

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

CREATING A CONNECTION FILE

To create a connection file, choose **File > New > Enterprise Database Connection File**.

Tip: If you need to be able to create a connection file without using a ReliaSoft desktop application (i.e., without taking up one of the available license seats), you can access this same feature from the ReliaSoft Admin tool.

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

Enter a name for the connection file, then choose the database type and version.

- For a SQL Server database:
  - Select **Encrypt communication** if you want to encrypt the connection between the application and the database
  - Select **Trust server certificate** if the server has a self-signed certificate.
  - Select **Use impersonation** if you want the 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 a Microsoft Active Directory® group that has a login. (See SQL Server Logins or Using Windows Impersonation.)

- For an Oracle database, enter the port, host and service identifiers, the database schema and the password for the enterprise database schema.
  - If you are including special characters in the password, you must include quotes around each special character when entering the password.

Note: If you want to encrypt the connection for an Oracle implementation, you must set the encryption type to either "requested" or "required" for the Oracle database. For more information, please consult the Oracle documentation (e.g., https://-docs.oracle.com/cd/B19306_01/network.102/b14268/asoconfg.htm#i1007808).

USING AN *.RSERP FILE TO CONNECT TO AN ENTERPRISE DATABASE

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

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

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

- **No access to the database**. If you have not been given access to the database, you will see a message stating that your account in the database is not active or not assigned to at least one security group. You will need assistance from someone who can create and update user accounts (see Managing User Accounts).
- **Windows authentication failed**. If your Windows login (domain and username) is different from what was specified for your user account, you will see a message to connect using alternative credentials. You will need assistance from someone who can enable the use of alternative credentials for your user account.
- **Cannot connect to the server or login failed**. Server-related issues may occur for several reasons, and you may see various messages pertaining to the situation. Common issues are:
  - You do not have a network connection or you may have entered the incorrect database name or server name in the connection file.
  - The SQL Server database resides in a different domain than the one you have used to log in to Windows and your network cannot recognize the domain name you have entered. Depending on your network configuration, you may be able to connect using either the "fully qualified domain name" (FQDN) or the IP address for the server.

- In SQL Server databases, a login issue may occur if your username is not associated with a SQL Server Login. (See SQL Server Logins or Using Windows Impersonation.)
- The server is not configured to allow remote connections, has certain firewall settings or is experiencing other issues.

Note: If you get an "error occurred when reading the connection file" message, the connection file may be corrupted or is using old encryption. You can create a new connection file by choosing **File > New > Enterprise Database Connection File**.

SQL Server Logins or Using Windows Impersonation

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

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

Your organization may choose to use any or all of these methods for your 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.

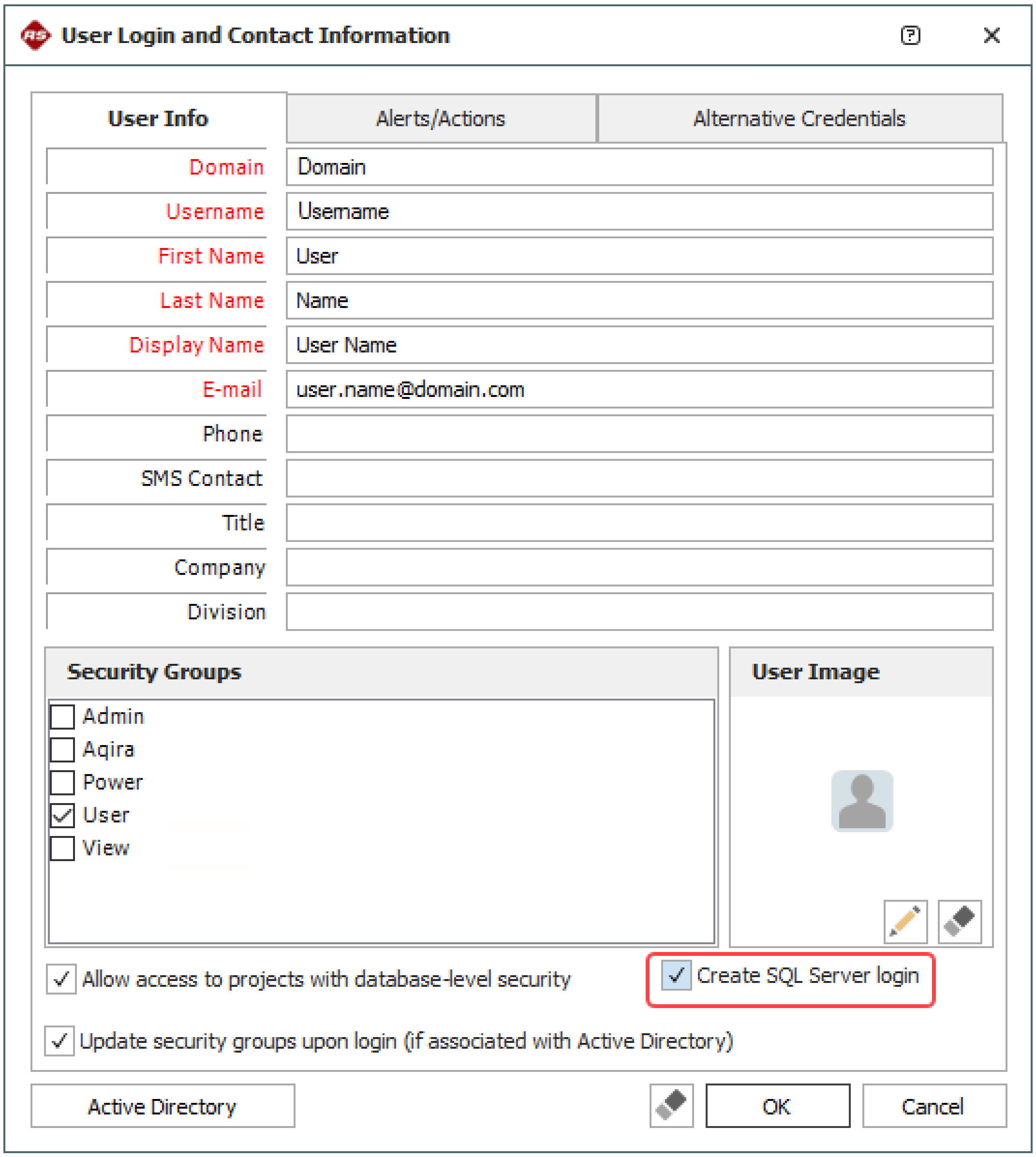
OPTION 1: CREATING INDIVIDUAL SQL SERVER LOGINS

If you choose to create individual SQL Server logins for some or all of the user accounts, you have two options. (For details and instructions on performing these tasks in SQL Server, see https://www.reliasoft.com/using-sql-enterprise-databases.)

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 ReliaSoft 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 ReliaSoft database. The additional authority would be added directly in SQL Server. Then, when any of these administrative users creates a new user account via the ReliaSoft application, the required SQL Server login can be created and the application database roles can be assigned automatically at the same time.

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



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

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

OPTION 2: USING A GROUP LOGIN

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

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

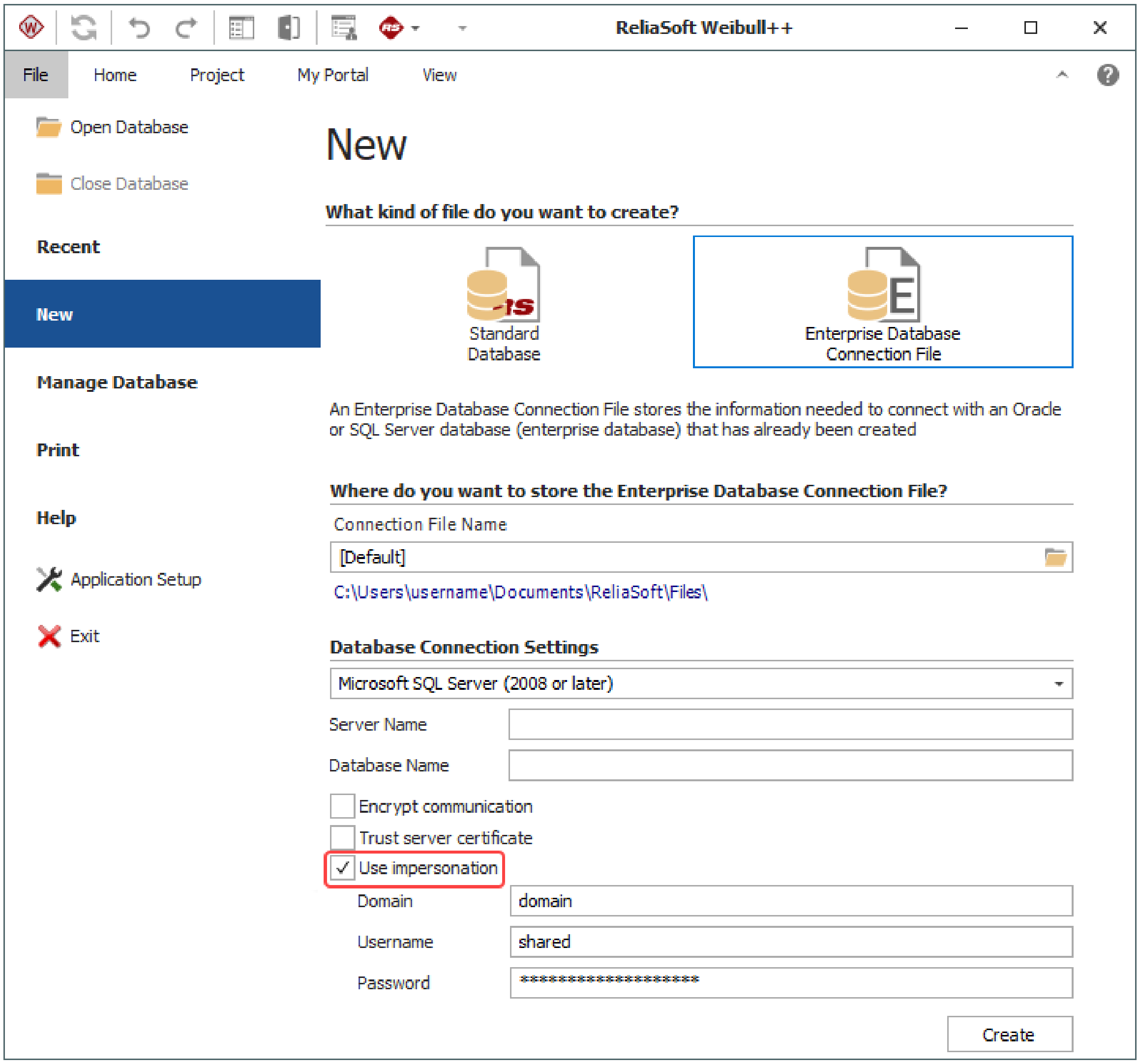
OPTION 3: USING WINDOWS IMPERSONATION FOR THE CONNECTION FILE

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

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

  1. Choose **File > New** then click **Enterprise Database Connection File**.
  2. Under **Database Connection Settings**, select **Microsoft SQL Server (2008 or later)** 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 ReliaSoft applications.)



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

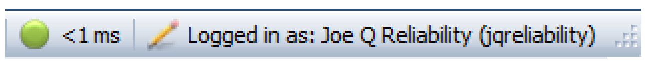
1. Choose **File > Open Database** and browse for the connection file.
2. Click **Open** to connect with the database.

After the first connection, this *.rserp file will be saved in the list of recent databases, 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 ReliaSoft applications, the user's actions will be governed by his/her own user account in the ReliaSoft applications. In other words, multiple users can connect with the database using the same enterprise connection file, but their activities within the ReliaSoft applications will be governed by the permissions established in their own individual user accounts, and any changes made to the analysis data will be recorded in the application under their own usernames.

Enterprise Database Connection Speed

If the ReliaSoft desktop application is exhibiting slow performance when you're connected to an enterprise database, the issue may be related to your network connection. The MDI status bar displays a connection speed indicator on the lower right side of the window.



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

< 30 ms: Good (acceptable performance)

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

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

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

> 150 ms: Extremely slow (will result in unacceptable performance and usability)

If a connection error occurs, the tooltip that is displayed when you point to the error message will show the associated IPStatus value.

Upgrading an Enterprise Database from a Previous Version

If you have an existing enterprise repository that was created in Version 8 or later of the ReliaSoft applications, you can transfer or convert all of the existing data into a new database that is compatible with the latest version.

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). In addition, you must be a member of the Admin group in the database to perform the task.

Upgrading an enterprise repository will also upgrade any library files and/or templates that are stored within that database.

Tip: If you need to be able to upgrade an enterprise database without using a ReliaSoft desktop application (i.e., without taking up one of the available license seats), you can access this same feature from the ReliaSoft Admin tool.

UPGRADING FROM VERSION 9 OR ABOVE

It is strongly recommended that you create a backup of the enterprise repository to be upgraded. The upgrade process converts the database itself to the latest version. ***This change cannot be undone***.

To upgrade the database, choose **File > Manage Database > Upgrade Enterprise Repository**.

In the window, specify the enterprise database that you wish to upgrade. Click **OK** to start the process.

UPGRADING FROM VERSION 8

You do not need to create a backup before upgrading a Version 8 enterprise repository. The upgrade process does not involve a direct conversion of data, but rather copies over all existing data into a new database. This complete switch to a new database is required in order to support the exclusive features in the latest version.

To upgrade the database, open the latest version of the ReliaSoft application and choose **File > Manage Database > Upgrade Version 8 Repository**.

Use the left side of the window to specify the Version 8 enterprise database that you wish to upgrade. Use the right side of the window to define the new Version 2021 enterprise database that the existing data will be copied into. Click **OK** to start the upgrade.

ReliaSoft Admin Tool

ReliaSoft desktop applications include an admin tool that allows a database administrator to set up and manage enterprise databases without requiring a software user license or an activated ReliaSoft desktop application.

To use the ReliaSoft Admin tool, your organization must have already established a database server with Oracle or Microsoft SQL Server, and you must have the permissions necessary to create databases on the server.

To access the tool, open the Windows Start menu and choose **ReliaSoft 2021 > Additional Tools > ReliaSoft 2021 Admin** from the programs list. Alternatively, you can open the Windows Start menu and type **ReliaSoft 2021 Admin** in the search bar.

The following features from the ReliaSoft desktop applications are available in the admin tool.

- Creating a new enterprise database
- Connecting to an existing enterprise database
- Upgrading an enterprise database from a previous version
- Manage user accounts:
  - Adding and editing user accounts
  - Importing users from Active Directory
  - Creating alternative credentials
  - Editing user login and contact info
  - Managing security groups
- Configuring the XFRACAS or SEP applications on a web server. For details, please consult the implementation guides for those web applications.

Projects

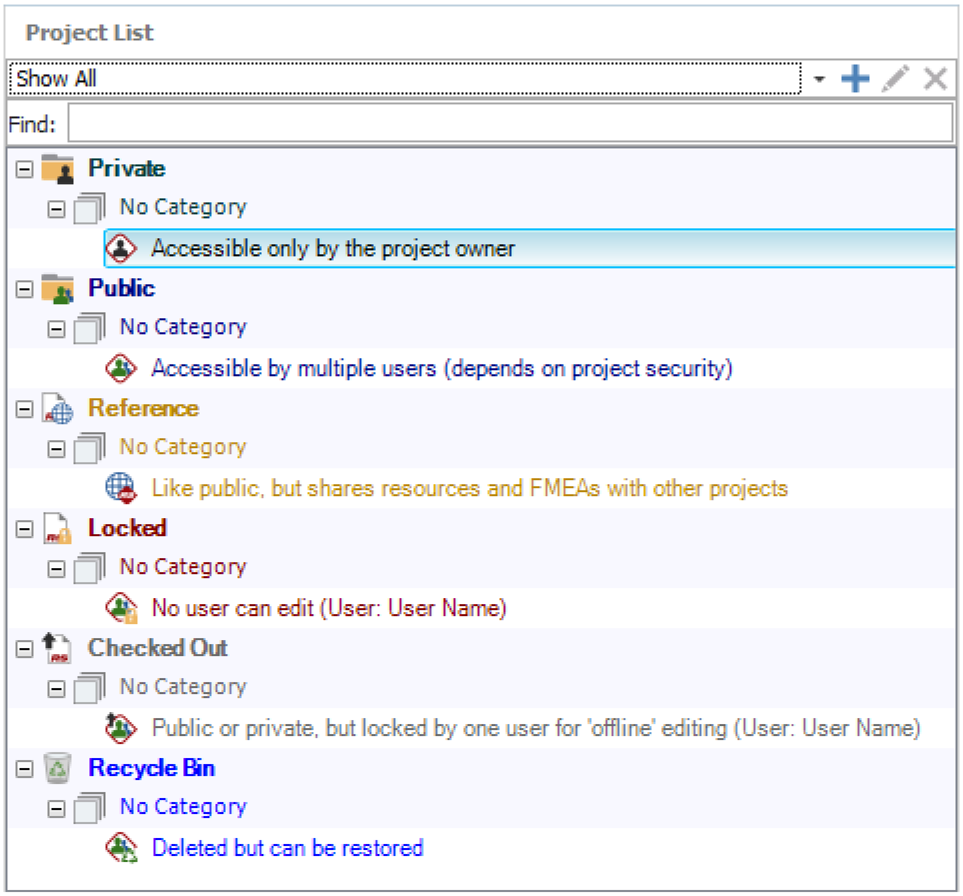
In ReliaSoft databases, *projects* serve as a way to keep related analyses together. Each project can contain analyses (e.g., folios, diagrams, plots, FMEAs, etc.), a Project Planner, resources that can be shared between analyses (e.g., models, tasks, actions, etc.) and attachments.

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

Creating and Managing Projects

This topic describes how to create and manage the projects that are accessible to you via the project list (**View > Project Manager > Show Project Manager**). This depends on your permissions in the database and any security that has been defined at the project level.

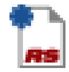
There are three types of projects: private, public or reference. In addition, a project may be locked or checked out at any given time. Each type of project is displayed under the appropriate heading in the project list:



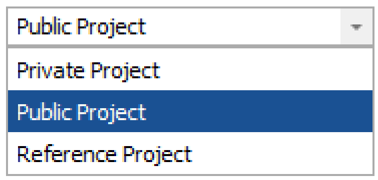
Tip: The Manage Projects window (**Project > Management > Manage Projects**) allows you to perform administrative tasks for multiple projects all at once, and to manage the private projects of other users. (See Manage Projects Window.)

Creating a New Project

When you create a new project, you must specify whether it will be public, private or reference. Select or right-click the appropriate heading in the project list and (i.e., Private, Public or Reference), and then choose **Project > Management > Create Project**.



The selected project type will be displayed at the bottom of the Project Properties window. If desired, you can change the selection before clicking **OK** to create the project.



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

Duplicating Projects

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

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

Deleting Projects

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

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

Note: You cannot delete a reference project if any of its resources are in use, including by projects in the recycle bin.

Project Properties

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

It can also be accessed from within the Manage Projects window.

The options available in this window will vary depending on which application you are using and whether the database has login security enabled. For MPC, see MPC's Project Properties (below). For all other ReliaSoft desktop applications, this may include:

- General Tab
- Security Tab (available only for public/reference projects in secure databases)
- Configurable Settings Tab (only in XFMEA/RCM++)

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

Project Owner: The user who has been identified as the project owner. In a secure database, the ability to edit the project properties is restricted to the project owner and to users with the relevant "manage all projects" permissions.)

Last Update: The time and date the project was last changed/updated, and the user who made the change. This considers any change to any of the analysis data in the project (not just the project properties).

General Tab

The General tab contains the following options:

- **Name** is the identifier for the project that will appear in the current project explorer and in many other windows and reports throughout the software. This field is required and cannot contain any of the following characters: \ / : " * ? < > |.

- **Description** and **Remarks** can contain more detailed notes or information about the project.

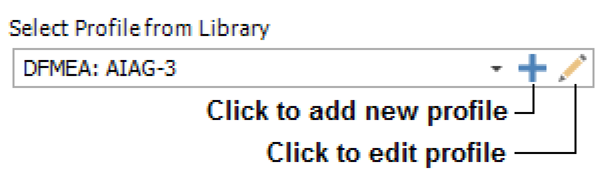
- **Proprietary Label** can contain a copyright or distribution statement, if desired. This label may appear in the footer of some reports generated in XFMEA/RCM++ or MPC.

- **Project Category** allows you to assign a project category that can be used to filter the projects displayed in the project list and in many other windows throughout the software.

In XFMEA/RCM++, 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++. (See Choosing the FMEA Structure in the XFMEA/RCM++ documentation.) Note that this option is not available when you have opened the window from the Manage Projects window and you are working with properties for multiple projects.

- **Select Profile from Library** displays a list of all profiles that have been defined in the active library. 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. If there is a mismatch in drop-down values or scales between the original profile and the new profile, you will be given an opportunity to specify replacement values to be used. (In a secure database, the ability to add and edit a profile is available only for users with the "Manage profiles and templates in XFMEA/RCM++" permission.)

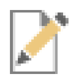
  

  Starting in Version 2021, you can update profiles for multiple projects when you have opened the Project Properties window from within the Manage Projects window. In this case, if there is a mismatch in drop-down values or scales between any of the original profiles and the new profile, you will be given the choice to clear properties wherever these values are used, to specify replacement values to be used, or to cancel the operation.

Configurable Settings Tab

The Configurable Settings tab is available for XFMEA/RCM++ only, and only when you are working with a single project's properties. It provides access to all of the configurable settings that have been defined for the current project, based on the profile that was selected on the General tab.

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



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.

In the **Enable Legacy Analyses** section, you can choose whether to allow users to create and use DVP&R or DRBFM analyses in the project. These settings are not part of the profile and may be different for each project. (See Design Verification Plans (DVP&Rs) and Design Reviews Based on Failure Mode (DRBFMs) in the XFMEA/RCM++ documentation.)

Security Tab

The Security tab is available only when you are working with a public or reference project in a secure database. For a full discussion on how to use these settings, see Planning Your Security Approach.

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

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 the **Name**, **Description**, **Remarks** and **Project Category**. In addition, you will be able to define the following:

- **Short Description** appears in the header of the Standard report template. It is not applicable for Dassault or Sukhoi reports.
- **Proprietary Label** appears in the footer of the Standard report template. It is not applicable for Dassault or Sukhoi reports.
- **MSG-3 Guidelines** displays a list of the available MSG-3 guidelines that can be used for the systems and powerplant analysis.
  - In most cases, the version that you select will not have any impact on the MSG-3 logic displayed in the interface and print-ready reports.
  - The only substantive difference occurs when you select MSG-3 Revision 2002.1 because this older version of the guideline 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.

Project Owner

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

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

Changing the Project Owner

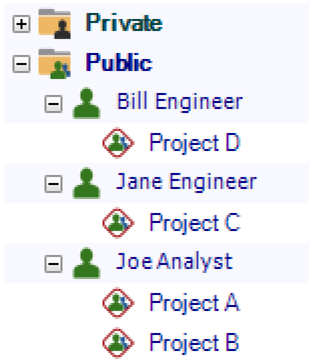
To change the owner, select the project in the project list and choose **Project > Security > Change Owner**.

In a secure database, this is available only for users with the applicable "Manage all projects" permissions.

Identifying the Project Owner

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

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



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

Project Owner: Bill Engineer
Last Updated: 9/20/2019 1:02 PM User: Bill Engineer

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

Name	Owner	Category	Security	Last Updated
Private				
Project E	Bill Engineer	No Category	Private	10/9/2019 9:04 AM
Public				
Project A	Joe Analyst	No Category	Database Level	10/9/2019 10:03 AM
Project B	Joe Analyst	No Category	Database Level	10/8/2019 3:41 PM
Project C	Jane Engineer	No Category	Database Level	10/8/2019 3:00 PM
Project D	Bill Engineer	No Category	Database Level	10/10/2019 11:06 AM
Reference				
Project F	Jane Engineer	No Category	Database Level	9/26/2019 1:13 PM
Locked				

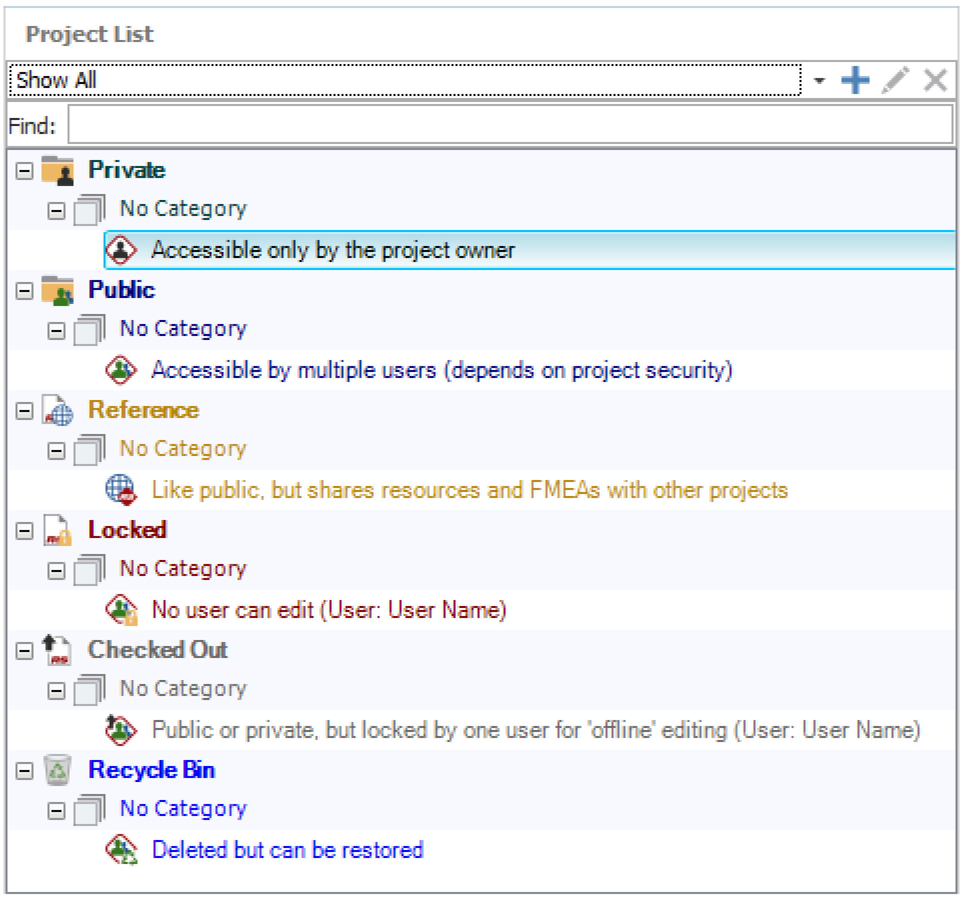
Public, Private and Reference Projects

In ReliaSoft databases, there are three types of projects you can create:

- A **public** project may be accessible to any user who has access to the database, depending on the security settings that have been implemented.

- A **private** project can be viewed and edited only by the project owner. The project list shows only your own private projects, while the Manage Projects window provides the ability to manage all users' private projects (including delete, lock/unlock or changing the project type).
- A **reference** project is like any regular public project, except that you can share its resources and FMEAs with other projects in the database. This gives you a pool of resources that can be used throughout the database by specified users (based on the security settings for the reference project), while allowing you to maintain fully functional analyses within the reference project itself. (For more information, see Local, Global and Reference Resources, and Linked FMEAs in the XFMEA/RCM++ documentation.)

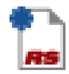
In the project list, each type of project is displayed under the appropriate heading, as shown in the following example.



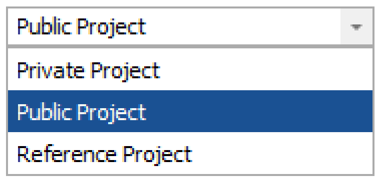
Note: In MPC, the Reference heading will appear only if reference projects already exist in the database (created by another ReliaSoft desktop application). You cannot create a new reference project or convert an existing project into a reference project in MPC.

Setting the Initial Project Type

When you create a new project, you must specify whether it will be public, private or reference. Select or right-click the appropriate heading in the project list and (i.e., Private, Public or Reference), and then choose **Project > Management > Create Project**.

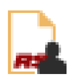


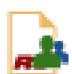
The selected project type will be displayed at the bottom of the Project Properties window. If desired, you can change the selection before clicking **OK** to create the project.

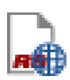


Changing the Project Type

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

 **Make Private**

 **Make Public**

 **Make Reference**

To change the project type for multiple projects simultaneously, use the Manage Projects window instead.

Locked and Unlocked Projects

In all ReliaSoft databases, a project can be *locked* to prevent users from editing the data. If a project is locked, it will be moved to the **Locked** heading in the project list and it cannot be edited by any database user unless it is unlocked again.

In a secure database, locking and unlocking a project are available only if the user a) is the project owner, b) has the "Lock or check out project" permission for the project or c) has the applicable "Manage all projects" permissions.)

Locking a Project

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

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

Unlocking a Project

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

Tip: The Manage Projects window allows you to select multiple projects and lock or unlock them all at once.

Check In and Check Out Projects

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

- An editable copy of the project will be saved into a standard database (*.rsr21) in the specified "Checked Out" folder on your local computer. This database will have the same name as the project.
- The original project will be set to read-only and moved under the **Checked Out** heading in the project list. This allows other users who would normally be able to access the project to see that it is currently being edited exclusively by one particular user. It also provides read-only access to the latest version of the project at the time it was checked out so those users can query, copy data, etc. while you're editing the project "offline."

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

IMPORTANT: If the project uses resources or FMEAs from a reference project (e.g., linked FMEAs, models, URDs) the links won't be maintained when you check out the project. Any linked resources/FMEAs will be replaced with local copies of the resources/FMEAs and stored directly within the project. See also Local, Global and Reference Resources.

Specifying the "Checked Out" Folder

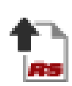
By default, projects will be checked out to the default Documents folder on your computer (e.g., My Documents\ReliaSoft\Files\Checked Out).

To change this location, open the Backup/Check Out Options page of the Application Setup window and browse for a new default path under **Check In/Out**. Keep in mind that anyone who has access to the folder will have full access to the checked out project; however, only the user who has the project checked out will have the ability to check in the project.

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

Checking Out a Project

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



In a secure database, this is available only if the user a) is the project owner, b) has the "Lock or check out project" permission for the project or c) has the applicable "Manage all projects" permissions.

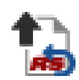
Checking in a Project

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

Note that when you check in a project, the software creates a restore point, which is an exact replica (i.e., a backup) of the project *before it was checked out*. This allows you to restore the project to its prior state when and if needed. The restore point will include a description of the user who checked out the project, as well as the date and time of the check out. (See Restore Points.)

Undo Check Out

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



In a secure database, this is available only if the user a) is the user who checked out the project, b) is the project owner or c) has the applicable “Manage all projects” permissions.

Manage Projects Window

The Manage Project window displays a list of all the projects that currently exist for the database and allows you to perform administrative tasks for multiple projects all at once. If you have created custom project filters, you can apply those same filters in this window.

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

In a secure database, this is available only to users with the applicable "manage all projects" permissions. Only the type of project that you have permissions for will be displayed. Projects that are under the Locked, Checked Out and Recycle Bin headings will be visible only if you have permissions for all three project types.

Tasks You Can Perform

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

- See a list of private projects created by other users (and perform any administrative tasks that may be needed for those projects).
- Change multiple public projects to private all at once, and vice versa.
- Change the owner for multiple projects all at once.
- Apply the same changes to the project properties for multiple projects all at once.
- Review and edit the security settings (if applicable) for multiple projects all at once.
- Delete, restore, lock and unlock, or undo check out for multiple projects all at once.
- Review and sort projects based on the user who made the last change to a project, the date/time of the last change and the last application that was used to make the change.

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

Tools

The Manage Project window contains the following commands:

PROJECT

Close closes the Manage Projects window.

Edit Project Properties allows you to view and edit the properties of the selected project(s).

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 or restore point.

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

Change Owner assigns a different database user to be the owner for the selected project(s). (See Project Owner.)

Project Security is available only for public and reference 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 Planning Your Security Approach.)

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/reference status edited. To unlock project(s), choose **Unlock Project**. (See Locked and Unlocked Projects.)

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 Public, Private and Reference Projects.)

Make Reference moves the selected project(s) into the Reference heading of the project list. To move reference project(s) to the Public heading, choose **Make Public**.

CHECK OUT

Undo Check Out discards all changes made to a checked out project and restores it to the state it was in at the time it was checked out. (See Check In and Check Out Projects.)

EXCEL

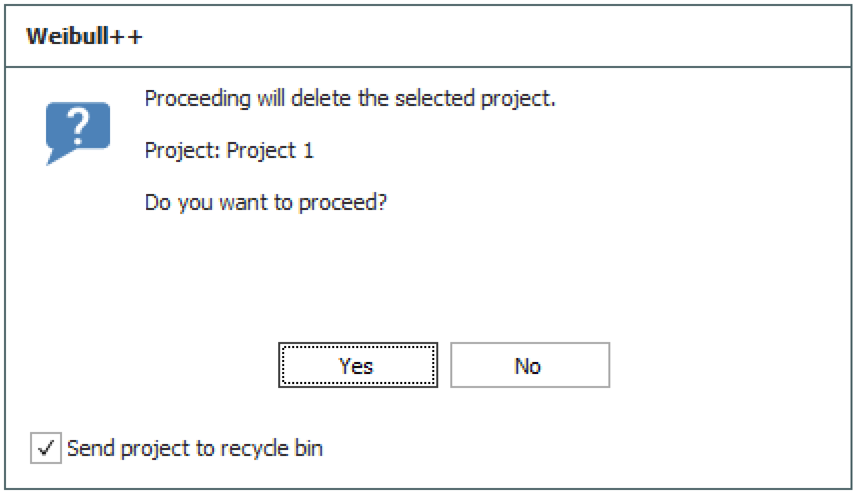
Send to Excel exports the data currently displayed in the Manage Projects window to an Excel spreadsheet.

Recycle Bin

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

Sending Projects to the Recycle Bin

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



If you want to move the project to the recycle bin, make sure the **Send project to recycle bin** check box is selected. If you want to immediately and permanently delete the project, clear the check box. *Remember that once you empty or delete a particular project from the recycle bin, there will be no way to get it back unless you have previously created a backup or restore point.*

Restoring and Deleting Projects from the Recycle Bin

In a secure database, the ability to restore and delete projects from the recycle bin is available only if the user a) is the project owner, b) has the "Delete project" permission, or c) has the applicable "manage all projects" permissions).

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

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

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

Security

Security Options

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

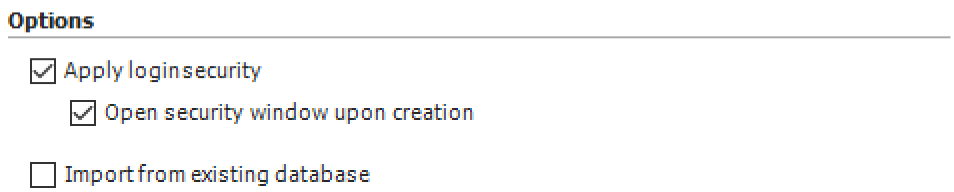
By default, each user in a secure database will have the same set of permissions for every public/reference project in the database (e.g., Jane Engineer has read-write access to all projects, Bill User has read-only access to all projects, and so on). Alternatively, you can configure the database to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

Tip: If you need to be able to manage an enterprise database without using a ReliaSoft desktop application (i.e., without taking up one of the available license seats), you can access the same security features from the ReliaSoft Admin tool.

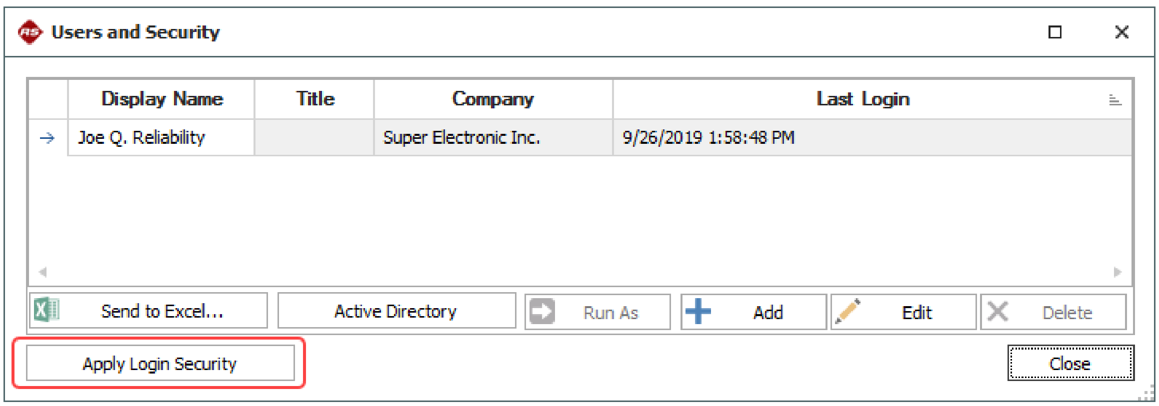
Applying Login Security

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

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



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



When you create an enterprise database or apply login security to a standard database, you will automatically be a member of the Admin security group, which has full permissions throughout the database. You can then use the Users and Security window to add/edit/delete other user accounts and assign them to appropriate security groups.

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

Planning Your Security Approach

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

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

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

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

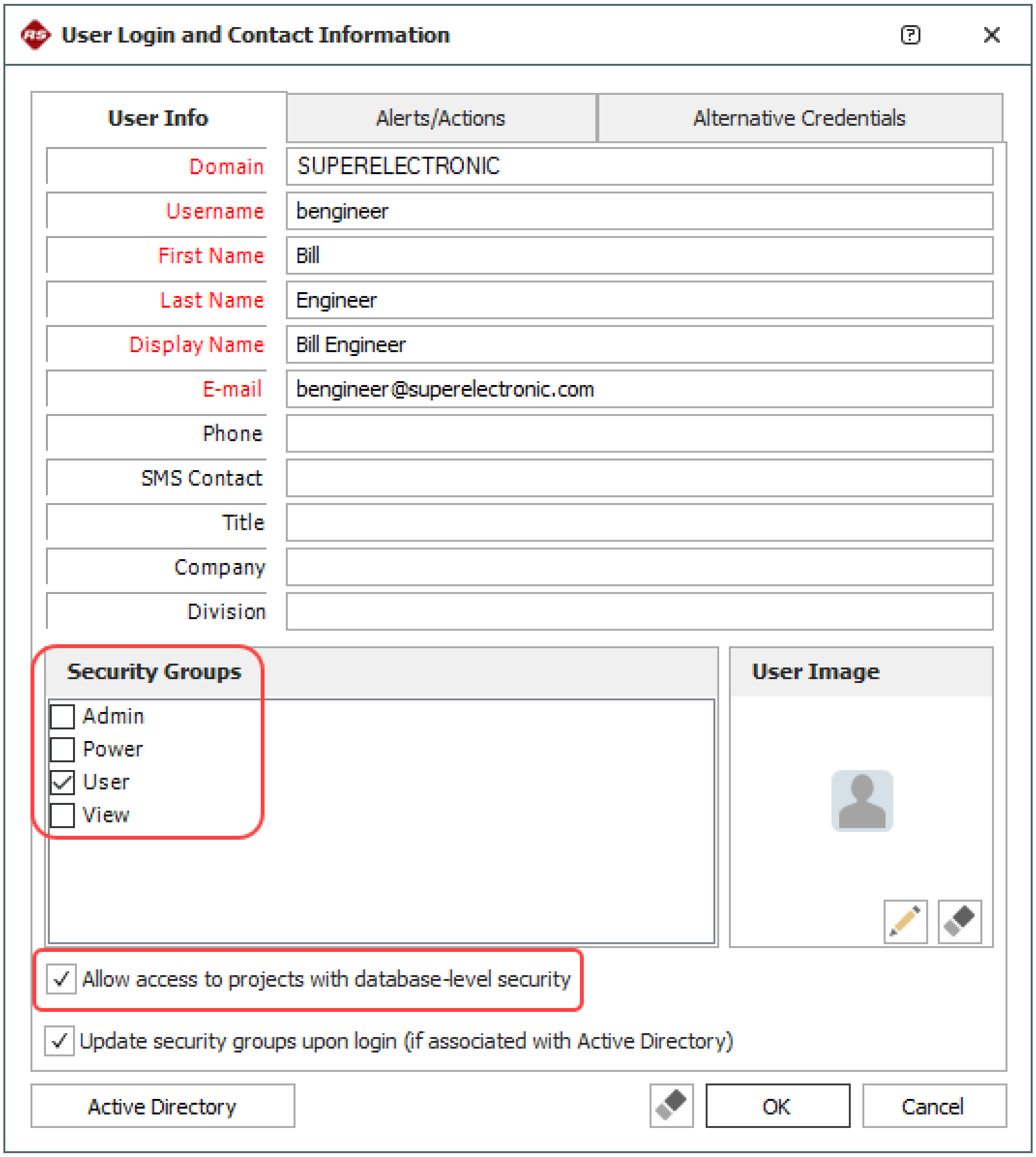
Same Permissions for All Public/Reference Projects

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

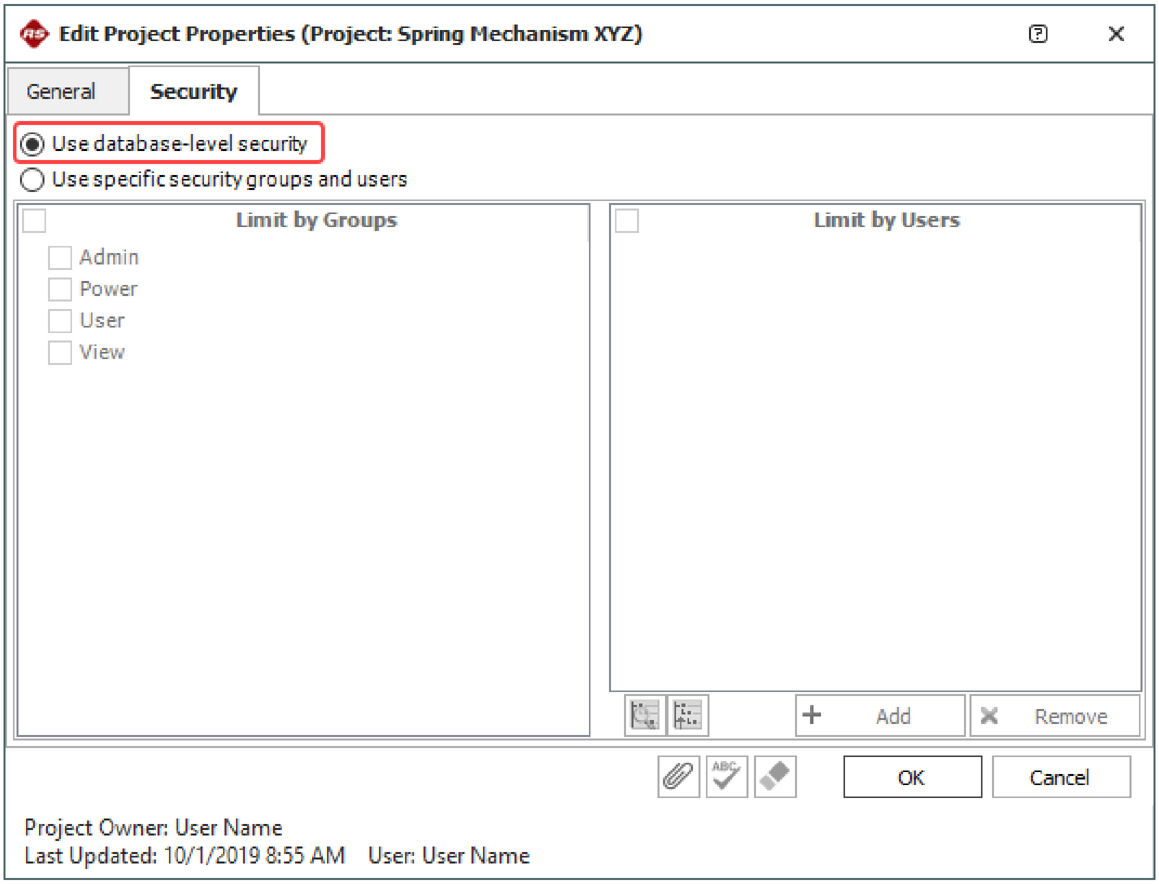
1. **User Accounts and Security Groups**: Assign each user account to an appropriate security group. This can be one of the four security groups that are created by default in each new ReliaSoft database — Admin, Power, User (Read/Write) or View (Read-Only) — or you can configure new or existing security groups to meet your particular needs.

   Select or clear the **Allow access to projects with database-level security** check box. If some of the projects in the database will continue to use database-level security, this allows you to specify whether each user will be able to access those projects.

   - If the option is selected (default), the user will be able to access any public/reference project that is set to use database-level security, with the combined permissions from any of the assigned security groups.
   - If the option is cleared, the user will only be able to access a project if it is specifically assigned to a security group that he/she belongs to, or if the user account is specifically assigned to the project.



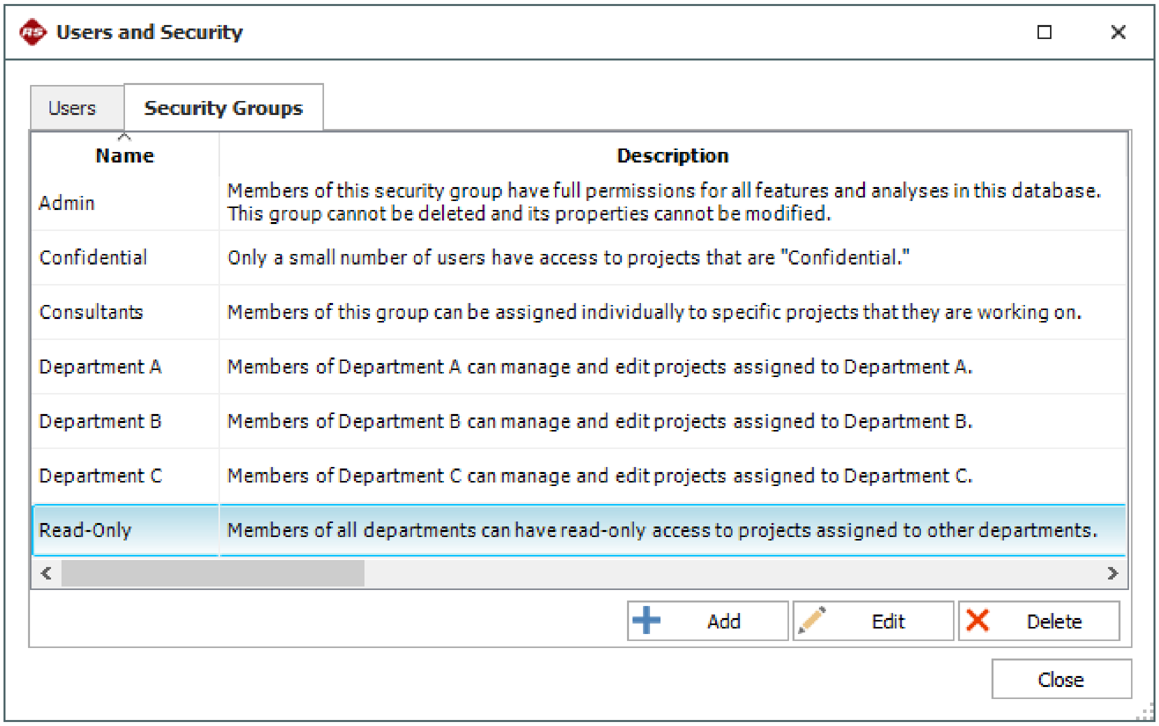
2. **Project Security**: Accept the default option on the **Security** tab of the Project Properties for all public and reference projects (**Project** > **Security** > **Project Security**). Note that if a user belongs to more than one security group (and the **Allow access to projects with database-level security** check box is selected for his/her user account), that user will have the combined permissions of those groups in any project that is set to database-level security.



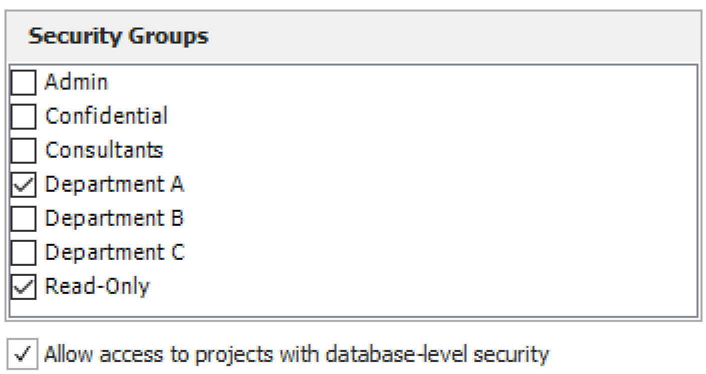
Different Permissions for Different Public/Reference Projects

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

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



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



If some of the projects in the database will continue to use database-level security, the **Allow access to projects with database-level security** check box gives you the option to decide whether each user will be able to access those projects.

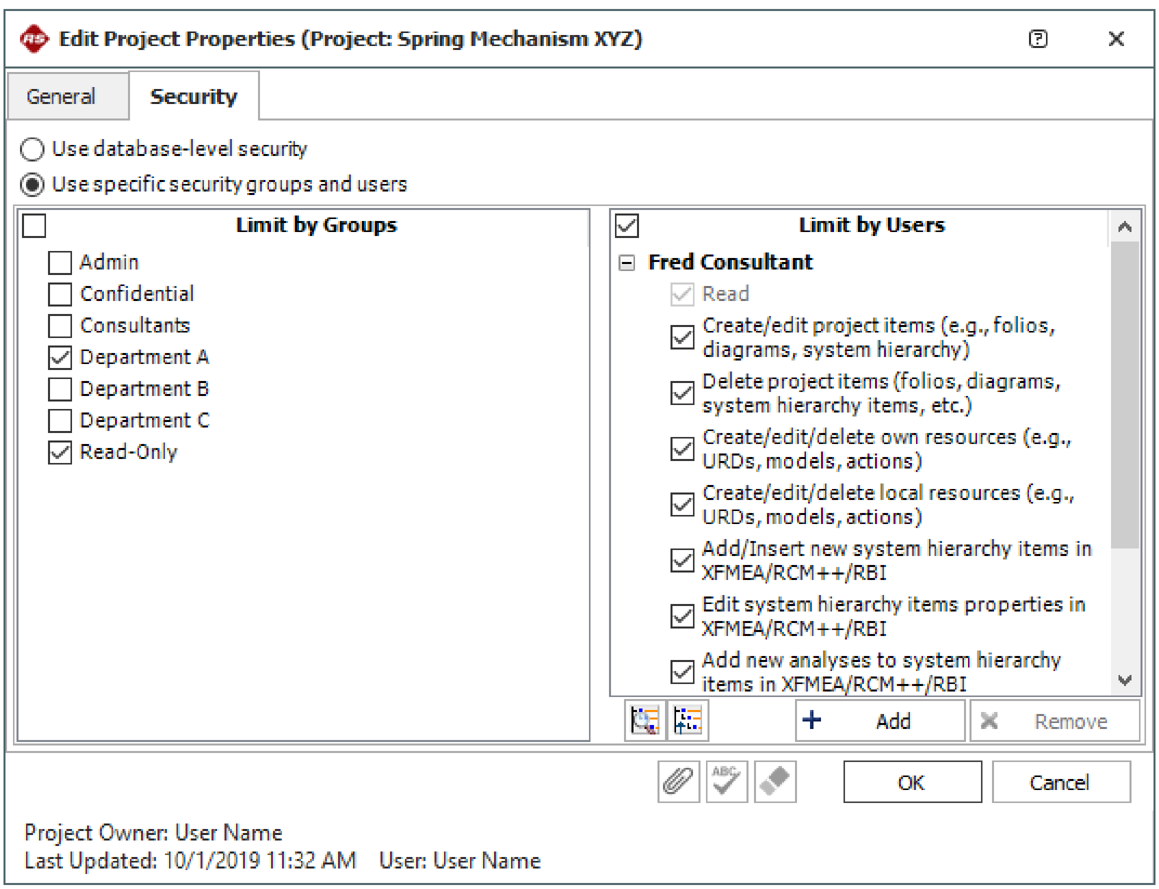
- If the option is selected (default), the user will be able to access any public/reference project that is set to use database-level security, with the combined permissions from any of the assigned security groups.

- If the option is cleared, the user will only be able to access a project if it is specifically assigned to a security group that he/she belongs to, or if the user account is specifically assigned to the project.

3. **Project Security**: Assign the appropriate security group(s) and/or specific user(s) for every public/reference project in the database.

   - When you assign a security group, every user who belongs to that group will be able to access the project with the permissions that are specified in the group.
   - When you assign a specific user, the combined permissions from all of the security groups that the user belongs to will be displayed. Use the check boxes to select which of those permissions will be in effect for that user in this particular project.

4. For the example shown below, users from Department A will have read/write access (because they belong to the "Department A" security group), users from Departments B and C will have read-only access (because they belong to the "Read-Only" security group) and Fred Consultant will have read/write access (because he belongs to the "Consultants" security group and has been specifically assigned to have those permissions in this project).



User Accounts

Managing User Accounts

Every person who will access a ReliaSoft database, be assigned to some role in an assigned action, and/or receive alerts via e-mail or SMS message must have a personalized user account in the database.

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

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

This topic describes how to use the Users and Security window to create, edit or delete/deactivate user accounts. The same window can also be used to enable login security for a standard database, and to manage security groups (permissions) for a secure database.

Starting in Version 2020, in enterprise databases, you can click the **Report** button at the bottom of the Users and Security window to generate an Excel spreadsheet that includes:

- All security groups and the permissions assigned to them.
- All users and the security groups they belong to.

BUILT-IN FIND/FILTER, CONFIGURATION AND GROUPING TOOLS

The Users tab of the Users and Security window offers the same filter, column configuration and grouping tools that are built in to other utilities that use a similar grid (e.g., the Analysis Explorer, Actions Explorer, etc.). For details about how to use each feature, see:

- Finding and Filtering Records
- Configuring Columns

- Grouping Panel

CREATING AND EDITING USER ACCOUNTS

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

You can also import user accounts from Microsoft Active Directory by clicking the **Active Directory** button. (See Importing Users from Active Directory.)

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

- In order to use Windows authentication, the user must be logged in to a computer with the same domain/username that is defined in the user account. If a user needs to connect to a database from a different domain, you can set up alternative credentials that will allow access without domain authentication. (See Creating Alternative Credentials.)
- In order to access the database, the user account must be assigned to at least one security group. (If you simply wish to send e-mail alerts to the user, a security group is not required.)
- For SQL Server databases, the username must be associated with a "SQL Server Login" that allows the database platform to recognize the user and give access to the application database. This can be accomplished with an individual login, a group login or Windows impersonation. (See SQL Server Logins or Using Windows Impersonation.)

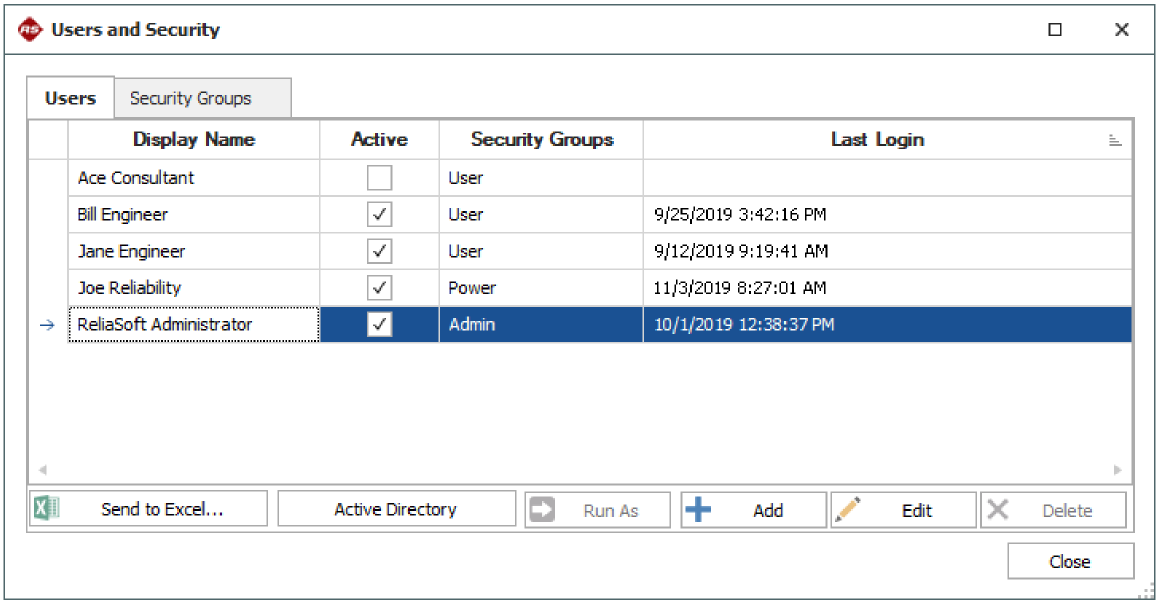
The **Run As** button below the table allows you to change your current connection to run as the user currently selected in the table. This allows you to test that user's permissions to be sure that they are set as you intended. Depending on the permissions that user has, you may have to close the database and reconnect to it in order to change back to your own account.

DELETING OR DEACTIVATING USER ACCOUNTS

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

If an account has already been logged in at least once (the **Last Login** column shows this information), attempting to delete the account will deactivate it instead. A deactivated account will not have access to the database.

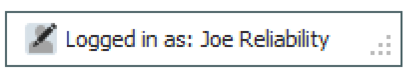
You can manually deactivate an account by clearing the check box in the Active column for the account. To reactivate the account and give the user access to the database again, select the check box in the Active column.

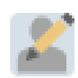


User Login and Contact Information

The User Login and Contact Information window contains the contact information, alerts preferences and action resource details for an individual user. In a secure database, it also includes access settings (security groups and alternative credentials).

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



or


To view or edit any user account in the database, choose **File > Manage Database > Users and Security**, then double-click any row in the Users table. (In a secure database, this is available only for users with the "Manage users and logins permission.")

All of the user account options are described below. Some of these settings cannot be modified when you are editing your own profile.

USER INFO TAB

- **Domain** and **Username** contain the credentials used by Windows authentication to identify the user and give him/her access to the database.
- **Contact Details**. Each user can update his/her own contact details, if desired. Note that the Display Name will appear in all projects, analyses and plots you create or modify in the database.

Click the **Active Directory** button to update the contact details based on information stored in Active Directory.

- **User Image**. Each user can save a profile photo, which will appear in locations such as the User Page of My Portal. Saved images are resized to 100 x 100 pixels.
- **Security Groups** control the user's access to the database. (See Managing Security Groups.)
- **Allow access to projects with database-level security**
  - If the option is selected (default), the user will be able to access any public/reference project that is set to use database-level security, with the combined permissions from any of the assigned security groups.
  - If the option is cleared, the user will only be able to access a project if it is specifically assigned to a security group that he/she belongs to, or if the user account is specifically assigned to the project.

  For more information, see Planning Your Security Approach.
- **Update security groups upon login (if associated with Active Directory)**
  - If the option is selected (default), the user's security groups will be assigned automatically based on his/her security group in Active Directory. (See Associating Security Groups with Active Directory.)
  - If the option is cleared, the user can be manually assigned to any of the available security groups.
- **Active**. Clear or select the check box to deactivate or activate the account. A deactivated account will not have access to the database.

ALERTS/ACTIONS TAB

- **Receive automated alerts**. Each user can choose to receive alerts via e-mail, SMS text message or ReliaSoft portal messages. (See Watches and Alerts and What is Your SMS Address?) Note that alerts via e-mail and SMS text are available only if a valid SMTP server has been defined for the database and the user account has an e-mail address/SMS contact defined.
- **When assigned to actions**. When a user account is assigned to an action (either as the Person Responsible or as part of the Team resource), the **Cost Category** and **Hours per Day** are used to calculate resource utilization and costs. (See Costs and Man Hours.)

ALTERNATIVE CREDENTIALS TAB

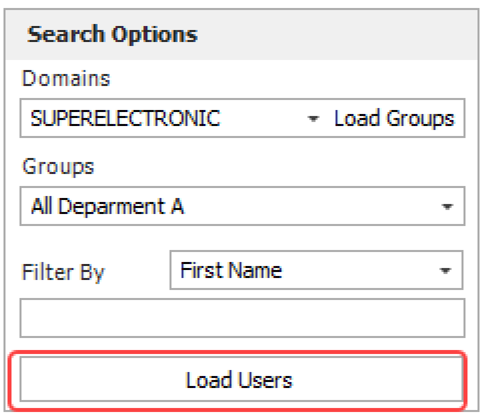
The Alternative Credentials tab is available only for secure databases. Select the check box to allow the user to bypass Windows authentication and connect to the database using an alternative username and password. (See Creating Alternative Credentials.)

Importing Users from Active Directory

If your organization uses Microsoft Active Directory, this topic explains how to import username and contact information from the directory to create new user accounts in the ReliaSoft database. (If you also wish to use Active Directory to manage the membership in security groups, see Associating Security Groups with Active Directory.)

To import users from Active Directory, choose **File > Manage Database > Users and Security**, click the **Active Directory** button, and then follow these steps:

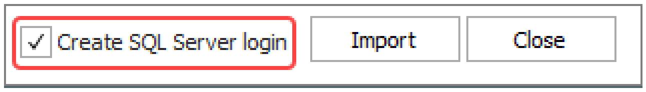
1. Enter or select the domain name in the **Domains** field. If your organization's directory is large, you may also choose to limit the search to specific groups by clicking **Load Groups** and then choosing an option from the drop-down list. You can also use the **Filter By** field to further limit the search. When you have specified the desired filters, click **Load Users** to update the table.



2. In the table of user accounts that match the filter criteria, select the check box for each user you want to create a account for. (The names of users who already have an account in the current database will appear grayed out.)

3. For secure databases only, use the **Import as Members Of** field to select the security group(s) that will be assigned to the new user account(s). If you skip this option now, you can assign security groups later for each individual account. (See Managing Security Groups.)

4. For SQL Server databases only, select the **Create SQL Server login** check box if you want to create an individual SQL Server login for each new user account. (See SQL Server Logins or Using Windows Impersonation.)



5. Click **Import** to create the account(s). After the process completes, the window will remain open to allow you to import additional users, if desired.

Creating Alternative Credentials

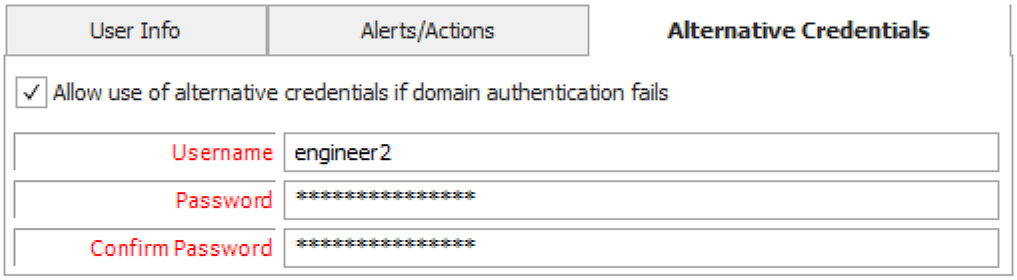
Secure databases use Windows authentication by default. This means that users must be logged in to a computer using the same domain and username specified for their user accounts. You can, however, allow specific users to bypass Windows authentication as needed. This may be useful for situations where:

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

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

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

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



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

Tip: The first time you connect to a database using alternative credentials, the application will automatically remember the login information on your computer. If you wish to clear the saved login, you can click the **Clear Alternative Credentials** link on the Other Common Settings page of the Application Setup.

User Groups

The User Groups window (formerly called "Notification Groups") allows you to create and manage groups of users that can be assigned throughout the current database for:

- Recipients for portal messages
- Monitors for assigned actions
- Team members in the Project Planning Resources

To access the window, choose **File > Manage Database > User Groups**.

In a secure database, this is available only for users with the "Manage users and logins" permission.

ADDING OR EDITING A GROUP

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

To add a new group, click **Add**.

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

The **Available Users** list shows all of the user accounts in the current database that are not yet assigned to the group. Double-click or use the buttons to move at least one user into the **Selected Users** list.

DELETING A GROUP

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

If the user group has been assigned to any existing actions, messages or project planning teams, the group will be removed automatically and those users will no longer be assigned in the affected records. ***There is no undo for delete***.

Managing Security Groups

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

As discussed in Planning Your Security Approach, there are two basic approaches you can use:

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

Note: If your organization has implemented an SEP web portal for an enterprise database and the site is configured to enable access by nCode Aqira users, a special "Aqira" security group will be created. This security group cannot be deleted, but you can modify the permissions that will be available to Aqira users.

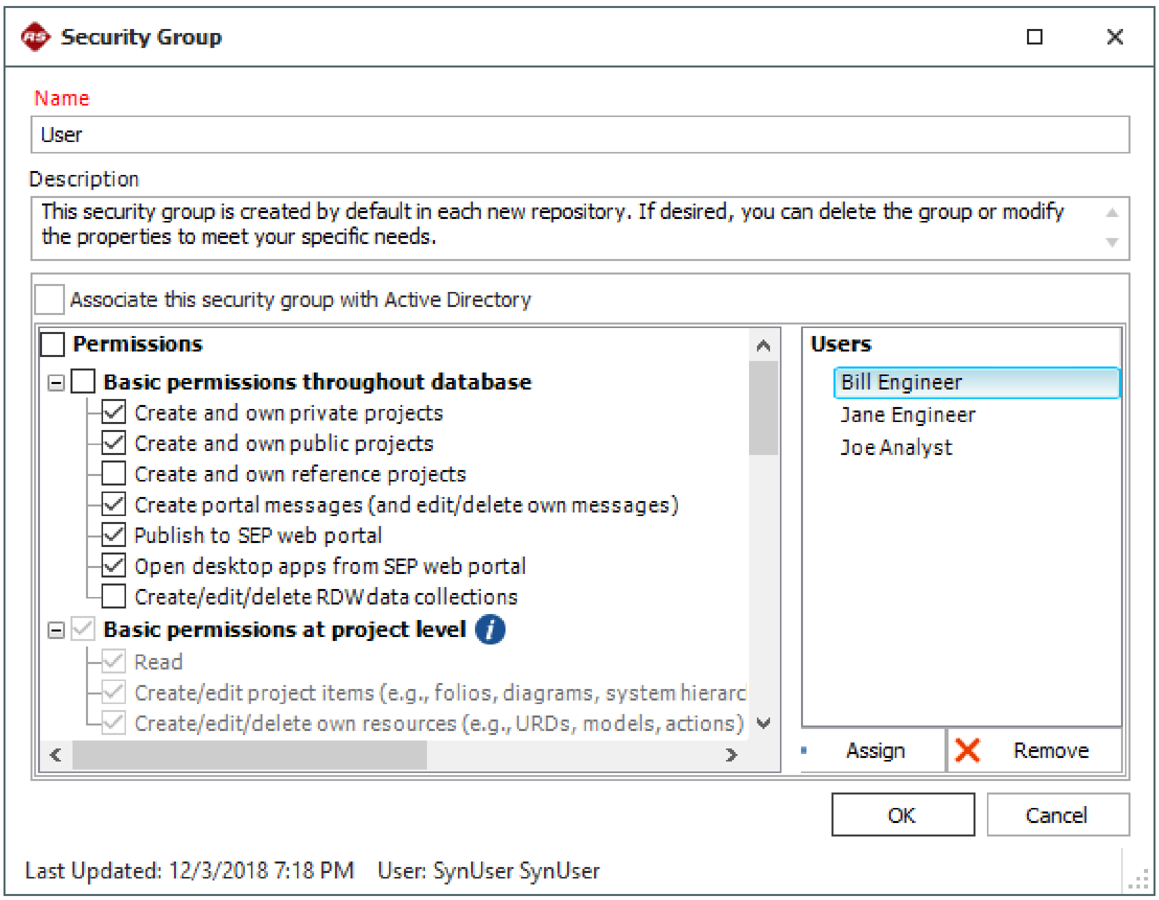
Creating, Editing or Deleting Security Groups

You can manage the security groups by choosing **File > Manage Database > Users and Security**. (In a secure database, this is available only to users with the "Manage users and logins" permission.)

In the Users and Security window, click the **Security Groups** tab to see all the security groups that have been created in the database. Use the **Add**, **Edit** or **Delete** buttons below the table to manage the groups.

When you edit a security group, the left side of the Security Group window allows you to choose the permissions, while the right side shows all the users currently assigned to the group.

If you have selected to **Associate this security group with an Active Directory group**, the list of assigned users can be updated automatically, or you will only be able to manually import/assign users who belong to the designated Active Directory group. (See Associating Security Groups with Active Directory.)



Starting in Version 2020, in enterprise databases, you can click the **Report** button at the bottom of the Users and Security window to generate an Excel spreadsheet that includes:

- All security groups and the permissions assigned to them.
- All users and the security groups they belong to.

Permissions

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

BASIC PERMISSIONS THROUGHOUT DATABASE

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

Create and own private projects	You can create and own *private* projects in the database that are accessible only to you. (Users with the "Manage all private projects" permission can still perform administrative tasks on all private projects, such as editing their properties, locking them, converting them to public projects, etc.)
Create and own public projects	You can create and own *public* projects that other users can view and edit (depending on the project security settings).
Create and own reference projects	You can create and own *reference* projects for sharing resources and FMEAs with other projects in the database (depending on the project security settings).
Create portal messages	You can create new messages and edit or delete the messages you have personally created via the Messages page in My Portal.
Publish to SEP web portal	You can publish your progress, results and analyses from a given project to the SEP web portal, making the information accessible from any web-enabled device.
Open desktop apps from SEP web portal	You can use the **Open** buttons on the FMEAs and analysis summary pages in the SEP web portal to open those applications using Remote ReliaSoft.
Create/edit/delete RDW data collections	You can create, edit and delete data collections in the Reliability Data Warehouse (RDW). This includes data extraction from XFRACAS as well as custom connections, and confers the ability to create dashboard layouts for custom connections. See Reliability Data Warehouse (RDW) in the Weibull++ documentation.

BASIC AND ADVANCED PERMISSIONS AT PROJECT LEVEL

A user can have these permissions in some projects but not others, depending on the project security settings. Regardless of the project security settings, these permissions are always automatically granted to the current project owner and anyone else that can manage the project.

Read	You can perform tasks that do not modify the data in the project (e.g., view the analysis, calculate metrics in a Quick Calculation Pad, export data, etc.).
Create/edit project items	You can create and edit items in a given project such as folios in Weibull++, diagrams in BlockSim, etc., as well as update the item properties. Starting in Version 2020, in order to create and edit system hierarchy items in XFMEA/RCM++, you must also have the "Add/insert new system hierarchy items in XFMEA/RCM++" permission.
Create/edit/delete own resources	You can create resources (e.g., URDs, models, etc.) and edit or delete any existing resources you have created.
Delete project items	You can delete any item in a given project (e.g., folios in Weibull++, diagrams in BlockSim, etc.). Starting in Version 2020, in order to delete system hierarchy items in XFMEA/RCM++, you must also have the "Delete system hierarchy items in XFMEA/RCM++" permission. This permission cannot be assigned unless you also have the "Create/edit project items" permission.
Create/edit project plans	You can create and edit project plans for a given project.
Create/edit/delete local resources	You can create, edit and delete any local resources in the project (not just the ones that you created).
Set project security	You can control who can view and edit a given project. This permission allows you to configure both project security settings and item permissions.
Edit project properties	You can use the Project Properties window to edit the name, description, category and other settings of a given project.

Lock or check out project	You can: • Lock and unlock a given project. • Check in and check out a given project.
Create restore points	You can utilize restore points for a given project, which are exact replicas of the project at a particular point in time (i.e., backups).
Delete project	You can delete a given project.
Add/insert new system hierarchy items in XFMEA/RCM++	You can add/insert new items into the system hierarchy in XFMEA/RCM++. See Building the System Hierarchy in the XFMEA/RCM++ documentation. This permission cannot be assigned unless you also have the "Create/edit project items" permission.
Edit system hierarchy items properties in XFMEA/RCM++	You can edit the item properties for system hierarchy items in XFMEA/RCM++. See Item Properties in the XFMEA/RCM++ documentation. This permission cannot be assigned unless you also have the "Create/edit project items" permission.
Add new analyses to system hierarchy items in XFMEA/RCM++	You can create new analyses for system hierarchy items in XFMEA/RCM++. See Associated Analyses and Diagrams in the XFMEA/RCM++ documentation. This permission cannot be assigned unless you also have the "Create/edit project items" permission.
Move system hierarchy items in XFMEA/RCM++	You can move system hierarchy items (move up, move down, promote, demote) in XFMEA/RCM++. See Building the System Hierarchy in the XFMEA/RCM++ documentation. This permission cannot be assigned unless you also have the "Create/edit project items" permission.
Delete system hierarchy items in XFMEA/RCM++	You can delete items from the system hierarchy in XFMEA/RCM++. This permission cannot be assigned unless you also have the "Delete project items" permission.

Activate/Manage change logs in XFMEA/RCM++	You can enable and manage change logs within a given project. Change logs can be created for FMEAs, DVP&R, Control Plan and P-Diagram analyses in XFMEA/RCM++. See Change Logs in the XFMEA/RCM++ documentation.
Deactivate change logs in XFMEA/RCM++	You can deactivate change logs within a given project.
Review change logs in XFMEA/RCM++	You can implement electronic approval tracking for change logs within a given project. See Electronic Approval Tracking in the XFMEA/RCM++ documentation.

ADMINISTRATIVE PERMISSIONS THROUGHOUT DATABASE

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

Manage users and logins	You can: • Use the Users and Security window to add user accounts to the database and define security groups, as well as create alternative credentials for user accounts. • Use the User Groups window to add and manage distribution groups for e-mail alerts, portal messages, action alerts and projecting planning resources. • Use the Database Logins window to view and export a history of database logins. • Use the Reset "In Use" Flags window to clear the "in use" status for selected database users (not just your own).
Manage project planning resources and working days	You can: • Use the Project Planning Resources window to manage the cost categories, teams, materials and facilities that are used for tracking resource utilization in Project Planner gates and actions. • Use the Working Days/Holidays window to specify the business days that will be used for Project Planner gates and actions.
Manage project/item categories	You can use the Project/Item Categories window to define the project and item categories that can be used for grouping and filtering projects and items in the database.

Manage other database settings	You can:
	• Use the Unit Settings window to define the unit and measurement settings available for use in any project within the database.
	• Use the Default Name Formats window to specify how default names for resources and blocks are created.
	• Use the Task Types window in RCM++ and MPC.
	   • In RCM++ , you can map the task types that are used only in RCM++ to the task classes in the universal reliability definition (URD). See Task Types in RCM++ in the RCM++ documentation.
	   • In MPC, you can modify the abbreviations used for MSG-3 task types. See Task Type Abbreviations in MPC in the MPC documentation.
	• Use the Database Settings window to set some default settings for existing and new databases. This includes enabling e-mail alerts for the database, activating history logs, etc.
	• Set up XFRACAS connection settings that allow ReliaSoft desktop applications to access the XFRACAS data stored in the database. This is available only for standard databases (*.rsr21).

Create/edit/delete global resources	You can create, edit and delete any global resources in the project (not just the ones that you created). In addition, you can transfer data from XFRACAS to the RDW. See Extracting Data from XFRACAS in the Weibull++ documentation.
Approve actions	You can review and approve actions, which are resources that allow you to track progress made in a project.
Manage all portal messages	You can edit or delete any messages that are visible to you via the Messages page in My Portal. This includes any messages for which you are the creator or one of the recipients.
Manage dash- board layouts	You can create and save layouts for use in the Dashboard Viewer in Weibull++ and BlockSim.

Manage profiles and templates in XFMEA/RCM++	This permission is available only in enterprise databases. You can: • Use the Profiles/Library Manager window in XFMEA/RCM++ to configure the predefined settings stored in the active library of the enterprise database. See Profiles/Library Manager in the XFMEA/RCM++ documentation. • Use the Templates Manager window in XFMEA/RCM++ to configure report templates stored in the enterprise database. See Templates Manager in the XFMEA/RCM++ documentation.
Manage Lambda Predict database settings	You can: • Use the FIDES Settings Manager window in Lambda Predict to configure the FIDES-related settings stored in the database. This applies to the FIDES prediction standard only (see FIDES Settings Manager in the Lambda Predict documentation). • Use the Custom Derating Standards Manager window in Lambda Predict to create, edit or delete user-defined derating standards stored in the database. See Creating Custom Derating Standards in the Lambda Predict documentation. • 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. See MIL-217 Custom Connection Types in the Lambda Predict documentation.

Manage MPC Settings	You can: • Use the Manage ATA Chapters window in MPC to define the systems and subsystems that are available for use in the database. See Managing ATA Chapters in the MPC documentation. • Use the Manage Major Zones window in MPC to define the zones and major sub-zones that are available for use in the database. See Managing Major Zones in the MPC documentation. • Use the Configurable Options for Systems and Powerplant Analysis window in MPC to enable or disable the display of some item properties and task properties fields. See Configurable Options for Systems and Powerplant Analysis in the MPC documentation. • Use the Configurable Options for Structural Analysis window in MPC to customize the settings for environmental deterioration (ED) and accidental damage (AD) analyses that can be performed for any structural item in MPC Plus. See Configurable Options for Structural Analysis in the MPC documentation. • Use the Configurable Options for Zonal and L/HIRF Analysis window in MPC to customize the settings for the standard zonal, enhanced zonal and L/HIRF analyses that can be performed for any zonal item in MPC Plus. See Configurable Options for Zonal and L/HIRF Analysis in the MPC documentation.
Manage all public projects	You have all the basic and advanced project-level permissions for public projects in the database. You can also change the project owner for any public project in the database.
Manage all reference projects	You have all the basic and advanced project-level permissions for reference projects in the database. You can also change the project owner for any reference project in the database.
Manage all private projects	You have all the basic and advanced project-level permissions for private projects in the database. You can also change the project owner for any private project in the database.

Associating Security Groups with Active Directory

If your organization uses Microsoft Active Directory (AD), you can give an AD group a specific set of permissions in the ReliaSoft database. The members of the AD group can then access the database based on the set of permissions associated with the group.

In addition, the software offers the option to automatically update the permissions of a user whenever that user is added or removed from an AD group. For example, if a user is moved from AD group A to AD group B, his/her user account can be automatically updated with the permissions associated with AD group B. The changes will take effect the next time the user connects to the database via any of the ReliaSoft desktop applications. (Note that for users who will connect only via the SEP web portal, you'll need to update their permissions manually either via a desktop application or the ReliaSoft Admin tool on the web server.)

Tip: Multiple security groups can be assigned to the same user account, if appropriate. For example, a user can be assigned to the "ABC Team" group (which is associated with Active Directory) and the "Read-Only" group (which is not). (See Planning Your Security Approach.)

Assigning Permissions to an Active Directory Group

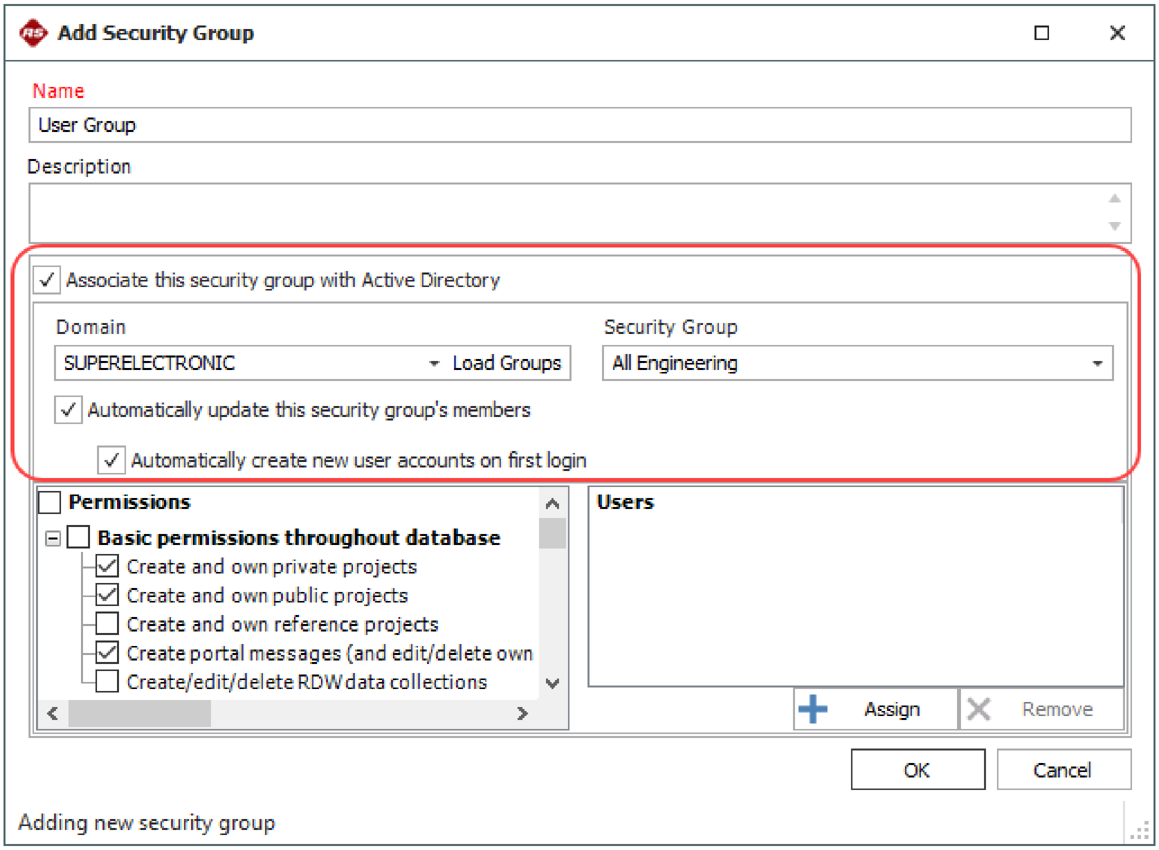
To associate a security group with an Active Directory group, choose **File > Manage Database > Users and Security** and click the **Security Groups** tab. Then double-click the security group you want to edit.

In the Edit Security Group window, select the **Associate this security group with Active Directory** check box, and then specify the domain name and Active Directory group to use. There are two additional options, which apply only when users log in from a ReliaSoft desktop application:

- **Automatically update this security group's members**. If a user is added to the AD group, his/her user account will automatically have this set of permissions the next time the user logs in to the database via any of the ReliaSoft desktop applications. Likewise, if the user leaves the AD group, his/her user account will no longer have this set of permissions.

  If you do not want the changes in Active Directory to be automatically applied to a particular user, clear the **Update security groups upon login** check box in that user's account in the ReliaSoft database.

- **Automatically create new user accounts on first login**. If a member of the AD group does not already have a user account, it will be created automatically (and assigned to this security group) the first time he/she tries to connect via any of the ReliaSoft desktop applications.



Creating/Updating Accounts Now

If you don't want to wait for all users to log in before creating/updating their user accounts, click the **Assign** button at the bottom of the window.

- To create new accounts, choose **Import users who are members of this Active Directory group**. This opens the Import Users from Active Directory window, but you can only choose users from the associated Active Directory group.
- To update the security groups for existing accounts, choose **Assign existing database users who are members of this Active Directory group**. This shows a list of existing users who also belong to the associated Active Directory group; you can select to update any or all of their accounts.

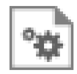
Setting Item Permissions

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

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

Project Explorer Items

In Weibull++, BlockSim or Lambda Predict, select the item in the current project explorer and choose **Project > Current Item > Item Properties**. (In a secure database, this is available only if the user a) is the project owner, b) has the "Set project security" permission, or c) has the applicable "Manage all projects" permissions.)



The settings are displayed on the Permissions tab of the properties window.

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

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

System Hierarchy Items

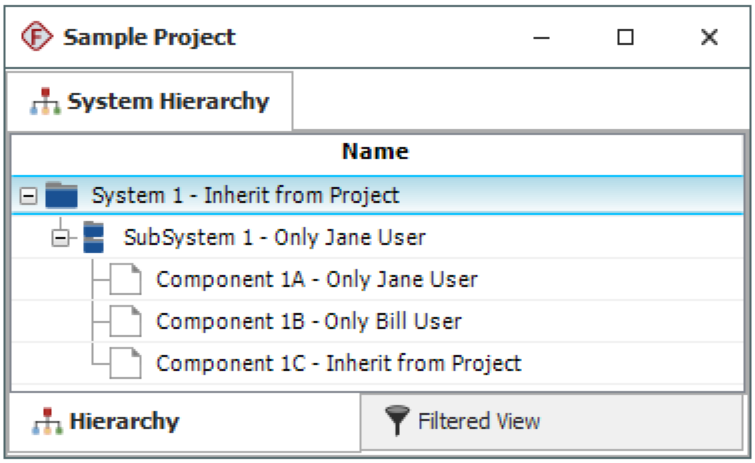
In XFMEA/RCM++, select the item and choose **System Hierarchy > Current Item > Item Permissions**. In MPC, select an item and choose **Security > Item Permissions** on the relevant ribbon tab. (In a secure database, this is available only if the user a) is the project owner, b) has the "Set project security" permission, or c) has the applicable "Manage all projects" permissions.)

In the Item Permissions window, you can choose to:

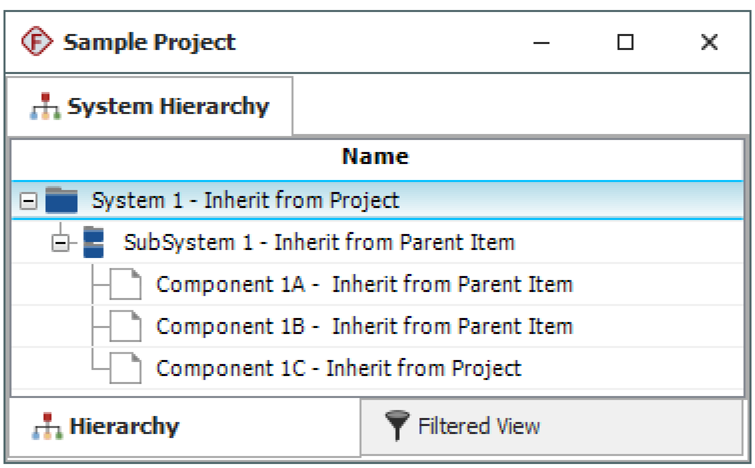
- Define the permissions for the item by selecting the **Restrict editing to selected project users** option and then selecting which specific users will be able to edit this particular item. All others can view, but not edit, the item. Only users with editing rights to the project will appear in the list.
- Set the item to inherit the same permissions as the next item above it in the system hierarchy by selecting the **Inherit from parent item** option.

- Remove item permissions and change the security settings back to the defaults by selecting the **Inherit from project** option.

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



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



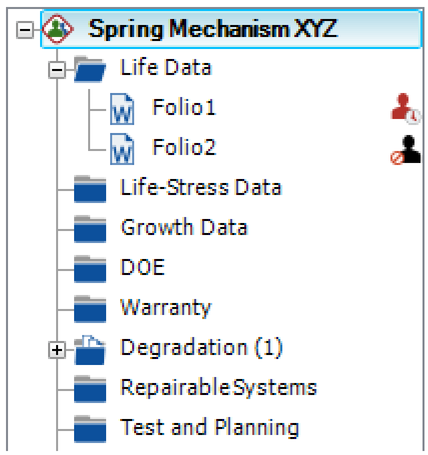
Status Indicators

ReliaSoft desktop applications include status indicators that tell you which project items (e.g., folios, diagrams, system hierarchy items, etc.) are in use or restricted.

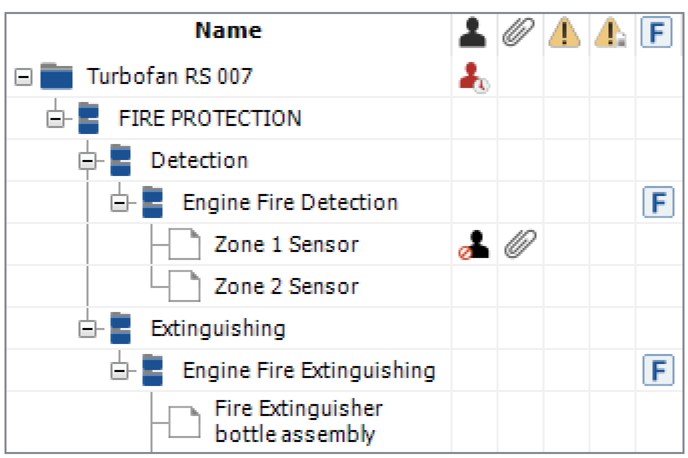
When you encounter an item that is read-only (e.g., OK button is disabled in the window or you are unable to type in the cells of a worksheet or table), you can check the status of the item to see if it is currently **in use** (i.e., you can't edit the item until the user closes the folio/diagram or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

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

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



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



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

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

Reset "In Use" Flags

Because ReliaSoft databases 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 reset the flags, choose **File > Manage Database > Reset 'In Use' Flags**.

(In a secure database, this is available only for users with the "Manage users and logins" permission.)

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 ReliaSoft application(s) that the user currently has connected to the database. You can select the check box for each user for whom you want to reset "in use" flags, then click **OK** to reset the flags.

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

Database Logins

The Database Logins window displays a record of the date and time the users in the database logged in. To open the Database Logins window, choose **File > Manage Database > Database Logins**. (In a secure database, this is available only for users with the "Manage users and logins" permission.)

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.

Database Settings

The Database Settings window contains settings that are shared by all users and analysis projects in the database. To access this window, choose **File > Manage Database > Database Settings**. (In a secure database, this is available only for users with the "Manage other database settings" permission.)

In a standard database, you can save the current settings as the default for each new standard database that you create from this computer by clicking the **Set as Default** button at the bottom of the window.

E-mail Settings Page

- **Enable Alerts via E-mail or SMS** configures the database to enable alerts via e-mail or SMS text message. (See Enable Alerts via E-mail or SMS.)
- **Action Alerts** apply to action resources. The database can be configured to auto-subscribe a "watch" for users based on their role(s) for a particular action. You can also specify the default text that will be used at the beginning of each action alert. (See Action Alert Preferences.)

Actions Page

This page allows you to automatically set the person who created an action to be the person responsible for it (see Person Responsible and Resources) and to hide the action start and completion dates in the FMEA worksheet view.

Starting in Version 2021, you can specify a default value for the **Planned Duration** field in actions. This value is a number of working days.

FMEA Change Logs Page

- **Activate change log when FMEA is created** configures the database to start a change log and an open revision when an FMEA analysis is created via any workflow (e.g., creating, copying, importing, etc.). This option is available only to members of the Admin group in the database. When this option is selected, you can define the version label that is used for the initial revision; this label does not have to be included in the list of predefined labels if such a list is defined (see next option below). If no custom label is defined, "First Revision" will be used by default. Version labels can be changed at any time.
- **Restrict version labels to a predefined list** allows you to limit the version labels used for FMEA change log revisions in XFMEA/RCM++ (see Change Logs in the XFMEA/RCM++ documentation) so that users must select labels from a predefined list. When this option is selected, you can define the list of labels on this page.

Attachments Page

Starting in Version 20.0.3, this page allows you to restrict the types of files that users can attach to or link to (see Attachments).

- **Do not allow users to attach links to files and URLs** prevents a user from adding file links and URL links to the database. In such cases, only embedded file attachments can be created.
- **Restrict files users can attach to the database** enables you to enter a list of the file extensions that will be permitted to be linked, embedded or attached. Enter the values, separated by semicolons, then click **Verify** (e.g., *.doc; *.docx; *.pdf; *.html).

IMPORTANT: For an existing database, these settings only apply going forward. Any attachment that does not meet the criteria can still be used or edited, but all new ones must match these criteria. Also note that these settings do NOT apply to Report Word Doc attachments in XFMEA/RCM++ and MPC, which can only be attachments with a *.doc or *.docx extension.

Other Settings Page

- **FMEA Structure (XFMEA/RCM++)** allows you to select the default FMEA structure for new projects created in the database, which determines how the software will display the effect and cause records in the FMEAs. If you don't specify a default, users will be prompted to select the structure for each new project they create in XFMEA/RCM++. (See Choosing the FMEA Structure in the XFMEA/RCM++ documentation.)
- **History Logs** allows you to turn the log on (or off) for all active projects, and stores a preference for whether the log should be activated by default for each new project. (See History Logs.)
- **Project Planner** allows you to enable the Project Planner for use in all projects in the database. Starting in Version 2020, the Project Planner is disabled by default.
- **Enable publish to SEP web portal** is available only in enterprise databases. If your organization has established an SEP web portal for the current database, select this check box to enable users to publish analysis summaries, ReliaSoft Workbooks and other information to the portal.
- **Delete All Portal Messages** deletes all portal messages for all users in the database. ***There is no undo for deleting portal messages***.

Unit Settings

ReliaSoft databases are pre-configured with a set of units for use with any analysis in any project within the database. The conversion factors are predefined, allowing you to easily express results in terms of other units. For example, you can enter data into a Weibull++ data sheet in terms of hours and then obtain results from the QCP in terms of years. Similarly, in RCM++ or BlockSim, you can define the duration of a maintenance task in terms of hours and specify the total operating time of a system in terms of months. In MPC, the units can be used to define task intervals.

IMPORTANT: Making changes to the existing units can have implications for all analyses throughout the database; therefore, it is best to set up units once, upon creation of the database.

To define units, choose **File > Manage Database > Unit Settings**. (In a secure database, this is available only to users with the "Manage other database settings" permission.)

If you're working in Weibull++, the Warranty Units tab will be available. It allows you to specify which of the configurable database units are equivalent to the built-in (not configurable) "warranty time units" used in some warranty folio formats. For details, see Manage Warranty Units in the Weibull++ documentation.

Setting SBU Equivalencies

The conversion factor of a unit is defined in relation to a system base unit (SBU). A base unit is a standard unit of measurement upon which other units can be based. Base units always have a conversion factor of 1. All other related units are converted back to the base unit.

For example, say that we choose kilograms (kg) as our SBU for mass. The conversion factor of related units, such as pounds (lb) and metric tons (T), must be entered in terms of the SBU:

Unit	SBU Equivalency (using kg as the base unit)
1 kg	1 kg
1 lb	0.4536 kg
1 T	1000 kg

Similarly, if we change the base unit to pounds (lb), the conversion factors must reflect the change:

Unit	SBU Equivalency (using lb as the base unit)
1 kg	2.2046 lb
1 lb	1 lb
1 T	2204.62 lb

By default, ReliaSoft databases use hours (hr) as the SBU for time, and miles (mi) as the SBU for length. To ensure that you obtain correct results when converting units, it is recommended that you set an SBU for each type of measurement (e.g., time, length, frequency, force, etc.) and define the conversion factors of all other units in terms of the relevant SBU.

Changing the Default Unit

You can set any unit as the default by selecting it in the **Use as Default** column. The software will automatically use the new default unit when you create a new analysis or resource in any project in the database. The change will not affect existing analyses and resources.

Creating and Deleting Units

To add a new unit, click the **Add** button below the table and follow the on-screen prompts to create the unit. The wizard will automatically calculate and apply the appropriate SBU equivalency based on your inputs.

To delete a unit, click the **Delete** button below the table. ***There is no undo for delete***.

Editing Existing Units

Click a cell to edit a unit's name, abbreviation, SBU equivalency and category. To apply your changes, click the **Save** button at the bottom of the window.

Note that if you will be using the usage format of the Weibull++ warranty folio, 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.

Built-in Find/Filter, Configuration and Grouping Tools

You can move a unit up or down its current position by dragging the row to the desired position, or by selecting it and using the **Move Up** or **Move Down** buttons.

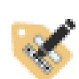
To show more tools, right-click any column heading. The window offers the same filter, column configuration and grouping tools that are built in to other utilities that use a similar grid (e.g., the Resource Manager). For details about each feature, see:

- Finding and Filtering Records
- Configuring Columns
- Grouping Panel

Default Name Formats

In all ReliaSoft desktop applications except MPC, the Default Name Formats window specifies the default names for new resources, as well as new blocks in BlockSim RBDs, fault trees, phase diagrams and PFS diagrams. For example, when you create a new model, the default name will be the word "Model" plus an increment to make the name unique (e.g., "Model," "Model_1," etc.). You can configure the database to use a different default name format if desired.

To open the window, choose **File > Manage Database > Default Name Formats**. (In a secure database, this is available only for users with the "Manage other database settings" permission.)



The **Item Name** column displays the basic name label for each type of resource or block. This will be represented by the \N code in the name format.

The **Name Format** column allows you to use any of the following elements to define the default name format for each type of resource or block:

- \N returns the label from the Resource Name column plus an increment to make the name unique (e.g., "Model," "Model_1," etc.).
- \U returns the name of the user who created the resource or block.
- \D returns the date when the resource or block was created.
- \S adds a sequential number that reflects the order in which the resource/block was added to the database.
- \T returns the ReliaSoft application ("tool") used to create the resource or block.
- \F returns the contents of all of the identifier fields except the category and comments fields.
- \G returns a summary of the properties of the resource or block. For example, for a node in a BlockSim diagram, \G returns the "k out of n" paths that are required.
- \I is only applicable for resources. It functions the same as \N unless there is another window, resource or analysis that it can "inherit" part of the name from. Some examples:
  - If you create a URD from BlockSim's block properties window, the default name will be "[Block Name]_URD."
  - If you create a metric based on a model, the default name will be "[Model Name]_Metric."
  - If you push a metric from a Project Planner gate, the default name will be "[Gate Name]_[Property Name]_Metric."

Tip: In BlockSim diagrams, you can use an asterisk (*) to insert the default block name. For example, if you enter `*System` for the block name, the diagram will display the default name for the block followed by the word "System." The asterisk also allows the block name to be updated dynamically when relevant properties change.

Project Planning Resources

In all ReliaSoft desktop applications except MPC, *project planning resources* can be assigned to actions in order to track the required costs and time in the Project Planner and anywhere else that actions are used. Because project planning resources are shared throughout the database, any change made to one is automatically applied to all actions in the database that use it.

- Cost Categories specify direct and/or per instance costs that can be assigned to any team member, facility or material.
- Teams identify groups of users who will work together on any given action.

- Facilities specify the cost and maximum utilization for facilities (e.g., test lab) or other resources for which the cost depends on the duration of use (e.g., rented test equipment).
- Materials specify the cost and quantity for materials (e.g., test units) or other resources for which the cost depends on the amount used rather than the duration (e.g., purchased usage data).

To open the Project Planning Resources window, choose **File > Manage Database > Project Planning Resources**. (In a secure database, only users with the "Manage project planning resources and working days" permission will be able to create and edit these resources; other users can select from the resources that have already been predefined.)

COST CATEGORIES

Cost categories can be created from the Project Planning Resources window, or when you are creating/editing a facility, material or individual user account. Enter the **Currency Symbol** to use for displayed costs at the bottom of the window.

- The **Direct Cost per Hour or Unit** is the per-hour cost when assigned to a team member or facility, or the per-unit cost when assigned to a material (e.g., the cost of a single light bulb).
- The **Cost per Instance** is a one-time cost that applies every time a team, facility or material is used.

TEAMS

Teams can be created from the Project Planning Resources window, or when you are defining an action. A team can include:

- Individual **Users** who have active user accounts in the database.
- **Groups** of active user accounts that have been predefined in the database.

Note that you cannot assign a cost category directly to a team. Instead, the team's cost/utilization is based on the cost category and maximum hours per day for each team member. Both of these properties are set on the **Alerts/Actions** tab of the User Login and Contact Information window.

FACILITIES

Facilities can be created from the Project Planning Resources window, or when you are defining an action.

- The **Cost Category** specifies the facility's hourly and/or per-use costs.
- The **Max Hours Per Day** is the number of hours of use if the facility is set to be 100% utilized for a given action (as specified in the action properties).

MATERIALS

Materials can be created from the Project Planning Resources window, or when you are defining an action.

- The **Cost Category** specifies the cost per unit (e.g., the cost for each test prototype), as well as any additional fixed cost (e.g., the cost for manufacturing a sufficient number of test prototypes).
- The **Quantity** that will be used (e.g., 100 light bulbs).

Working Days/Holidays

In all ReliaSoft desktop applications except MPC, actions and Project Planner gates contain important dates like start dates and due dates. You can specify which dates will be considered working days for all actions and gates in the database. This way, an action's cost/time estimates, as well as its duration, will be based solely on working days.

To define the working days for the database, choose **File > Manage Database > Working Days/Holidays**. (In a secure database, this is available only for users with the "Manage project planning resources and working days" permission.)

In the window that appears, use the **Define Work Week** area on the left side to specify which days of the week will be considered regular working days (e.g., Monday-Friday).

Define Holidays

On the **Define Holidays** tab, you can define three different types of holidays.

- **Fixed Date - Recurring** holidays occur every year on the same calendar date (e.g., New Year's Day is always January 1).
- **Variable Date - Recurring** holidays occur every year, but not always on the same date (e.g., Thanksgiving is always the fourth Thursday of November).
- **Fixed Date - Non-recurring** holidays occur once and do not happen again. For example, a specific Monday could become a non-working day due to weather conditions.

In the Holiday Properties window, select **Move to adjacent working day if necessary** to make sure the holiday will be observed when a) it occurs outside the work week, but b) there is a regular working day immediately before/after it. In this case, the holiday is observed on the nearest working day. (If there is a working day immediately before *and* after the holiday, it is observed on the following day.)

View Observed Holidays

The **View Observed Holidays** tab allows you to easily examine the names and dates of all holidays that will be observed during the work week, and can also help you make sure your working days/holidays settings are correct. Specify the years you want to view in the **Displayed Range in Years** area.

XFRACAS Connection

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. There are several ways in which you might wish to share data between XFRACAS and the analyses that you perform in ReliaSoft desktop applications:

- Use the Reliability Data Warehouse in Weibull++ to extract data from XFRACAS incidents. (See Reliability Data Warehouse in the Weibull++ documentation.)
- Share system configuration and failure mode data between XFRACAS and XFMEA/RCM++. (See Import or Sync from XFRACAS in the XFMEA/RCM++ documentation.)

When you are working with an enterprise database, the data from XFRACAS and ReliaSoft desktop applications can be stored in the same SQL Server or Oracle database.

When you are working with a standard database (*.rsr21), you will need to establish a connection to an enterprise database that contains XFRACAS data. To do this, choose **File > Manage Database > XFRACAS Connection**. (In a secure database, this is available only for users with the "Manage other database settings" permission.)

In the XFRACAS Connection Settings Window, select **Connect to an enterprise database with XFRACAS data** then choose Microsoft SQL Server or Oracle.

- For a SQL Server database, enter the server name and database name.
  - **Encrypt communication**. Secures the connection information between the ReliaSoft application and the enterprise database.
  - **Trust server certificate**. Select this option if the server has a self-signed certificate.
  - **Use impersonation**. Configures the connection setting to impersonate a Windows user account with a SQL Server login that is shared by multiple users. (See SQL Server Logins or Using Windows Impersonation.) If you do not use impersonation, then each user who uses the features that utilize XFRACAS must have an account in the external database that contains the XFRACAS data.
- For an Oracle database, 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.

Managing and Restoring Data

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 the minimum database maintenance recommendations are posted on the ReliaSoft website:

See https://www.reliasoft.com/using-sql-enterprise-databases for SQL Server.

See https://www.reliasoft.com/using-oracle-enterprise-databases 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 maximum file size is ~ 2GB, maximum number of concurrent users is 255, etc. In addition, some specific database vulnerabilities are discussed in a Microsoft publication at http://support.microsoft.com/kb/283849/EN-US/. As this 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 our 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 computer 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 Backup/Check Out Options page of the Application Setup, the ReliaSoft desktop 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 (*.rsr21) 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 by at any time by choosing **File > Manage Database > 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 Technical Support 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, we 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.

Restore Points

In all ReliaSoft desktop applications, you can create and manage *restore points* for any analysis project. Within the context of these applications, 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.

Create a Restore Point

You can create a restore point for a 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.

In a secure database, this is available only if the user is the project owner or has the "Create restore points" permission.

IMPORTANT: When you create a restore point, any global, reference and FMEA 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 those resources in the interim.

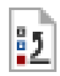
Restore a Project

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

- **Overwrite existing project** uses the restore point to roll back the current project to the 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** uses the restore point to create a new project. You may enter a unique name for the new project. When the restore process completes, the new project will be accessible from the project list.

Manage all Restore Points in the Database

To manage all the restore points that have been created in the database, choose **File > Manage Database > Restore Points**. (In a secure database, this is available only for users who have all three "Manage all projects" permissions.)



This opens the Restore Points window, which displays the details for each of the restore points.

- To create a new restore point, click the **Create** button then select the project you want to back up. 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.

Desktop Application Interfaces

IN THIS CHAPTER

Most ReliaSoft desktop applications have their own Multiple Document Interfaces (MDIs) that provide full-featured support for the relevant analysis methods.

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

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

Ribbon and Backstage View

The ribbon at the top of the MDI 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 , 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.

Project Manager and My Portal

The Project Manager allows you to manage the projects within the current database and, for many ReliaSoft applications, it also allows you to manage all of the folios, diagrams, reports, attachments, etc. in the currently open project(s). 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.)

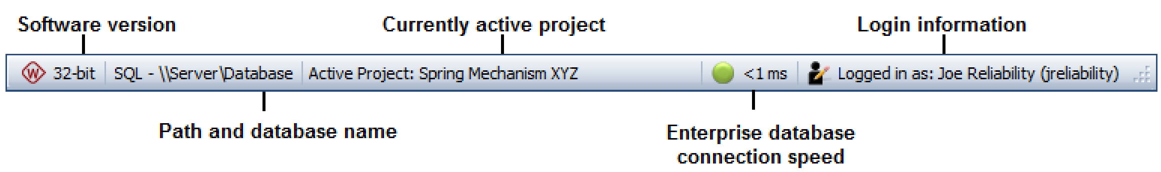
My Portal provides quick access to messages, assigned actions and other information that may be of personal interest to you while working in the ReliaSoft applications. This panel is docked on the right side of the MDI and unpinned by default. ("Unpinned" means that only a small tab will be visible in the interface unless you move the pointer over the tab to make the entire panel visible.)

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 Show, Tile, Dock and Pin Panels.)

Tip: If desired, you can change the overall color scheme used in the MDIs for all ReliaSoft desktop applications on your computer. Use the **Skins** drop-down list on the Common Settings page of the Application Setup to select the style you prefer.

Status Bar

As an example, the following picture shows the status bar at the bottom of the MDI for an enterprise database:



For more information about the connection speed indicator, see Enterprise Database Connection Speed.

Show, Tile, Dock and Pin Panels

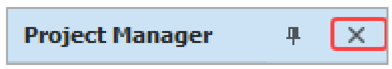
In all ReliaSoft desktop applications, the MDI includes a Project Manager panel and a My Portal panel. You can decide whether each panel will be displayed and how it will be positioned in the interface at any given time.

Show or Hide

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

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

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

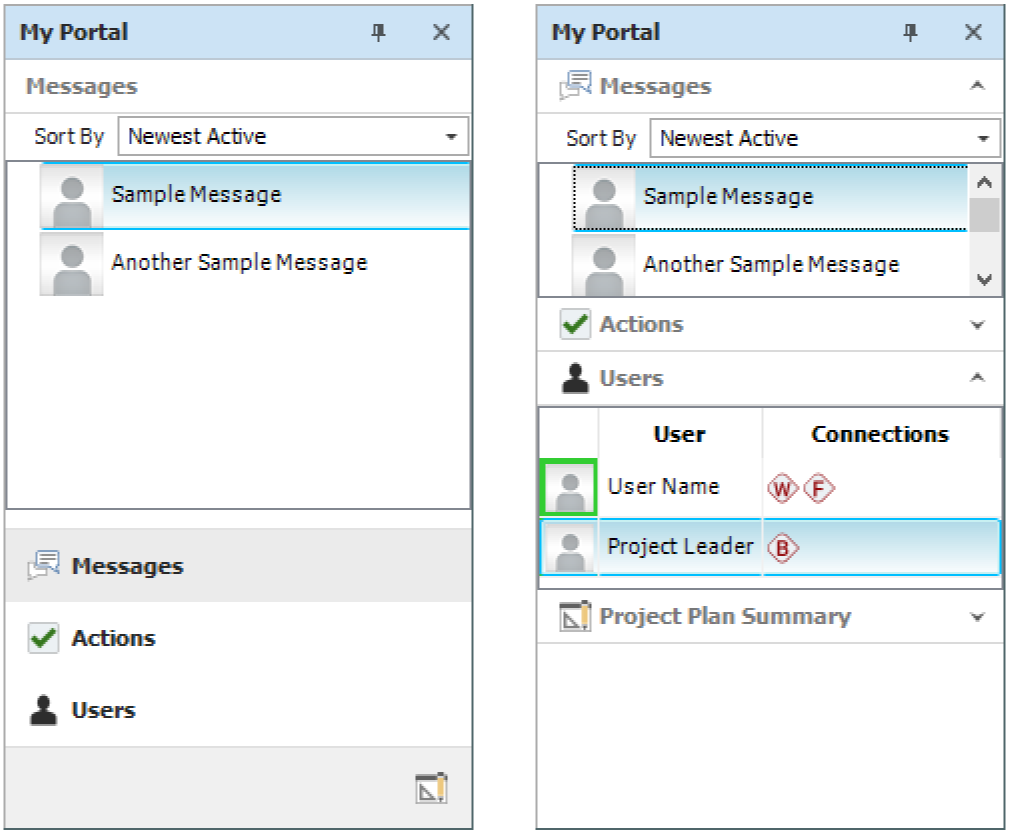


Tile or View One Page at a Time

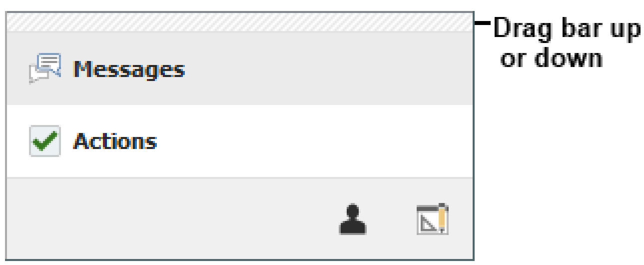
In all ReliaSoft desktop applications, My Portal consists of four pages. You have the option to view one page at a time (and use large buttons or small icons at the bottom of the panel to switch between pages) or tile the pages so that more than one page can be visible at the same time. To specify your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

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

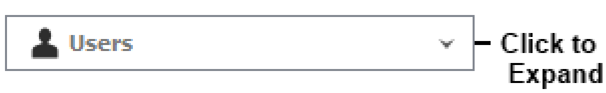
As an example, the first picture shows the My Portal panel when it is not tiled. The second picture shows the tiled panel, with two of the four pages expanded to be visible at the same time.



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



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



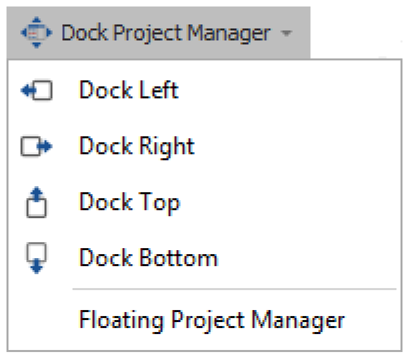
Dock or Float

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

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

There are several ways to set the docking position:

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

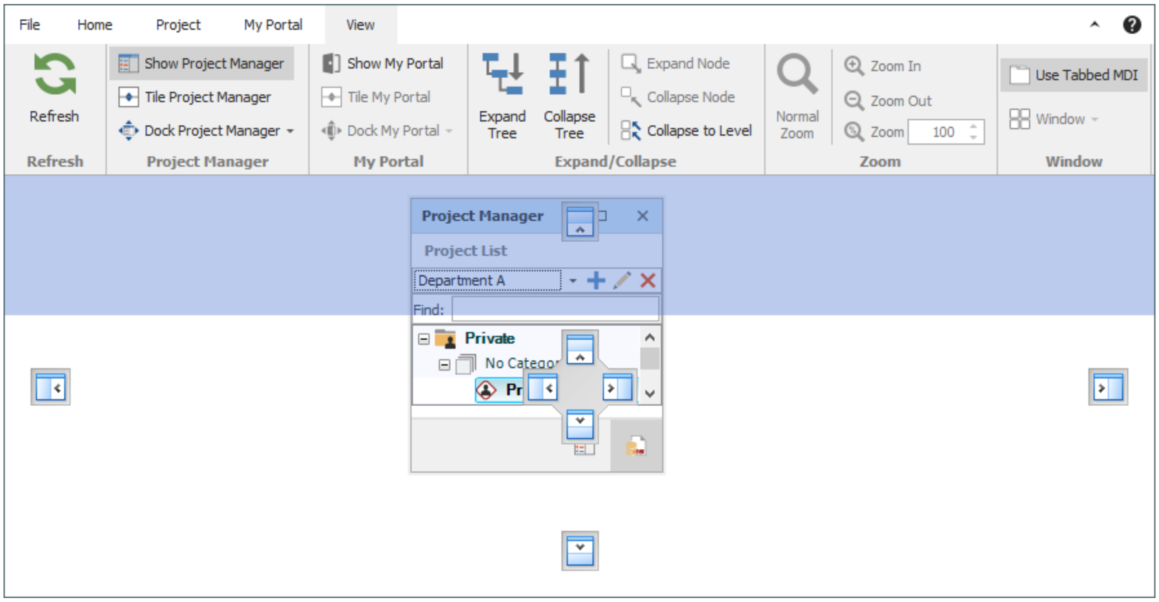


  The same options are available when you choose **View > My Portal > Dock My Portal**.

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

    or

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



If you prefer to float the window, you can:

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

    or

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

Pinned or Unpinned

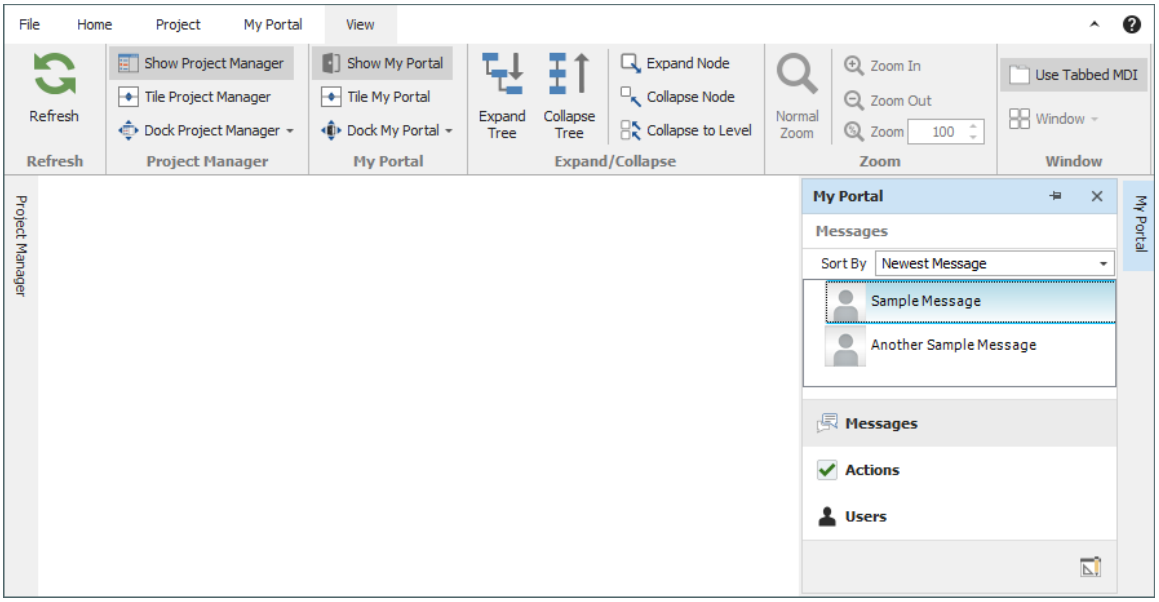
When docked, both panels can also be toggled between pinned and unpinned states by clicking the pushpin icon in the panel's title bar. When the pushpin in the icon is vertical , the panel is pinned and will be displayed at all times.

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

The behavior for unpinned panels is as follows:

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

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



Project Manager

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

- In all ReliaSoft desktop applications, the project list displays all the projects in the current database. (For XFMEA/RCM++ and MPC, this is the only page available in the Project Manager).
- In Weibull++, BlockSim and Lambda Predict, the Open Project(s) page of the Project Manager (i.e., the current project explorer) displays all the analysis folios, reports and other items in the current project(s) that apply to the software you are using.

There may be up to five nodes in the project list, indicating the sharing status of the projects: **Private**, **Public** or **Reference** projects, **Locked** projects and **Checked Out** projects. Additionally, the Recycle Bin is shown in the project list.

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.

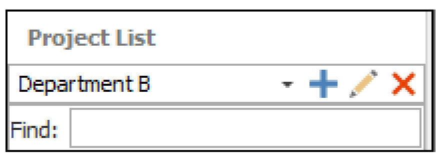
Opening a Project

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 ReliaSoft application but you will see only 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 project list can utilize the same project filters that are available in many other locations throughout ReliaSoft desktop applications. For example, with the filter shown below, the project list will show only projects belonging to Department B.

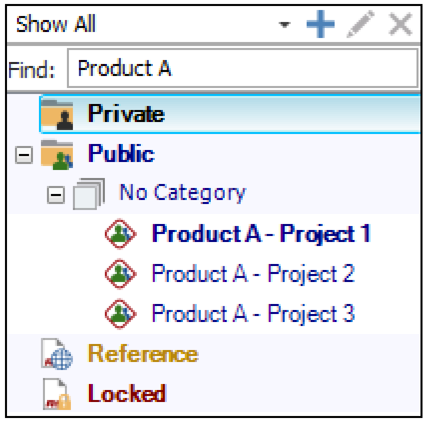


This feature might be particularly useful for an enterprise database that may be used to store analysis projects for the entire organization in a single centralized location. In such cases, the number of projects displayed in the project list could become overwhelming and these filters provide the ability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

To remove the filter, select **Show All** 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.



Current Project Explorer

In all ReliaSoft desktop applications except XFMEA/RCM++ and MPC (which use the system hierarchy to manage all of the different analyses in a given project), the Open Project(s) page of the Project Manager allows you to manage all of the relevant *project items* (e.g., folios, diagrams, etc.) in the project(s) that are currently open.

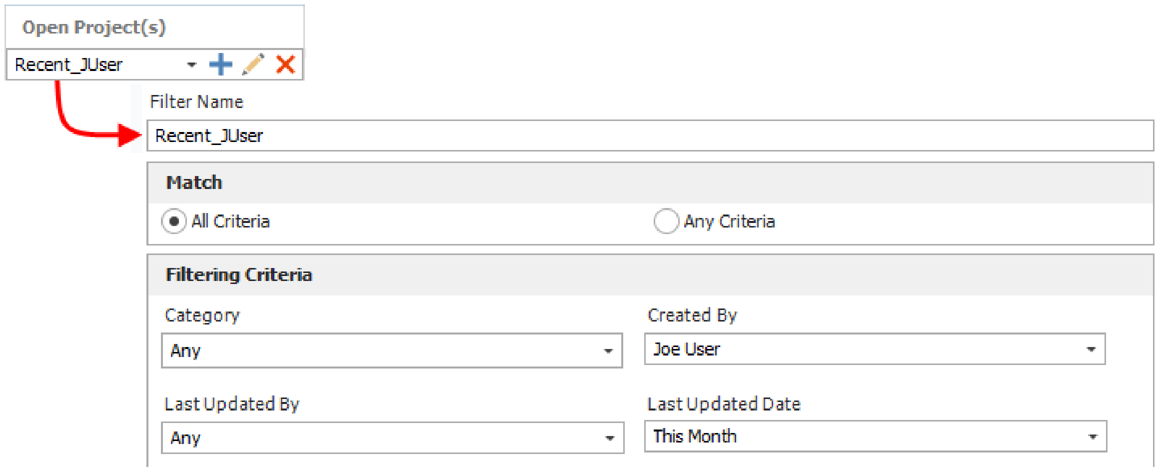
Applications that use the current project explorer can have multiple projects open at the same time. (See Working with Multiple Projects.)

Note: Remember that each project may contain items from any ReliaSoft desktop application but you will see only 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 Project Items

The current project explorer can utilize the same item filters that are now available in many other locations throughout ReliaSoft desktop applications. (This provides more flexibility than the **Filter based on creator** feature in prior versions.)

For example, with the filter shown below, the current project explorer will show only items created by Joe User and updated within the current month.



To remove the filter, select **Show All** from the drop-down list.

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 ReliaSoft 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. (See Status Indicators.)

Adding Items

You can add items to the project by right-clicking the relevant folder and choose an appropriate command on 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.

Sub-Folders and Drag and Drop

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** on 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.

Working with Multiple Projects

ReliaSoft desktop applications that use the current project explorer (i.e., Weibull++, BlockSim and Lambda Predict) can have multiple projects open at the same time. To enable this functionality, select the **Allow multiple projects (project explorer)** check box on the Common Settings page of the Application Setup.

Tip: There is a separate setting that controls the ability to have multiple projects open in XFMEA/RCM++.

When you have more than one project open, the projects will be shown in the current project explorer with the most recently opened project at the top. Note that the icon shown beside each project indicates whether it is a public, private or reference project and, when applicable, whether it is locked or checked out.

	Unlocked, Not Checked Out	**Locked**	**Checked Out**
Public Project			
Private Project			
Reference Project			N/A

In addition to the convenience of being able to view analyses from multiple projects, having multiple projects open allows you to transfer data between the projects quickly and easily. Specifically:

- For any item that can be imported/exported between projects, you can simply drag the item (analysis, multiplot, report, etc.) from the source project to the appropriate folder in the destination project. This opens the Export window, where you can just click **Export** to finish the export, or add more items to export before you finish the process.
- In Weibull++, data can be copied from an analysis in one project and pasted to an analysis in another project.
- In BlockSim, blocks can be copied from a diagram in one project and pasted to a diagram in another project, according to the following rules:
  - If the block uses resources that are local to the source project, they will also be copied and added to the destination project.
    - If a resource of the same name exists in the destination project, the name of the copy will be incremented (e.g., if "Model1" already exists, the new copy might be renamed to "Model1_1").
    - If a model is associated with a data source (e.g., a model published from a Weibull++ folio), the model is copied into the destination project as a model with the association removed.
  - If the block uses resources that are global or in a reference project, they will not be duplicated. They will retain their original reference to that global/reference resource.
  - Subdiagram blocks cannot be copied and pasted across projects.

Note: The FMRA view is not available in BlockSim when you have multiple projects open.

My Portal

In all ReliaSoft desktop applications, *My Portal* provides quick access to messages, assigned actions and other information that may be of personal interest to you while working in the ReliaSoft application. (When you first activate the software, My Portal will be "docked" on the right side of the MDI and "unpinned." Move your mouse pointer over the tab to make the panel visible at any time. For other options, see Show, Tile, Dock and Pin Panels.)

If your organization chooses to implement a web-based SEP portal for an enterprise database, you can also access many of these same features — as well as selected metrics, reports and other details from specific analyses — from any web-enabled device.

Portal Messages

The ReliaSoft desktop application's messages feature provides a convenient way to communicate with other database users. Messages are displayed in a "timeline" format similar to social media websites like Facebook®.

These messages are displayed both in My Portal (in any ReliaSoft desktop application) and in the SEP web portal (if it is implemented for your database).

All database users will be able to view their own messages. In a secure database, the following permissions are required to create, edit or delete messages:

- **Create portal messages** allows you to create new messages, and edit or delete the messages you have personally created.
- **Manage all portal messages** allows you to edit or delete any messages that are visible to you. This includes any message for which you are the creator or one of the recipients.

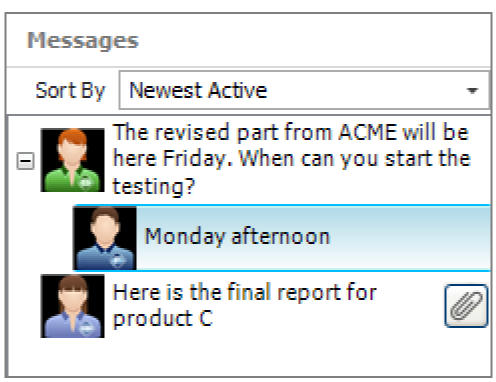
Message Display Options

The messages timeline will display all messages that were created by or addressed to you. You can sort the message threads in two ways:

- **Newest Active** considers the dates/times of both the main messages and the replies.
- **Newest Message** considers only the dates/times of the main messages.

Create, Edit, Delete or Reply to Messages

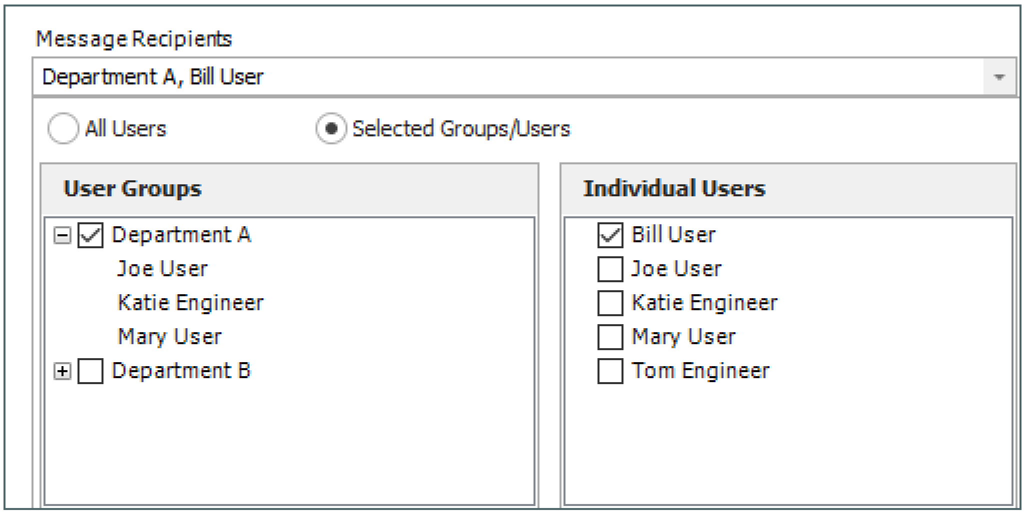
In ReliaSoft desktop applications, you can access the message commands from the My Portal tab of the ribbon, and by right-clicking inside the Messages page of the panel.



Note that the purpose of the **Refresh Messages** command is to update the display with any new messages that were recently added or changed by another database user.

Message Recipients

You can specify whether a message will be visible to all database users or only to specific users or groups. If you select Selected Groups/Users, you can choose from two lists: user groups that have been predefined in the database and/or individual user accounts.



If a valid SMTP server has been defined for the database, you can select the **Also send this via e-mail** check box to send an e-mail copy of the message to the recipient(s). (See Enable Alerts via E-mail or SMS.)

Actions in My Portal

Your team can use actions to track specific assignments that need to be performed. These versatile resources can be used multiple times in different locations, if appropriate. This may include actions in Project Plans, FMEAs or Test Plans, as well as actions that are independent of any particular analysis or plan.

This topic discusses how to view and manage actions via My Portal in any ReliaSoft desktop application. If you want to view/manage all action records, see Actions Explorer or Resource Manager. For information about the action properties and other general considerations that apply wherever the action may be used, see Action Resources.

Actions Displayed in My Portal

The Actions page in My Portal (and in the SEP web portal) display action records that:

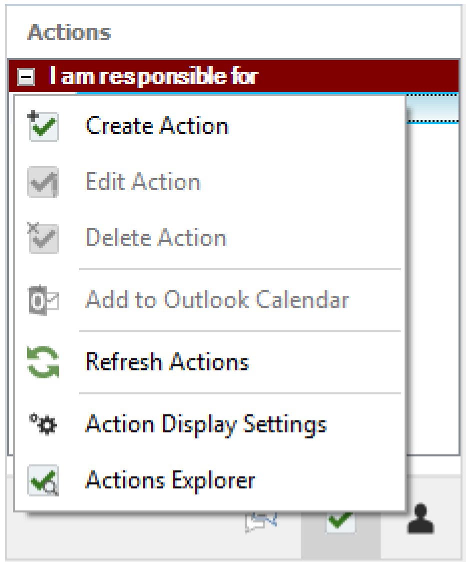
- Need to be reviewed by you.
- Are relevant to you in some way and are:
  - **Past Due** – the completion date has passed.
  - **Due Next 7 Days** – the completion date occurs within the next 7 days.
  - **Past Start Date** – the start date has passed.
  - **Start Today** – the start date is today.
  - **Start Tomorrow** – the start date is tomorrow.
  - **Start Next 7 Days** – the start date occurs within the next 7 days.
  - **In Progress** – the actual start date has been entered and the action is not yet "Past Due" or "Due Next 7 Days."

If the action is not used in the Project Planner, this will be based on the *planned* start/completion dates. But if the action is used in a plan, it's based on the *expected* dates (which can shift automatically if there are recorded delays in prior gates/actions).

Also note that if an action fits under more than one heading, it will display under the more urgent one. For example, if the action is both due in the next 7 days and past the planned start date, it will show under "Due Next 7 Days."

Create, Edit or Delete Actions from My Portal

In ReliaSoft desktop applications, you can access the portal actions commands from the **My Portal** tab of the ribbon, and by right-clicking inside the **Actions** page of the panel.



Tip: If an action doesn't fit the My Portal display criteria for your computer/username (see details below), the new record won't be displayed in your portal after you create or edit it. In such cases, you can use the Actions Explorer or Resource Manager to access the record.

Add to Outlook Calendar

The **Add to Outlook Calendar** command will be enabled if Microsoft Outlook is installed on your computer.

It launches Outlook's interface for creating a new calendar event and automatically populates the subject and date. You can modify the details as needed 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.

Refresh Actions

The **Refresh Actions** command updates the display with any new actions that were recently added or changed by another database user.

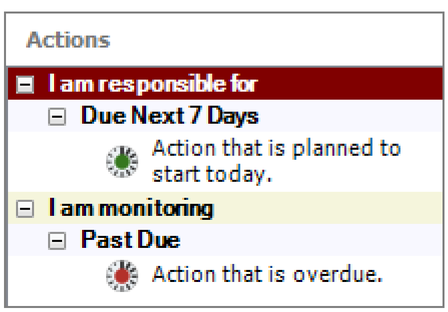
Action Display Settings

To specify your personal preferences for displaying actions in My Portal (stored per computer/username), right-click inside the panel and choose **Action Display Settings**.

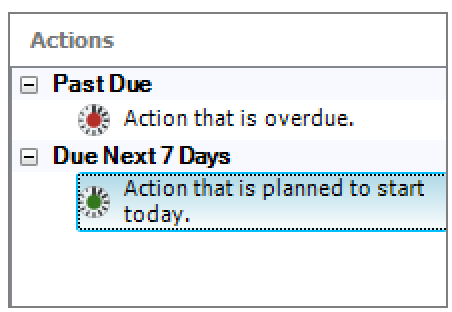
An action may be relevant to you in one or more of the following ways. Use **Show Actions** to specify which actions to display.

- **I am responsible for** – you are assigned in the **Person Responsible** field.
- **I am a team member in** – you belong to the team assigned in the **Team** field.
- **I need to review/approve** – you are assigned in the **Reviewer** field.
- **I am monitoring** – you are assigned in the Action Monitors window, or you have personally subscribed to “watch” the action.
- **I am the creator** – you are listed in the **Created By** field.

If you select **Display ‘Show Actions’ labels in portal**, the panel will also include headings that indicate how the action is relevant to you.



With the 'Show Actions' labels



Without the 'Show Actions' labels

Finally, you can choose to **Sort by Start Date** or **Sort by Due Date** within each node of the display.

Tip: If an action is relevant to you in more than one way, it will display if any applicable relevance check box is selected. If you are using relevance headings, the action will display under the first selected heading that fits. For example, if you are both the person responsible and the creator, and all relevance options are selected, the action will display under "I am responsible for." Alternatively, if only the creator relevance is selected, it will display under "I am the creator."

Action Explorer

The **Actions Explorer** command opens a flexible utility that allows you to explore all of the action resources that are stored in the current database. You can use it to filter, sort, add, edit or delete actions. (See Actions Explorer.)

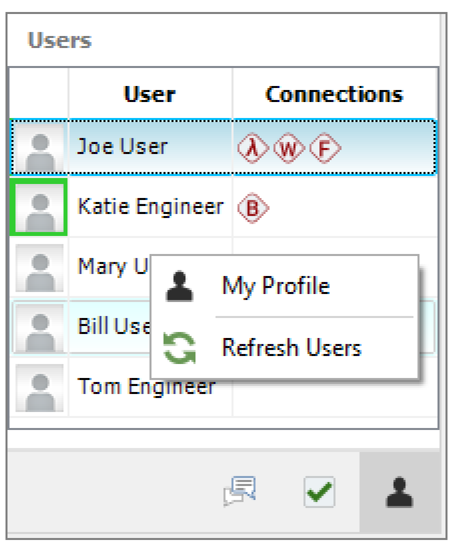
Note that when you return to the Actions page in My Portal, you will need to use the **Refresh Actions** command to view any changes.

Database Users List

The database users list shows all active user accounts in the database, and identifies which of those users are currently connected to the database with any of the ReliaSoft applications.

This list is displayed both in My Portal (in any ReliaSoft desktop application) and in the SEP web portal (if it is implemented for your database).

In ReliaSoft desktop applications, you can access the users list commands from the **My Portal** tab of the ribbon, and by right-clicking inside the **Users** page of the panel. Similar commands are available in the SEP web portal, if applicable.



My Profile opens the User Login and Contact Information window, which allows you to edit the contact information associated with your user account.

Refresh Users updates the list to reflect any recent changes in user activity.

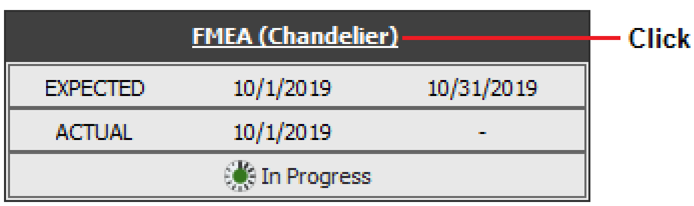
Project Plan Summary in My Portal

In all ReliaSoft desktop applications except MPC, the Project Plan Summary page in My Portal provides a summary of the plan for the current project, if applicable.

The summary in My Portal always reflects the entire plan for the current project, based on the actual date/time from your computer. If you want to see the summary for a particular gate or action, or to estimate what the statuses would be for a different date, you must use the Project Planner window.

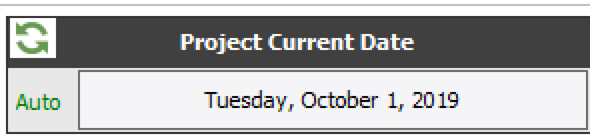
- Choose **Project > Management > Project Planner**.

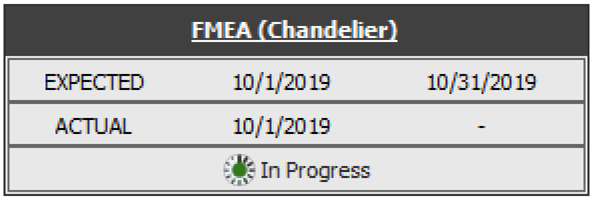
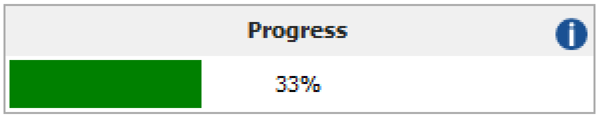
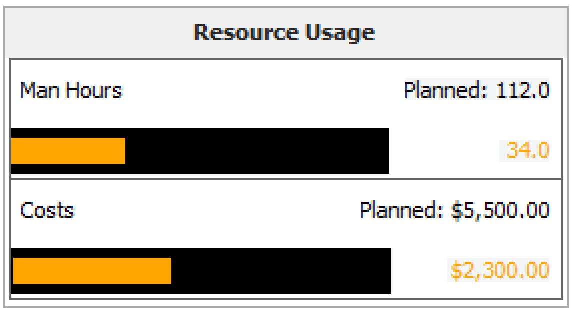
- Click the link in the project name bar at the top of the summary in My Portal.

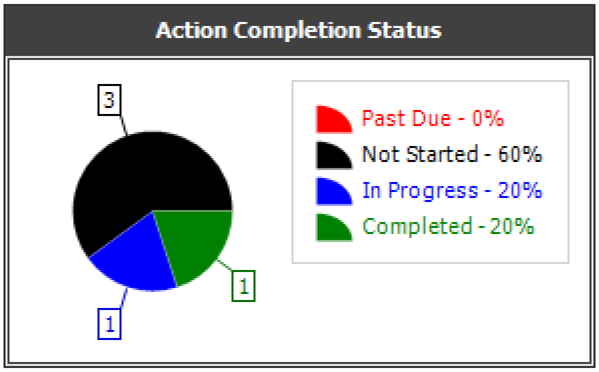


How to Read the Summary

The plan summary in My Portal provides the following information at-a-glance. (For details, see Project Plan Summary Panel.)

	By default, this shows the current date, which is used to determine the status (e.g., "Missed Start Date"), resource usage to date, etc. Use **Refresh** to obtain the latest information from the Project Planner.

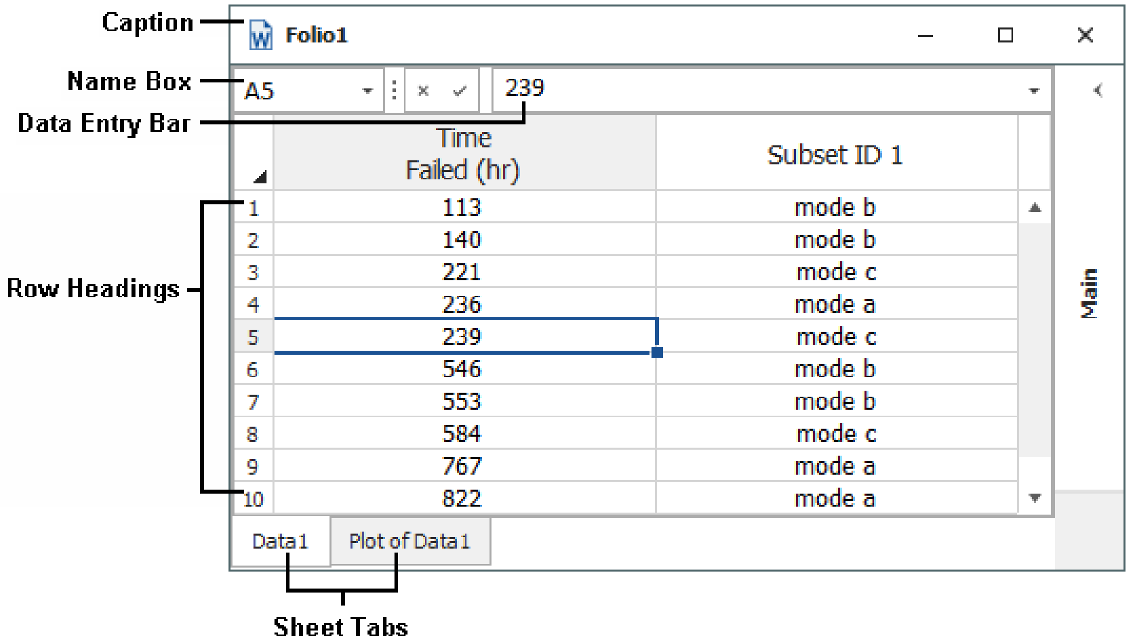
	The top bar displays name of the project that the summary applies to. Click the link to open the project in the Project Planner. The next two rows show the expected dates and any actual dates, and the last row shows the status.
	This is the percentage of the total duration of dependent actions/gates that is complete. For example, if Action 1 (duration = 3 days) is complete and Action 2 (duration = 1 day) is incomplete, the progress is 75%.
	For resource usage, the black bars show the planned man hours and costs. The colored bars show either the expected usage to date (estimated based on the number of days in progress), or the actual usage if complete. (See Costs and Man Hours.) •Yellow = In progress and usage to date is still within plan •Orange = In progress and usage to date has exceeded plan •Green = Complete and actual usage was within plan •Red = Complete and actual usage exceeded plan

	Displays only if actions have been assigned to the project or gate (either directly or via dependent gates). •Not Started = Actions that have not started and can complete on time •In Progress = Actions that are in progress and can complete on time •Completed = Completed actions that have been approved, or don't require approval •Past Due = Actions that have not completed and can no longer complete on time

Folios

Data Sheets

The following picture shows the basic components of a data sheet using one particular format as an example.



- **Caption** shows the name of the folio and the name of the sheet that is currently displayed.
- **Name Box** shows the location of the currently selected cell by listing the column letter and row number that intersect at the cell's location.
- **Data Entry Bar** shows the contents of the selected cell, which may be text, a numerical value or a formula. For text or numerical values, you can enter/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. Each data sheet contains up to 65,536 rows. The application's speed of execution is inversely proportional to the number of data rows in the current data set.
- **Column Headings** indicate the titles of the columns. To rename a heading, right-click it and choose **Rename Column**. When applicable, you can save the column titles and use them as the default headings for all new data sheets of the current type. (See Default Column Headings.)
- **Sheet Tabs** represent the different sheets within the folio. To rename a sheet, double-click the tab. The number of folios that can be opened at once depends on the amount of memory of your system (up to a maximum of 256 for some folio types).

Default Column Headings

In life data, life-stress and growth data folios, creating a new data sheet automatically applies default column headings that are generally appropriate for the analysis. If there are certain types of data that you frequently use (e.g., miles-to-failure, cycles-to-failure, etc.), you can customize the column headings to always use your preferred column titles.

To modify the defaults for a particular data type, first rename the column headings in an existing data sheet. Right-click the heading and choose **Rename Column** on the shortcut menu. Then save the new titles as the defaults. For a life data folio, choose **Life Data > Format and View > Set Headers as Default**. (For a life-stress folio, the command is in the Life-Stress Data tab; for a growth data folio, it's in the Growth tab.)

These preferences are saved for the current computer/username, and will be applied the next time you create a new data sheet for that particular data type.

To apply the new default headings to an existing analysis, open the data sheet and choose [**Life Data/Life-Stress Data/Growth Data**] **> Format and View > Apply Default Headers**.

If you ever need to restore the default column headings back to the shipped settings, choose **File >Application Setup**, then click **Reset Application Settings** or **Reset All Settings** on the Reset Settings page.

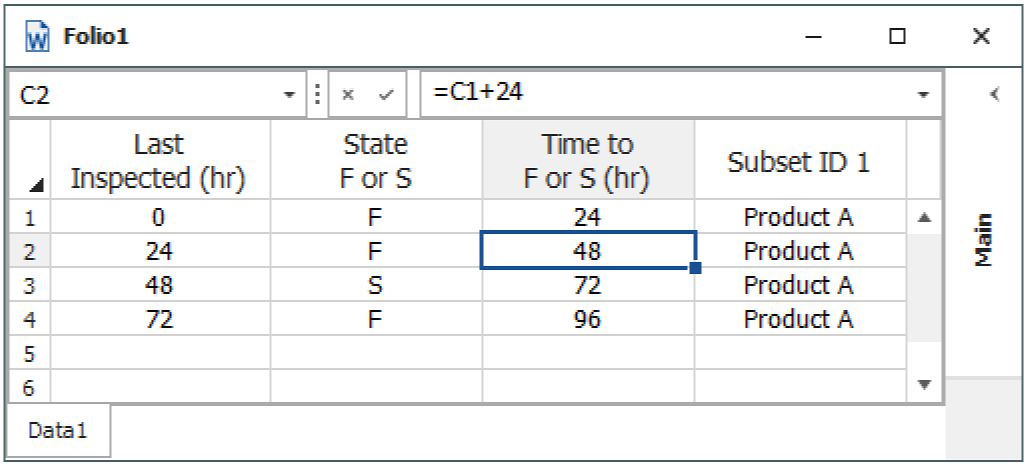
Resizing Columns

Column widths are set to span evenly by default. If you want to resize and have the changes saved with the folio, right-click a column heading and choose **Column Width Style > Default**.

To resize a column, drag the edge of the heading to the desired position, or right-click it and choose **Column Width**.

Entering Formulas in the Data Sheet

Data sheets in Weibull++ 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 life data folio, or in the Classification column in a growth data folio). For example, in a life data 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 be 24+24 = 48.

Change Units

In Weibull++, you have the ability to define the 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 units that will be available for use in any project within the database and set up the conversion factors.

The units are displayed in the heading of the relevant column in the data sheet. If you wish to change the units of an existing data sheet, click the **Change Units** icon on the Main page of the control panel.

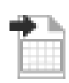
In the Change Units window, the units that are currently being used in the data sheet will be displayed at the top of the window. In the **Change Units To** field, you can choose any other defined unit to be used in the data sheet. In the **Conversion Options** area, specify what will happen when the 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 life data, life-stress data and growth data 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.

General Spreadsheets

General Spreadsheets can be inserted into a Weibull++ life data folio, life-stress data folio or growth data folio. They provide the same spreadsheet capabilities that are available in the spreadsheet module in ReliaSoft Workbooks, but 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.

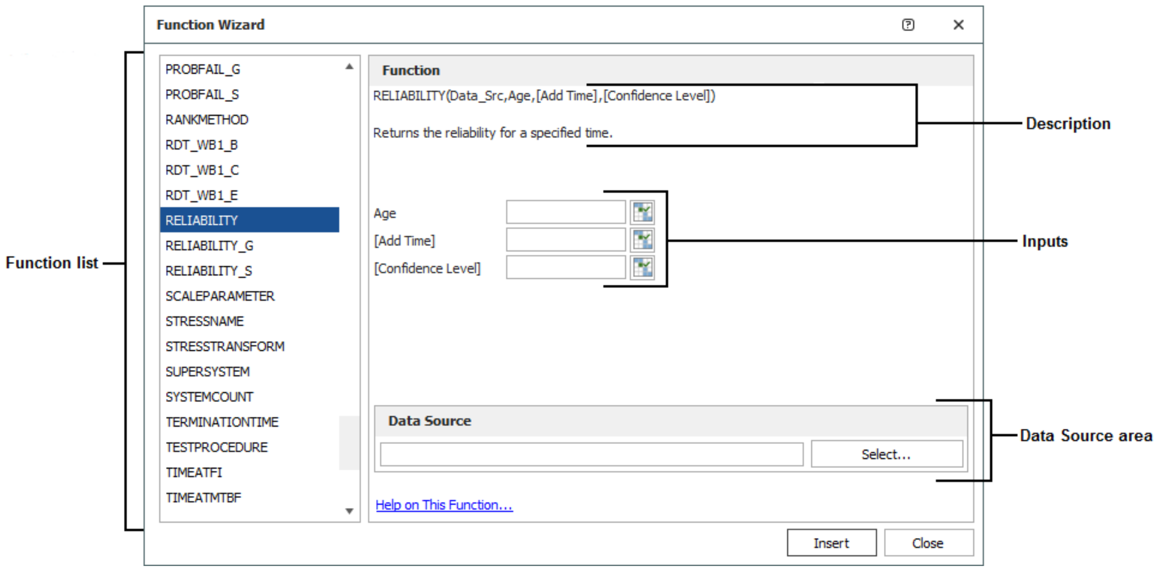
To add a General Spreadsheet to a folio, right-click the data sheet tab area (the area at the bottom of the window that shows the name of the data sheets in the folio) and choose **Insert General Spreadsheet** on the shortcut menu. This command also appears on the ribbon tab for the particular type of data folio. For example, to insert a general spreadsheet into a Weibull++ life data folio, you can choose **Life Data > Folio Sheets > Insert General Spreadsheet**.



Inserting Data Source Functions

To build and insert functions that utilize a referenced analysis (data source), click a cell in the general spreadsheet and choose **Sheet > Sheet Actions > Function Wizard**.

The following picture shows the most complex configuration as an example:



For the function arguments:

- Brackets [ ] indicate that the input is optional.
- You can use cell references as inputs. For example, instead of entering 1000 for the Age 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.)
- You can also use variable names as inputs. (See Defined Names.)

For the data source:

- Click the **Select** button and then select which data sheet in the folio to use as the data source.
- Any time you make changes to the data source, the general spreadsheet will need to be recalculated to reflect the most current results. Choose **Sheet** > **Format and View** > **More Settings** > **Recalculate Formulas**. The general spreadsheet may return "N/C" if the data source needs to be recalculated.

Inserting Math Functions

To add math, date, logic and other functions, click a cell in the general spreadsheet and choose **Sheet** > **Sheet Actions** > **Insert Function**.

Select a function from the drop-down list and click **OK**. You can enter inputs for the function arguments by either selecting the cells in the sheet or typing them directly into the appropriate field.

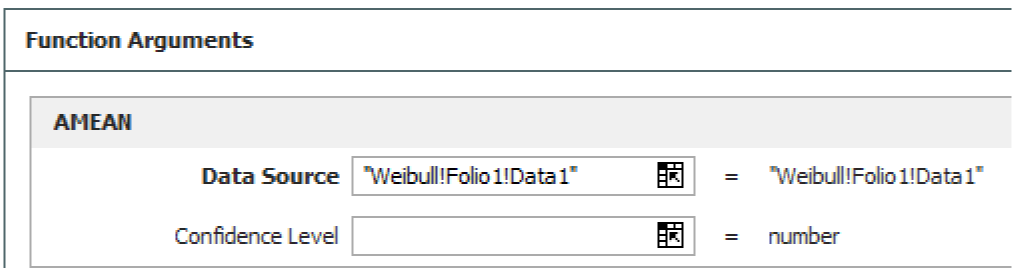
REFERENCING A DATA SOURCE

Some functions (e.g., `AMEAN`, `D_COUNT`, etc.) require you to reference a particular data source.

To reference a data source, use the following syntax, with the quotes and exclamation marks (!). You can only reference sheets in the same folio.

```
"Application!Folio!DataSheet"
```

In the following example, the data source is a data sheet called "Data1" in the Weibull++ folio called "Folio1".



Control Panels

Many of the analysis folios, diagram sheets and other interfaces in ReliaSoft desktop applications utilize a control panel that allows you to make required inputs, initiate the desired analysis or simulation and view/access applicable results. This topic describes some features that are common to most control panels. (For XFMEA/RCM++, MPC and Lambda Predict, see System Panel and Analysis panel instead.)

Switching Between Pages

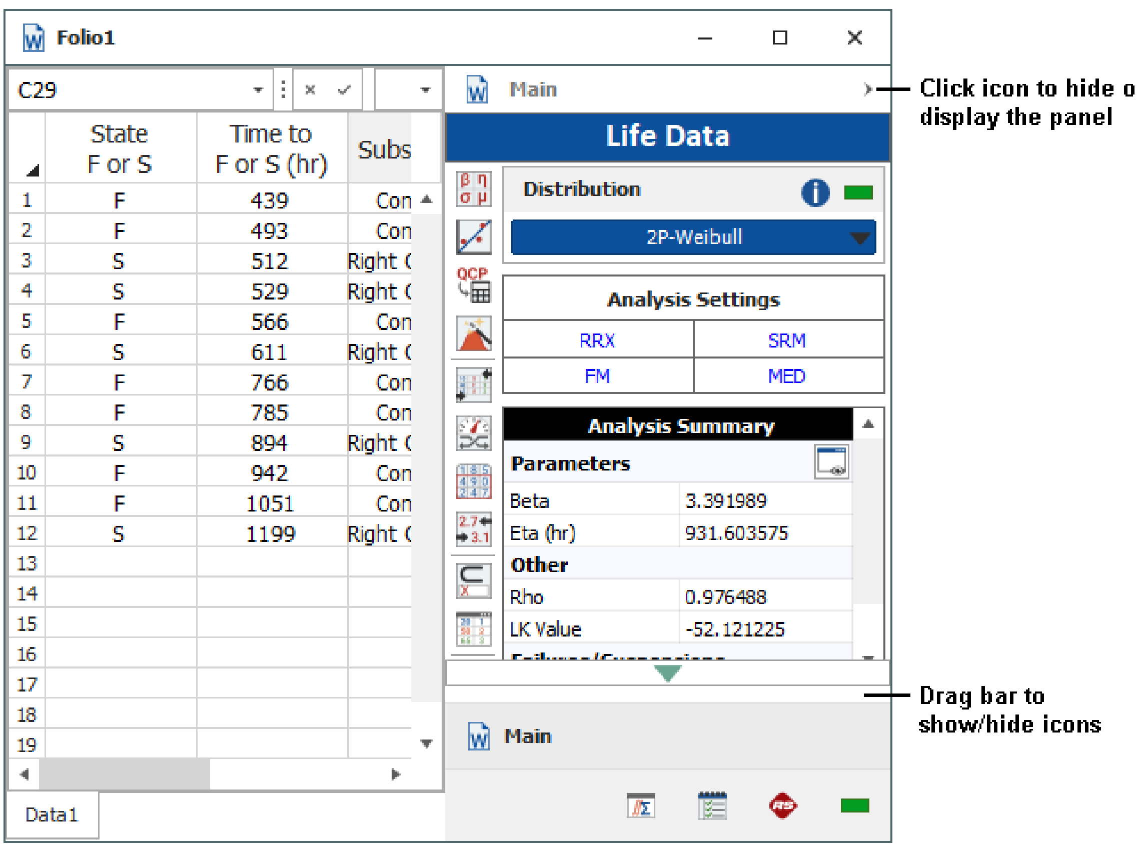
Many control panels and navigation panels contain multiple pages that vary depending on the type of analysis. You can switch between pages by clicking either the large buttons or the small icons at the bottom of the panel.

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.

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.

When the control panel is hidden, only the title bar and page icons will be visible on the right side of the window (as shown next using a Weibull++ folio as an example). When you click the bar, the active page will be displayed temporarily. When you click anywhere outside of the control panel page, it will be automatically hidden again.



Identifiers Page

The Identifiers page of the control panel allows you to enter identifying details (e.g., category, name, part number, etc.) that will be associated with the analysis and any models or SEP summaries that are published from the analysis. This will help you to find, filter and group analyses and resources throughout the database.

In folios that can have multiple data sheets, the Folio Identifiers are edited via the Item Properties window, and the Data Sheet Identifiers can be entered directly in the control panel. Use an asterisk (*) if you want the data sheet's identifiers to be the same as the folio's (see Identifiers).

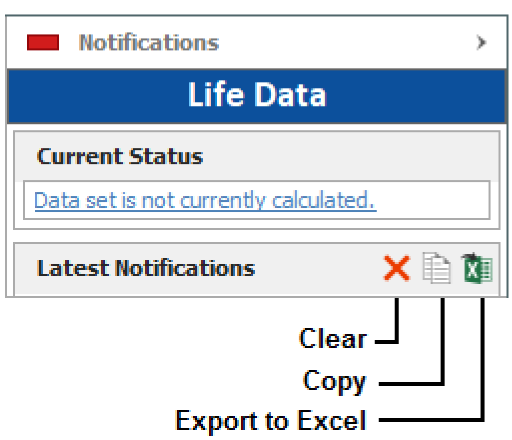
Publishing Page

The Publishing page allows you to view and manage information from the current analysis that is shared throughout the ReliaSoft desktop applications.

- **Model/Fitted Model** – publishes a model based on the current analysis. See Publishing Models.
- **SEP Summary** – publishes a summary of the current analysis to the SEP web portal. This is visible only if the enterprise database is configured for SEP and you have the "Publish to SEP web portal" permission. See Publishing to SEP.
- **Metrics** – shows all of the metric resources that are associated with the current analysis. See Showing Metrics in Folios/Diagrams.

Notifications Page

When applicable, the Notifications page provides information on the **Current Status** of the analysis, along with any warnings or errors. 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. The **Latest Notifications** area displays any warnings and errors generated during analysis.



Utilized Resources Page

In BlockSim only, the Utilized Resources 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 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 unless some block in the diagram belongs to them.

This list is not updated automatically; if you have made changes to the diagram, click the **Refresh** icon to update the list.

Double-click a resource in the list to open its properties window for viewing or editing. Click the **Resource Manager** icon to view/manage all of the resources available in the project.

System Panel and Analysis Panel

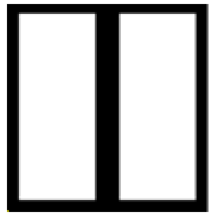
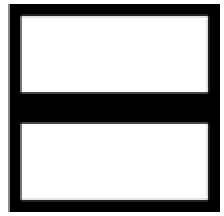
Some ReliaSoft applications use two convenient panels to manage all of the analysis information in a particular project (XFMEA/RCM++ and MPC) or in a particular prediction folio within the project (Lambda Predict).

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.

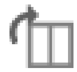
This topic discusses the ways in which you can configure the layout of these two panels to fit your particular workspace preferences.

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**.

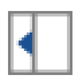
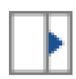


Resize a Panel

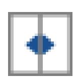
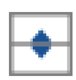
You can resize the panels by dragging and dropping the vertical (or horizontal) separator into the desired position.

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 

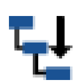
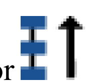
When you wish to return to the two-panel layout, choose the **Split Panels** command.

 or 

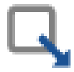
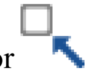
Expand or Collapse Nodes

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.

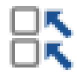
To expand or collapse all branches at the same time, choose **Expand Tree** or **Collapse Tree**.

 or 

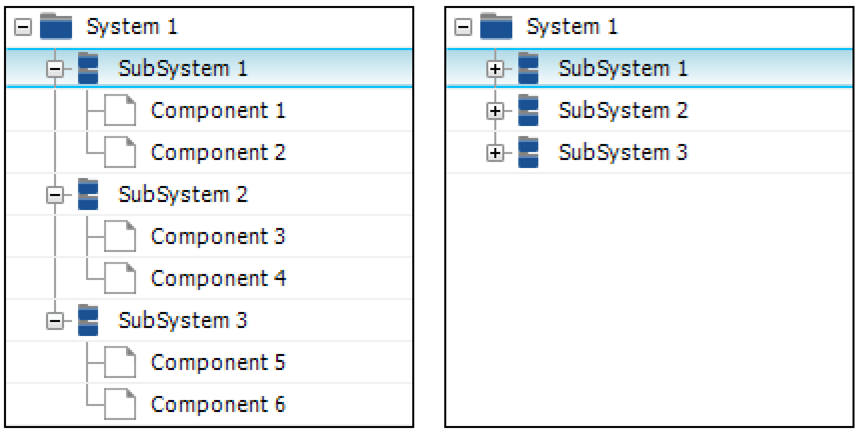
To expand or collapse a specific branch, you can click the + or - icons, or select the item and choose **Expand Node** or **Collapse Node**.

 or 

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 heading 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.

Printing

ReliaSoft desktop applications offer different printing utilities for data sheets, diagrams, plots and ReliaSpft Workbooks.

Data Sheets and Diagrams

To print a data sheet or diagram, choose **File > Print** or press **CTRL+P** on the keyboard. You will see a preview of the printed page, and can choose to print or to access page setup options.

When printing an RBD or fault tree in BlockSim, clicking the **Print** button on this page will open the Print Diagrams window, where you will be able to choose whether to print only the **Current** diagram (and possibly its subdiagrams on separate pages) or to **Choose multiple** diagrams from the current project to print. If you choose to print the current diagram and its subdiagrams, you can select **Print subdiagram page number** to include the starting page number for each subdiagram in the caption of the relevant subdiagram block.

To print an XFMEA/RCM++ diagram (e.g., FMEA block diagrams, etc.), choose **Home > Print > Print**.

Plots

To print a plot, choose **File > Print** or press **CTRL+P** on the keyboard.

To print a plot displayed in the Plot Viewer (as seen in XFMEA/RCM++ and Lambda Predict), click the **Print** icon on the plot control panel.

ReliaSoft Workbooks

To print ReliaSoft Workbooks, choose **View > Print > Print Preview**. For more printing options, you can send the report to Microsoft Word or Excel and use the print utilities in those applications. Choose **Home > Report > Send to Excel** or **Send to Word**.

Reports Generated in Word or Excel

In XFMEA/RCM++, MPC and Lambda Predict, reports are generated in Microsoft Word or Excel. You will need to use the print utilities built in to those applications.

Other Features

For other features that offer the print functionality (e.g., RS Draw, Resource Manager, etc.), the **Print** icon will be displayed in the window itself.

Page Setup - Plots and Diagrams

The Page Setup for plots and diagrams is accessible via the print window (see Printing).

Maximize Printing Space

The following options are available on the Page tab or Margins tab of the Page Setup.

- **Orientation**. The Portrait orientation works best for small charts and diagrams. If the image is wider than it is tall (which is the case for most plots), select the Landscape orientation.
- **Keep aspect ratio** (available for plots). Select this option to maintain the proportional relationship between the width and height of the image. Clear the check box if you want to stretch the image to fill the paper.
- **Scaling** (available for diagrams) allows you to shrink or enlarge a diagram to fit a specified number of pages. For example, to print a diagram to four pages in a rectangle, you would enter the values 2 by 2 pages. The software will automatically adjust the size of the diagram image to fit to the specified pages.
  - **Print without interior margins** is used for printing images that span multiple pages, so that the pages can be assembled into a whole picture. It removes as much margin space as possible from the interior edges of each page, where it meets the other pages.
  - **Print empty pages** prints any blank pages within the print area; otherwise, the blank pages are omitted from the printout.
- **Margins**. By default, the margins will be printed based on the **System's default** option, which is the measurement system specified in your computer's region and language settings. You can change the unit to inches or centimeters as desired.

Headers/Footers

To add custom headers/footers for printing, click the **Header/Footer** tab of the Page Setup.

To insert variable information such as date, time, number of pages, etc., use the command icons in the window to insert the appropriate field codes (or use the codes shown in the table below).

Field Code	Description
&A	Inserts the sheet name. (Does not apply to plots in XFMEA/RCM++ and Lambda Predict.)
&D	Inserts the current date.
&T	Inserts the current time.
&P	Inserts the page number.
&N	Inserts the total number of pages.

&ZA	Inserts the name of the application (e.g., Weibull++).
&ZC	Inserts the company name.
&ZU	Inserts the user name.
&ZP	Inserts the project name.
&ZI	Inserts the name of the item. (Does not apply to plots in XFMEA/RCM++ and Lambda Predict.)
&ZL	Inserts the path to the current database.

Page Setup - Data Sheets

The Page Setup for data sheets is accessible via the print window (see Printing). For the spreadsheet module in ReliaSoft Workbooks, choose **View > Print > Page Setup**.

Maximize Printing Space

The following options are available on the Page tab or Margins tab of the Page Setup.

- **Orientation**. The Portrait orientation works best when the data has more rows than columns. If the data sheet is wider than it is tall, select the Landscape orientation.
- **Scaling** allows you to shrink the font size by a certain percentage to fit the data sheet into one page, or enlarge it to span the data sheet across multiple pages.
- **Paper size** and **Print quality**. The options depend on the type of printer you use.
- **First page number** allows you to specify any number to be the starting page number.
- **Margins** adjusts the widths of the margins on the paper.
- **Center on page** allows you to center the data sheet in the printout horizontally, vertically or both.

Print Order and Other Options

The Sheet tab of the Page Setup provides more printing options:

- **Print area** allows to specify a range of cells to print. Recommended if you want to print only a specific selection of data and not the entire data sheet. For example, you could enter the range `A1:B10` to print only cells A1 through B10. To print the entire data sheet, clear this field.

- **Print titles**. If your data sheet is more than one page long, you can repeat row or column titles on each printed page to make it easier to read your data. For example, to repeat the column titles of a data table that has several rows spanning multiple pages, you could enter the range `$1:$1` into the **Rows to repeat at top** field, where `$1:$1` is the location of the column title in the table.
- **Gridlines** prints the gridlines of a data sheet.
- **Draft quality** prints the data sheet at a lower quality for faster printing.
- **Row and column headings** prints the row numbers and column headings of the data sheet.
- **Comments**. If you've added comments to the data sheet, you can choose to print them along with the data sheet. Choose whether to print the comments at the end of the sheet or as they are displayed in the sheet.
- **Cell errors as** allows you to specify how any cell errors in the data sheet should be displayed in the printouts. Choose whether to print them as displayed, <blank>, dashes (--) or as #/NA
- **Page order**. If your data sheet spans multiple pages, you can specify the order in which the pages are numbered and printed.
  - **Down, then over** prints from the top down, then left to right.
  - **Over, then down** prints from left to right, then from the top down.

Headers/Footers

The Header/Footer tab of the Page Setup provides several options for adding custom headers/footers to the printed page.

- **Header** and **Footer** drop-down lists allow you to select a predefined header/footer format. The software includes several predefined formats to choose from.
- **Different odd and even pages** and **Different first page** gives you the option to apply different headers/footers for odd and even pages, or choose to apply a different header/footer to the first page only. Select the check box for the desired option and then click the **Custom Header/Footer** button to edit the headers (see the section below). To apply the same header/footer to all pages in the document, clear the check boxes for both options.
- **Scale with document** auto-adjusts the font size to scale with the data sheet.
- **Align with page margins** auto-adjust the position of the header/footer to be consistent with the page margins.

EDIT THE HEADERS/FOOTERS

You can edit the headers/footers for the current printing session. Note that the edits are discarded when you switch to different header/footer format or when you exit the print preview.

To edit an existing header/footer, choose it from the drop-down list and click the **Custom Header/Footer** button. Alternatively, you can select **none** from the drop-down list and then click the button.

To insert variable information such as date, time, number of pages, etc., use the command icons in the window to insert the appropriate field codes (or use the codes shown in the table below).

Format Code	Description
&[Page]	Inserts the page number.
&[Pages]	Inserts the number of pages.
&[Date]	Inserts date.
&[Time]	Inserts time.
&[Path]&[File]	Inserts the file path.
&[File]	Inserts the file name.
&[Tab]	Inserts the sheet name.
&[Picture]	Inserts a selected picture. Click the **Format Picture** icon on the Header/Footer window to set the image size and other properties.

Page Setup - Word Processing Module

To access the Page Setup for the word processing module in ReliaSoft Workbooks, choose **View > Print > Page Setup**.

Maximize Printing Space

The following options are available on the Margins tab or Paper tab of the Page Setup.

- **Margins** adjusts the widths of the margins on the paper.
- **Orientation**. The Portrait orientation works best when the data has more rows than columns. If the data sheet is wider than it is tall, select the Landscape orientation.
- **Paper size**. The options depend on the type of printer you use.

Layout

The Layout tab of the Print Setup provides more printing options:

- **Section** divides the report into several sections, where each section can have different formatting elements such as the page orientation, margins, headers/footers, etc. The easiest way to insert section breaks is to exit the Page Setup and choose **Page Layout > Page Setup > Breaks**.
- **Headers and footers**. You can choose to apply different headers/footers for odd and even pages, or choose to apply a different header/footer to the first page only. To apply the same header/footer to all pages in the document, clear the check boxes for both options.To configure and save header/footer information into the report, exit the Page Setup and choose **Document > Header & Footer > Header** or **Footer**.To set up a header/footer that applies only to the current printing session (i.e., the information will be lost when you exit the preview), exit the Page Setup and choose **View > Print > Print Preview**. In the preview window, click the **Header/Footer** icon on the ribbon. To insert variable information such as date, time, number of pages, etc., use the command icons in the window to insert the appropriate field codes (or use the codes shown in the table below).

Format Code	Description
[Page #]	Inserts the page number.
[Pages #]	Inserts the number of pages.
[Page # of Pages#]	Inserts date.
[Date Printed]	Inserts the current date.
[Time Printed]	Inserts the current time.
[User Name]	Inserts the username of the current user.

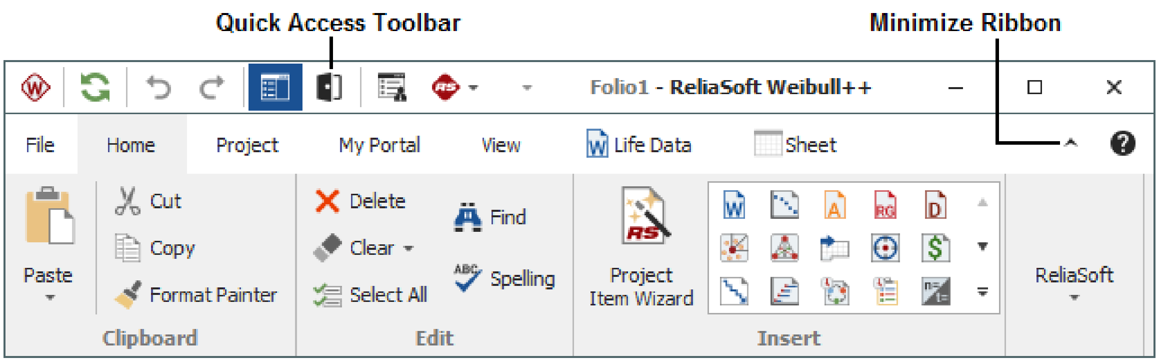
Weibull++ Ribbon

IN THIS CHAPTER

In ReliaSoft desktop applications, the *ribbon* is divided into tabs that relate to a task or activity, such as managing the projects in a database or performing a specific type of analysis. Depending on the current activity, some commands may appear dimmed or will not appear at all.

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.

To minimize the ribbon and maximize your workspace, click the arrow on the right-side of the ribbon (labeled "Minimize Ribbon" in the picture below). The ribbon will be expanded when you click a tab and then minimized after you click a command.



Quick Access Toolbar

The Quick Access Toolbar is a customizable feature of the ribbon that allows you to access frequently used commands without changing tabs.

To add your favorite ribbon commands to the Quick Access Toolbar, right-click the command and select **Add to Quick Access Toolbar**.

To remove a command from the Quick Access Toolbar, right-click the command and select **Remove from Quick Access Toolbar**, or click the **Customize Quick Access Toolbar** icon and clear the check box for the command.

To reposition the Quick Access Toolbar, click the **Customize Quick Access Toolbar** icon and select to show it either below the ribbon or above the ribbon.

To reset the Quick Access Toolbar to default settings, click the **Reset All Settings** button in the Application Setup (**File** > **Application Setup**).

Backstage View (File Tab)

The *Backstage view* (File tab) is the first view you will see when you start a ReliaSoft desktop application.

Save As (available only when a standard database is currently open) saves a copy of the standard database to another pathname/filename and/or file type. You can save the file as a new standard database (*.rsr21) or compressed standard database (*.rsgz21).

Pack and E-mail (available only when a standard database is currently open) sends a compressed version of the standard database via e-mail.

Open Database opens an existing standard database, enterprise database or compressed standard database, from the current version or any previous version. You can also open even earlier file types, depending on the application you are currently using.

Close Database 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 (*.rsr21 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 Common Settings page of the Application 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 (*.rsr21) or connection file for access to an enterprise database (*.rserp).

Manage Database provides a variety of tools and configurable settings for the database. The settings are shared by all users and all analysis projects in a database.

Print (available only in Weibull++ and BlockSim) generates a print preview of the current data sheet, diagram or plot, and sends it to the printer.

Help provides a variety of resources to help you use the application more effectively. In addition, it displays information about the application, including the Compile Release Version (CRV) and License information.

Application Setup allows you to set your personal 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.

Exit closes the current database (if any) and shuts down the application.

Manage Database

The Manage Database section of the Backstage View contains the following commands. The available commands depend on the current application and the database type.

Users

Users and Security controls which users can access the database and their permissions.

User Groups create and manage groups of users that can be assigned to actions, messages, etc. throughout the current database.

Database Logins displays a list of users who have connected to the database and allows you to export this information to Excel.

Reset "In Use" Flags allows you to reset the flags that indicate the "in use" status of projects or items within a project.

Lambda Predict Database Settings

Available only in Lambda Predict:

FIDES Settings Manager defines the settings related to the FIDES prediction standard such as process audits, pi factors, categories, etc. (See FIDES Settings Manager in the Lambda Predict documentation.)

Custom Derating Standards Manager allows you to add, edit, delete or import/-export custom derating standards. (See Creating Custom Derating Standards in the Lambda Predict documentation.)

MIL-217 Custom Connections defines custom connection types for use with the MIL-HDBK-217F prediction standard. (See MIL-217 Custom Connection Types in the Lambda Predict documentation.)

XFMEA/RCM++ Libraries

Available only in XFMEA/RCM++:

Profiles/Library Manager configures the predefined settings of the profiles stored in the software library. Profiles allow you to customize the look, drop-down lists, rating scales and other settings of a particular project in order to fit the needs of the analysis. (See Profiles/Library Manager in the XFMEA/RCM++ documentation.)

Templates Manager allows you to manage the configurable templates that can be used for customized reports, saved queries or importing/exporting data via Excel. (See Templates Manager in the XFMEA/RCM++ documentation.)

Settings and Predefined Options

Database Settings allows you to enable alerts via e-mail or SMS, activate history logs, and other settings.

Unit Settings defines the unit and measurement settings that will be available for use in any project within the database.

Project/Item Categories defines the categories that can be used to filter and group data throughout the database.

Default Name Formats (not applicable for MPC) specifies the default names for new resources, as well as new blocks in BlockSim RBDs, fault trees and phase diagrams.

Project Planning Resources (not applicable for MPC) defines the cost categories, teams, materials and facilities that are used for tracking costs, man hours and resource utilization in actions and Project Planner gates.

Working Days/Holidays (not applicable for MPC) specifies the business days when project planning resources can be utilized.

Dashboard Layout Manager (not applicable for MPC) shows all of the predefined dashboard layouts that will be available for any user to view for a particular data set.

Task Types (applicable only in RCM++ and MPC) maps the task types used in RCM++ (e.g., Restoration, Failure Finding, etc.) to the corresponding task classes in the universal reliability definition (e.g., Preventive, Inspection, etc.) so that simulation and cost calculation results are accurate. (See Task Types in RCM++ in the XFMEA/RCM++ documentation.) In MPC, it allows you to modify the abbreviations used for the MSG-3 task types. (See Task Type Abbreviations in MPC in the MPC documentation.)

XFRACAS Connection is available only for standard databases (*.rsr21) in Weibull++ and XFMEA/RCM++. It allows you to define connection settings for an enterprise database.

Database Tools

New Enterprise Database opens a wizard that leads you through the steps to create a new enterprise database (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.

Upgrade Enterprise Repository converts an existing Version 9 or later enterprise database to a Version 2021 enterprise database. ***This cannot be undone***. It is recommended that you create a backup of the database before performing the upgrade.

Upgrade Version 8 Repository copies all of the existing data from a Version 8 enterprise database to a new Version 2021 enterprise database.

Restore Points manages all of the project backups that are stored in the current database.

Compact and Repair is available only in a standard database (*.rsr21). It helps to reduce the size of the database and protect against data loss and corruption. (See Backups and Database Maintenance.)

Import from Version 5 is available only in XFMEA/RCM++. It allows you to import data from a Version 5 XFMEA/RCM++ database. This command is available only when you are connected to an enterprise database and have the "Manage users and logins" permission and the applicable "Manage all projects" permissions.

Help Center

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**.

Using This Application

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 computer.

Quick Tour opens the Quick Tour within the application's help file. The Quick Tour provides a brief overview of the application's features, including basic steps for performing the most common analysis types. This command is available only if the current application offers a Quick Tour.

Online Resources

reliasoft.com opens the main page of the reliasoft.com website.

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.

reliawiki.org opens the ReliaWiki website, which is both a resource portal and a wiki for professionals in reliability engineering and related fields.

Examples

Open Examples Folder provides access to a set of example projects that are designed to help you explore various software features.

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 Technical Support provides the contact information you will need to obtain technical support.

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 Manager**. The License Manager displays additional information about which products you have currently registered and/or your contact information on file with us.
- **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.

Home Tab

The Home tab contains the following commands:

Clipboard

Paste

- **Paste** pastes the contents of the Clipboard to the current folio or spreadsheet.
- **Paste Special** opens the Paste Special window, which allows you to paste specific cell contents or attributes (such as formulas, formats, or comments) from the Clipboard.

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 on the Clipboard can be pasted into this and other applications. If you are copying a plot, the way the plot is copied will depend on your selection in the **Copy Plot Graphic** setting in the Application Setup window.

Format Painter copies the format properties of text in a sheet and applies it to other text. To use the Format Painter, first select the text with the format properties that are to be copied, then choose **Format Painter** and click the text to which the format properties are to be applied.

Edit

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.

Select All selects all of the cells in the currently active sheet.

Find searches the active data sheet for specified text/values and, if desired, replace the original with new text/values. You can specify the order in which to search (either **By Rows** or **By Columns**), and whether to look in equations (**Formulas**) or within the results of equations (**Values**). Select the **Match Case** check box to limit the results to text that have the same case. Select the **Find Entire Cells Only** check box to limit the results to text/numbers that are identical to the search term.

To replace every instance of a text string, click **Replace** and in the **Replace With** field, type the text that will replace the original (if you don't type anything, the tool will delete the original). Click **Find Next** to search for the text. When a match is found, you can either click **Replace** to replace the original with the new text, or click **Find Next** to leave the original as-is and find the next occurrence. To replace all instances without review, click **Replace All**.


Spelling activates the Spell Check utility, which allows you to check the spelling within the current sheet.

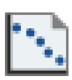
Insert


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.

Insert Gallery displays icons of the analyses, plots and diagrams that you could add to the project. Click the **More** arrow to view the full gallery.

Life Data


Life Data adds a life data folio for life data analysis. It 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.


Non-Parametric LDA adds a non-parametric life data analysis (LDA) folio. It allows you to analyze life 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.

Life-Stress Data


Life-Stress Data adds an Accelerated Life Testing life-stress data folio for accelerated life testing data analysis. It allows you to capture life data 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.

Growth Data

Growth Data adds a Reliability Growth data folio for growth data analysis.

Designs

Standard Design adds a DOE standard design folio for identifying important factors (i.e., "screening") and determining the best settings for these factors (i.e., "optimization").

Robust Design adds a DOE robust design folio that includes all the functionality of the standard design folio, with the addition of some features specific to the robust design methodology.

Mixture Design adds a DOE mixture design folio for when the factors in an experiment are components in a mixture, and when you wish to determine the best proportions to use for each component.

Free Form adds a free form folio that allows you to analyze data from an experiment by manually entering the factor level combination and observed response for each test run.

Optimization opens the Select Optimization Tool window, which provides tools that allow you to examine factor level combinations and evaluate how well they meet your response goal(s).

Warranty

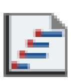
Warranty adds a warranty data analysis folio for capturing and analyzing failure data from fielded products, in order to predict future warranty returns (product failures) and plan for warranty fulfillment needs such as the number of spares to stock.

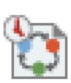
Degradation

 **Degradation** adds a degradation analysis folio. If you have Accelerated Life Testing on your computer, all four options will be available:

- **Nondestructive Degradation folio** allows you to estimate the point at which a product is expected to fail based on the level of degradation of a physical or performance characteristic (e.g., depth of tire tread, voltage of battery, propagation of crack size) over time. Multiple measurements are taken per test unit over time, and the folio analyzes the extrapolated failure times using standard life data analysis to obtain reliability predictions about the product.
- **Destructive Degradation folio** is used in cases where measuring the level of degradation of a specimen is not possible without invasive techniques or destroying the specimen (e.g., corrosion in a chemical container, strength measurement of adhesive bond). Unlike the nondestructive degradation analysis folio, only one measurement is taken per test unit and multiple samples are required at different points in time.
- **Accelerated Nondestructive Degradation folio** is used for cases where the specimen is subjected to elevated stress conditions. The elevated stress conditions accelerate the degradation of the product, reducing the test time required to obtain the measurements. The folio analyzes the extrapolated failure times using accelerated life testing data analysis to obtain reliability predictions about the product at normal operating conditions.
- **Accelerated Destructive Degradation** is similar to the Weibull++ destructive degradation folio except that accelerated stress levels are used during testing to further reduce the test time required to obtain the predicted failure times.

Repairable Systems

 **Non-Parametric RDA** adds a non-parametric recurrent event data analysis (RDA) folio. The folio is similar to a parametric RDA folio, except that it 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.

 **Parametric RDA** adds a parametric recurrent event data analysis (RDA) folio for capturing and analyzing 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.

 **Event Log** adds an event log folio, which 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.

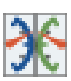
Test and Planning

 **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.

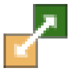
 **Target Reliability** adds a target reliability estimator folio, which generates plots to help you select a reliability goal that will minimize cost, maximize profit and maximize the return on an investment.

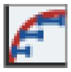
 **Linearity and Bias** adds a linearity & bias folio that is used to estimate a measuring device's *accuracy* (i.e., the extent to which the device underestimate or overestimates, across the device's expecting operating range).

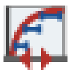
 **Gage R&R** adds a gage R&R folio that is used to estimate a measurement system's *precision* (i.e., the variation in measurements that can be expected when the same part is measured repeatedly).

 **Gage Agreement** adds a gage agreement folio that is used to compare the accuracy and precision of two systems.

 **ALTA Test Plan** adds an ALTA test plan folio, which 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.

 **Maintenance Planning** adds a maintenance planning folio, which generates a cost vs. time plot that helps you determine the cost-effective time to replace aged components.

 **Continuous Growth Planning** adds a continuous growth planning folio to the project.

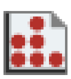
 **Discrete Growth Planning** adds a discrete growth planning folio to the project.

Repairable Systems Test Design opens the Repairable Systems Test Design folio that can assist you in designing a demonstration test.

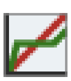
Comparison

 **Stress-Strength Comparison** adds a stress-strength comparison folio for comparing two data sets to determine the probability of failure based on the probability of stress exceeding strength.

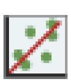
 **Life Comparison** adds a life comparison folio for comparing two data sets to determine whether the lives of the units in the first data set will outlast the lives of the units in the second.

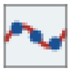
 **One-Way ANOVA** adds a DOE one-way ANOVA folio that offers a simple way to evaluate the effect of a single factor on a single response.

Mission Profile

 **Mission Profile** adds a mission profile folio to the project.

Regression Tools

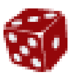
 **Multiple Linear Regression** adds a multiple linear regression folio, which allows you to enter and analyze your own experimental data without using one of the DOE design types provided in available in Weibull++.

 **Equation Fit Solver** adds a non-linear equation fit solver folio, which 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++.

Diagram

 **Diagram** adds a reliability block diagram (RBD). In Weibull++, 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.

Simulation

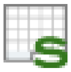
 **Monte Carlo** opens the Monte Carlo utility, which allows you to generate a data set that fits a specified life distribution or model. The generated data set can then be analyzed like any other data set and used in a variety of applications, such as in the analysis of probabilistic design models, or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

- **Monte Carlo** generates a data set for life data analysis.
- **Stress-Dependent Monte Carlo** generates a data set for accelerated life testing data analysis.
- **Repairable Systems Monte Carlo** generates a single data set containing values that are distributed according to the Crow-AMSAA (NHPP) model with specified beta and lambda parameters.

 **SimuMatic** opens the SimuMatic utility, which allows you to generate several data sets all at once and automatically perform an 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.

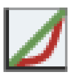
- **SimuMatic** performs basic life data analysis on each of the generated data sets.
- **Stress-dependent SimuMatic** performs accelerated life testing data analysis on each of the generated data sets.

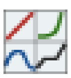
- **Repairable Systems SimuMatic** generates several sets of data at once and automatically perform a large number of reliability analyses on them.

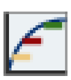
 **Simulation Worksheet** adds a simulation worksheet that allow you to vary values that are used in simulating a BlockSim simulation RBD or an event analysis flowchart. This allows you to investigate the effect of one or more settings on the simulation results.

Reports and Plots

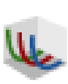
 **ReliaSoft Workbook** adds a ReliaSoft Workbook, which is a custom reporting tool that contains both spreadsheet and word processing modules.

 **Overlay Plot** adds an overlay plot that allows 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.

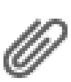
 **Side-By-Side Plot** adds a side-by-side plot that allows you to display different plots of a single data set all in a single window for easy comparison. For life data analysis, you can select to plot the data set using multiple distributions or plot types. For Accelerated Life Testing, you get a third option to plot the data set using multiple life-stress relationship models.

 **Multi-Phase Plot** adds a new Multi-Phase Plot to the project. This allows you to plot data from multiple test phases together, along with data from a growth planning folio, if desired.

 **3D Plot** adds a 3D plot that allows you to graph functions with three variables such as the reliability at a given time and stress level.

 **3D Overlay Plot** adds a 3D overlay plot. This type of plot is similar to regular overlay plots but displayed in 3D space.

Attachments

 **Attachment** adds linked or attached files to the project.

ReliaSoft

Quick Tools

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. 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.

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.

Quick Parameter Estimator (W++) opens the life data 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.

Quick Parameter Estimator (ALTA) opens the Accelerated Life Testing 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.

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. This command is not available in MPC.

Batch Properties Editor opens the Batch Properties Editor, which allows you to perform batch editing of most types of local resources in the current project. In BlockSim, it also allows you to edit the blocks used in the diagrams. This command is not available in MPC.

Actions Explorer allows you to explore all of the action resources that are stored in the current database. You can filter and sort by date, status, person responsible, relevancy to you, etc. (See Actions Explorer.)

Analysis Explorer allows you to explore all of the different analyses that are stored in the current database. You can filter, group and sort the analyses in a flexible grid, and also present the information in a wide variety of dashboard charts. (See Analysis Explorer.)

Reliability Data Warehouse allows you to access data from ReliaSoft's XFRACAS or other external data sources for life data analysis. (See Reliability Data Warehouse (RDW).)

Launch Application provides a drop-down list of the other ReliaSoft applications that are installed on your computer. When you click an icon, the same database will be opened automatically in the new application.

Project Tab

Unless otherwise indicated, these commands apply to the project that is currently selected in the project list. 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.

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.

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.

Transfer Project is available only in XFMEA/RCM++. It allows you to create a new project with data transferred from the selected project. (See Transfer Projects in the XFMEA/RCM++ documentation.)

Task Numbering is available only in MPC. It allows you to specify whether or not you want to reuse task numbers within the project. (See Task Numbering in MPC in the MPC documentation.)

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.

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 Import, Export and Data Conversion.)

Configurable Settings is available only in XFMEA/RCM++. It allows you to view/-modify the configurable settings for the current project. (See About Configurable Settings in the XFMEA/RCM++ documentation.)

Restore Point 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.

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.

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.

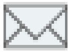
Delete Project permanently deletes the selected project. ***There is no undo for delete*** unless you have a stored backup or restore point.

History Log is available only if you are the project owner or have the applicable "manage all projects" permissions. 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 ReliaSoft application.

Project Planner opens the Project Planner, which provides full project planning and management capabilities for the current project. This command is not available in MPC.

Attachments is available only in XFMEA/RCM++ and MPC. It adds linked or attached files to the selected project. In other ReliaSoft desktop applications, attachments commands are available in the current project explorer.

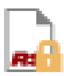
E-mail Project

 **Pack and E-mail** compresses a copy of the selected project to a *.rsgz21 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.

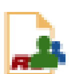
Security

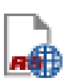
 **Project Security** is available only for public projects in secure databases. It opens the Project Properties window with the Security tab active, where you can specify the user accounts that can view/modify the selected project.

 **Change Owner** allows you to assign a different database user to be the owner for the selected project.

 **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 Locked and Unlocked Projects.)

 **Make Private** moves the selected public project into the Private heading of the project list. (See Public, Private and Reference Projects.)

 **Make Public** moves the selected private or reference project into the Public heading of the project list.

 **Make Reference** moves the selected public project into the Reference heading of the project list. This command is not available in MPC.

Current Item

The following commands are available when you select a specific item (a diagram, report, multiplot, etc.) in the current project explorer. This group is not available in XFMEA/RCM++ or in MPC.

Edit brings the selected item to the front of the windows and activates it as the current control. If an attachment is selected, the command opens the attachment in the appropriate application, if that application is installed on your computer.

Rename allows you to rename the selected item.

Duplicate creates a copy of the selected item within the project. The duplicate will have the same name as the original with an increment number added to the end (e.g., RBD1_1, RBD1_2, etc.). In BlockSim, you can choose to **Duplicate With Resources**, which also creates duplicate resources in the Resource Manager window.

Delete deletes the selected item. ***There is no undo for delete***.

Item Properties is available only for project explorer items (e.g., folios, diagrams, plots, etc.). It opens the Item Properties window, which allows you to view and edit the item's Identifiers, item settings (if applicable) and item permissions (in secure databases).

Save Locator Link is a link file (similar to a Windows shortcut) that provides quick access to the specific analysis. In an enterprise database, you can choose to **Save** or **E-mail** the file. In a standard database, the only option is to save the file. In XFMEA/RCM++, these commands are available in the System Hierarchy tab. In MPC, they are in the Systems, Structures, Zones or L/HIRF tab.

Data Management

The following commands are available only in BlockSim.

Mirror Group Manager opens the Mirror Group Manager window, which allows you to add, view and edit groups of mirrored blocks within the project that is currently open/visible. (See Mirroring (Using Blocks in Multiple Locations) in the BlockSim documentation.)

Maintenance Group Manager opens the Maintenance Group Manager window, which allows you to add, view and edit the maintenance task groups available to the project that is currently open/visible. (See Maintenance Groups.)

Flow Group Manager opens the Flow Group Manager window, which allows you to add, view and edit the flow groups available to the project that is currently open/visible. (See Flow Groups in the BlockSim documentation.)

Reports

MRBR is available only in MPC. It allows you to generate a maintenance review board report in Microsoft Word. (See Maintenance Review Board Reports in the MPC documentation.)

My Portal Tab

The My Portal tab contains commands related to using the My Portal feature.

Messages

Create Message opens the Message window, which allows you to compose and send messages within the database.

Edit Message opens the Message window, which allows you to view and edit all of the properties of the message.

Reply to Message opens Message Reply window, which allows you to respond to a message sent to you.

Delete Message deletes the current message.

Actions

Create Action opens the Action window and adds an action to the project.

Edit Action opens the Action window, which allows you to view and edit all of the properties of the action record.

Delete Action deletes the current action record.

Add to Outlook Calendar 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.

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.

Tile Project Manager tiles the project list and current project explorer so they are both displayed simultaneously in the Project Manager panel. This command is not available in XFMEA/RCM++ or MPC.

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.

Tile My Portal tiles the Messages, Actions and Users 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.

Workspace Layout

The commands in this group are available only in XFMEA/RCM++, MPC and Lambda Predict.

Change Orientation toggles between the two possible orientations for the System panel and the Analysis/Properties panel. These panels can be side-by-side or one on top of the other.

Hide System [Panel/Hierarchy] hides the System panel in the Project window. When you want to show the panel again, choose **View > Workspace Layout > Split Panels**.

Hide [Analysis Panel/Properties] hides the Analysis/Properties panel in the Project window. When you want to show the panel again, choose **View > Workspace Layout > Split Panels**.

Split Panels splits the Project window into two equal panels, with 50% of the available space used for the System panel and 50% of the available space used for the Analysis/Properties panel.

Properties View is available only in Lambda Predict.

Tree View organizes the properties depending on whether they are physical properties (e.g., part quality level, case type, etc.) or application properties (e.g., ambient temperature, applied voltage, etc.). Depending on the prediction standard and component you are working with, the properties may also be organized according to failure mechanisms or type of data.

Pi Factor View organizes the item properties based on the pi factors that they contribute to so you can see how the values affect the failure rate calculations. The relevant pi factors will vary depending on the item's failure rate model. (See the Tree View and Pi Factor View topic in the Lambda Predict documentation.)

Expand/Collapse

The commands in this group apply to the hierarchical view that currently has focus. Depending on the application, this can be the project list, the current project explorer, the System panel, the Properties panel (Lambda Predict only) or the Analysis panel if it is using a hierarchical view.

Expand Tree expands the entire tree in the hierarchical view.

Collapse Tree collapses the entire tree in the hierarchical view.

Expand Node expands the selected branch in the hierarchical view.

Collapse Node collapses the selected branch in the hierarchical view.

Collapse to Level collapses the hierarchical view to the level of the item that is currently selected. For example, if you have selected a cause in the FMEA hierarchy view and choose this command, all branches in the FMEA hierarchy view will be collapsed to the cause level and the nodes for controls and actions will be hidden.

Zoom

Normal Zoom sets the degree of magnification to 100%.

Zoom In increases the degree of magnification.

Zoom Out decreases the degree of magnification.

Custom Zoom allows you to specify the degree of magnification.

Window

Use Tabbed MDI maximizes the windows to fill the full space available in the MDI and shows a tab for each open window. If you clear this command, the open windows will be displayed as separate windows that can be resized and moved around within the MDI.

Window

Cascade cascades all open project windows inside the MDI.

Tile Horizontally horizontally tiles all open project windows inside the MDI.

Tile Vertically vertically tiles all open project windows inside the MDI.

Arrange Icons automatically organizes the icons for all open, minimized project windows inside the MDI, sending them to the bottom left corner.

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).

FMRA

The commands in this group are available only in BlockSim. (See Failure Modes and Reliability Analysis (FMRA) in the BlockSim documentation.)

Show FMRA opens the FMRA view, which displays the FMRA (failures modes and reliability analysis) hierarchy that is associated with the project.

Disassociate FMRA deletes all of the diagrams and blocks associated with the current FMRA hierarchy and closes the FMRA view.

Show

The commands in this group are available only in XFMEA/RCM++.

Show FMRA enables/disables the FMRA tab in the System panel.

Application-Specific Tabs

3D Plot Tab

The 3D Plot tab is visible when you view a 3D plot sheet.

Plot

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

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

Actions

Copy Plot Graphic copies the current plot to the Clipboard as a bitmap (*.bmp) graphic.

3D Plot Setup opens the 3D Plot Setup, which offers several options for customizing the 3D plot such as editing axis titles, line styles, grid lines, etc.

Export Plot Graphic opens the Save As window, which allows you to save the current plot graphic as a *.jpg, *.gif, or *.png.

Display

Show Legend displays information about the objects shown on the plot.

Center on Canvas centers the 3D plot horizontally and vertically on the plot sheet.

Auto Spin auto-rotates the plot horizontally.

Normal Zoom sets the degree of magnification to 100%.

Zoom In and **Zoom Out** increase or decrease the degree of magnification.

Custom Zoom allows you to specify the degree of magnification.

Data Tab

The Data tab is visible when you view a DOE design folio. Depending on which type of design folio you view, the following commands may be available.

Analysis

Calculate analyzes the data for each response that is selected to be included in the analysis. To exclude a response from the analysis, clear the check box in its column heading.

Plot creates a plot based on the analysis. If you click this icon before the current data set has been analyzed, an analysis will be performed automatically.

Prediction opens the Prediction window, which allows you to enter values for each factor and see predicted results for the selected response.

Optimization adds an optimal solution plot to the optimization folio that is linked to the design folio. If no linked optimization folio exists, the software will create one.

Overlaid Contour Plot adds an overlaid contour plot to the optimization folio that is linked to the folio. If no linked optimization folio exists, the software will create one.

Design

Select Terms opens the Select Terms window, which allows you to specify the terms (i.e., factors, factorial interactions, blocks and quadratic effects) to be considered for each response.

Alter Data Type is available only for designs that are configured to accept life data. It opens the Alter Data Type window, which is used to specify the type of life data response (e.g., right censored data) that is used in the current experiment.

Select Transformation opens the Select Transformation window, which allows you to select a transformation to apply to each response.

Variability Analysis is available only for two level factorial and Plackett-Burman designs with standard life data. It opens the Variability Analysis window, which allows you to select the response column(s) that you want to see variability analysis for and which factors to consider.

Fold Design is available only for designs that require two-level factors (i.e., two level factorial and Plackett-Burman designs). It opens the Fold Design window, which allows you to replicate the design with runs that use the opposite levels for that factor.

Optimal Design opens the Optimal Design window, which allows you to optimize a built design by taking into account a) the specific effects that are under investigation, b) limitations on the number runs that can be performed and c) constraints on the factor level combinations that can be used during the experiment.

Augment Design presents options where you can augment the design by adding individual runs or replicates. See Augmenting Designs.

Set Shape Parameter applies to reliability DOE designs only. If you already have a reliable estimate for the life distribution's shape parameter, you can enter it here. This can reduce the uncertainty in your results, since the analysis will solve for fewer variables.

Simulation Worksheets

Transfer to Simulation Worksheet transfers the factor settings to a new simulation worksheet.

Folio Sheets

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

Delete Sheet deletes the current sheet within the folio.

Format and View

Visible Columns opens the Visible Columns window that allows you to choose which columns to hide/show in the Data tab.

Display

Actual Values displays the actual values for the factors.

Coded Values displays the coded values for the factors. Which coded values are used (e.g., whether 0 or -1 represents the low level) is dependent on the design type.

Amount applies to mixture designs only.

- **Amount** displays the component factor values as amounts (e.g., 2 pounds).

- **Proportion** applies to mixture designs only. It displays the component factor values as proportions (e.g., 0.2 = 20% of the total mixture used in the run).

- **Pseudovalue** applies to mixture designs only. It displays the component factor values as scaled values between 0 and 1, where 0 is the minimum possible value for the factor (given the design settings) and 1 is the maximum possible value.

ReliaSoft

Publish Model allows you to use the results of the growth analysis as a resource object and make it available for unlimited use elsewhere within the project, in any ReliaSoft application that has a need for that type of object.

Publish SEP Summary publishes a summary of the current analysis to the SEP web portal. This is visible only if the enterprise database is configured for SEP and you have the "Publish to SEP web portal" permission (see Publishing to SEP).

Degradation Tab

The Degradation tab is visible when you view a nondestructive degradation analysis folio data sheet for degradation analysis.

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), and then performs life data analysis or accelerated life testing analysis on the extrapolated times.

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.

Quick Calculation Pad opens the Quick Calculation Pad, which allows you to calculate results identical to the calculations you can perform in a life data folio QCP or a life-stress data folio QCP.

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

Distribution Wizard helps you select a distribution that best fits a data set. See the life data version or life-stress data version of the Distribution Wizard.

Transfer Life Data

Note: If you are using the Accelerated Life Testing degradation folio, the icons show the Accelerated Life Testing version, and the extrapolated data are transferred to a life-stress data folio.

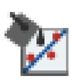

Transfer Life Data to New Folio transfers the extrapolated failure and suspension times to a new life data folio.

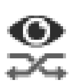

Transfer Life Data to Selected Folio transfers the extrapolated failure and suspension times to an existing life data folio.

ReliaSoft


Publish Model publishes the chosen distribution as a model for use throughout the project, in any ReliaSoft application where models are used. See Publishing Models.

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.


Switch to Unit View/Switch to Inspection View toggles between the views for a nondestructive data sheet. The Unit View displays the information about each measurement in separate rows for each unit and inspection time. The Inspection View displays measurements for multiple units at each inspection time.

Add/Remove Columns

- For life data nondestructive folio data sheets, this allows you to add or remove subset ID columns when using the Inspection View.
- For Accelerated Life Testing nondestructive data sheets, this allows you to add or remove columns for stress levels and, when using the Inspection View, add or remove subset ID columns.

Design Tab

The Design tab is visible is available when creating an experiment design in a design folio.

Responses

Add Response adds a new response to the design.

Remove Response removes the currently selected response from the design.

Factors

Add Factor adds a new factor to the design.

Remove Factor removes the currently selected factor from the design.

Import/Export

Import Template allows you to load the settings from a design that was previously saved.

Export Template allows you to save all your current design settings for use in future designs.

General

Build Designbuilds the experiment based on the current settings.

Clear Settings resets all the settings on the Design tab.

Undo Changes reverts to the settings that were originally used.

Change Orientation allows you to display the navigation and input panels side-by-side or with one panel above the other.

Destructive Degradation Tab

The Destructive Degradation tab is visible when you view a destructive degradation folio data sheet for degradation analysis.

Analysis

Calculate fits the degradation measurements directly to the chosen degradation model and life distribution.

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.

Quick Calculation Pad opens the Quick Calculation Pad, which allows you to calculate results identical to the calculations you can perform in a life data folio QCP or a life-stress data folio QCP.

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

Distribution Wizard helps you select a distribution that best fits a data set. See the life data version or life-stress data version of the Distribution Wizard.

ReliaSoft

Publish Model publishes the chosen distribution as a model for use throughout the project, in any ReliaSoft application where models are used. See Publishing Models.

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.

Diagram Tab

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

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.

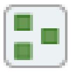
Transfer to BlockSim is available only if BlockSim is activated on your computer. It exports the blocks into a new BlockSim simulation diagram. All the blocks must be associated with a published model before they can be exported.

Blocks

Add Block adds a new standard block to the current diagram. The block represents a data sheet in a life data or life-stress data folio.

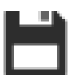
Add Node adds a new node (*k*-out-of-*n* switch) to the current diagram.

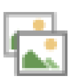
 **Add Junction** adds a non-failing block to the diagram. Because diagrams must flow from a single starting point to a single ending point, you may need to use junction blocks as starting and/or ending blocks (see Diagram Analysis Constraints). They can also be used anywhere within the diagram that you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

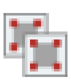
 **Auto Arrange** automatically arranges the blocks in the diagram so that all blocks are evenly spaced and centered.

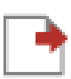
Settings

 **Connect Blocks** allows you to create connectors between blocks in the current diagram. When this command is selected, click the source block, hold down the left mouse button and drag a line from the source block to the destination block. When the crosshairs are located above the destination block, release the mouse button to create a connector. To stop adding connectors and return to the normal mode, click the diagram or clear the Connect Blocks option (i.e., by choosing the command again).

 **Save Diagram** The application always saves the diagram upon calculate or close. This command provides the option to manually save the diagram at any time while you're editing. This can be useful if you're working across a network in an enterprise database.

 **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 *.wmf, *.png, *.gif or *.jpg format. You will be able to use the exported graphic in any application, provided that the application supports the file format.

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.

Properties

Block Properties opens the data sheet that is associated with the block.

Line Bend alters the bend style of the selected connectors. If a custom style is used, the options for removing bend points will become available.

Equation Fit Solver Tab

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

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.

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 create, edit, and select equation templates for use in the analysis.

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

Event Log Tab

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

Analysis

Calculate converts the log entries into failure/repair times and then fits distributions to the converted data. It then 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++ life data 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.

Format Tab

The Format tab is visible when you are working with a reliability block diagram (RBD).

Format

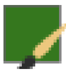
Select Font sets the font properties for the selected blocks.

Font Color sets the font color for the selected text.

Fill Color sets the fill color for the selected text.

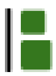
Styles

 **Diagram Style** opens the Diagram Style window, which allows you to customize the appearance of the current diagram and set the default properties for connectors and new blocks that are added to the diagram.

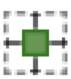
 **Block Style** opens the Block Style window, which allows you to customize the appearance of the selected block or annotation.

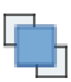
 **Connector Style** opens the Connector Style window where you can customize the appearance of the selected connectors.

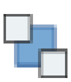
Alignment and Spacing

 **Align Blocks** allows you to easily adjust the position of multiple blocks. The selected blocks will be aligned with the first block selected, except for **Align to Grid**, which moves each selected block to its nearest point on the grid. (See Arranging and Resizing Blocks.)

 **Make Spacing Equal** allows you to easily adjust the spacing between multiple blocks. The selected blocks will be moved so that they are equally spaced over the original size of horizontal and/or vertical area. (See Arranging and Resizing Blocks.)

 **Make Same Size** allows you to resize multiple blocks of different sizes. The selected blocks will be resized to match the height and/or width of the first block selected. See Arranging and Resizing Blocks.

 **Bring to Front** moves the currently selected block(s) or connector(s) in front of other objects on the diagram.

 **Send to Back** moves the currently selected block(s) or connector(s) behind other objects on the diagram.

Growth Data Tab

The Growth Data tab is visible when you view a growth data folio data sheet.

Analysis

Calculate estimates the parameters of the selected model and analysis method, 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.

Quick Calculation Pad opens the Quick Calculation Pad, which allows you to calculate results, such as the cumulative MTBF and the expected number of failures, based on the analyzed data sheet.

Statistical Tests Report performs statistical tests on the active data set and displays the results in the Results Panel. These tests may include the Chi-Squared, Cramér-von Mises, Laplace Trend and Common Beta Hypothesis tests, depending on the data type.

Event Report opens an Event Report for the current analysis in the Results Window. The Event Report gives information on individual failure modes. Event Reports are available for all multi-phase data types and for the Multiple Systems with Event Codes data type.

Crow Extended

Effectiveness Factors is available only if the data sheet contains Projections columns. It opens the Effectiveness Factors window, which allows you to define the effectiveness factors for each BD mode to estimate the fractional decrease in failure intensity.

Convert Modes opens the Convert Modes window, which allows you to convert any classification and mode in the data set to another classification and mode.

Mode Processing opens the Mode Processing window where you can extract the first failure time for each unique BC mode and/or unique BD mode in a data set.

Test for Fix Effectiveness opens the Test for Fix Effectiveness utility, which allows you to assess whether or not corrective actions have been effective across phases. This command is available for all multi-phase data types, for the Multiple Systems with Event Codes data type and for any data set analyzed using Change of Slope calculations.

Advanced Systems

The following commands are visible when you view the data sheet in Advanced Systems View.

Edit Systems

- **Copy System** copies the selected system in the Advanced Systems View Explorer.
- **Paste System** pastes the copied system as a new system in the Advanced Systems View Explorer. The copied system will be named as "Copy of (name of copied system) [increment]." Thus, if you create a single copy of a system named "System A," the copy will be named "Copy of System A 1."
- **Add System** adds a system to the Advanced Systems View Explorer.
- **Delete System** deletes the selected system from the Advanced Systems View Explorer.
- **Select All** selects all systems in the Advanced Systems View Explorer for inclusion in the analysis.
- **Clear All** clears the check boxes for all systems in the Advanced Systems View Explorer so that none are selected for inclusion in the analysis.
- **Expand All** expands the hierarchy in the Advanced Systems View Explorer to show the start and end times for each system.
- **Collapse All** collapses the hierarchy in the Advanced Systems View Explorer to hide the start and end times for each system.
- **Move Up** moves the selected system up one place in the Advanced Systems View Explorer.
- **Move Down** moves the selected system down one place in the Advanced Systems View Explorer.

- **Rename** allows you to change the name of the selected system in the Advanced Systems View Explorer. When selected, edit the name in the text box that appears and press **ENTER**.

 **System Results**

- **Current System** displays the calculated results for the currently selected system.

- **All Systems** displays the calculated results for all systems.

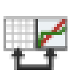
Options

Alter Parameters opens the Alter Parameters window, which allows you to alter the values of the corresponding calculated parameters. No recalculation is performed. This command is available for any data set calculated using any model except the Crow Extended and Crow Extended - Continuous Evaluation. Furthermore, for data sets calculated with the Crow-AMSAA (NHPP) model, this functionality is not available for Change of Slope calculations.

 **Auto Group Data** opens the Auto Group Data window, which allows you to specify intervals to group the data in the current data sheet. This command is available only for the Failure Times, Multi-Phase Failure Times and Fleet data types.

 **Interval GOF** opens the Interval Goodness-of-Fit Test window, which allows you to evaluate the model's fit to the combined equivalent system (for developmental data) or superposition system (for fielded systems data) and, if necessary, omit individual systems from the combined analysis. This command is available for the following data types: Multiple Systems - Concurrent Operating Times, Multiple Systems with Dates, Multiple Systems with Event Codes, Repairable Systems and Fleet.

 **Batch Auto Run** opens the Batch Auto Run window, which allows you to quickly extract data subset(s) from an existing data set. This command is available only for the Multiple Systems - Concurrent Operating Times, Multiple Systems with Dates, Multiple Systems with Event Codes, Repairable Systems, Fleet, Multi-Phase Failure Times, Multi-Phase Mixed Data and Multi-Phase Grouped Failure Times data types.

 **Link Mission Profile** allows you to select a Mission Profile in the project that you want to group the data sheet by. The data in the data sheet will be transferred to a new data sheet in

the folio and grouped according to the specified Mission Profile. This command is available only for the Failure Times data type.

Folio Sheets

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

Insert Additional Plot Sheet 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.

Delete Sheet deletes the current sheet within the folio.

Move or Copy Sheet 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.

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

Format and View

Insert Columns allows you to insert additional columns into a growth data folio data sheet. The types of columns available for insertion vary depending on the data type.

Delete Columns allows you to delete columns from a growth data folio data sheet. The types of columns that can be deleted vary depending on the data type.

Set Headers as Default saves the current column headings as the defaults for new data sheets of the same type. To reset the default headings, click **Reset Application Settings** in the Application Setup.

Apply Default Headers applies the currently saved defaults to the column headings of the active data sheet.

Switch to Normal View/Switch to Advanced Systems View toggles between systems views for data types with multiple systems. The Advanced Systems View displays the data one system at a time and provides a navigation panel to select which system you wish to view. This view allows you to view the calculated results for each system individually and for the "equivalent" system (for reliability growth analyses) or the "superposition" system (for fielded systems analyses). In addition, this view gives you the ability to remove individual systems from consideration in a particular analysis. The Normal View displays the data for multiple systems all together in the same data sheet.

Transfer Data

Transfer to New Data Type opens the Transfer to New Data Type window, which allows you to select a data type into which you would like to transfer the data. This command is available only for the Multiple Systems - Concurrent Operating Times, Multiple Systems with Dates, Multiple Systems with Event Codes, Repairable Systems and Fleet data types.

Transfer to Life Data folio opens the Transfer to Life Data folio window, which allows you to transfer the current Reliability Growth data set to a life data folio. Please note that you can transfer data only after the parameters have been calculated.

ReliaSoft

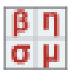
Publish Model allows you to use the results of the growth analysis as a resource object and make it available for unlimited use elsewhere within the project, in any ReliaSoft application that has a need for that type of object.

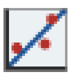
 **Publish SEP Summary** publishes a summary of the current analysis to the SEP web portal. This is visible only if the enterprise database is configured for SEP and you have the "Publish to SEP web portal" permission (see Publishing to SEP).

Life-Stress Data Tab

The Life-Stress Data tab is visible when you view an Accelerated Life Testing life-stress data folio data sheet for accelerated life testing analysis.

Analysis

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

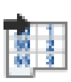
 **Plot** creates a new sheet in the folio that provides a choice of applicable plot types. For life-stress data folios, this includes plots such as use level probability, reliability vs. time, life vs. stress, etc.

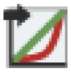
Quick Calculation Pad opens the Accelerated Life Testing Quick Calculation Pad, which allows you to calculate results, such as the mean life and the probability of failure, based on the currently active data sheet.

 **Distribution Wizard** opens the Distribution Wizard, which helps you select the life distribution that will best fit your data.

 **3D Plot** creates a 3D plot in the project, allowing you to graph functions with three variables such as the reliability at a given time and stress level. The 3D plot is saved under the **Multiplots** heading in the current project explorer.

Folio Sheets

 **Insert Data Sheet** inserts a new blank data sheet into the currently active folio and opens the Accelerated Life Testing 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, allowing 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.

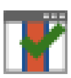
 **Select Sheet** allows you to select which sheet in the current folio you want to be active.

 **Move/Copy Sheet** allows you to move or copy any of the sheets within the currently active folio. In the Move/Copy window, select the sheet you wish to move and 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. ***There is no undo for delete***.

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.

 **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).

Alter Parameters provides 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 **Alter One Parameter (and Recalculate)** allows you to alter one of the parameter values and have the software estimate the other parameter values that would result in a good fit for the model. See also Alter Parameters.

 **Stress Transformation** opens the Stress Transformation window, which allows you to specify a transformation equation for each stress that will be used in the analysis.

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.

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 Accelerated Life Testing 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.

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.

Set Headers as Default saves the current column headings as the defaults for new data sheets of this type. To reset the default headings, click **Reset Application Settings** in the Application Setup.

Apply Default Headers applies the currently saved defaults to the column headings of the active data sheet.

Add/Remove Columns opens the Add or Remove Columns window, which allows you to add/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 identical data points. Grouping data significantly speeds up calculations when you have a very large data set.

ReliaSoft

Publish Model publishes the chosen distribution as a model for use throughout the project, in any ReliaSoft application where models are used. See Publishing Models.

Publish SEP Summary publishes a summary of the current analysis to the SEP web portal. This is visible only if the enterprise database is configured for SEP and you have the "Publish to SEP web portal" permission (see Publishing to SEP).

Life Comparison Tab

The Life Comparison tab is visible when you view a life comparison folio.

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.

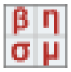
Select Data Sheets allows you to select the data sheets to compare.

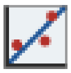
Parameter Estimator opens the Target Reliability Parameter Estimator window, which allows you to explore how you could attempt to change one of the distribution parameters in order to meet a specified target reliability. The plot and reliability estimate will be based on the altered distributions and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

Life Data Tab

The Life Data tab is visible when you view a life data folio data sheet for life data analysis.

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 life data folios, this includes plots such as reliability vs. time, failure rate vs. time, probability plot, etc.

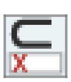
Quick Calculation Pad opens the Weibull++ Quick Calculation Pad, which allows you to calculate results, such as the mean life and the probability of failure, based on the analyzed data sheet.

 **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).

 **3D Plot** creates a 3D plot in the project, allowing you to graph functions with three variables such as the reliability at a given time and stress level. The 3D plot is saved under the **Multiplots** heading in the current project explorer.

Options

 **Batch Auto Run** opens the Batch Auto Run window, which allows you to easily extract data from an existing data set based on their subset ID.

 **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.

Alter Parameters provides 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 **Alter One Parameter (and Recalculate)** allows you to alter one of the parameter values and have the software estimate the other parameter values that would result in a good fit for the model. See also Alter Parameters.

Fill Median Ranks is available only if the data type is free-form. 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 identical data points. Grouping data significantly speeds up calculations when you have a very large data set.

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.

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.

Insert Additional Plot inserts an overlay plot in the currently active folio, allowing you to display 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 allows you to select which sheet in the current folio you want to be active.

Move or Copy Sheet allows you to move or copy any of the sheets within the currently active folio. In the Move/Copy window, select the sheet you wish to move and 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. ***There is no undo for delete***.

Format and View

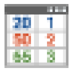
Alter Data Type opens the Weibull++ Folio Data Sheet Setup window, which allows you to change the data entry columns in the current data sheet. Note that when you change the setup, you may lose information already entered in the data sheet. For example, switching the data sheet from one that supports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

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.

Set Headers as Default saves the current column headings as the defaults for new data sheets of the same type. To reset the default headings, click **Reset Application Settings** in the Application Setup.

Apply Default Headers applies the currently saved defaults to the column headings of the 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. 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 font colors.

- **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.

ReliaSoft

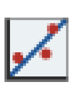
 **Publish Model** publishes the chosen life distribution as a model for use throughout the project, in any ReliaSoft application where models are used. See Publishing Models.

 **Publish SEP Summary** publishes a summary of the current analysis to the SEP web portal. This is visible only if the enterprise database is configured for SEP and you have the "Publish to SEP web portal" permission (see Publishing to SEP).

Maintenance Planning Tab

The Maintenance Planning tab is visible when you view the Maintenance Planning folio.

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.

Mission Profile Tab

The Mission Profile tab is visible when you view a mission profile data sheet.

Mission Profile

Insert Profile Sheet adds a new profile sheet to the current mission profile.

Delete Profile Sheet deletes the current profile sheet from the mission profile.

Validate Mission Profiles sorts each profile sheet by the Cumulative Time column and checks that all necessary data have been entered.

Plot Mission Profiles creates a Mission Profile plot for all profile sheets in the current folio.

Transfer Convergence Points to Profile Sheets adds all convergence points entered on the convergence points sheet to each profile sheet in the mission profile.

Non-Parametric LDA Tab

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

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.

Quick Calculation Pad opens the 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++ life data folio QCP.

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.

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++ life data 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++ life data folio.

ReliaSoft

Publish Model publishes the chosen life distribution as a model for use throughout the project, in any ReliaSoft application where models are used. See Publishing Models.

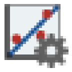
Optimization Tab

The Optimization tab is visible when working with an optimization folio.

Actions

Calculate allows you to specify new optimization settings.

Plot is available only for dynamic overlaid contour plots. It updates the current plot to reflect the changes that have been made.

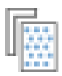
 **Plot Setup** opens the Plot Setup window, which allows you to customize most aspects of the plot appearance including the titles, axis numbers and line styles.

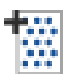
 **Select Responses & Factors** is available only for optimal solution plots. It allows you to choose the factors (i.e., columns) and responses (i.e., rows) to be displayed in the plot.

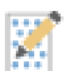
 **Select Data Source** allows you to select the source data sheet for the optimization folio.

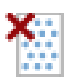
Solutions

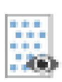
The following commands are available only for the optimal solution plot.

 **Duplicate Solution** creates a copy of the currently selected solution and adds it to the Solutions list. You can then modify the solution as needed.

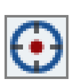
 **Create Solution** allows you to enter a name for the solution and specify the factor values to be used for the solution. Your new solution will be added to the solutions list.

 **Edit Solution** allows you to manually adjust the factor values for the currently selected solution. Note that if you edit an optimal solution generated by the software, your changes will be used to create a new custom solution and the original solution will remain unchanged.

 **Delete Solution** removes the currently selected solution from the Solutions list.

 **View Solutions** opens the Solutions window, which shows each solution, including factor settings, the associated block (if any) and the predicted response(s) at those settings.

Tools

 **Add Optimization Tool** opens the Select Optimization Tool window, which allows you to add an additional optimization tool of a type that is not already present in the folio to the folio. Note that each optimization folio can have only one optimization tool of each type and this command is unavailable if the folio already contains each type of tool.

Plot Tab

The Plot tab is visible when you view a plot sheet.

Plot

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

Quick Calculation Pad is available only for applicable plots. It opens the Quick Calculation Pad (QCP), which allows you to obtain reliability results based on the currently active folio or diagram.

3D Plot creates a 3D plot in the project, allowing you to graph functions with three variables such as the reliability at a given time and stress level. The 3D plot is saved under the **Multiplots** heading in the current project explorer. This option is not available for reliability growth data folios.

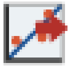
Actions

Copy Plot Graphic copies the plot to the Clipboard as a graphic. If you will be pasting copied plots into any one of the spreadsheets built in to ReliaSoft applications (e.g., ReliaSoft Workbooks or General Spreadsheets), choose **Metafile Optimized for ReliaSoft Spreadsheet**. If you will be pasting them into external applications, choose **Bitmap** or **Metafile Optimized for External Use**.

Copy Plot Data exports the plot data to the Clipboard. You can then paste that data into another application. The exact information depends on the plot type.

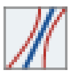
Plot Setup opens the Plot Setup window, which offers several options for customizing a plot such as editing axis titles, line styles, grid lines, etc.

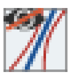
RS Draw launches ReliaSoft Draw, which is a graphics editor that offers advanced options for annotating, inserting images, drawing images on a plot, and more.

 **Export Plot Graphic** opens the Save As window, which allows you to save the current plot graphic as a *.wmf, *.png, *.gif, or *.jpg file.

 **Show/Hide Plot Items** provides options for displaying or hiding from view certain plot items such as data points, lines, probability scales, etc.

Confidence Bounds

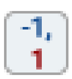
 **Confidence Bounds** are available if the associated analysis includes calculations for confidence bounds. It opens the Confidence Bounds Setup window, which provides options for defining the properties of the bounds that will be displayed on the plot.

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

Display

The following commands are available only for DOE design folios.

 **Actual Values** displays the actual values for the factors.

 **Coded Values** displays the coded values for the factors. Which coded values are used (e.g., whether 0 or -1 represents the low level) is dependent on the design type.

 **Amount** applies to mixture designs only.

 **Amount** displays the component factor values as amounts (e.g., 2 pounds).

 **Proportion** applies to mixture designs only. It displays the component factor values as proportions (e.g., 0.2 = 20% of the total mixture used in the run).

Pseudovalue applies to mixture designs only. It displays the component factor values as scaled values between 0 and 1, where 0 is the minimum possible value for the factor (given the design settings) and 1 is the maximum possible value.

Options

The following commands are available only for reliability growth data folios.

Plot Modes applies to Individual Mode MTBF/FI/Reliability plots. See Plot Modes window.

Plot Systems applies to System Operation plots for multiple systems. See Select Systems to Plot window.

Plot Beta Bounds applies to Beta Bounds plots. See Beta Bounds window.

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.

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 applies to parametric RDA folios only. It 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).

Sheet Tab

The Sheet tab is visible when you view data sheets, general spreadsheets or DOE simulation worksheets.

Format and View

Format Cells opens the Format Cells window, which provides several ways to change the way data are displayed in a cell or range of cells:

- **Number tab** sets the numbering format of a cell. Anything you type in the cell will be automatically formatted based on your selection. For example, if you select the number format and specify 3 decimal places, typing the value 1.2 in the cell will automatically display it as 1.200.
- **Alignment tab** sets the horizontal, vertical alignment and indentation text in a cell. Includes text control options such as wrap text, shrink to fit, merge cells and text direction.
- **Font tab** sets the font, style, color, size and effects of the selected text.
- **Border tab** applies borders around the selected cell(s) and controls the line style, thickness and color of the border lines.
- **Fill tab** sets the background color of the selected cell(s). You can also apply a pattern style and choose a pattern color.

Recalculate Formulas forces a recalculation of all formulas in the spreadsheet.

Defined Names applies only to general spreadsheets. It 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 sheet or function. Names can help make formulas in the spreadsheet easier to understand and maintain.

Sheet Actions

The following commands apply only to general spreadsheets.

Function Wizard opens a function wizard that helps you to build functions that utilize a referenced analysis (data source).

Insert Function opens a function wizard that helps you to build functions that return results from mathematical values (e.g., sine, pi or averages).

Chart Wizard allows you to create new plots and charts based on the current spreadsheet. To use, select a range of cells for which you would like to create a chart, then click **Chart Wizard** and select the area where you want to place the chart.

AutoSum allows you to compute the sum of the values in a range of selected cells. If a cell range is not selected, the command computes the sum of the values in the cells directly above the current cursor location.

Sort allows you to reorder the selected data either in ascending or descending order. Note that you may need to be careful when sorting certain types of data in order to prevent errors in analysis.

- If your analysis takes into account the order position of a data point (1st, 2nd, 3rd, etc. position), sorting may result in an inaccurate analysis.
- If the range of cells that you wish to sort contains blank cells, the blank cells may be put in the first position (ascending order) or the last position (descending order).

Rows/Columns

Delete deletes the selected cells and moves the remaining cells based on the specified direction. Note that when the cells are deleted, the information inside the cells is also deleted.

Insert inserts new cells at the current cursor location and moves the existing cells based on the specified direction. If a range of cells is selected in the sheet, then the same number of cells will be inserted.

Transfer Data

Send to Excel exports all sheets to an Excel file.

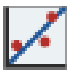
Send to Word exports the current sheet to a Word file.

SimuMatic(A) Tab

The SimuMatic(A) tab is visible when you view a Life-Stress Data SimuMatic results folio.

SimuMatic

 **Stress-Dependent SimuMatic** allows you to change your simulation settings and replace your current simulated data sets with new ones.

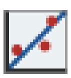
 **Plot** creates a probability plot of all the data sets.

SimuMatic(R) Tab

The SimuMatic(R) tab is visible when you view a Repairable Systems SimuMatic results folio.

SimuMatic

 **Repairable Systems SimuMatic** allows you to change your simulation settings and replace your current simulated data sets with new ones.

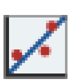
 **Plot** creates a probability plot of all the data sets.

SimuMatic(W) Tab

The SimuMatic(W) tab is visible when you view a Life Data SimuMatic results folio.

SimuMatic

 **SimuMatic** allows you to change your simulation settings and replace your current simulated data sets with new ones.

 **Plot** creates a probability plot of all the data sets.

 **Transfer Life Data to Selected Folio** transfers the simulated data set to an existing Weibull++ life data folio, with one data sheet for each parameter of the model that was used to generate the data.

Transfer Life Data to New Folio transfers the simulated data set to a new Weibull++ life data folio, with one data sheet for each parameter of the model that was used to generate the data.

Stress-Strength Tab

The Stress-Strength tab is visible when you view a stress-strength comparison folio.

Analysis

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

Select Data Sheets 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.

Test and Planning Tab

The Test and Planning tab is visible when you view a growth planning folio.

Analysis

Calculate calculates the inputs and generates the growth planning results.

Plot creates a new sheet in the folio that provides a choice of applicable plot types that show the results that are expected if you implement the plan.

Quick Calculation Pad opens the Quick Calculation Pad, which allows you to calculate a variety of results that are expected if you implement the plan.

Show Results allows you to see a summary of the inputs and results in the Results window.

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.

Reliability Demonstration Test

Calculate implements the selected test design method based on the specified inputs.

Get Distribution updates the shape parameter and demonstrated time in the RDT sheet based on data from an existing Weibull++ life data folio in the current project.

Show RDT Table is available only for the Parametric Binomial test design method and for Reliability Growth test design folios. For life data analysis, this command creates a parametric binomial table and plot based on the target metric and life distribution that you specified on the RDT sheet. For growth analysis, this creates an RDT table and plot that allows you to compare different test scenarios.

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.

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

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.

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

Expected Failure Time

Get Failure Model updates the expected failure model (i.e., parameters and time) based on data from an existing Weibull++ life data 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.

Test Plan Tab

The Test Plan tab is visible when you view an ALTA test plans folio.

Analysis

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

Warranty Tab

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

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 calculated separately.

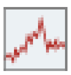
Plot creates a new sheet in the folio that provides a choice of applicable plot types. In warranty analysis folios, this includes plots such as expected failures vs. period, reliability vs.

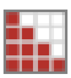
time, probability plot, etc. You can plot the data in the warranty analysis folio with or without a forecast.


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++ life data folio QCP.

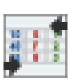

Distribution Wizard opens the Distribution Wizard, which helps you select the distribution that best fits your data based on the selected parameter estimation method (i.e., RRX, RRY or MLE). In warranty analysis folios, the ranking in the Distribution Wizard is based on the converted failure/suspension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

Tools


Forecast opens the Forecast Setup window, which allows you to specify the time periods at which 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.


Restore Guide Colors applies only if you have manually changed the colors of cells in the Returns sheet of a Nevada chart format warranty folio. It returns the cells to their original colors, which are used in the Statistical Process Control feature of the Nevada chart format to indicate the following:

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


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++ life data 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++ life data folio in order to perform life data analysis.

Life Data Analysis Folios

IN THIS CHAPTER

Weibull++ Life Data Folio

The Weibull++ life data folio for life data analysis is the most basic folio in the application. 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 metrics such as probability of failure, reliability, mean life or failure rate. The ReliaWiki resource portal has more information on the basics of life data analysis at http://www.reliawiki.org/index.php/Life_Data_Analysis_Reference_Book.

Weibull++ Life Data Folio Setup

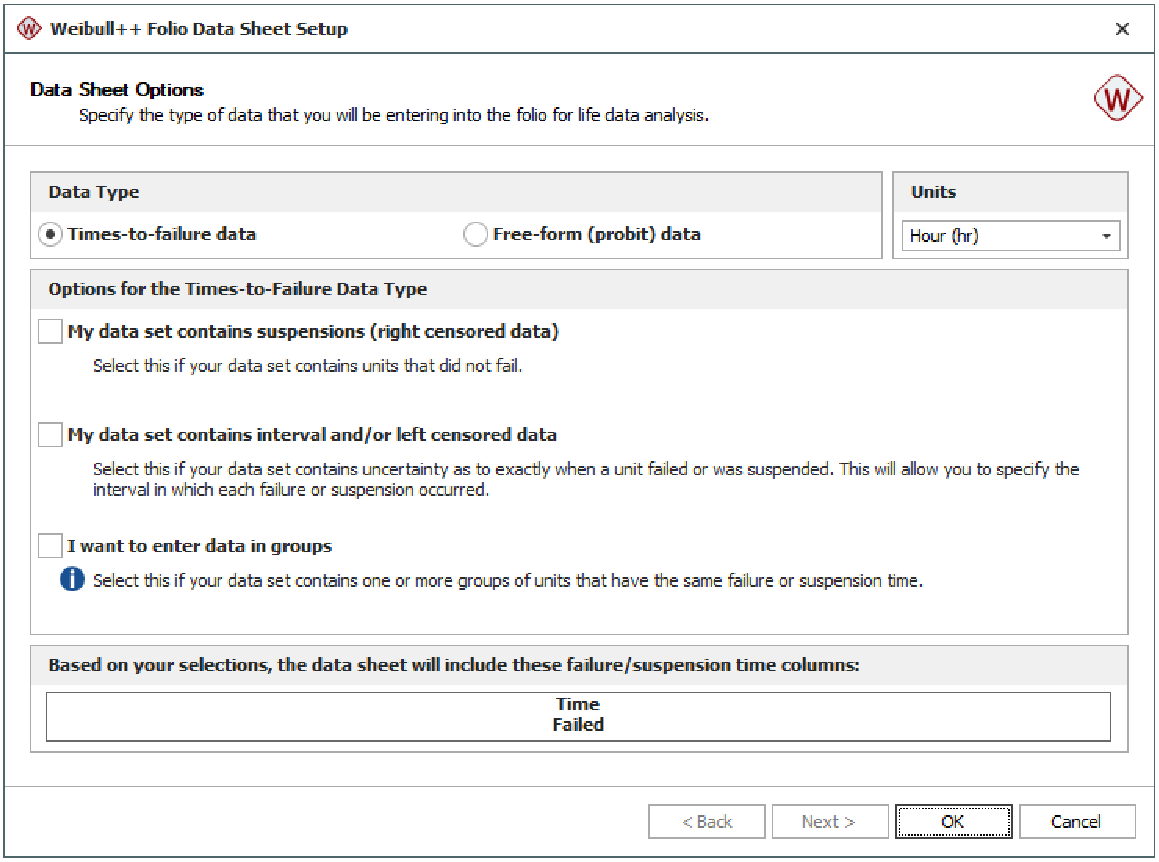
To add a Weibull++ life data folio to a project, choose **Home > Insert > Life Data** or right-click the **Life Data** folder in the current project explorer and choose **Add Life Data** on the shortcut menu.

A setup window will prompt you to select a data type to use in the folio and a unit appropriate for your data (the unit may be measured in mileage, distance, weight, etc.; see Unit Settings).

There are two types of data available: 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. (See Weibull++ Data Types.)

Any of the times-to-failure data types can also be entered as grouped data. The ReliaWiki resources portal has more information on grouped data analysis at: http://www.reliawiki.org/index.php/Grouped_Data_Parameter_Estimation.

The data type you select determines the data entry columns that will appear in the data sheet. 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, as shown in the following example.



Weibull++ Data Types

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.

In a Weibull++ life data folio, the data type you select determines the data entry columns that appear in the data sheet. The following sections describe each data type and demonstrate how to enter data in a Weibull++ life data folio data sheet.

Tip: You can change the data entry columns for an existing Weibull++ data sheet anytime by choosing **Life Data > Format and View > Alter Data Type**.

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 life data 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
1	12	failed unit
2	24	failed unit
3	32	failed unit
4	56	failed unit
5	80	failed unit
6	104	failed unit

Note: In this example and in the rest that follow, the Subset ID column is used for explanatory purposes and is not intended to demonstrate how the column would normally be used.

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 in 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	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

TIMES-TO-FAILURE WITH SUSPENSIONS (RIGHT CENSORED DATA)

The term *suspension* describes the units that 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 in 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
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	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

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 in 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
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.

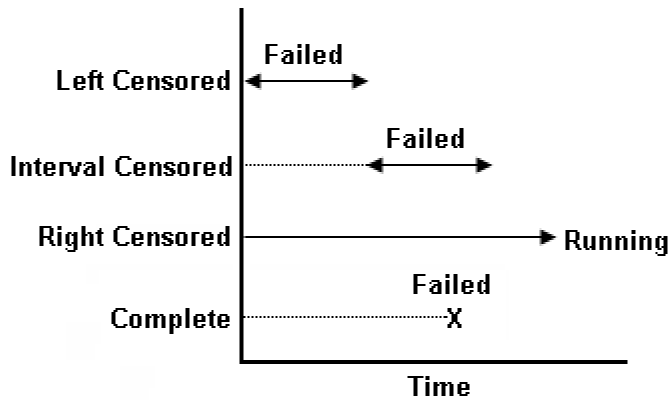
	Number in State	Last Inspected (hr)	State End Time (hr)	Subset ID 1
1	3	0	150	3 left censored units
2	1	150	300	1 interval censored unit
3	7	300	450	7 interval censored units
4	3	450	600	3 interval censored units
5	1	600	750	1 interval censored unit
6	1	750	900	1 interval censored unit

TIMES-TO-FAILURE WITH ALL TYPES OF CENSORED 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 (hr)	State F or S	State End Time (hr)	Subset ID 1
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.



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 in 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
1	100	28.9	at 100 hr, the prof of failure is 28.90%
2	200	31.38	at 200 hr, the prof of failure is 31.38%
3	300	49.99	at 300 hr, the prof of failure is 49.99%
4	400	68.61	at 400 hr, the prof of failure is 68.61%
5	500	87.05	at 500 hr, the prof of failure is 87.05%
6	600	89.2	at 600 hr, the prof of failure is 89.20%

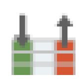
Subset IDs

By default, Weibull++ life data folio data sheets include a **Subset ID** column for logging any pertinent information or comments about the data. The column can contain values or text up to 30 characters, including spaces.

The information in the subset ID column 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 analysis methods in Weibull++ that are affected by subset IDs:

- Competing failure modes (CFM) Analysis
- Batch Auto Run
- Auto Group Data
- Specify Points

To insert additional subset ID columns to a data sheet, choose **Life Data > Format and View > Add/Remove Columns**.



In the window, click the **Add Subset ID Column** button. To remove a subset ID column from the list, clear the check box next to the column's name.

Life Data Folio Control Panel

The life data 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 topic focuses on the Main page of the life data 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 Control Panels.

Control Panel Main Page

- 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.
  - The nine distributions are: Weibull, exponential, lognormal, normal, generalized gamma, gamma, logistic, loglogistic and Gumbel. The ReliaWiki resource portal has information on the statistical characteristics of each distribution at: http://www.reliawiki.org/index.php/Life_Distributions.

    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.

  - 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 then calculate the Weibull parameters of those subpopulations. The overall reliability of the full data set is then computed by obtaining the sum of the proportional reliability contributions of each subpopulation.
    - **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.
    - **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) and the failure mode responsible for each failure can be clearly identified. 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.

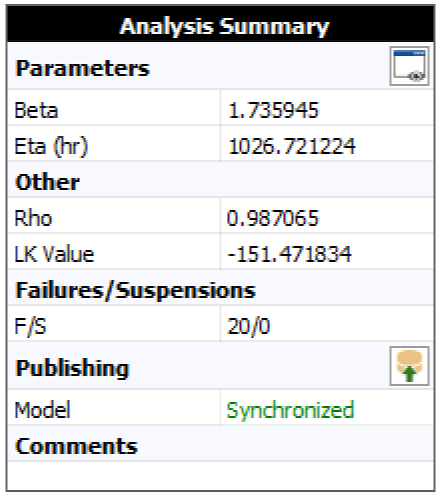
- The **Analysis Settings** area provides a summary of the settings that will be used to fit the distribution. Click the options displayed in blue text to toggle between the available options.

Analysis Settings	
RRX	SRM
FM	MED

The following table serves as a guide to the abbreviations. These settings are also available on the Analysis page of the control panel (see Life Data Folio Analysis Settings for a description of these settings). Note that the available options will vary depending on the data type, distribution and analysis method you have selected.

Analysis Method	**Rank Method**
Rank Regression on X (RRX)	Standard Ranking Method (SRM)
Rank Regression Y (RRY)	ReliaSoft Regression Method (RRM)
Non-Linear Rank Regression (NLRR)	
Maximum Likelihood Estimation (MLE)	
Confidence Bounds Method	**Rank Method**
Fisher Matrix Confidence Bounds (FM)	Median Ranks (MED)
Likelihood Ratio Confidence Bounds (LRB)	Kaplan-Meier Estimator (K-M)
Bayesian Confidence Bounds (BSN)	
Beta-Binomial Confidence Bounds (BB)	

The **Analysis Summary** area displays the calculated parameters and other results that may be used to evaluate how well the chosen distribution or analysis method fits the data set. As an example, the following picture shows the analysis results for a particular data set.



Depending on your application settings, additional information may also 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) compares 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 **Pnz** (percent non-zero) is automatically calculated and displayed in the results if the data set contains failures at $t = 0$. 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.
- The number of failures and suspensions. This includes failures at $t = 0$ but excludes suspensions at $t = 0$.
- The model status (as shown in the previous picture) is displayed if the result of the analysis is associated with a published model. The status "Synchronized" indicates that the

published model reflects the latest results from the analysis. If the 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." (See Publishing Models.)

Tip: 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. 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.

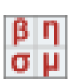
ADDITIONAL RESULTS

The following tools may also be used in conjunction with your own engineering knowledge about the model in order to further evaluate the fit of the distribution to the data set.

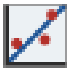
- For a rank regression analysis, the **Probability Plot** shows you how well the solution line tracks the plotted data points. This allows you to visually assess the fit of a distribution. This method can be used only when the parameters have been calculated using rank regression.
- **Confidence Bounds** 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.
- The **Goodness of Fit Results** utility uses three different tests to determine how well the parameters of the distribution fit the data. Note that it can only be used to evaluate data sets that do not contain censored data.

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.

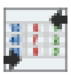


Calculate estimates the parameters of the chosen distribution, based on the current data set and the specified analysis settings. This tool is also available by choosing **Life Data > Analysis > Calculate**.

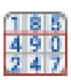
 **Plot** creates a new sheet in the folio that provides a choice of applicable plot types. For life data folios, this includes plots such as reliability vs. time, failure rate vs. time, probability plot, etc. This tool is also available by choosing **Life Data > Analysis > Plot**.

QCP opens the Weibull++ 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**.

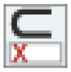
 **Distribution Wizard** opens the Distribution Wizard, which helps you select the distribution that best fits your data based on the selected parameter estimation method (i.e., RRX, RRY or MLE). This tool is also available by choosing **Life Data > Analysis > Distribution Wizard**.

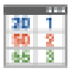
 **Alter Data Type** opens the Weibull++ Folio Data Sheet Setup window, which allows you to change the data entry columns in the current data sheet. Note that when you change the setup, you may lose information already entered in the data sheet. For example, switching the data sheet from one that supports 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**.

 **Change Units** opens the Change Units window, which allows you to change the units for the values in the current data sheet.

 **Auto Group Data** opens the Auto Group Data window, which allows you to group together identical data points. 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**.

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**.

 **Color Data Values** applies different font colors to the entries in the data sheet for easy visualization. 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 font colors.

- **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: 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.

 **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**.

Weibull++ Life Data Folio Analysis Settings

The Analysis page of the Weibull++ life data 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 topic provides a brief description of all the settings. For the background theory of each analysis method, the ReliaWiki resource portal provides more information at: http://www.reliawiki.org/index.php/Parameter Estimation.

The Analysis page of the Weibull++ life data folio control panel contains the following settings:

- **Analysis Method** sets 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 that:
  - 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 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.

  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.

- **Rank Method** sets 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.

- **Use RS Regression Method** uses ReliaSoft's 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 exact times-to-failure data, there is no difference in the results between the SRM and RRM methods. The ReliaWiki resource portal provides more information on the RRM method at: http://www.reliawiki.org/index.php/ReliaSoft's Alternate Ranking Method.

- **Confidence Bounds Method** sets 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.) The ReliaWiki resource portal provides more information on the background theory of confidence bounds at: http://www.reliawiki.org/index.php/Confidence Bounds.

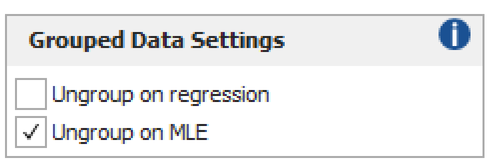
  - The Fisher Matrix method is the standard way of calculating confidence bounds and it is the default method in the software. For data sets with a large number of data points, there is no significant difference in the results between the Fisher Matrix method and the likelihood ratio method; however, 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.

- **Sort before calculation** sorts the failure/suspension times in the data sheet in ascending order. This will not affect the results of the calculation because, in life data folios, all data points are assumed to be independent and identically distributed (i.i.d.).

- **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.

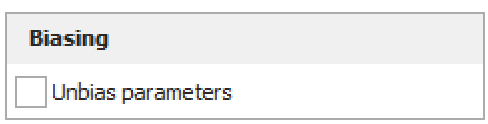


When using rank regression with grouped data, the software will plot only the data points that correspond to the highest median rank position in each group. For example, for two groups of data with 10 units each, the software will plot only two data points: the 10th rank position out of 20 and the 20th 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. 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.

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 failure data, you have to be cautious about ungrouping the data. In grouped 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, for a group of 10 units with a failure time of 100 the software will assume that the 10 units failed between 0 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.

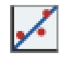
- **Biasing** is available when you select the 2-parameter Weibull distribution with MLE analysis, and only if the Bayesian confidence bounds method is not selected.



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. The ReliaWiki resource portal provides more information about the correction factors at: http://www.reliawiki.org/index.php/The_Weibull_Distribution#Unbiased_MLE.

Life Data Analysis Plots

To create a plot in a Weibull++ life data folio, choose **Life Data > Analysis > Plot** or click the icon on the Main page of the control panel.



Tip: You can add additional plot sheets to the folio by choosing **Life Data > Folio Sheets > Insert Additional Plot**. The additional sheets can function as overlay plots to display results from multiple data sheets in the current folio on a single plot.

The following is a description of the different types of plots that can be created in a Weibull++ life data folio. (For general information on working with plots, see Plot Utilities.)

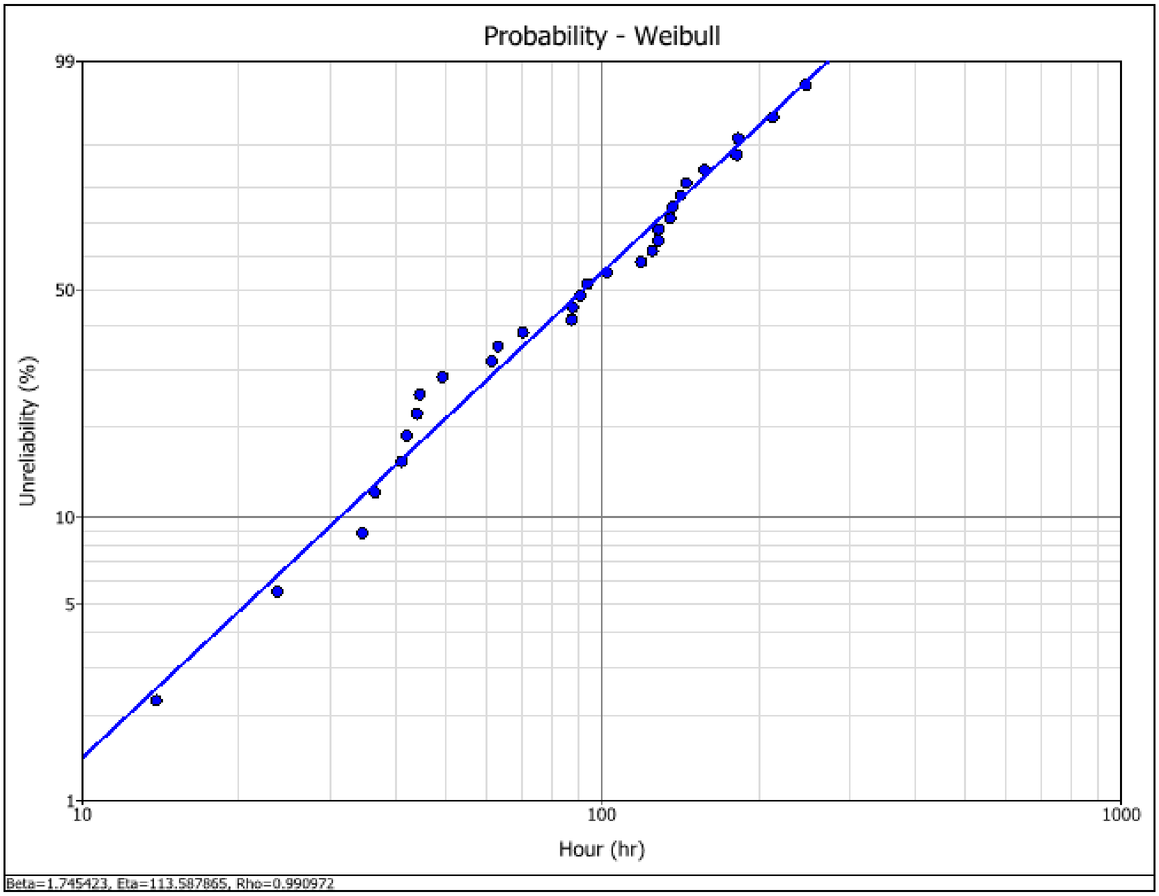
- The **Probability** plot shows the trend in the probability of failure over time. The plotting positions of the data points are determined by the failure/suspension times in the data set (x-axis) and their corresponding unreliability estimates (y-axis).

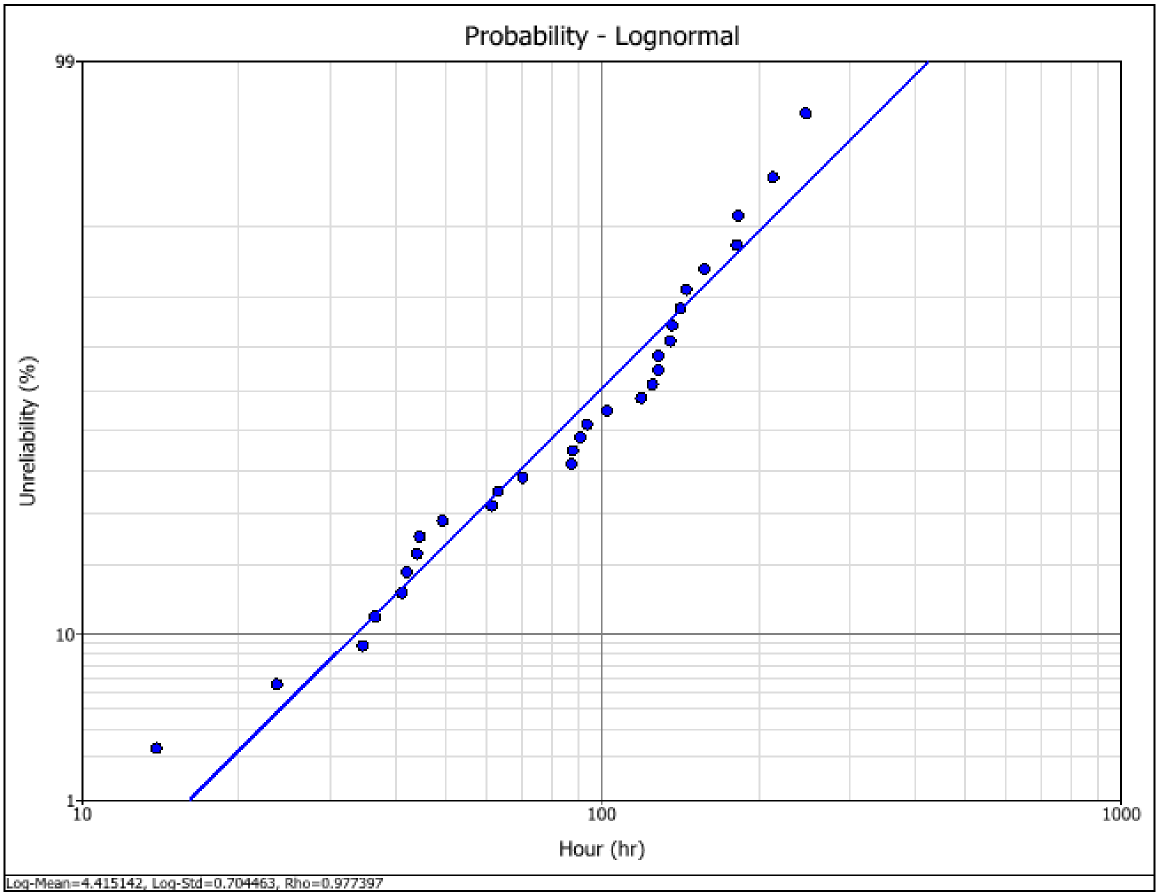
  Note: Unlike the probability plots for other distributions, the y-axis in an exponential probability plot always indicates the reliability instead of the unreliability. This tradition arose from the time when probability plotting was performed "by hand." The exponential reliability model starts with 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 following examples 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.


Probability - Weibull
99
50
10
5
1
10
100
1000
Unreliability (%)
Hour (hr)
Beta=1.745423, Eta=113.587865, Rho=0.990972



- 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). In Weibull++, contour plots are typically used to compare data sets. (See Contour Plots.)
- The **Failures/Suspensions (F/S) 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. Use the **Time Interval** field

in the control panel to define the length of the interval, then choose any of the following 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 on the control panel to show 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 show a bar chart of the number of suspensions in each interval.

- The **Failures/Suspensions Pie** chart displays the ratio and proportion of failures and suspensions to the whole data set.
- The **Failures/Suspensions Timeline** plot is similar to a horizontal bar chart. 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.

Contour Plots

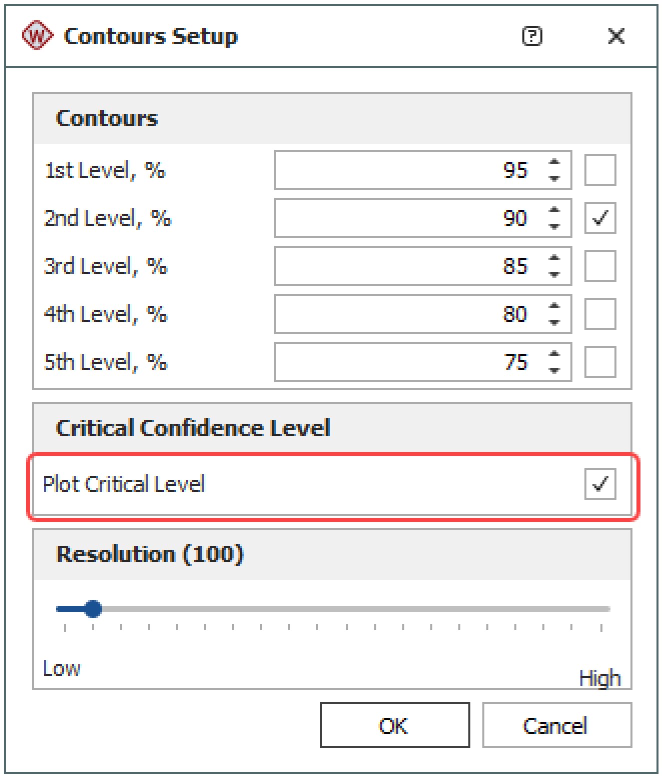
In Weibull++, contour plots are available only for the 2- and 3-parameter Weibull, normal and lognormal distributions. The first time you select a contour plot from the **Plot Type** drop-down list, the Contours Setup window will appear. This window allows you to select: a) the number of contour lines 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.

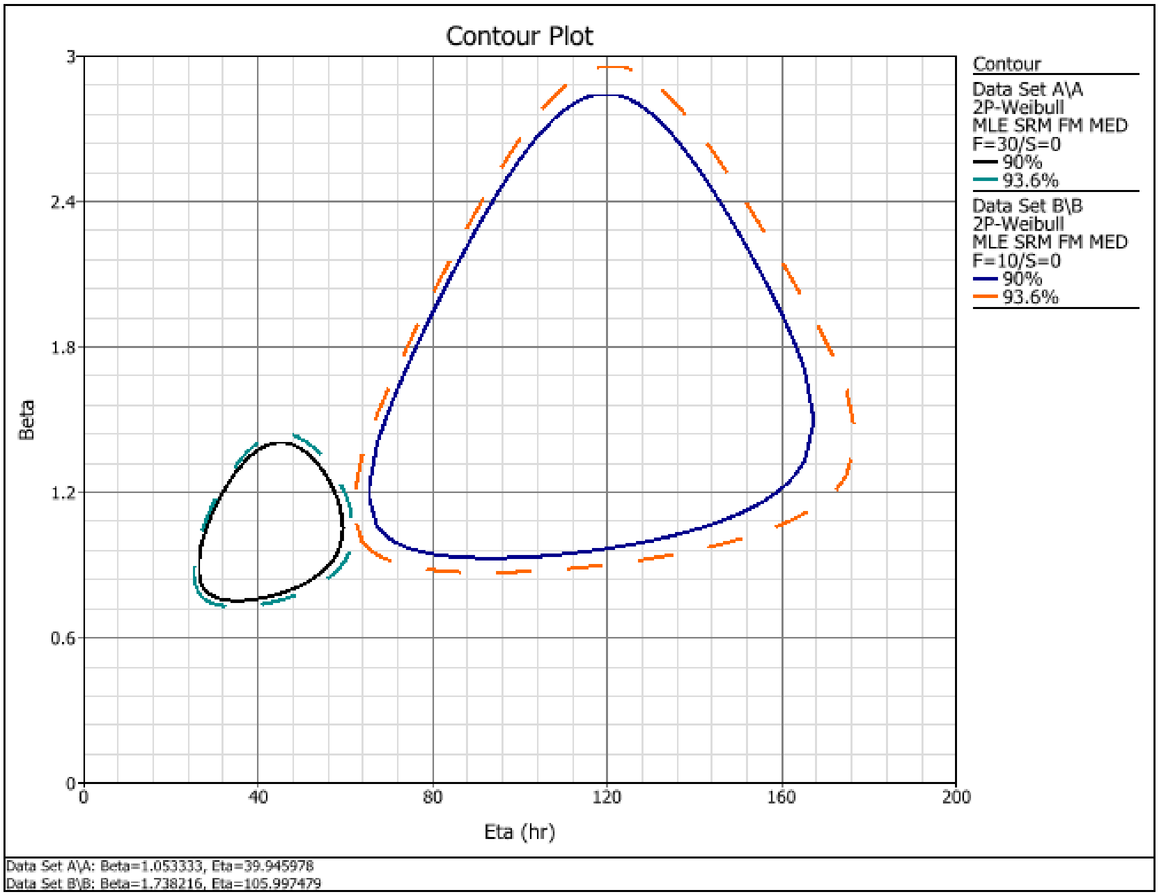
To change the level of detail at which the contours are plotted (and the smoothness of the lines), adjust the **Resolution** slider. You can click and drag the slider or, for fine adjustments, click the slider and then use the arrow keys. The number of points plotted to create the lines is displayed beside the field name. Note that higher resolutions will take longer to plot.

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 93.6%. (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.



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 topic 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.

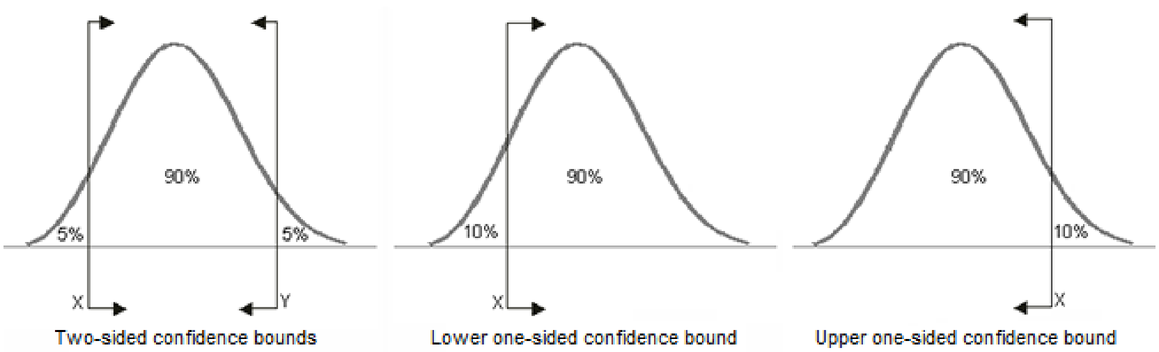
- Life Data Analysis
- Accelerated Life Testing Data Analysis
- Recurrent Event Data Analysis

How to Use the QCP

1. Choose a metric in the **Calculate** area. (Note: To calculate the **Parameter Bounds**, you must first select a type from the **Bounds** drop-down list.)

2. If applicable, use the **Units** drop-down list to specify the units for 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 life data 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.

3. If applicable, use the **Bounds** drop-down list to specify what type of confidence bounds to calculate. The following graphics illustrate the types. The ReliaWiki resource portal provides more information on the background theory of confidence bounds at: http://www.reliawiki.org/index.php/Confidence Bounds.



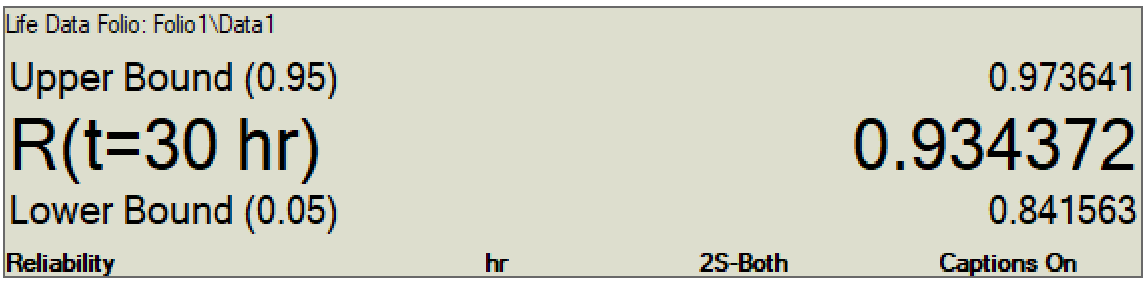
If you select **Both One-Sided**, the QCP will calculate both the lower one-sided bound and the upper one-sided bound.

Note: Confidence bounds calculation options are disabled when the underlying data type is free-form data.

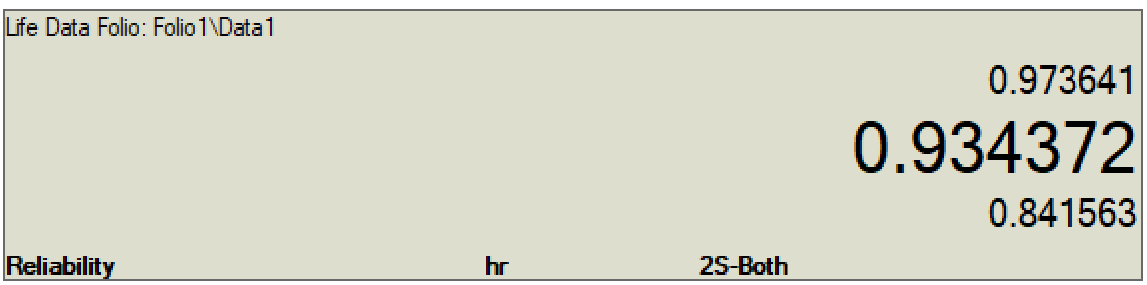
4. Make any required inputs in the **Input** area.

5. Click **Calculate**. The calculated value(s) will always be displayed in the results area at the top of the window, and clicking the **Report** button displays a summary of the current calculation input/output in the Results window. You can also use the **Options** drop-down list to configure other settings, including:

- **Precision** sets the number of decimal places displayed in the results.
- **Scientific Notation** sets the point at which numbers will be converted to normalized scientific notation. For example, setting this to 3 means that all numbers with a value of 1,000 or more will be converted to normalized scientific notation (e.g., 1.0E+3).
- **Set Display Font** allows you to change the font style and size that is used to display the calculated value(s) in the results.
- Select **Captions** if you want additional information to be displayed in the results area along with the values (a green light in the button indicates that this option is selected). For example, if you have selected to calculate the reliability for a given 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.



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.

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 Quick Calculation Pad (QCP).

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 duration 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 duration of the interval in the **Mission Additional Time** field.

For example, a product may have a 10% probability of failure for 3 years. If the product has operated for 2 years without failure, the conditional probability of failure for an additional year (for a total of 3 years of operation) may be 5%.

Life

Reliable Life

Calculates the estimated time at which a specified reliability value will be achieved. Enter the reliability goal in the **Required Reliability** field. For example, a goal of 90% reliability with a reliable life of 4 years means that if 100 identical units are fielded, then 90 of them will be still be operating at the end of 4 years.

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++ life data 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++ life data 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) folios.

For more complex repairable system analyses, see ReliaSoft's BlockSim software.

Mean Remaining Life

Calculates the expected remaining life given that the product, component or system has survived to time *t*. Enter the time at which you wish to calculate the mean remaining life in the **Mission End Time** field.

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.

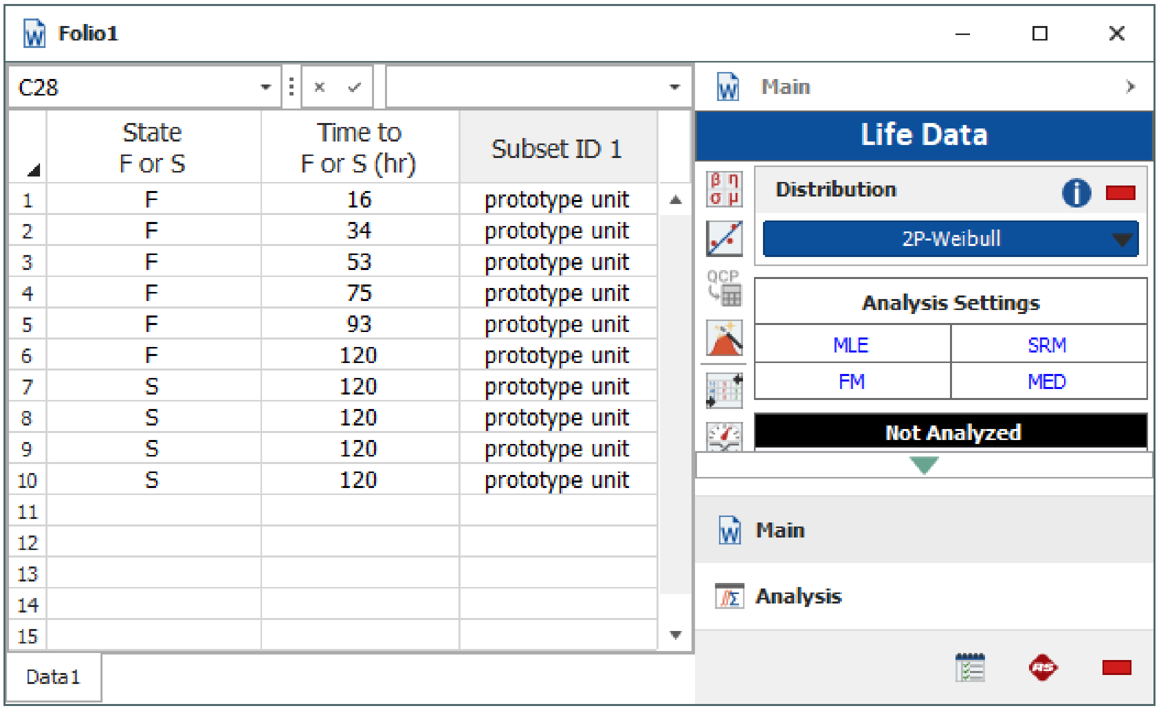
Life Data Analysis Example

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.

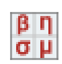
Create a Life Data Folio

1. Choose **Home > Insert > Life Data**. In the 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.
2. 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.)



Analyze the Data Set

3. Choose **Life Data > Analysis > Calculate** or click the icon on the Main page of the control panel.



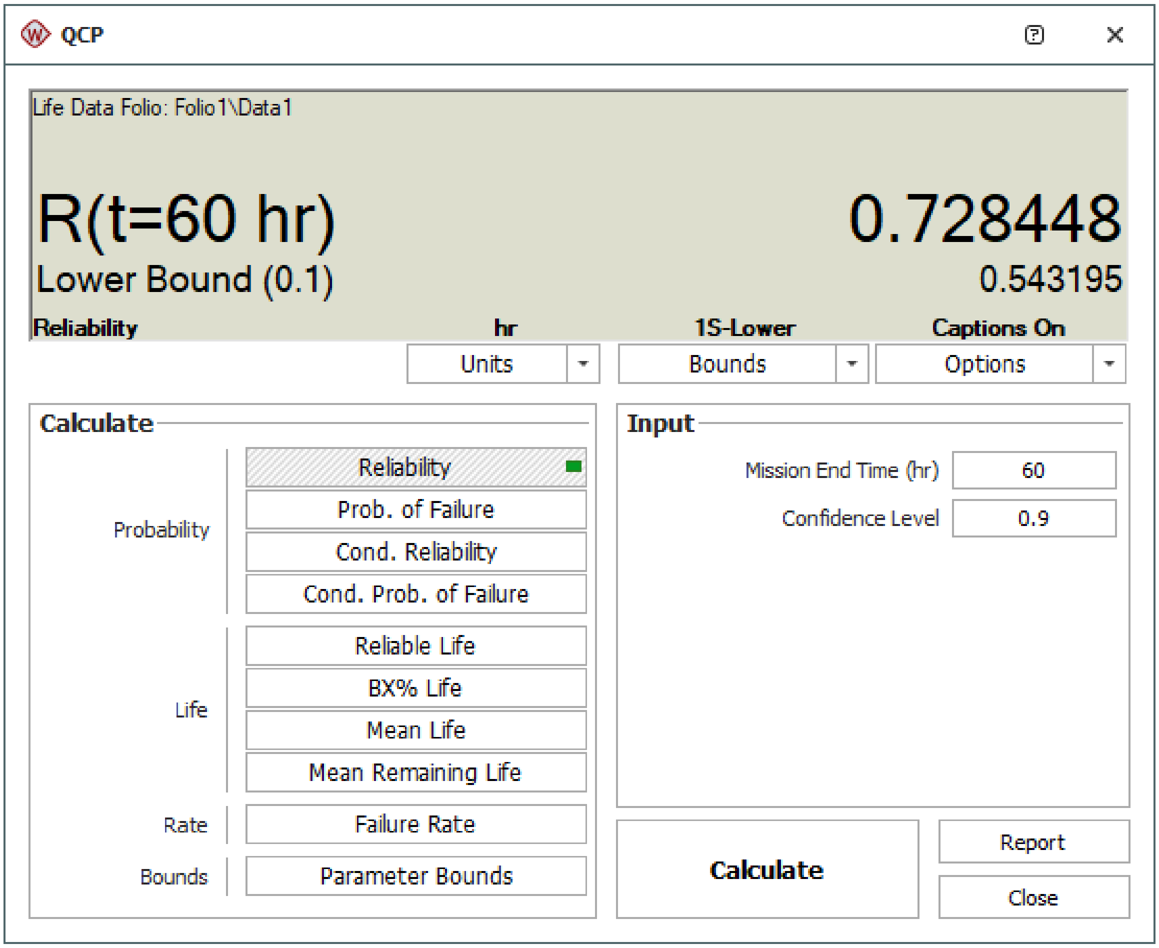
The parameters of the distribution will be automatically computed and displayed in the **Analysis Summary** area of the control panel. The parameters are beta = 1.5070 and eta = 128.6414.

4. 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.

5. In the QCP, choose to calculate the **Reliability** with **lower 1-sided** confidence bounds. Select **Hour** for the time units and then make the following inputs:

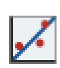
- Mission End Time = **60**
- Confidence Level = **0.9**

6. 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% at the end of 60 hours of operation.



Create a Reliability vs. Time Plot

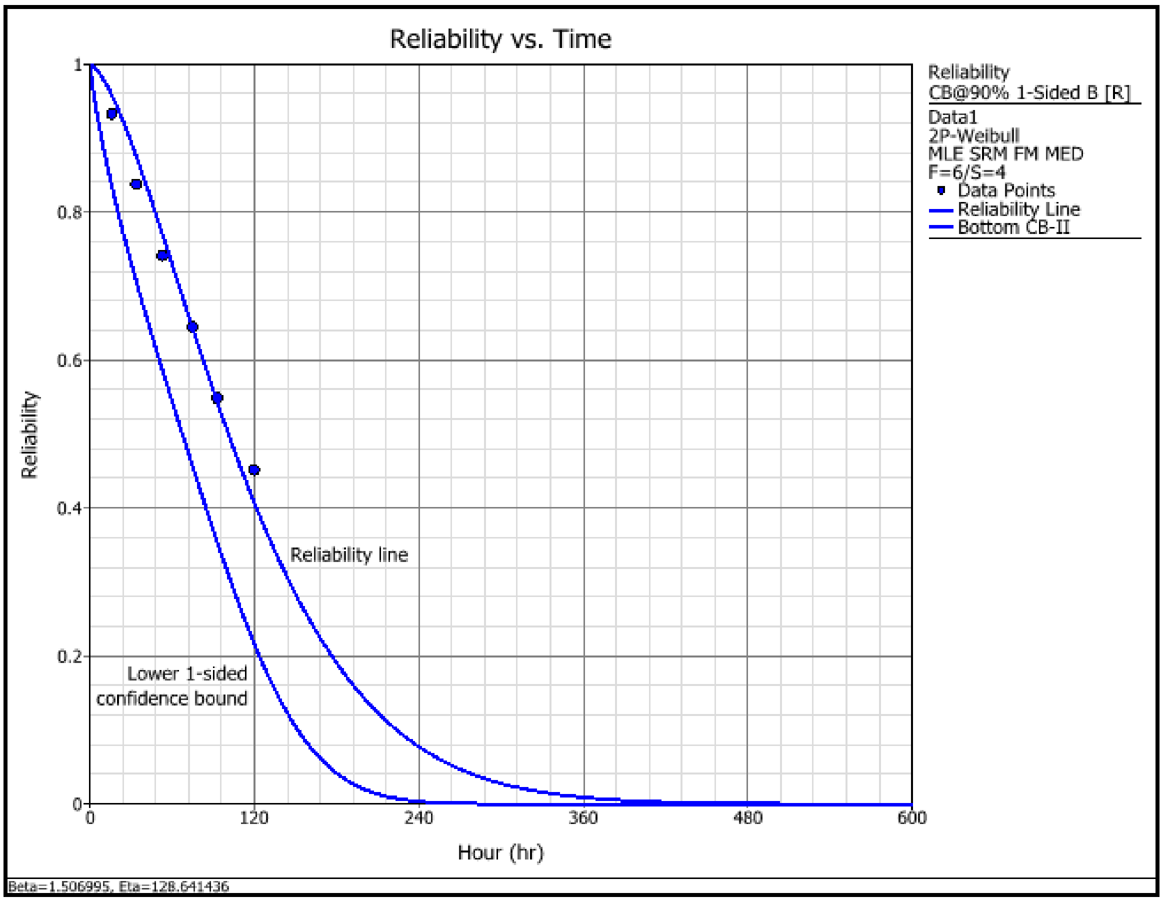
7. To visualize the reliability of the device over time, close the QCP and then create a plot by choosing **Life Data > Analysis > Plot**, or by clicking the icon on the Main page of the control panel.



8. Show the confidence bounds on 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 (with annotations added). 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.



Tools

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.

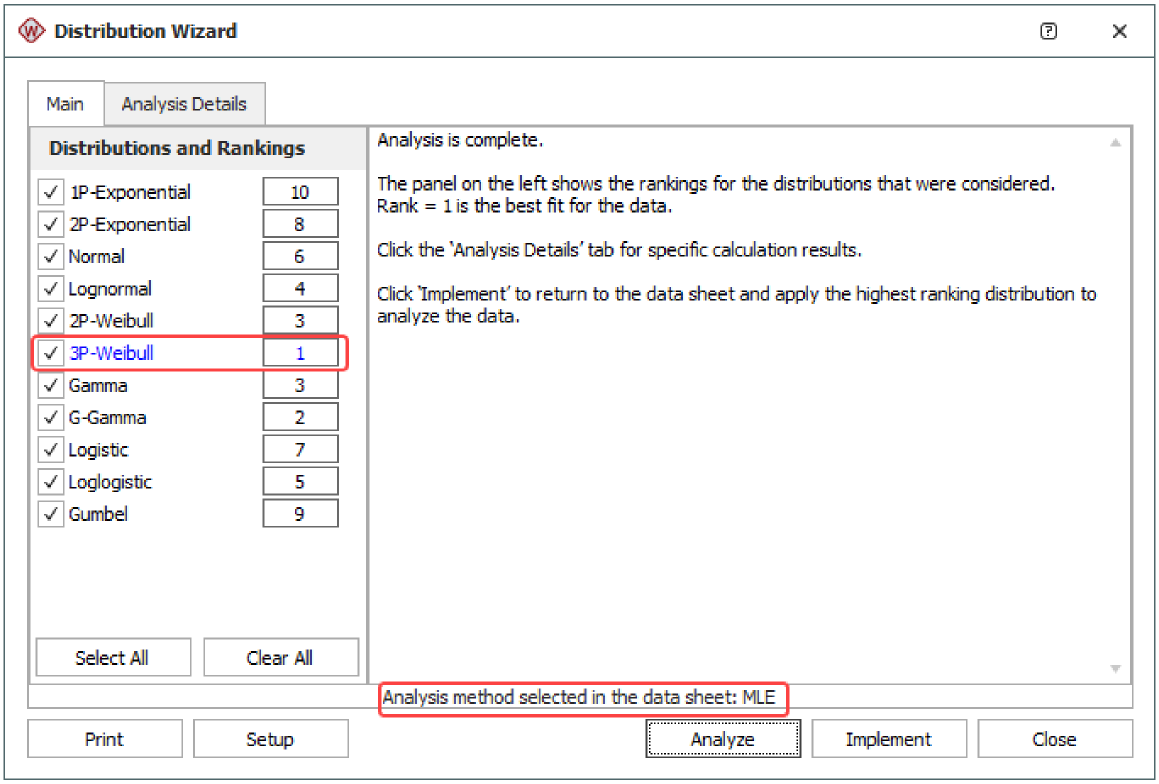
COMPARING DISTRIBUTIONS

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. Then 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. (The available distributions will vary depending on which folio you opened the tool from.) 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 3P-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.

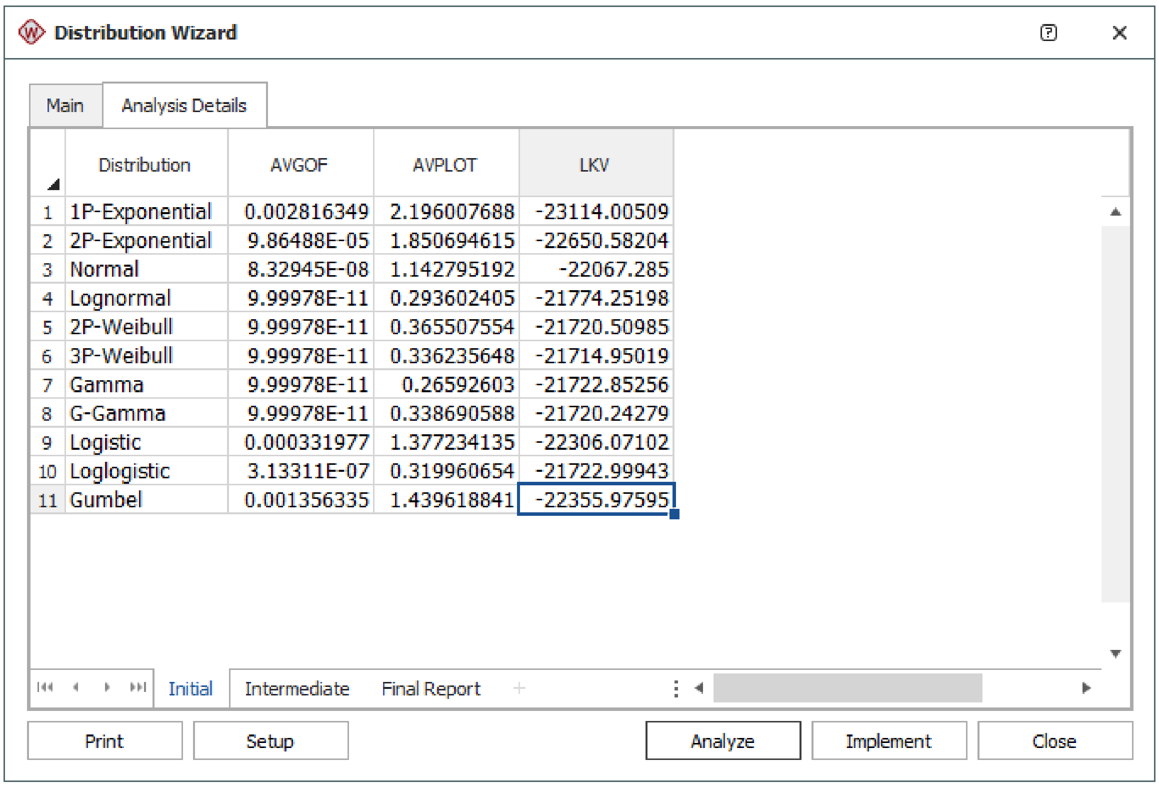
GOODNESS OF FIT TESTS

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 test (GOF)** tests for statistical difference (the difference between the expected and obtained results).

- The **Correlation coefficient test (PLOT)** measures how well the plotted points fit a straight line.
- The **Likelihood value test (LKV)** computes the value of the log-likelihood function, given the parameters of the distribution.

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	0.002816349	2.196007688	-23114.00509
2	2P-Exponential	9.86488E-05	1.850694615	-22650.58204
3	Normal	8.32945E-08	1.142795192	-22067.285
4	Lognormal	9.99978E-11	0.293602405	-21774.25198
5	2P-Weibull	9.99978E-11	0.365507554	-21720.50985
6	3P-Weibull	9.99978E-11	0.336235648	-21714.95019
7	Gamma	9.99978E-11	0.26592603	-21722.85256
8	G-Gamma	9.99978E-11	0.338690588	-21720.24279
9	Logistic	0.000331977	1.377234135	-22306.07102
10	Loglogistic	3.13311E-07	0.319960654	-21722.99943
11	Gumbel	0.001356335	1.439618841	-22355.97595

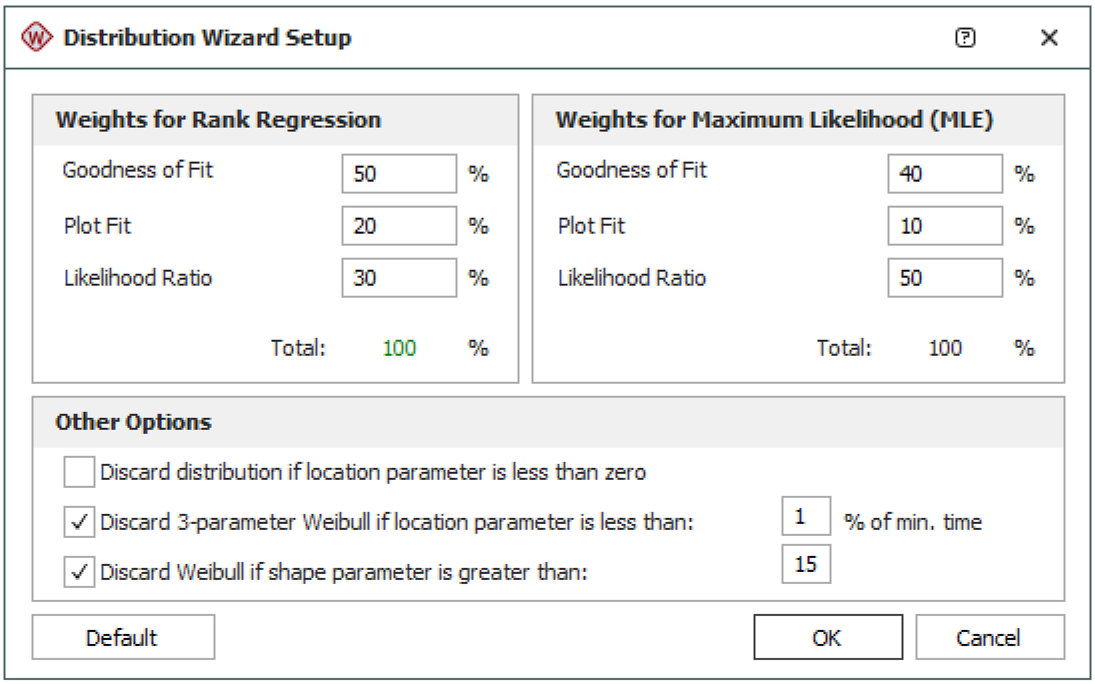
The **Intermediate** sheet displays the ranking of the distributions based on the values from the Initial sheet. The **RAVGOF** column contains the ranking of the GOF test, the **RAVPLOT** column contains the ranking of the PLOT test and the **RLKV** column contains the ranking of the LKV test.

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

When you open the Distribution wizard from the Weibull++ life data folio, you have the option to change the weights 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 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.

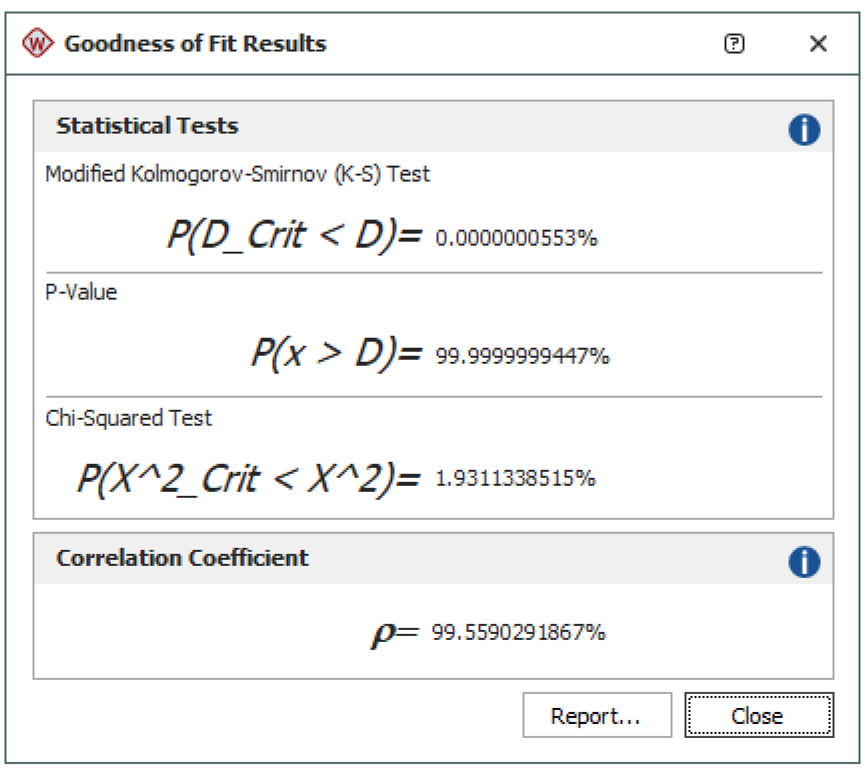
- **Discard 3-parameter Weibull if location parameter is less than (_)% of min. time** will exclude the Weibull distribution from the ranking if the value of the location parameter is smaller than the first time-to-failure. The default setting is for a value that is less than 1% of the first time-to-failure, 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.

Goodness of Fit Results

The Goodness of Fit Results utility performs statistical 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.

To use the utility, first calculate the parameters of the distribution of your data set, and then choose **Life Data > Options > Goodness of Fit Results**.

The following picture shows sample results for a particular data set.



- 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 *χ^2_Crit* for chi-squared) is less than the calculated statistic value (*D* statistic for K-S and *χ^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.

Specify Points

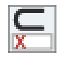
The Specify Points feature enables you to experiment with possible alternative scenarios by allowing you to adjust the way that the line is fitted to the points for a rank regression analysis. The ranks will be calculated based on all of the data points, but the regression line (which determines the 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, first identify the data points that you wish to include in the regression line/parameter estimation. In the Weibull++ life data folio data sheet, assign the selected data points to a specific subset ID, as shown in the following example.

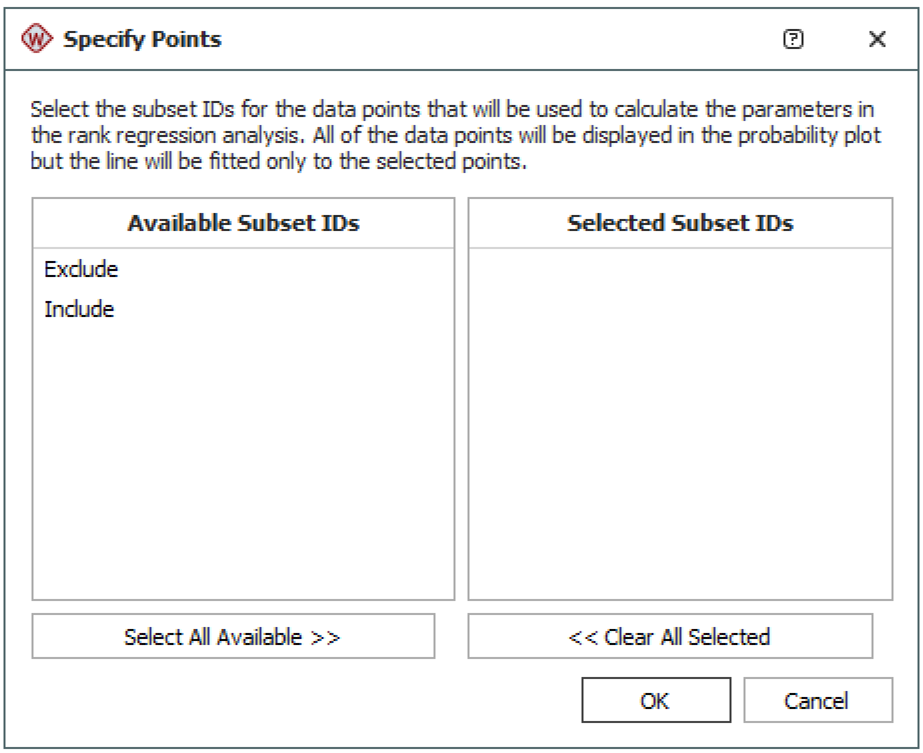
	Time Failed (hr)	Subset ID 1
1	21.691	Exclude
2	27.2772	Exclude
3	32.6591	Exclude
4	87.4316	Include
5	95.2312	Include
6	108.8436	Include
7	124.4518	Include
8	152.9014	Include
9	166.4217	Include
10	220.5643	Include

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.

Choose **Life Data > Options > Specify Points** or click the icon on the Main page of the control panel.



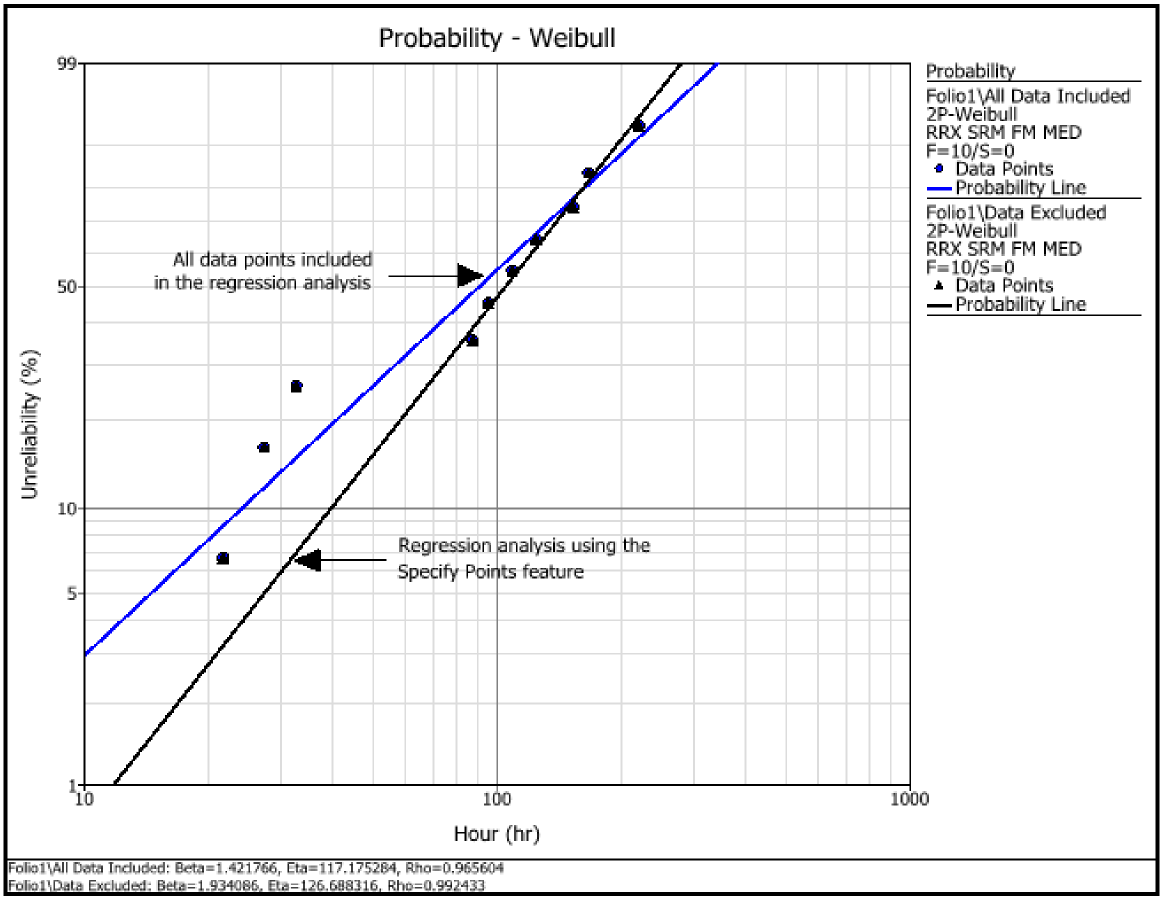
You will be presented with a list of all the subset IDs in your data sheet as shown next. If you have more than one subset ID column in your data sheet, a drop-down list will be available to allow you to choose which column to use in the analysis.



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.

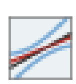
The overlay plot shown next illustrates the changes to the estimates of the parameters between the cases where the three data points are either included or excluded from the regression analysis.



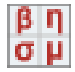
Fill Median Ranks

The Fill Median Ranks feature is available only for data sheets that have been configured to support free-form data. 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.

The ReliaWiki resource portal has more information on median ranks at: http://www.reliawiki.org/index.php/Parameter_Estimation.

Special Analysis Methods

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.

The ReliaWiki resource portal provides more information about the mixed Weibull distribution at http://www.reliawiki.org/index.php/The_Mixed_Weibull_Distribution.

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.

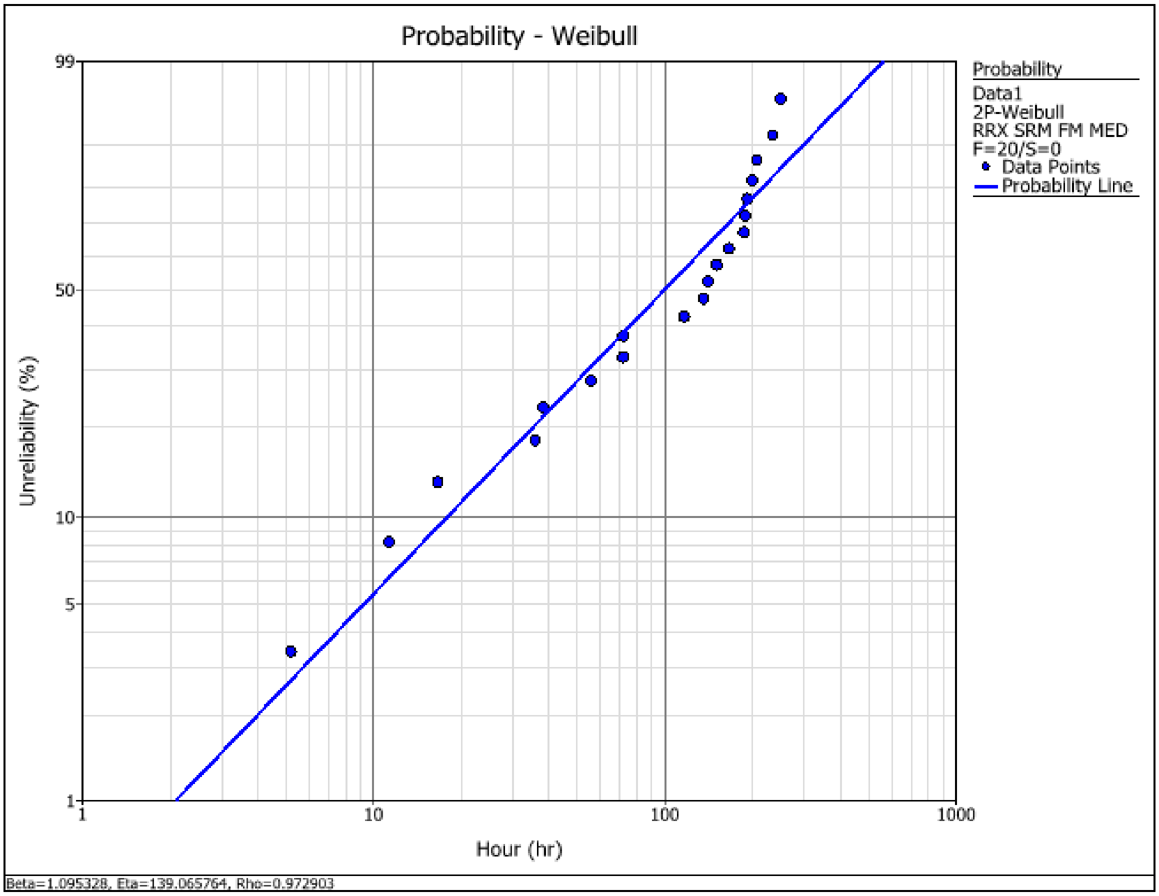
EXAMPLE

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	140.1
11.3	150.5
16.6	166.0
35.9	186.7
38.2	188.8
55.7	191.3
71.8	198.8
72	206.7
116.4	234.3
135.8	249.8

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++ life data folio.

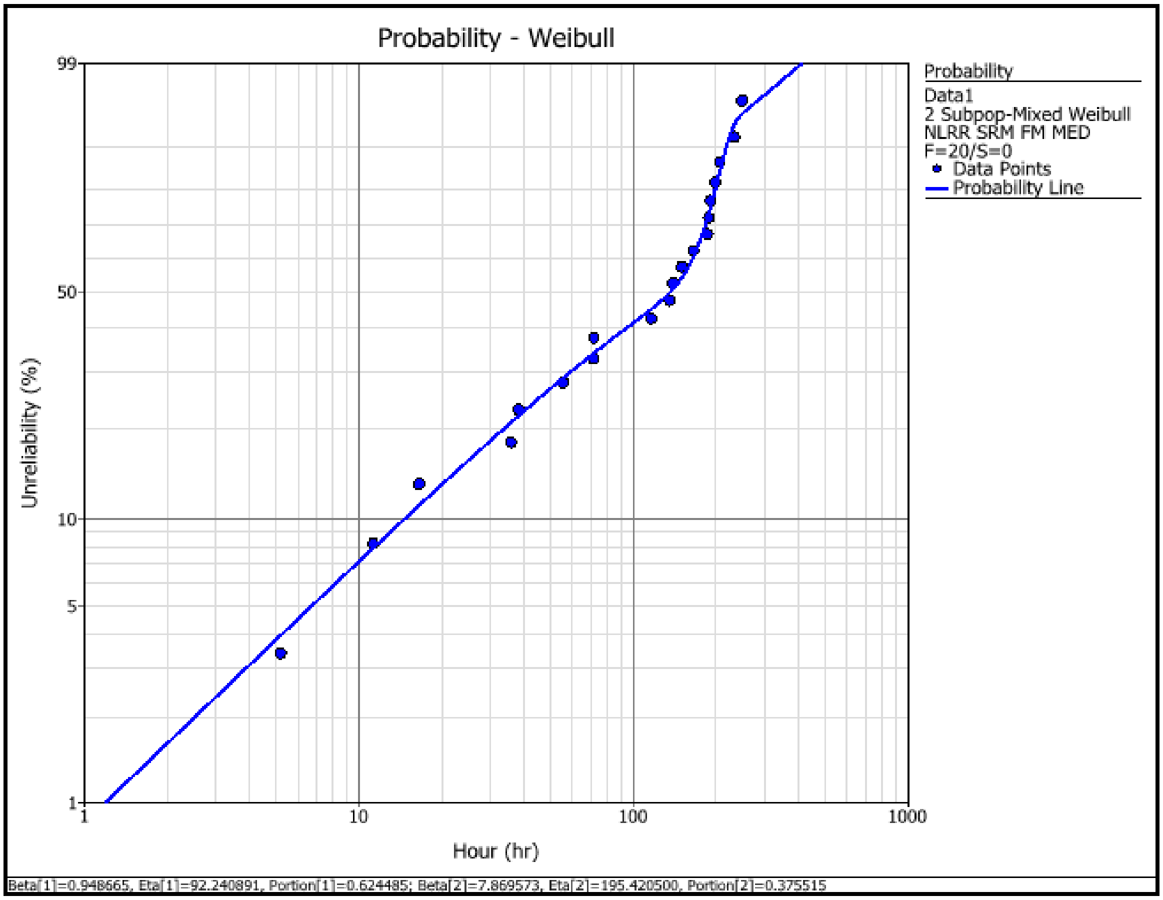
2. On the Main page of the control panel, click the **Distribution** drop-down list. Choose **Mixed Weibull** and then choose the number of subpopulations to consider (for this example, choose **2 Subpop-Mixed Weibull**). Click **Calculate**.

   Weibull++ will automatically fit a 2 subpopulation mixture model with Weibull parameters and portion values. In the results shown next, **Beta[1]** and **Eta[1]** are the parameters that were calculated for the first subpopulation. The result showing Portion[1] = 0.6245 means that subpopulation 1 comprises 62.45% of the data points.

Analysis Summary	
Parameters	
Beta[1]	0.948665
Eta[1] (hr)	92.240891
Portion[1]	0.624485
Beta[2]	7.869573
Eta[2] (hr)	195.420500
Portion[2]	0.375515
Other	
Rho	0.987811
LK Value	-111.765246
Failures/Suspensions	
F/S	20/0
Comments	

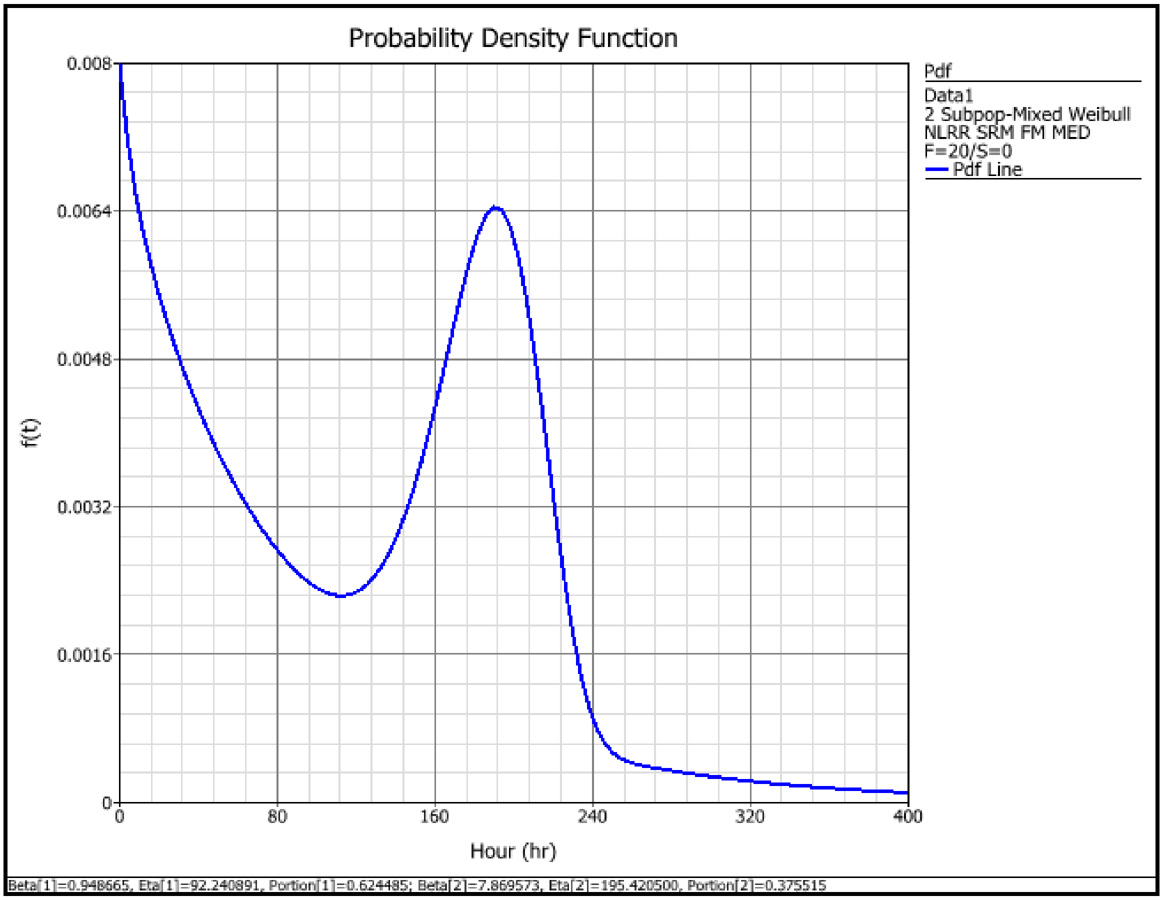
Note: The subpopulation numbers are ordered based on the value of the scale parameter (eta). The order is used only to identify the subpopulation and does not affect the analysis.

3. To view the probability plot of the data set, click the **Plot** icon on the control panel. The plot shows that 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).



To visualize the mixed Weibull distribution, choose **Pdf Plot** in the Plot Type drop-down list. The plot shows that 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.



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.

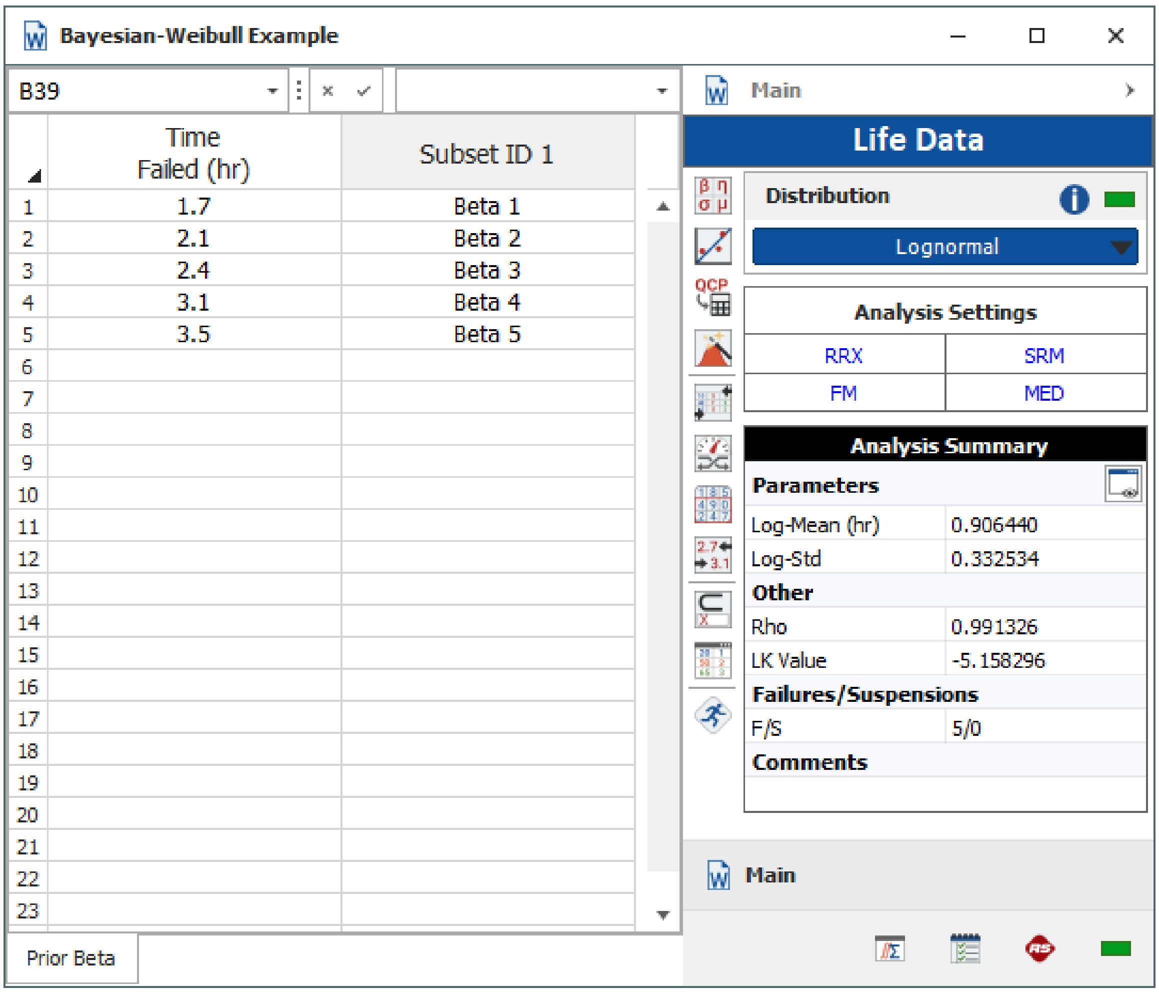
The prior information in the analysis is in the form of a distribution fitted to a range of likely beta values. In the context of life data analysis, the beta parameter is used to define the shape of the Weibull distribution, which reflects the failure behavior of a product (i.e., whether the failure 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.

The ReliaWiki resource portal provides more information about Bayesian-Weibull analysis at: http://www.reliawiki.org/index.php/Bayesian-Weibull Analysis.

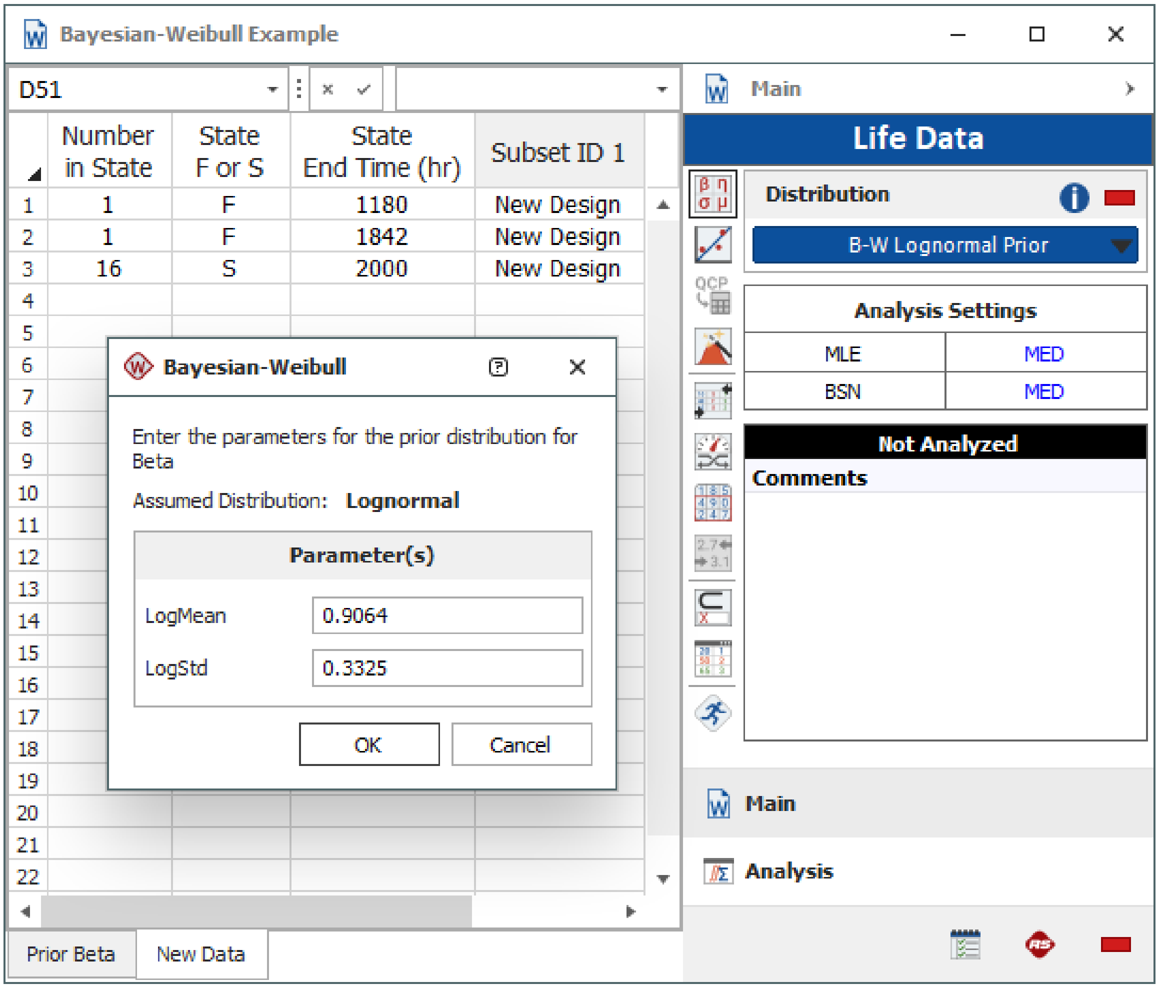
EXAMPLE

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 first need to obtain the prior distribution of beta in order to describe the failure behavior 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.

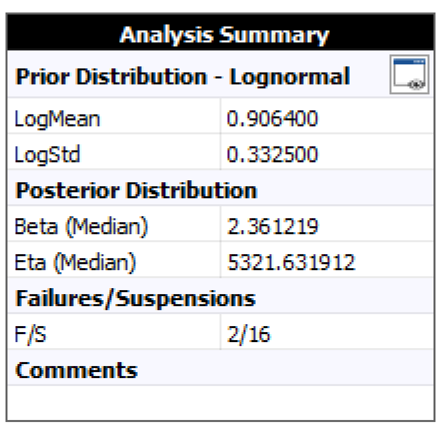


Next, collect the current times-to-failure data of the new design and enter the information in a new data sheet, as shown next. On the Main page of the control panel, click the **Distribution** drop-down list and choose **Bayesian-Weibull > B-W Lognormal Prior**. Click **Calculate**. A window similar to the one shown in the inset will be displayed.



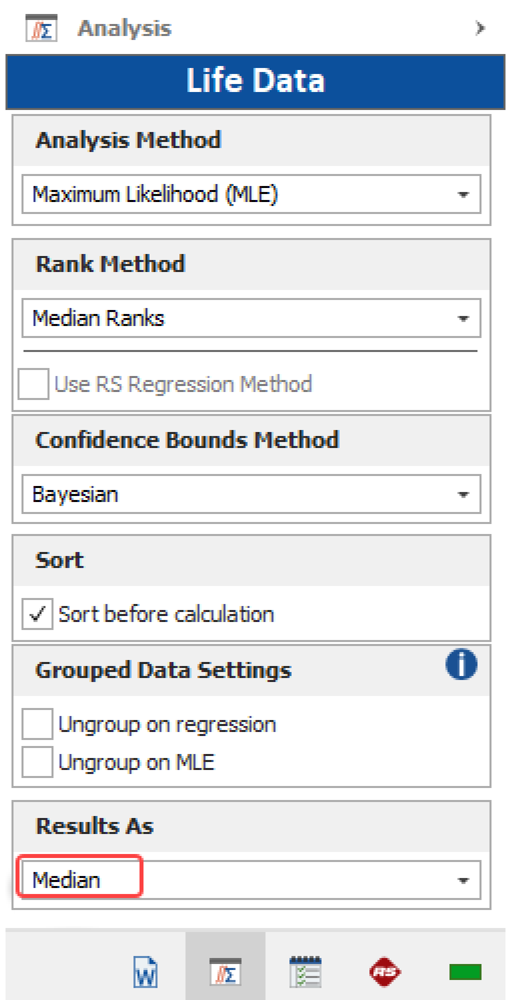
Enter the calculated parameter values of the prior distribution of beta: LogMean = 0.9064 and LogStd = 0.3325. Click **OK**.

The following figure shows the results of the analysis.



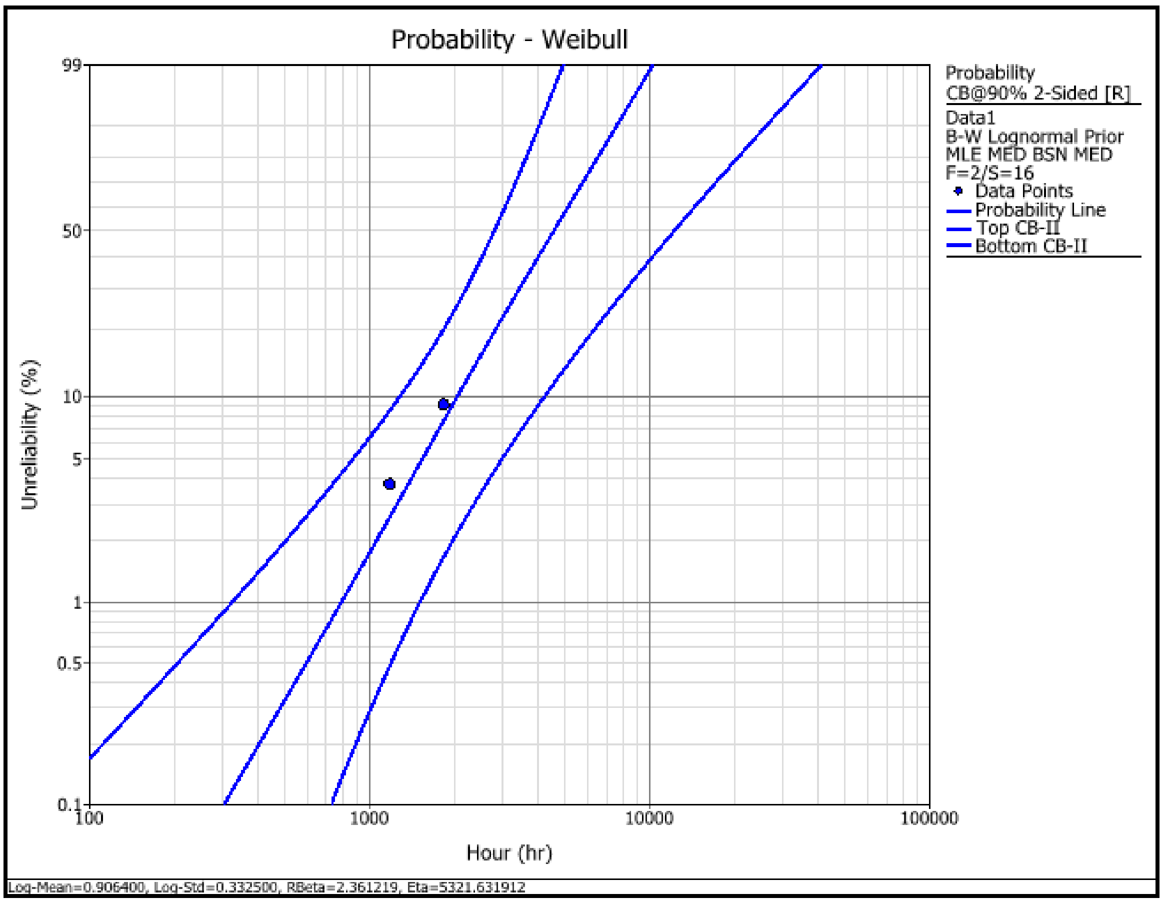
Note that instead of a single calculated value for each of the parameters, the parameters of the Bayesian-Weibull distribution are distributed. This result is known as a *posterior distribution*. In the example, the result showing that Beta (Median) = 2.3612 means that the median value of the posterior 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.

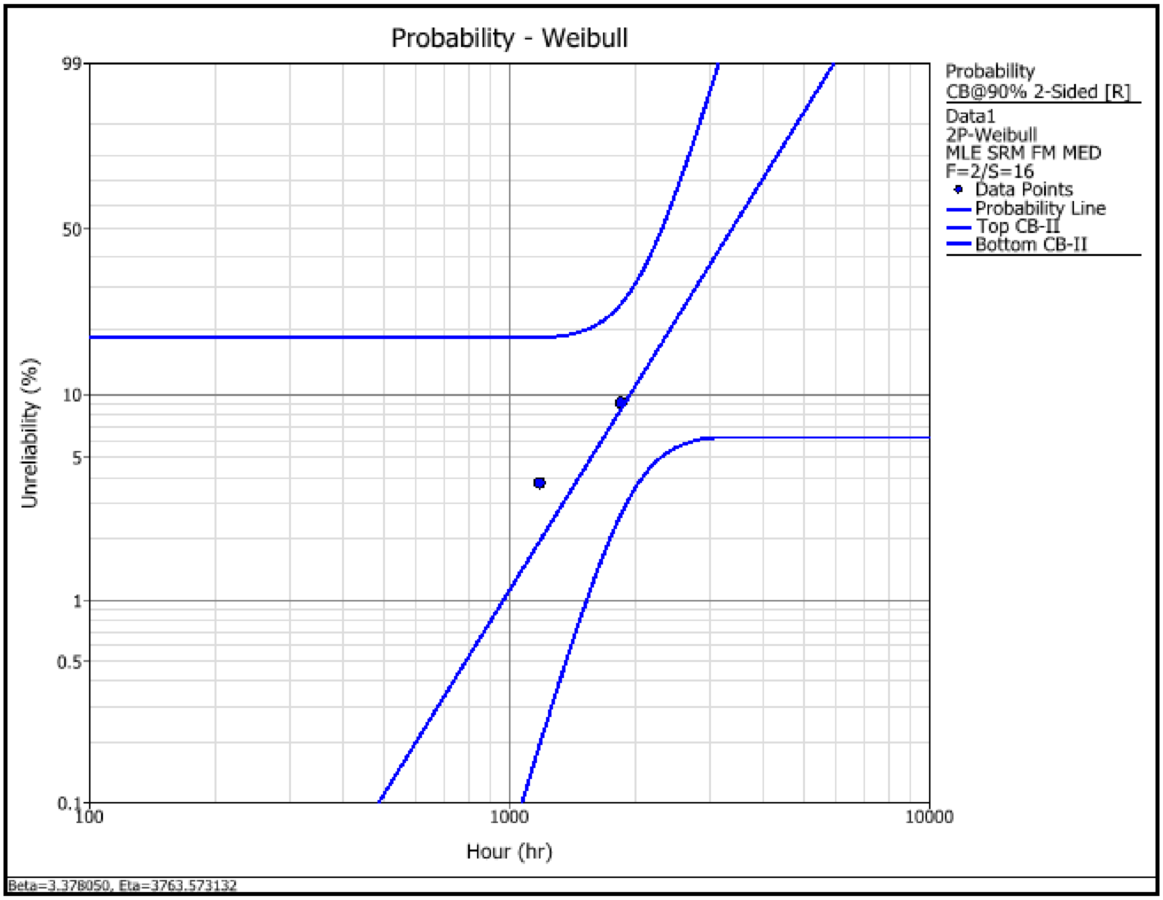


To obtain point estimates of a metric, such as the probability of failure or the reliability at a given point in time, use the Quick Calculation Pad (QCP). The QCP will automatically compute the metric of interest using the median value of the posterior distribution (or the mean, if you switch the setting 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

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:

$$R(t) = R_1(t) \cdot R_2(t) \cdot \ldots \cdot R_n(t) \tag{1}$$

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++ life data 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.

The ReliaWiki resource portal provides more information about competing failure modes analysis at: http://www.reliawiki.org/index.php/Competing_Failure_Modes_Analysis.

Tip: When it is not possible to categorize your data set into distinct modes, mixed Weibull analysis may be more appropriate.

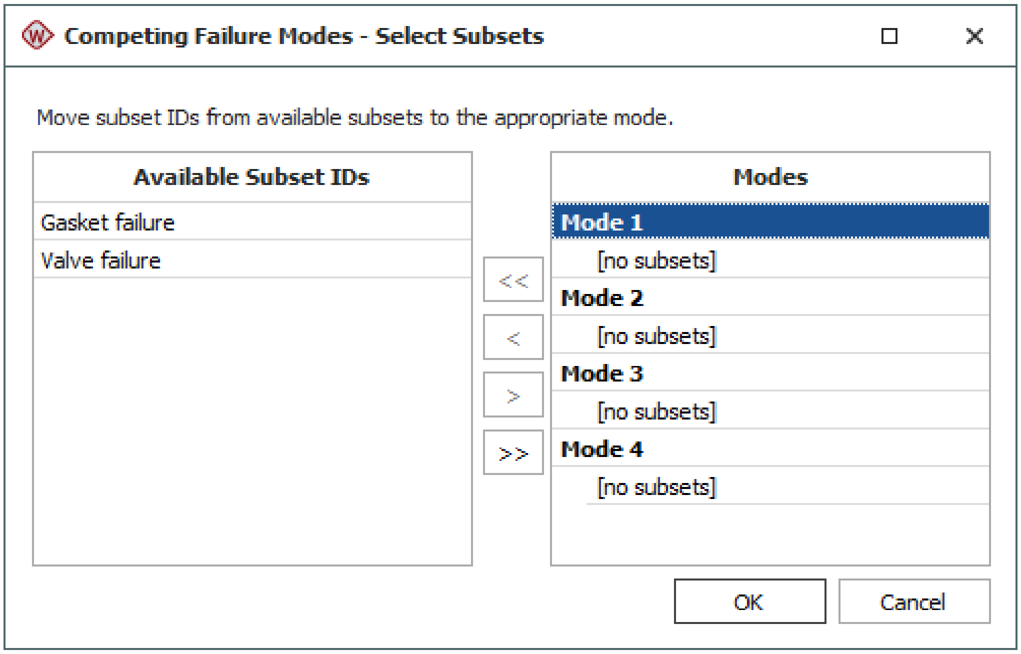
EXAMPLE

Consider a water pump that is considered failed when either its gasket or its valve fails. To use CFM analysis in a Weibull++ life data 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
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 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. If you have more than one subset ID column in your data sheet, a drop-down list will be available to allow you to choose which column to use in the analysis.



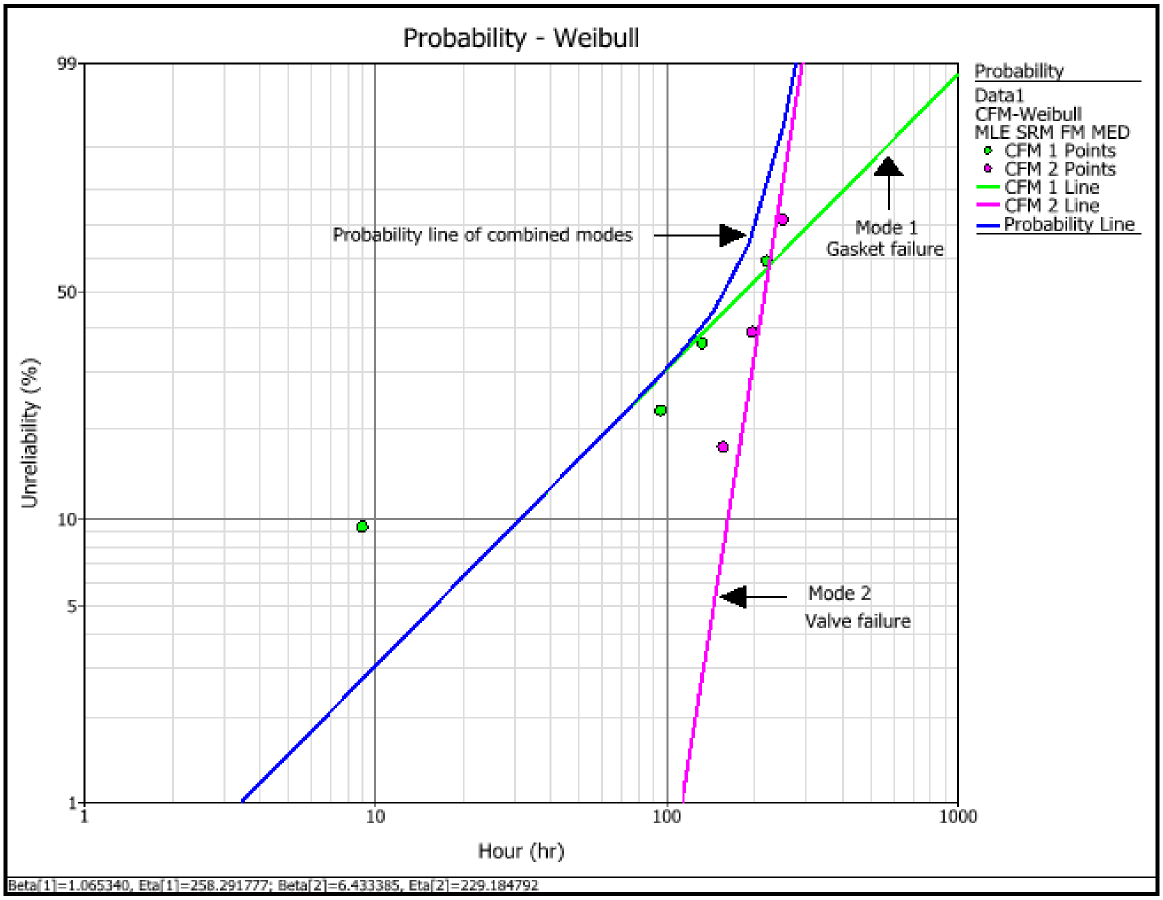
You can define up to four modes to include in the CFM analysis, and each mode can be associated with one or more subset IDs. (For example, consider a product with components A, B and C, where components B and C are identical, used in two places in the product and subjected to identical stresses. In this 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.)

First, select the subset ID(s) in the **Available Subsets** column. Then select a mode from the **Modes** column and 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.

Analysis Summary	
Parameters	
Beta[1]	1.065340
Eta[1] (hr)	258.291777
Beta[2]	6.433385
Eta[2] (hr)	229.184792
Other	
LK Value [1]	-26.302222
LK Value [2]	-16.186679
Comments	

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.



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++ life data 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."

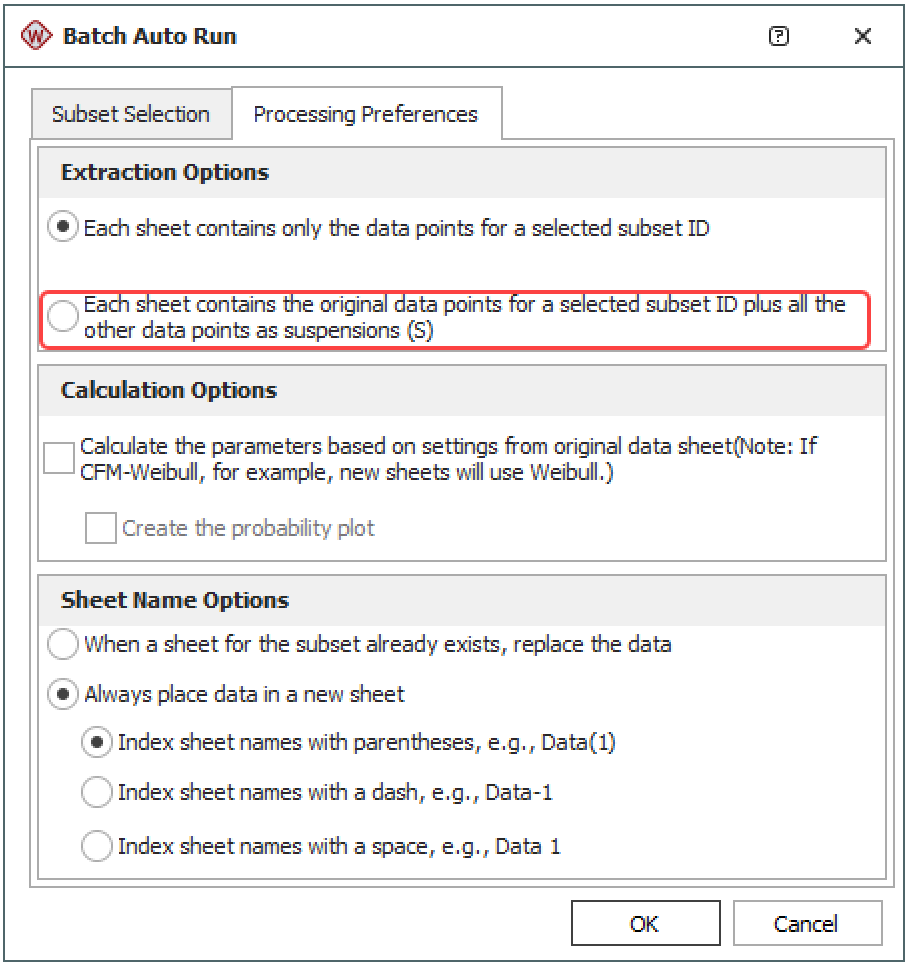
Example

Using the same water pump system in the previous example, we'll create an RBD that illustrates the relationship between the failure modes.

The first step is to analyze separately 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.

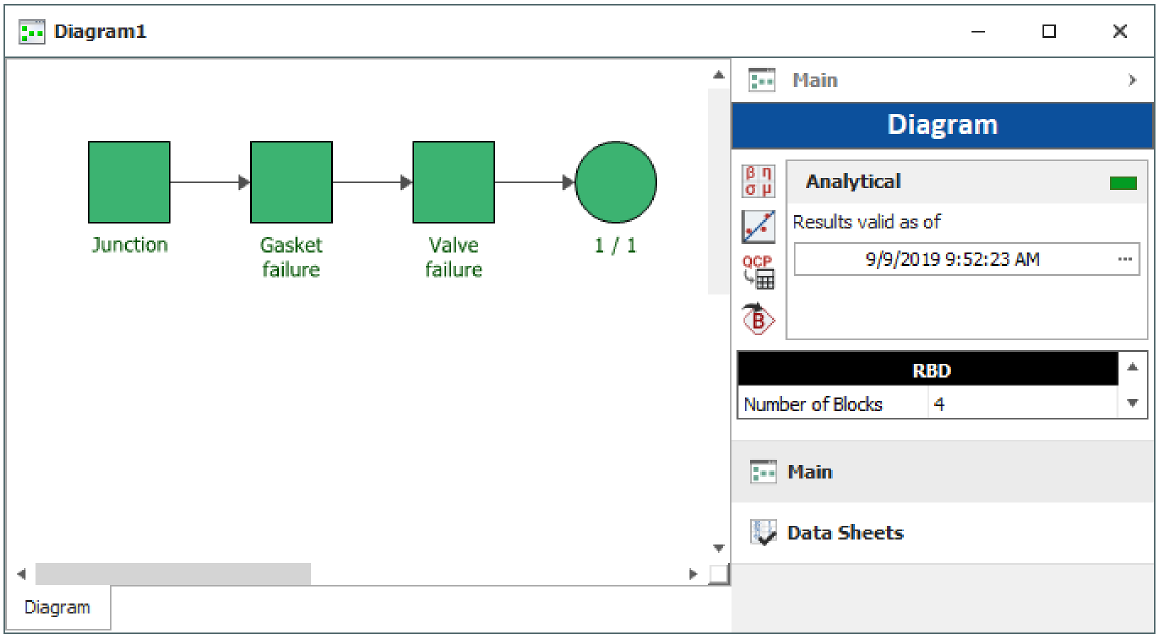
2. In the Batch Auto Run window, click the **Select All Available** button at the bottom of the window to automatically include all the modes in the batch auto run process.
3. Click the **Processing Preferences** tab and select the option shown next.



3. 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 both data sheets).

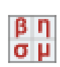
4. The next step is to describe the relationships between the failure modes. Add an RBD to the project by right-clicking the RBD folder in the current project explorer and choosing **Add Diagram** on the shortcut menu.

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 Building Diagrams in Weibull++.

1. Add a junction block by choosing **Diagram** > **Blocks** > **Add Junction**. The junction block serves as a unique starting point for the diagram.
2. Add blocks by choosing **Diagram** > **Blocks** > **Add Block**. You will be asked to select the data sheet of the failure mode that the block will represent.
3. Add nodes by choosing **Diagram** > **Blocks** > **Add Node**. The nodes act as switches that the diagram paths move through.
4. Add arrows by choosing **Diagram** > **Settings** > **Connect Blocks**.
5. 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).

Fractional Failure Analysis

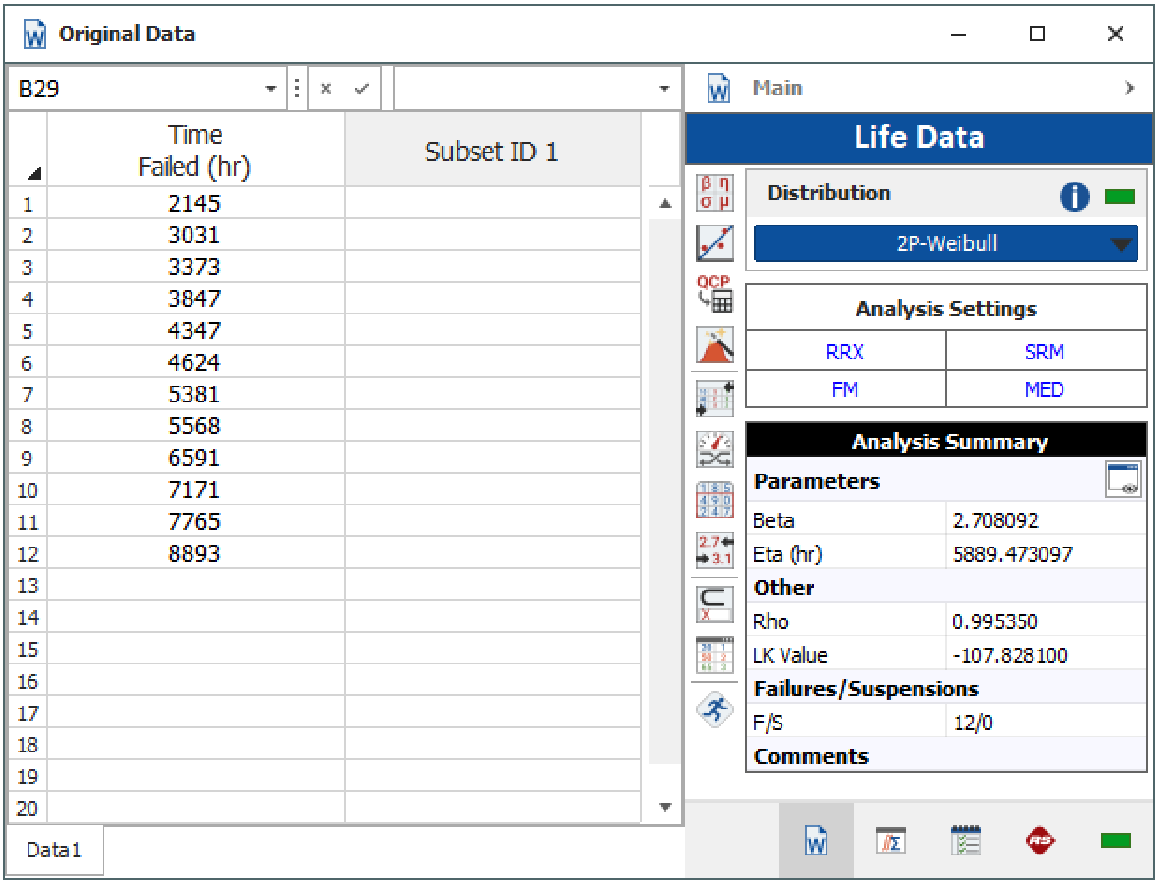
You can use fractional failure analysis in Weibull++ life data folios to account for the effectiveness of an assumed corrective action or product redesign in your analyses. For example, if the fix for a failure is expected to be 70% effective, you can enter the failure count as 0.3 (30%) in the data sheet. This allows you to analyze a product's reliability, B10 life or other metrics, assuming an implemented fix.

EXAMPLE

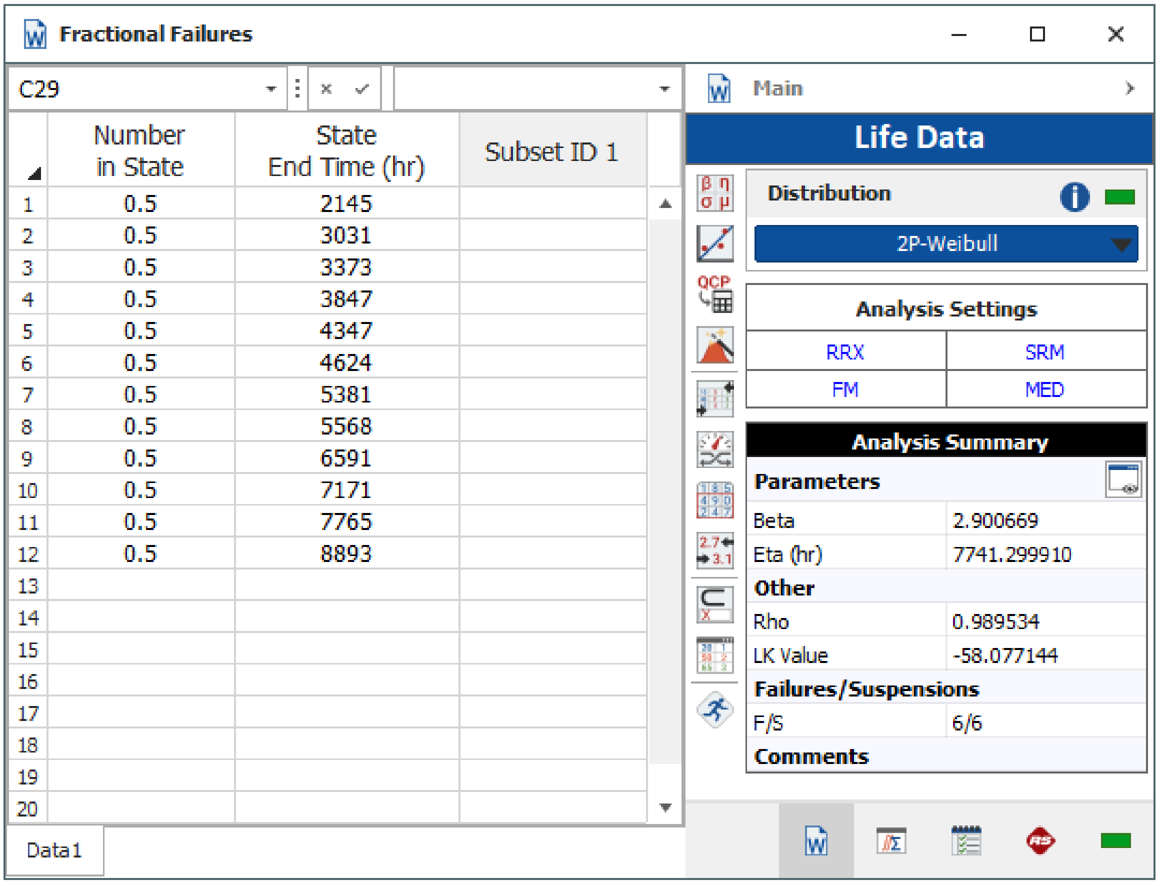
*The data set used in this example is available in the example database installed with the Weibull++ software (called "Weibull21_Examples.rsgz21"). To access this database file, choose **File > Help**, click **Open Examples Folder**, then browse for the file in the Weibull sub-folder.*

The name of the project is "Standard - Fractional Failures."

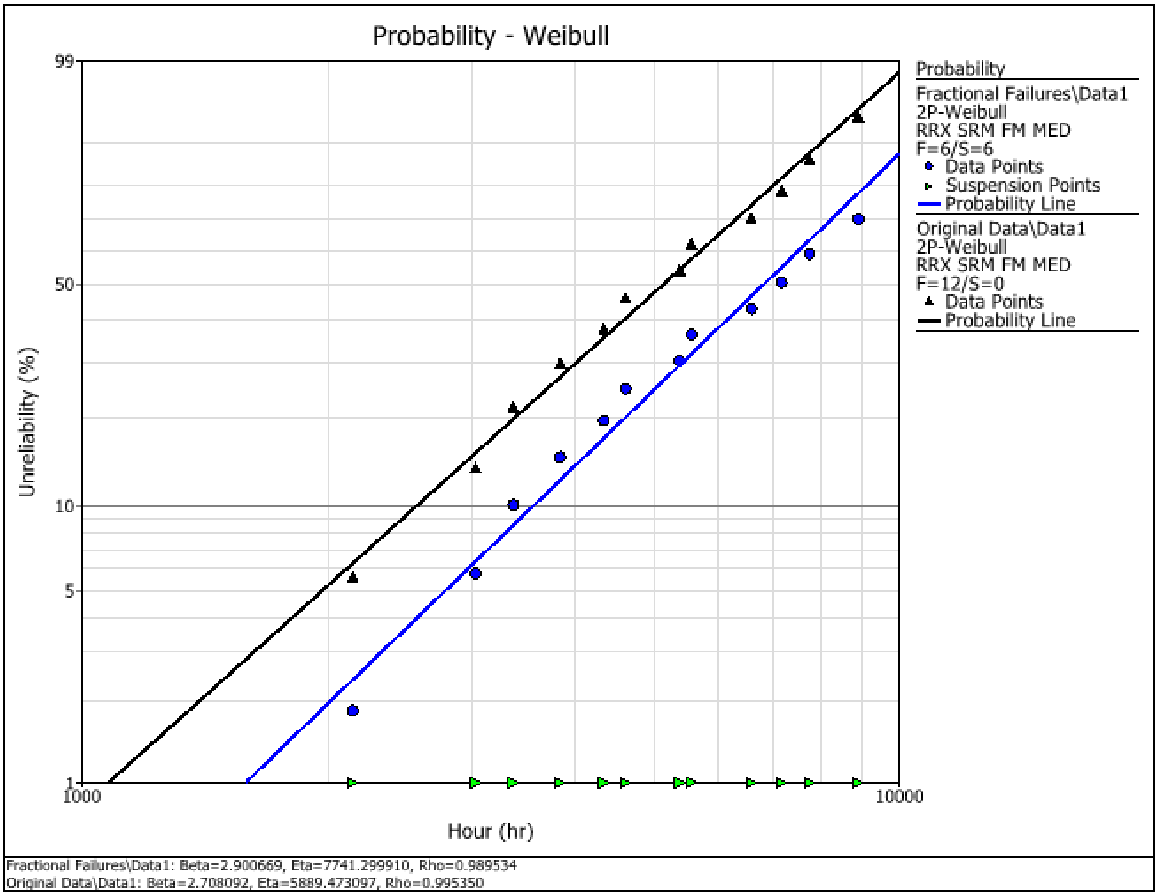
A product underwent a reliability test. The following picture shows the data set as it was entered in Weibull++.



The analysts believe that the planned design improvements will yield 50% effectiveness. To estimate the reliability of the product based on the assumptions about the repair effectiveness, they enter the data in groups, counting a 0.5 failure for each group, as shown next.



The overlay plot, shown next, compares the changes to the estimates in the parameters between the original data set and the data set that includes the repair effectiveness.



Non-Parametric LDA Folio

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.

For more information on non-parametric vs. parametric analysis, please read "*Data Analysis and Reporting*" at: http://www.weibull.com/.

Non-Parametric LDA Folio Data Sheet

To create a non-parametric LDA folio, choose **Home > Insert > Non-Parametric LDA**, or right-click the **Life Data** 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. The ReliaWiki resource portal has more information on these analysis methods at: http://www.reliawiki.org/index.php/Non-Parametric Life Data Analysis.

Kaplan-Meier Analysis

The following example shows the data sheet for the Kaplan-Meier analysis. 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.

	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

Actuarial-Simple Analysis

The following example shows the data sheet for the Actuarial-Simple 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.

	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

Actuarial-Standard Analysis

The following example shows the data sheet 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.

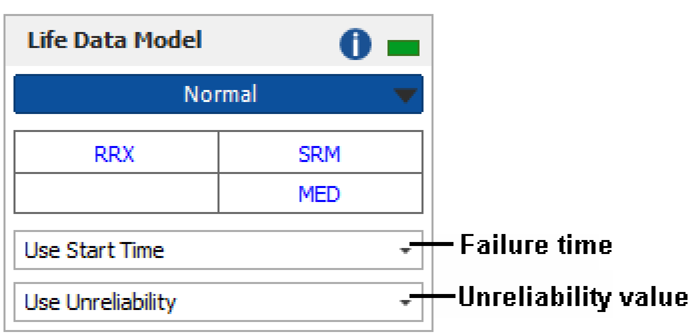
The data sheet contains information about the start time and end time of each interval, as well as the number of units in each interval that are either failures or suspensions. The **Available Units** column shows the number of units at the start of the interval and the **Adjusted Units** column shows 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.

	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

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 topic 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 Control Panels.

- 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++ life data folio, except for the two drop-down lists shown next:



The two options allow you to choose which values will be fitted with the chosen life distribution.

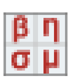
- **Failure time**: If you use the Kaplan-Meier analysis, the failure times are the end times of the intervals. If you use either the Actuarial-Simple or Actuarial-Standard analysis, each failure time may be the start time of the interval, end time of the interval, or the average between the start time 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.

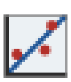
- The **Non-Parametric Results** area allows you to view the reliability estimates calculated by the non-parametric analysis and the free-form data set that will be used for parametric analysis. Click the **Non-Parametric Results (...)** button to open the Results window, which allows you to view, edit 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 click the Quick Calculation Pad (QCP) icon on the control panel 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 the 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++ life data folio QCP. This tool is also available by choosing **Non-Parametric LDA > Analysis > Quick Calculation Pad**.

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**.

Change Units opens the Change Units window, which allows you to change the units for the values in the current data sheet.

Transferring Data to a Weibull++ Life Data 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++ life data folio. Choose **Non-Parametric LDA > Transfer Life Data > Transfer Life Data to 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 life data 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 life data folio's QCP, or to generate other graphical plots in the life data folio's plot sheet. If you use the same analysis settings in both the life data folio and non-parametric LDA folio, then you will obtain the same results from the QCPs in both folios.

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 "Weibull21_Examples.rsgz21"). 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.

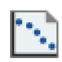
Units that are reported to have malfunctioned during the test period are marked as failed (F), while units that did not exhibit any problems are marked as suspensions (S). At the end of the 41-week test, the analyst's log shows the following information. The objective is to create a Reliability vs. Time plot in order to predict the failure rate behavior of the product.

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
1	S	26
1	F	28
1	F	30
1	S	32
2	S	35
1	S	39
1	S	41

Create a Non-Parametric LDA Folio

1. On the Home tab, click the **Non-Parametric LDA** icon in the Insert gallery.



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.

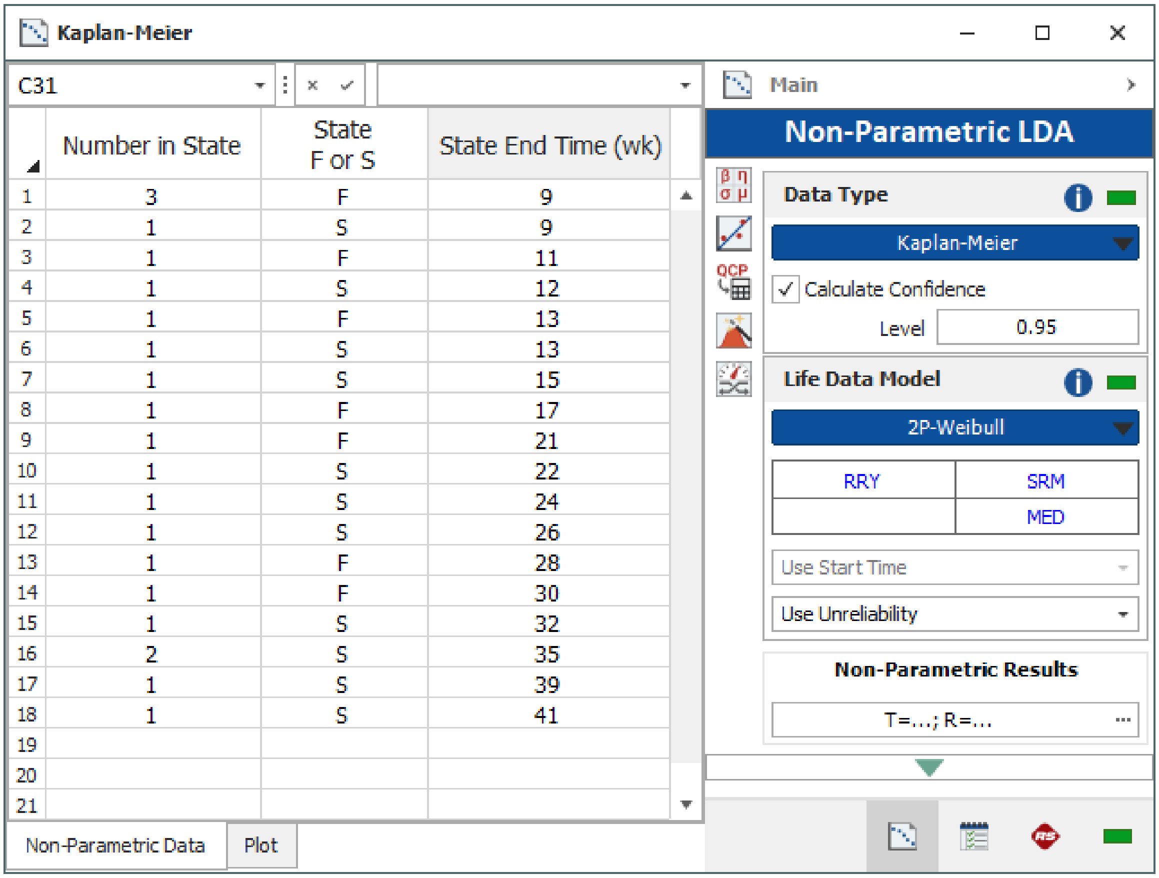
   In the Change Units window, change the time unit to **Week**. Click **OK**.

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 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:

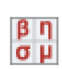
- **2P-Weibull**.
- **RRY** (Rank Regression on Y).
- **Use Unreliability**. This means that the free-form data set will be based on the unreliability estimates calculated from the non-parametric analysis, rather than the upper or lower confidence bound.

  The following picture shows the completed setup.



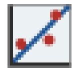
Analyze the Data Set

6. Choose **Non-Parametric LDA > Analysis > Calculate** or click the icon on the control panel.

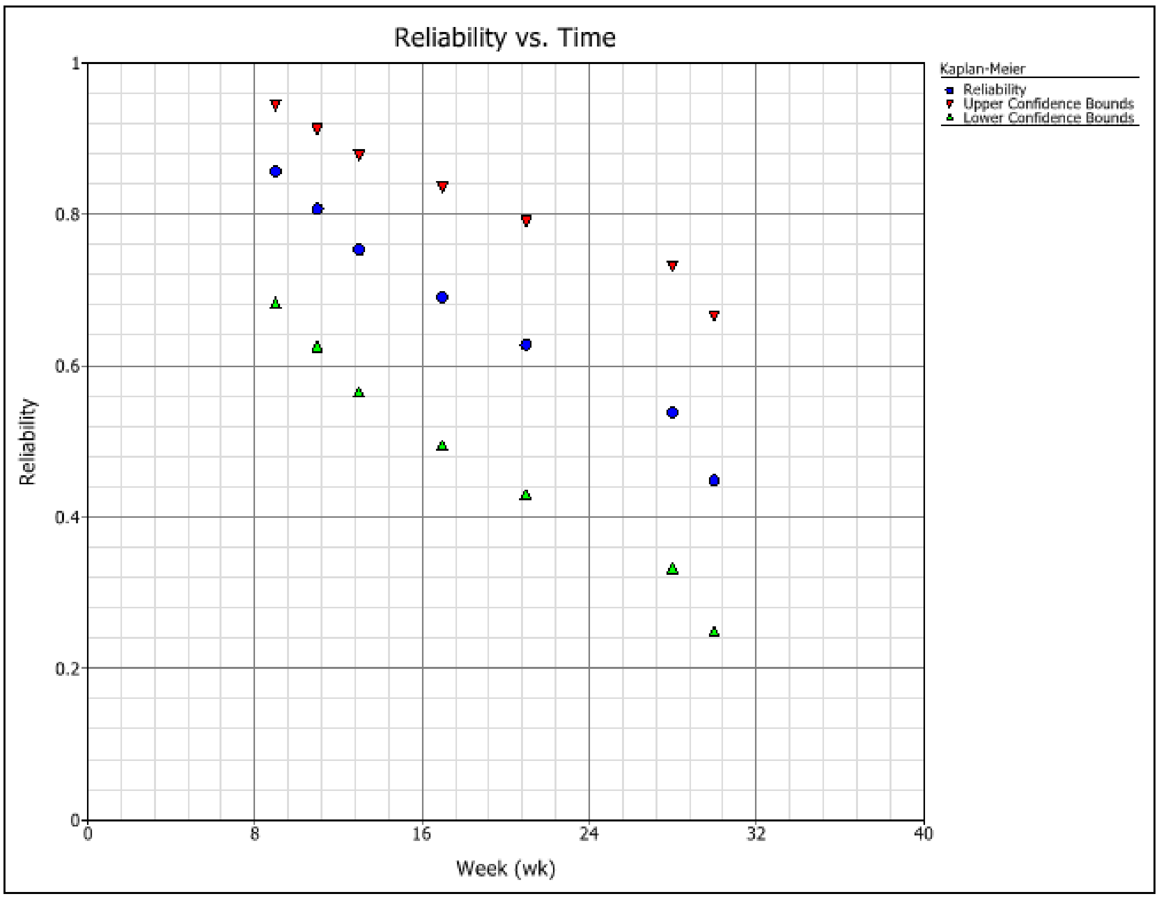


   The software will perform a non-parametric analysis on the data set, and then automatically perform a separate standard life data analysis based on the settings specified in the Life Data Model area.

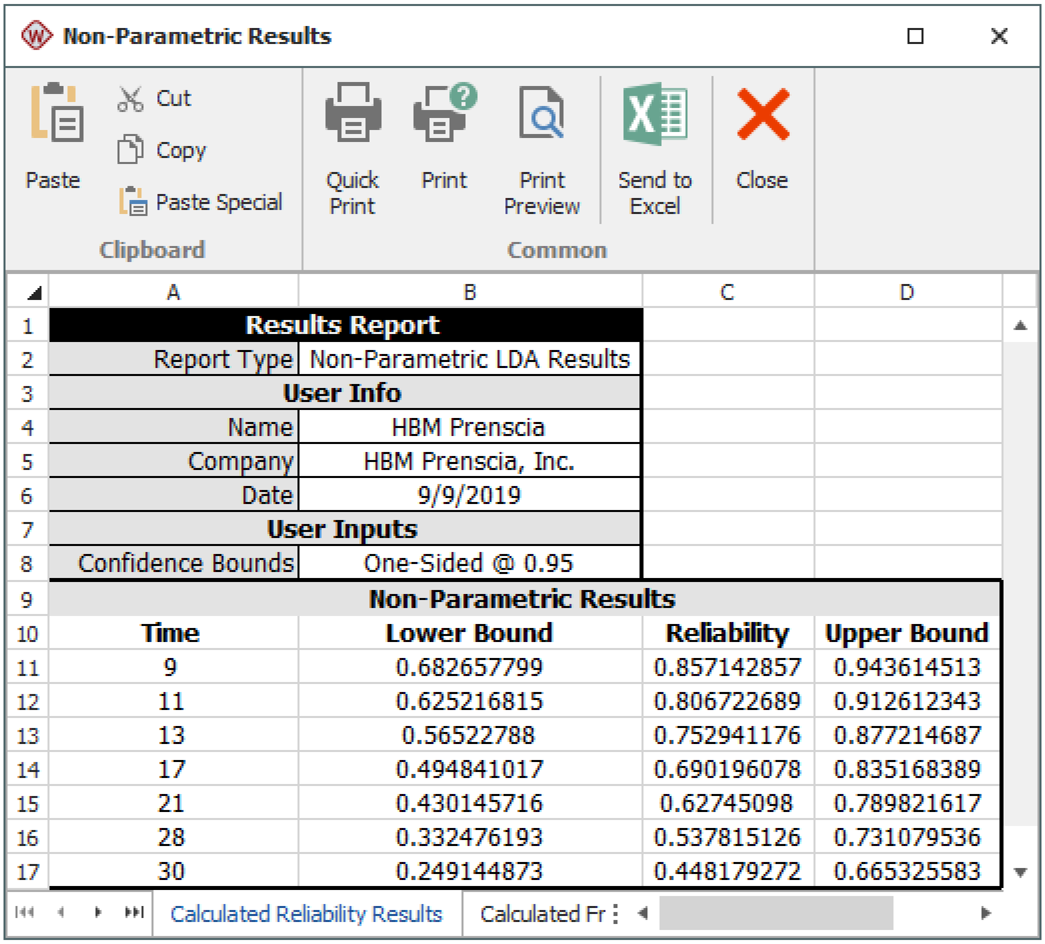
7. To create the Reliability vs. Time plot, choose **Non-Parametric LDA > Analysis > Plot** or click the icon on the control panel.



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. Click the **Non-Parametric Results (...)** button on the control panel to open the Results window, as shown next. 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%.



Results Report			
Report Type	Non-Parametric LDA Results		
User Info			
Name	HBM Prenscia		
Company	HBM Prenscia, Inc.		
Date	9/9/2019		
User Inputs			
Confidence Bounds	One-Sided @ 0.95		
Non-Parametric Results			
Time	Lower Bound	Reliability	Upper Bound
9	0.682657799	0.857142857	0.943614513
11	0.625216815	0.806722689	0.912612343
13	0.56522788	0.752941176	0.877214687
17	0.494841017	0.690196078	0.835168389
21	0.430145716	0.62745098	0.789821617
28	0.332476193	0.537815126	0.731079536
30	0.249144873	0.448179272	0.665325583

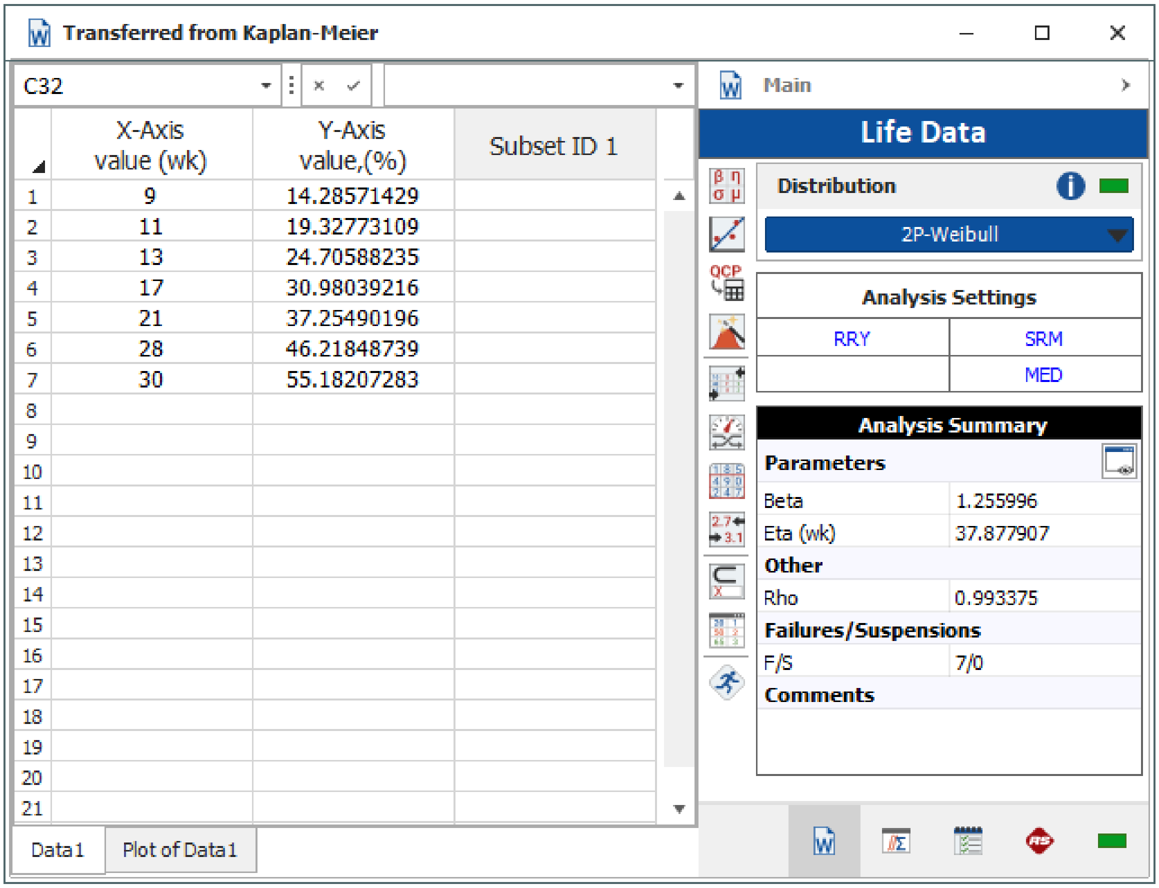
Evaluate the Life Data Model

To assess whether the assumed distribution (that was chosen in the Life Data Model area of the control panel) provides a good fit, transfer the results of the non-parametric analysis to a Weibull++ life data folio.

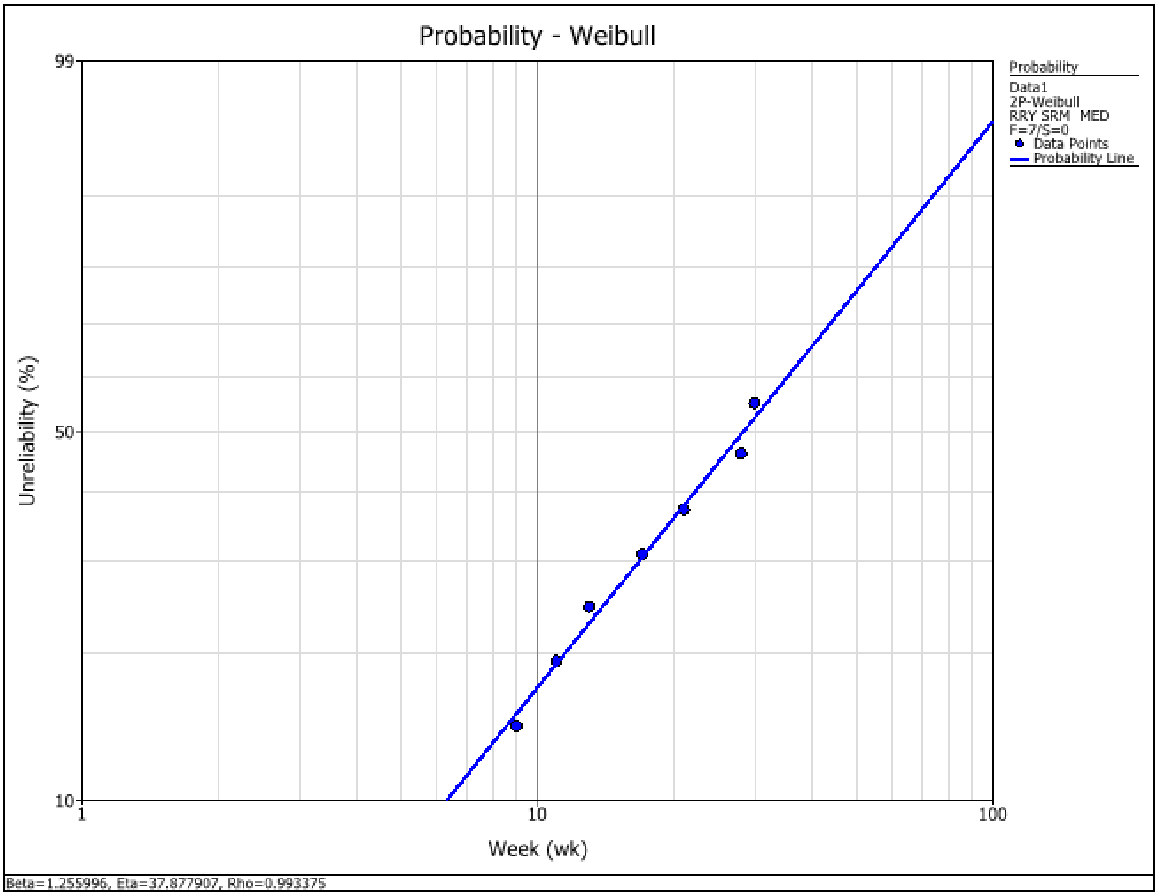
9. Choose **Non-Parametric LDA > Transfer Life Data > Transfer Life Data to New Folio**.

   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 life data 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.



10. 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).



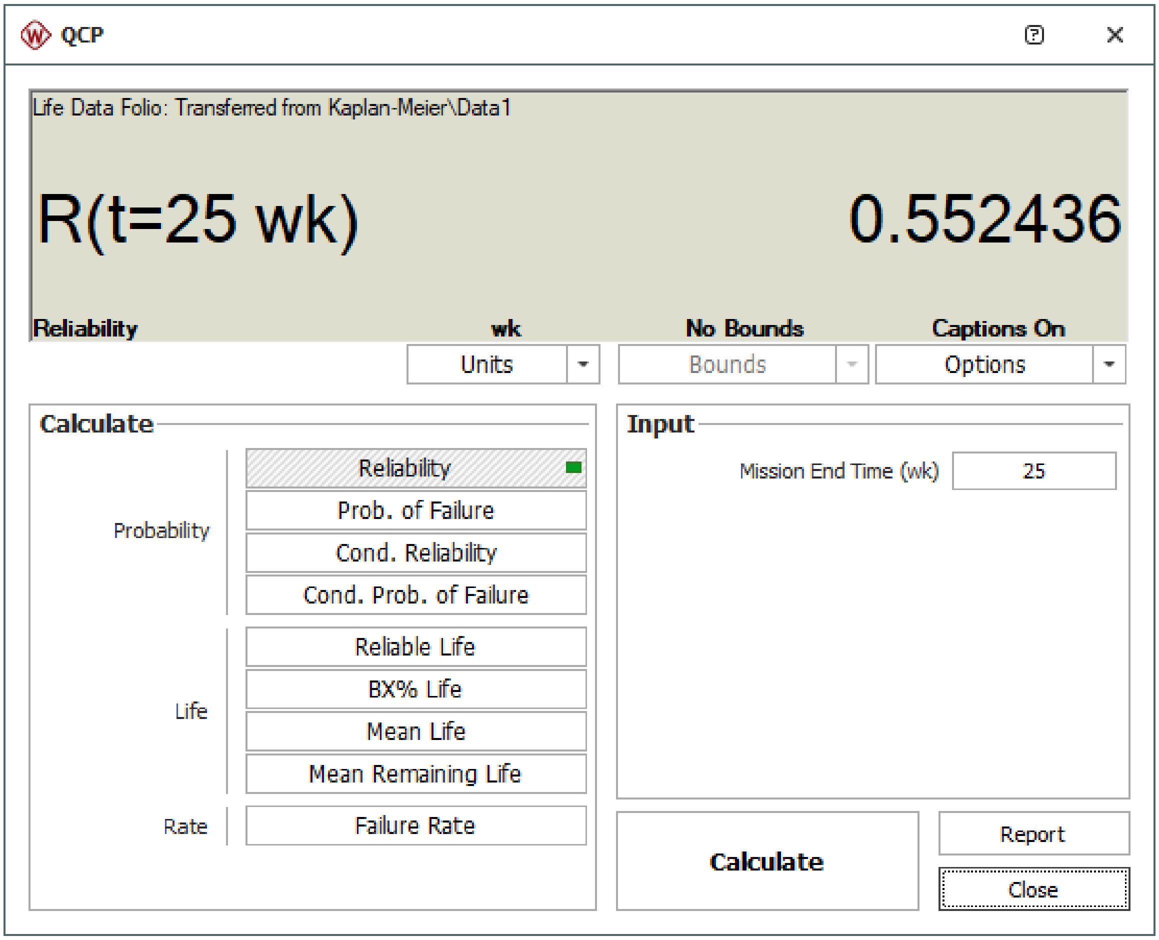
Obtain Results for Other Points in Time

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 life data 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 life data folio and non-parametric LDA folio, then you will obtain the same results from the QCPs in both folios.

1. Click the **QCP** icon on the control panel.

2. 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 Folios

Warranty claims data typically consist of quantities of units sold and quantities returned within a specified warranty period. 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 in order to detect and correct potential product quality problems in the field, as well as plan for warranty fulfillment needs such as repair costs and the number of spares to stock.

The ReliaWiki resource portal has more information on warranty analysis at: http://www.reliawiki.org/index.php/Warranty_Data_Analysis.

Warranty Analysis Folio Setup

To add a warranty analysis folio to a project, choose **Home > Insert > Warranty**, or right-click the **Warranty** folder in the current project explorer and choose **Add Warranty** on the shortcut menu.

In the setup window, select a format to use with the folio. You must 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 can convert the format of an existing warranty analysis folio to a another format. To do so, open the existing folio and choose **Warranty > Tools > Convert Data**, and then choose the format to convert to. (The data in the existing folio must already be analyzed in order for the command to become available.)

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.

Below is 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

Control Panel Settings

This section focuses on the warranty folio control panel, which contains most of the tools you need to perform a warranty analysis. For more information about control panels in general, see Control Panels.

The warranty control panel 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.
- The **Suspensions** page is available only for the usage format. It contains the settings for estimating the amount of usage for fielded units.

CONTROL PANEL MAIN PAGE

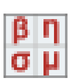
- The **Life Data Model** area contains the options for performing life data analysis. When you click **Calculate**, Weibull++ converts the warranty claims data into failure/suspension data and then performs a life data analysis based on the settings you specified in this area.
  - **Use Subsets** gives 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 the **Results Report** icon in the **Analysis Summary** area.
- The **Suspend After** check box allows you to specify a period beyond which the data are considered incomplete. For example, consider a shipment that has been in the field for 14 months. If the warranty period is 12 months, then the failures entered in the data sheet for months 13 and 14 are taken into account, but all remaining units in that shipment period are suspended at 12 months. 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: a) specify the warranty period to be in days, month or years, or b) consider a warranty period based on usage (e.g., 100K miles or 10 years, whichever comes first).
  - This option is not available for the times-to-failure format.

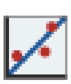
- The **Analysis Summary** area displays the results of the life data analysis. This includes the parameters of the selected distribution, the rho and/or LK value (when applicable), and the number of failures and suspensions in the converted data set (includes failures at $t = 0$, but excludes suspensions at $t = 0$). Click the **Results Report** icon  to open the Results window, which shows the results in a worksheet that you can copy or print.
- The **End of Observation Period** setting is 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 of the control panel, which contains the options for estimating the amount of usage accumulated by suspensions.

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.



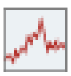
Calculate converts the warranty claims data to failure/suspension times and then estimates the parameters of the chosen life distribution based on the converted data. If you select the **Use Subsets** option on the Main page of the control panel, the parameters of the distribution of each subset ID will be calculated. This tool is also available by choosing **Warranty > Analysis > Calculate**.

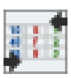

Plot creates a new sheet in the folio that provides a choice of applicable plot types. In warranty analysis folios, this includes plots such as expected failures vs. period, reliability vs.

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**.

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++ life data folio QCP. This tool is also available by choosing **Warranty > Analysis > Quick Calculation Pad**.

 **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**.

 **Folio Setup** is available only for the Nevada chart format. It opens the Nevada Chart Setup window, which allows you to change the time periods of the warranty claims data in the current data sheet.

 **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. 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 Nevada Chart Setup.
- 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.

Warranty Data Formats

Nevada Chart Format

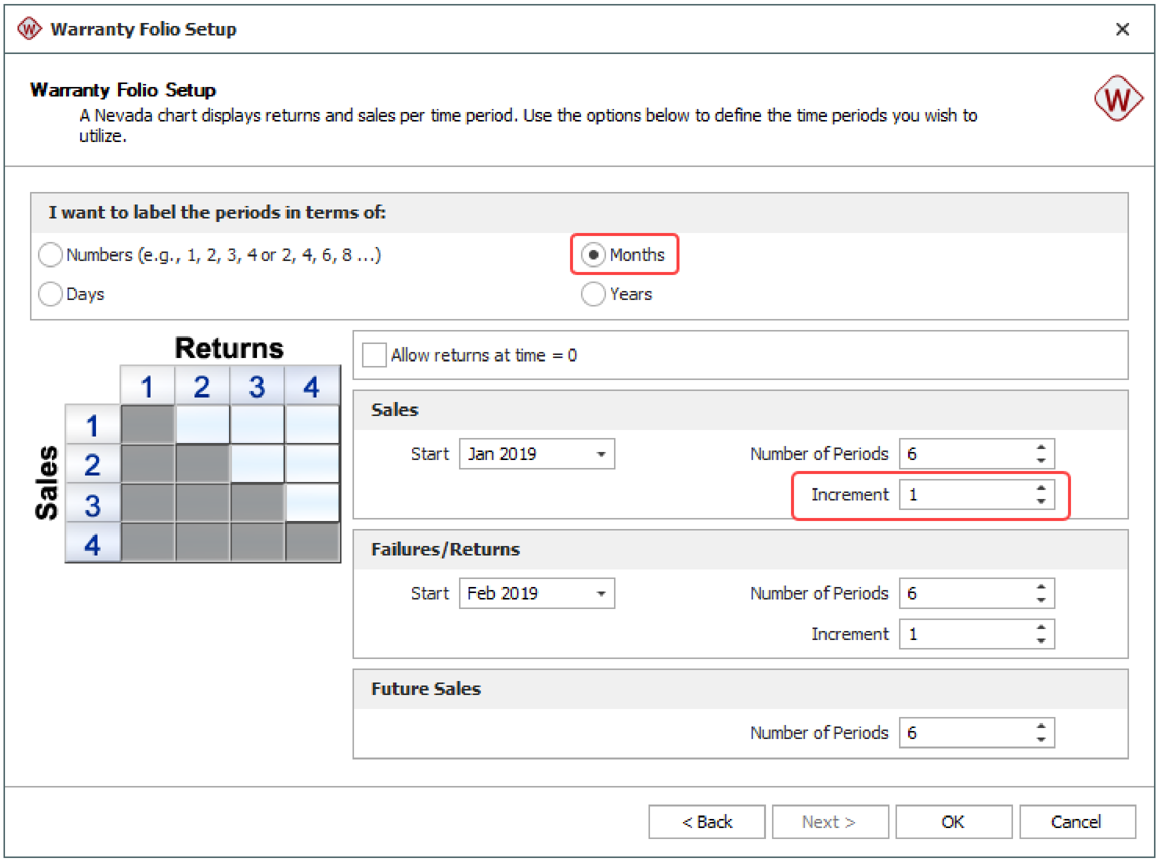
NEVADA CHART FORMAT

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.

Nevada Chart Setup

The setup window of the Nevada chart format allows you to define the time periods of interest. The time unit may be measured in days, months, years, or by using a unit-less number.

- The **Sales** and **Failures/Returns** areas allows you to define the number of sales and return periods. The **Increment** field indicates the length of the time period. In the following picture, 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 **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.
- The **Allow Returns at Time = 0** check box 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.

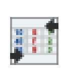


Time Units

In the Nevada chart format, you can change the time periods for an existing Nevada chart data sheet at any time. Note, however, that changing the time period only changes the name of the unit; the data set is not converted to the new unit.

There are two ways to change the time periods:

- If the periods are labeled in terms of days, months or years, choose **Warranty > Tools > Data Sheet Setup** or click 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, 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 units to use. The units that are available in the drop-down list are defined at the database level.

Data Sheets

The Nevada chart folio consists of two data sheets: Sales and Returns.

In the Sales data sheet:

- 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 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 **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.

	Quantity In-Service	Subset ID
Jan 19	100	Batch 1
Feb 19	150	Batch 2
Mar 19	200	Batch 3
Apr 19	250	Future Sales
May 19	250	Future Sales
Jun 19	250	Future Sales

In the Returns data sheet:

- The column headings represent the warranty periods
- The row headings represent the sales periods

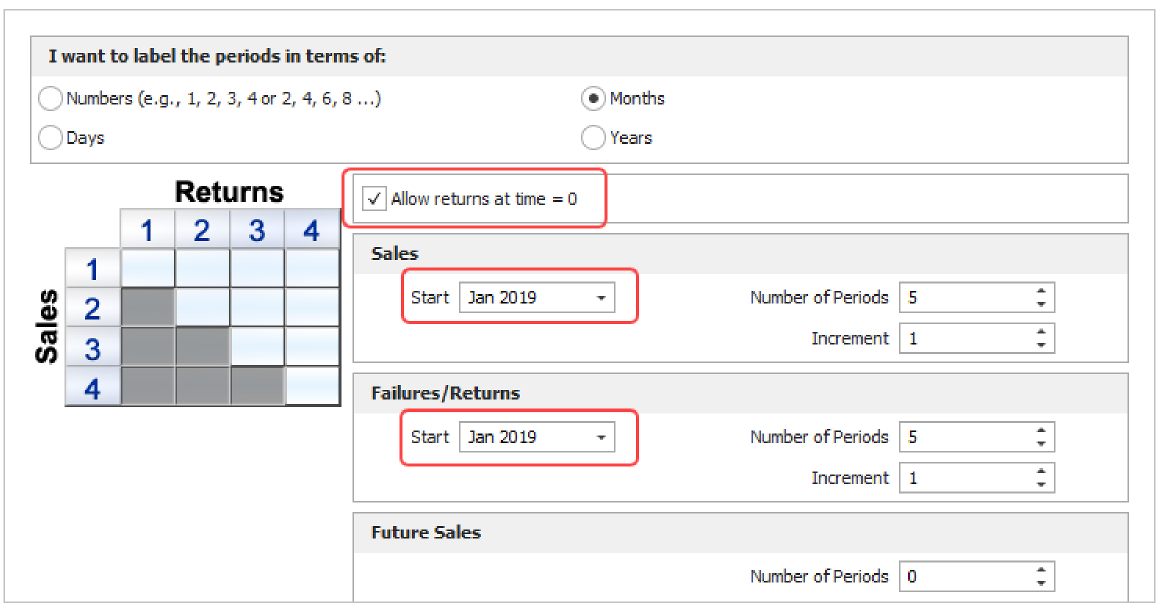
In the example shown here, there were 3 returns in February from the batch that was in-service in January. In March, 5 more units from the batch that was in-service in January were returned and 2 units from the batch that was in-service in February were also returned. The rest of the chart can be read in a similar manner.

	Feb 19	Mar 19	Apr 19
Jan 19	3	5	7
Feb 19		2	4
Mar 19			5

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.

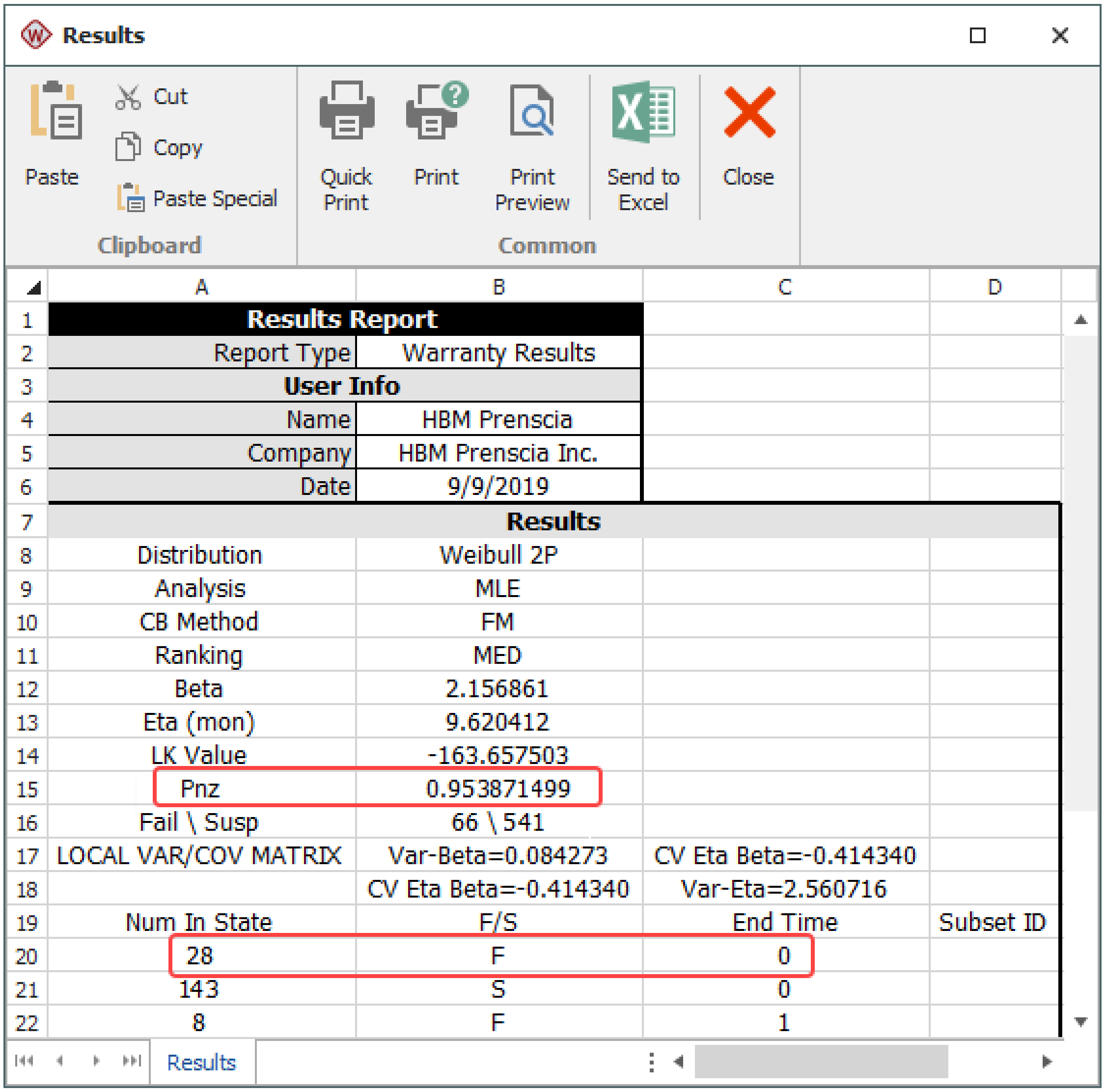
To include zero-time failures in the Nevada chart warranty analysis, select the check box during setup and set the return start date to be the same as the sales start date, as shown in the following example.



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. In this example, 5 units from the January sales were damaged during shipping and 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.

	Jan 19	Feb 19	Mar 19	Apr 19	May 19
Jan 19	5	2	6	5	3
Feb 19		3	1	5	4
Mar 19			5	3	7
Apr 19				8	2
May 19					7

When you calculate the parameters and view the results report, the converted data set will contain a certain number of data points with failures at time = 0, as shown in the next example. **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 **Fail/Susp** field shows the total number of failures and suspensions in the data set (including failures at $t = 0$ but excluding suspensions at $t = 0$).



	A	B	C	D
1	Results Report			
2	Report Type	Warranty Results		
3	User Info			
4	Name	HBM Prenscia		
5	Company	HBM Prenscia Inc.		
6	Date	9/9/2019		
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	0.953871499		
16	Fail \ Susp	66 \ 541		
17	LOCAL VAR/COV MATRIX	Var-Beta=0.084273	CV Eta Beta=-0.414340	
18		CV Eta Beta=-0.414340	Var-Eta=2.560716	
19	Num In State	F/S	End Time	Subset ID
20	28	F	0	
21	143	S	0	
22	8	F	1	

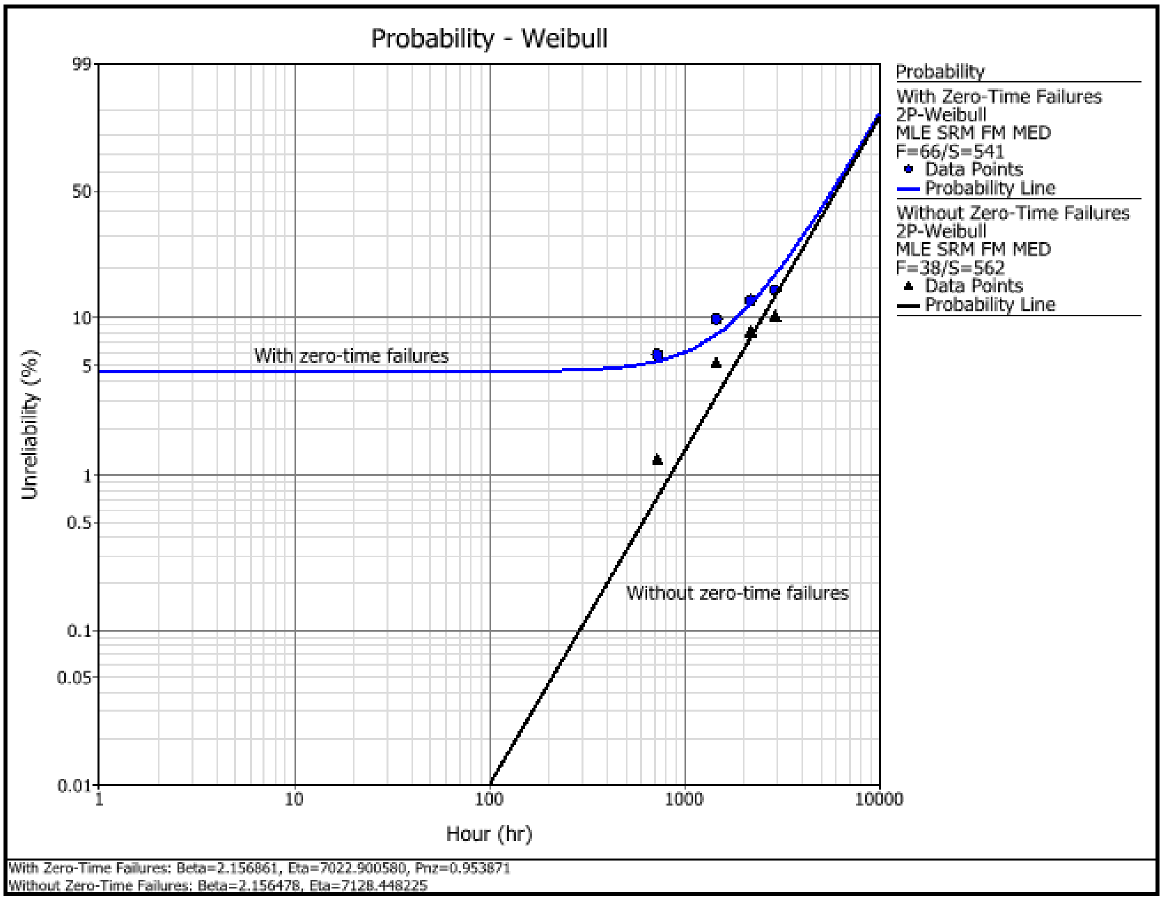
The distribution parameters are calculated without the zero-time failures, but any subsequent reliability calculations for the full data set will be multiplied by the Pnz value, such that:

$$R'(t) = Pnz \times R(t)$$

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.

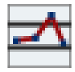


STATISTICAL PROCESS CONTROL

The Nevada chart format includes a statistical process control (SPC) method for analyzing warranty returns data. The SPC method evaluates the discrepancy between the actual number of failures and the number of failures predicted by the distribution parameters. It 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.

To use SPC, click the **Statistical Process Control** page icon on the control panel.



1. Select the **Calculate chi-squared values** check box, and then enter the desired control values into the **Critical** and **Caution** fields in decimal form.

   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. The values must be greater than zero and less than one. By default, the critical value is 1% and the caution value is 10%.

2. Select the **Color-code Returns sheet** check box to highlight the results. You can select to evaluate the results **By sales period** (results are color-coded by rows) or **By returns period** (results are color-coded by columns).

   - 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.

3. If you have manually changed the colors of any cells, click the **Restore Guide Colors** button to return the cells to their original colors.

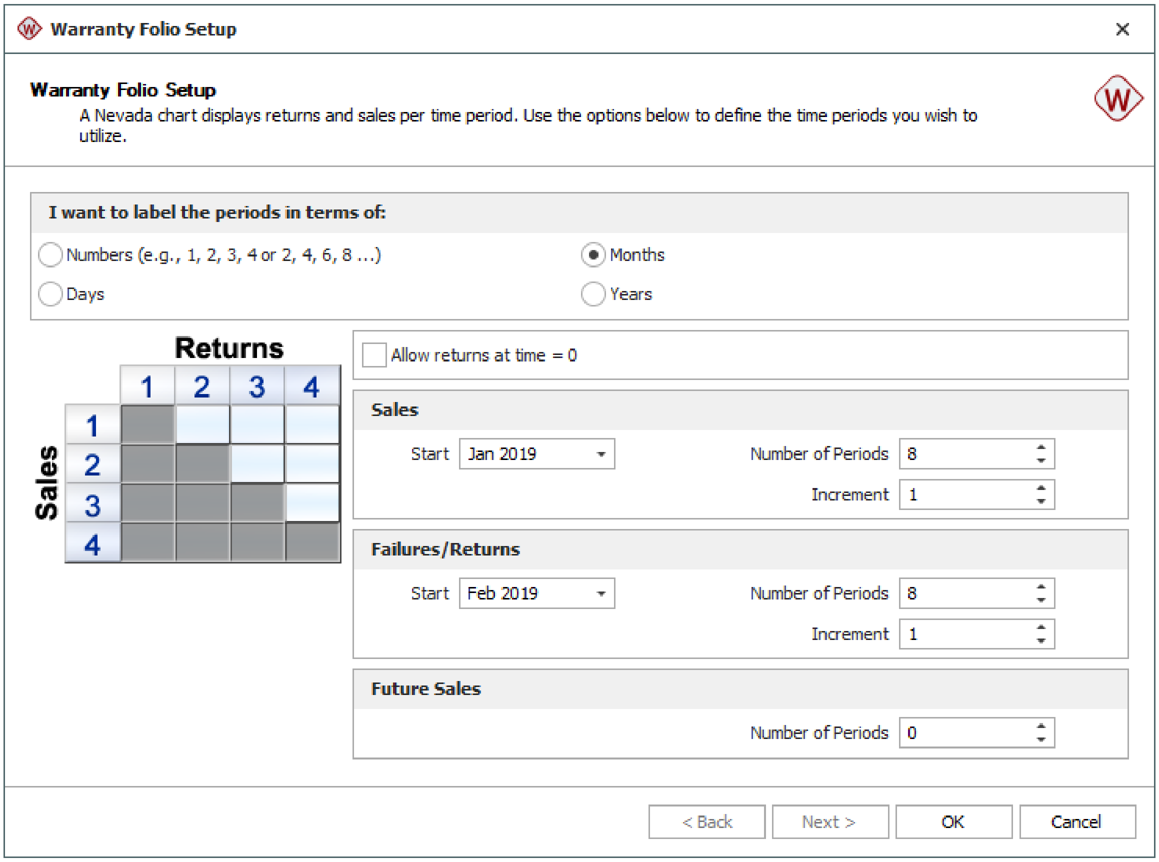
Tip: If you wish to perform a manual evaluation of the table of chi-squared values, you can use Weibull++'s Quick Statistical Reference (QSR) tool to look up chi-squared values for different control values and degrees of freedom. You can access the QSR by choosing **Home > ReliaSoft > Quick Tools > Quick Statistical Reference**.

EXAMPLE OF A NEVADA CHART WARRANTY ANALYSIS

The data set used in this example is available in the example database installed with the Weibull++ software (called "Weibull21_Examples.rsgz21"). 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 "Warranty - Nevada Chart Format with Statistical Process Control."

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.



	Quantity In-Service	Subset ID
Jan 19	1150	Supplier 1
Feb 19	1100	Supplier 1
Mar 19	1200	Supplier 2
Apr 19	1155	Supplier 1
May 19	1255	Supplier 1
Jun 19	1150	Supplier 1
Jul 19	1105	Supplier 2
Aug 19	1110	Supplier 1

	Feb 19	Mar 19	Apr 19	May 19	Jun 19	Jul 19	Aug 19	Sep 19
Jan 19	2	4	5	7	12	13	16	17
Feb 19		3	4	5	3	8	11	14
Mar 19			2	3	5	7	23	13
Apr 19				2	3	4	6	7
May 19					2	3	3	4
Jun 19						2	3	3
Jul 19							2	12
Aug 19								2

Analyze the Data Set Using the Statistical Process Control (SPC) Method

1. On the Main page of the control panel, choose the **2P-Weibull** distribution and the **MLE** parameter estimation method.

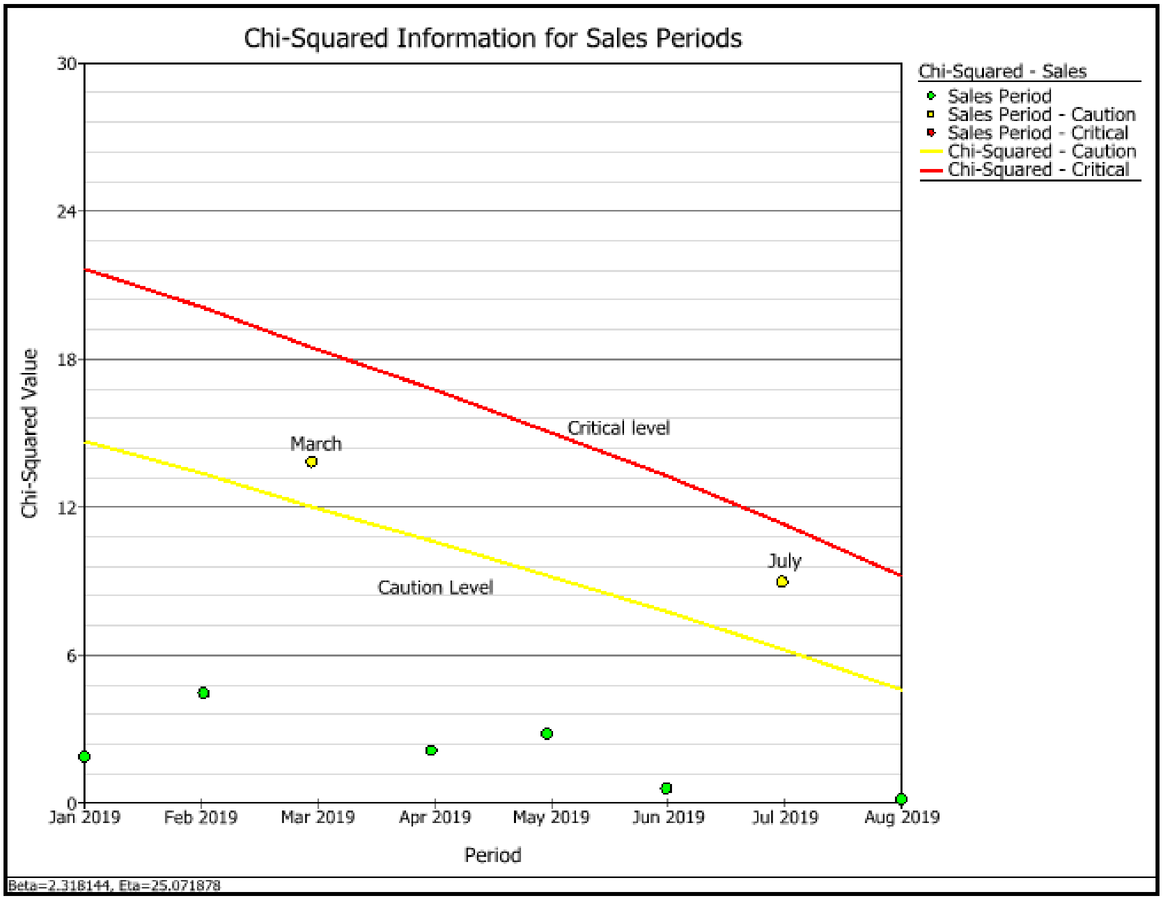
2. On the Statistical Process Control page of the control panel, select the **Calculate chi-squared values** check box. Make sure that the critical value is **0.01** and that the caution value is **0.1**.

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**. The plot shows that 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.

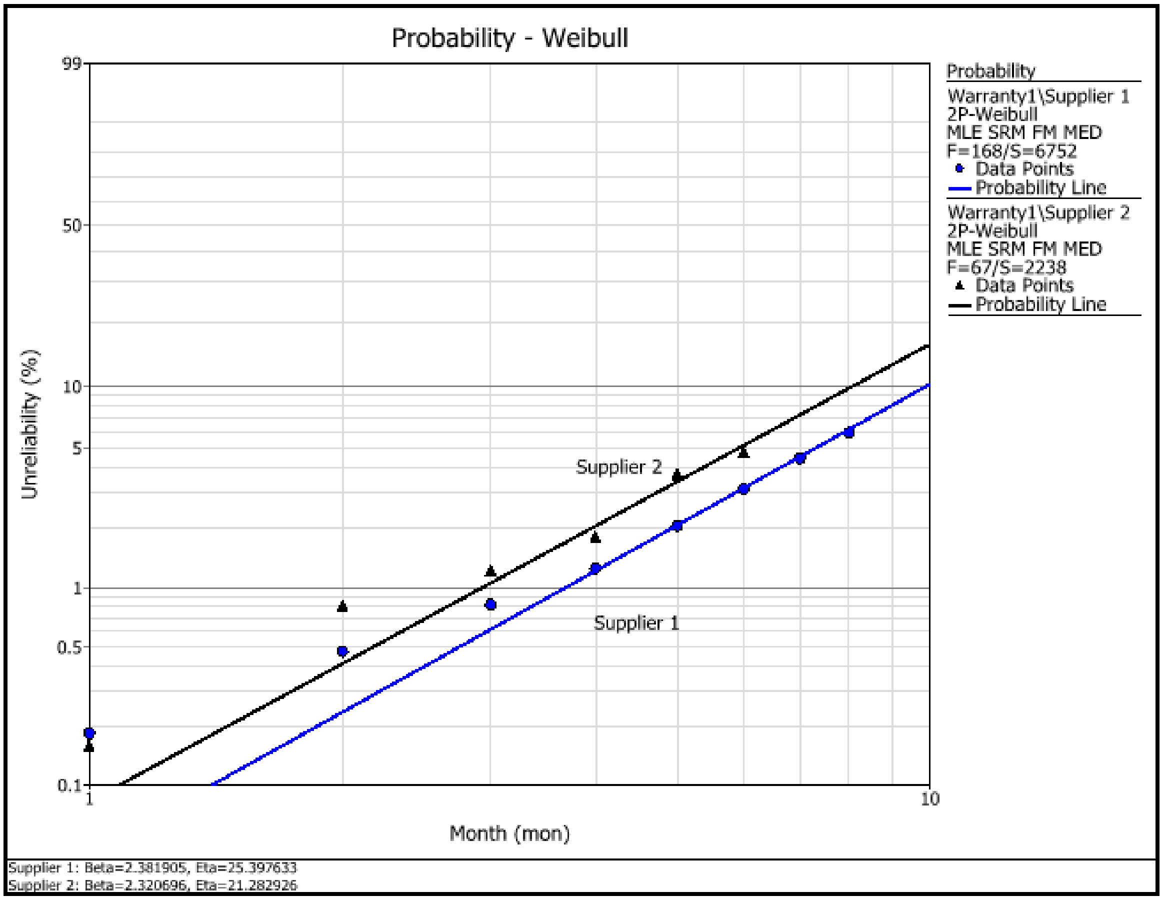


Analyze the Data Set Based on Suppliers

6. 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 the2P-Weibulldistribution and theMLEparameter 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. The plot shows that the material from Supplier 2 tends to have a higher probability of failure compared to the material from Supplier 1.



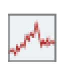
9. To determine the statistical difference between the two data sets, create a contour plot. Click the **Plot Type** drop-down list on the control panel and choose **Contour Plot**. In the Contour Setup window, select to plot the contours at the 90% confidence level and click **OK**.

   The resulting plot shows that 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.

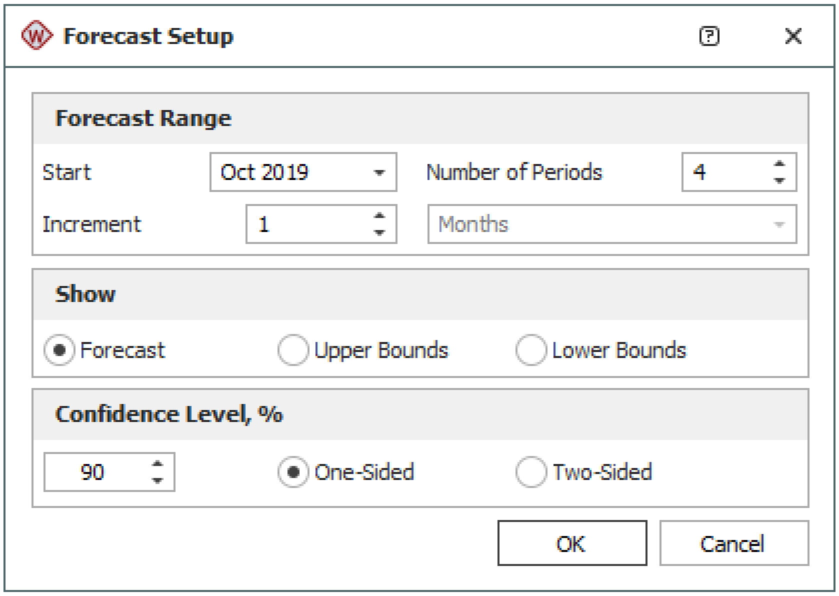
Generate a Warranty Forecast

The next step is to generate a warranty forecast to obtain the expected number of product returns within a specified warranty period.

10. 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**.

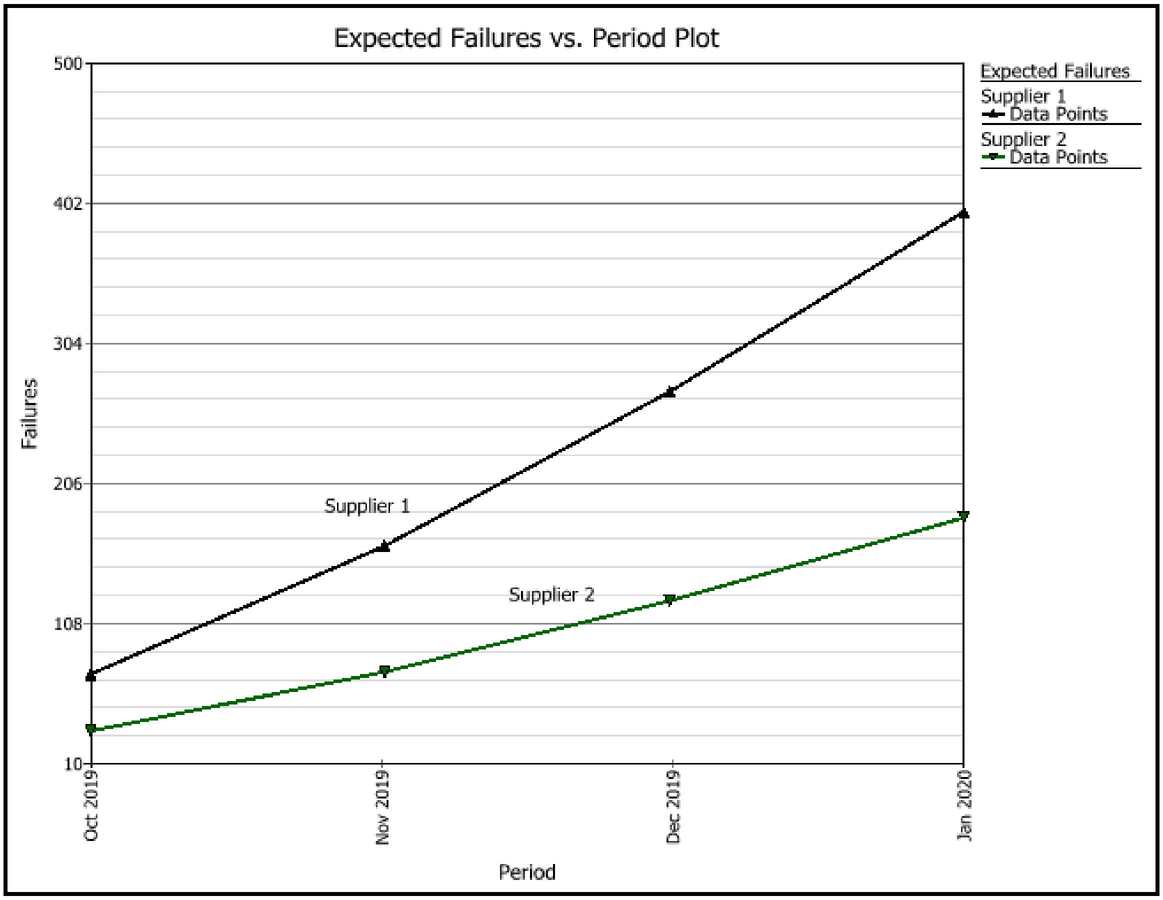


The following figure shows the Forecast sheet with the warranty period starting in October 2019 and ending in January 2020. The results show that the highest number of product returns is expected to come from the batch that was in-service in March.

	Subset ID	Oct 19	Nov 19	Dec 19	Jan 20
Jan 19	Supplier 1	22	25	28	31
Feb 19	Supplier 1	18	21	24	27
Mar 19	Supplier 2	26	30	35	39
Apr 19	Supplier 1	13	16	19	22
May 19	Supplier 1	11	14	17	21
Jun 19	Supplier 1	7	10	13	16
Jul 19	Supplier 2	7	11	15	19
Aug 19	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.



Times-to-Failure Format

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.

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 units to use. The units that are available in the drop-down list are defined at the database level. When you select new units, the appropriate columns in the data sheet will be automatically configured for the units you selected, and the new unit will be displayed next to the results in the control panel.

Data Sheets

The times-to-failure format consists of two data sheets: Data and Future Sales.

In the data sheet:

- Data entry is similar to entering data in a Weibull++ life data folio.
- 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.

	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

In the Future Sales sheet:

- The **Quantity In-Service** column is for recording the projected number of units to be sold. This information is used in the forecast analysis.
- The **Time** column is for logging the anticipated manufacturing/sales/shipment period.

	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			

EXAMPLE OF A TIMES-TO-FAILURE WARRANTY ANALYSIS

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 for 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.

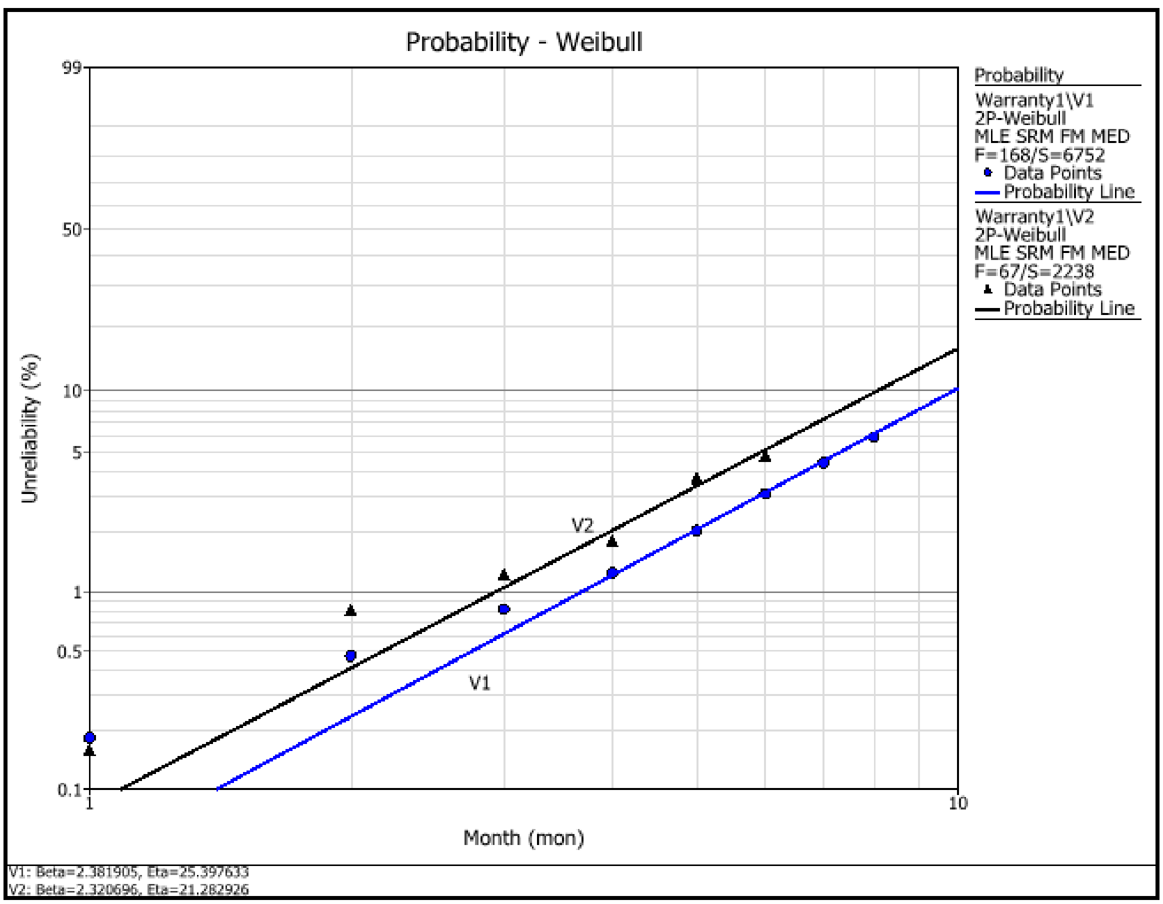
	Number in State	State F or S	State End Time (mon)	Subset ID
1	13	F	1	V1
2	1108	S	1	V1
3	17	F	2	V1
4	20	F	3	V1
5	1142	S	3	V1
6	20	F	4	V1
7	1243	S	4	V1
8	27	F	5	V1
9	1133	S	5	V1
10	24	F	6	V1
11	30	F	7	V1
12	1052	S	7	V1
13	17	F	8	V1
14	1074	S	8	V1
15	4	F	1	V2
16	15	F	2	V2
17	1091	S	2	V2
18	5	F	3	V2
19	7	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

Analyze the Data for Each Product Version

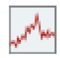
1. On the Main page of the control panel, select the **Use Subsets** option. Then for each subset ID, choose the **2P-Weibull** distribution and the **MLE** analysis method. Click **Calculate**.
   - The parameters for V1 are estimated to be: beta = 2.3819 and eta = 25.3976.
   - The parameters for V2 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. It shows that V2 has a higher probability of failure compared to V1.



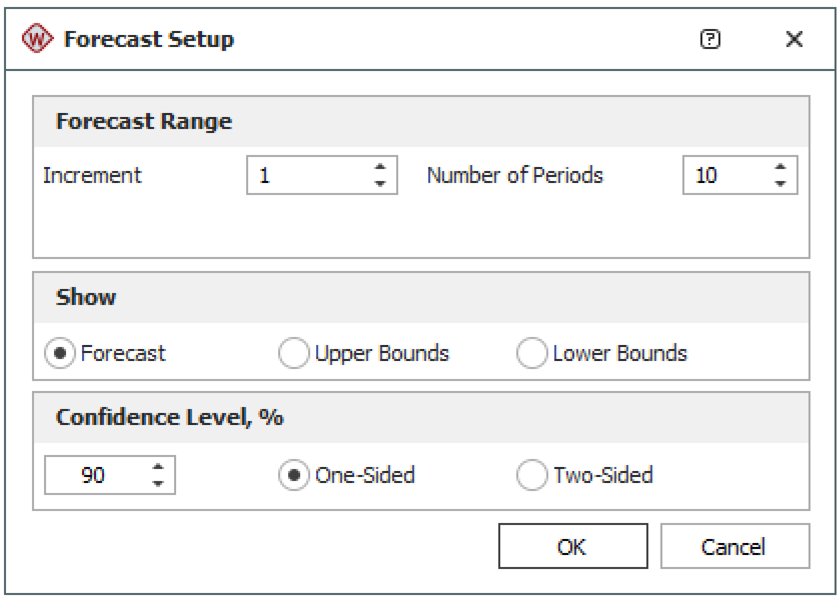
Generate a Warranty Forecast

The next step is to generate a warranty forecast to predict the expected number of product returns.

3. Return to the data sheet and 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**.



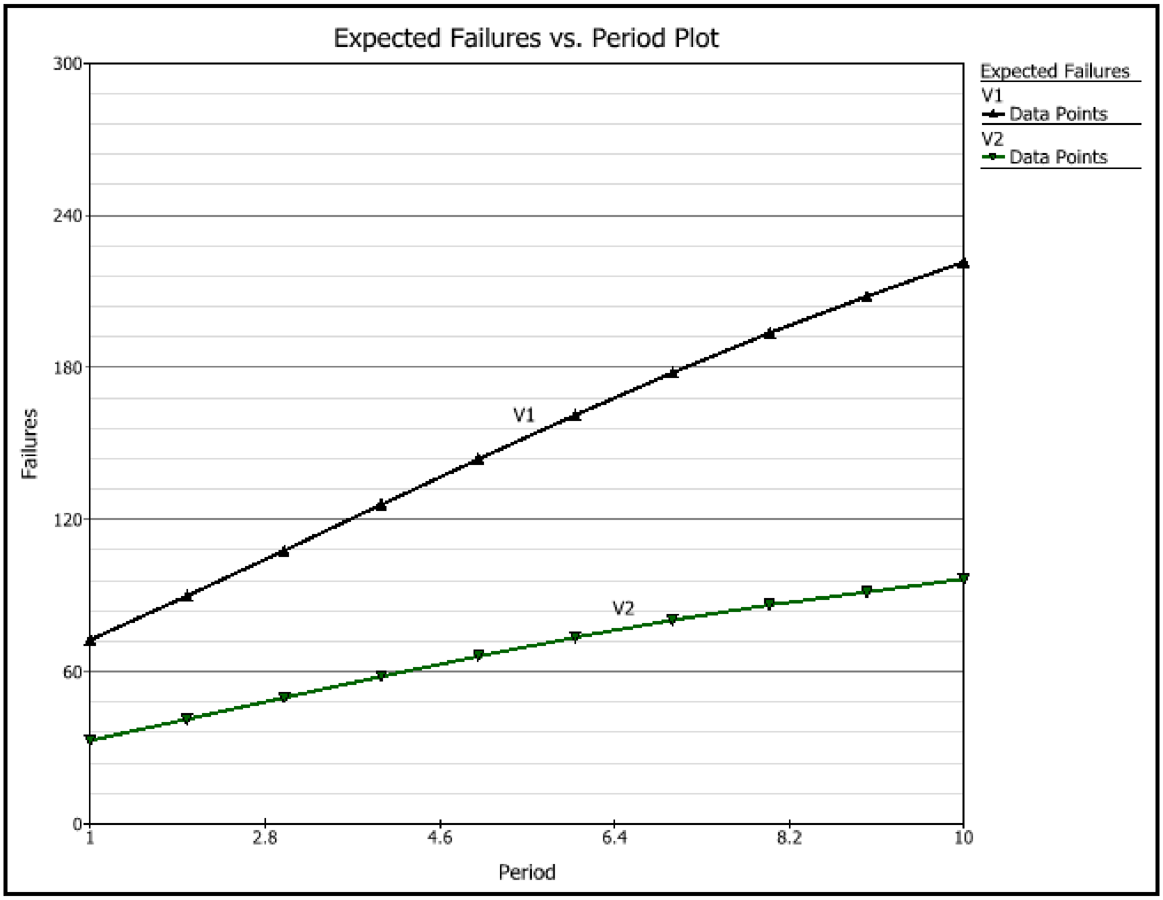
5. On the control panel of the Forecast sheet, select the **Show Subset ID** check box. The following figure shows the results (showing only the first 8 columns).

	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**

Note: Complete data set is not shown.

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.



Dates of Failure Format

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.

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.

Data Sheets

The dates of failure format consists of three data sheets: Sales, Returns and 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.

In the Sales sheet:

- 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.

	Quantity In-Service	Date In-Service	Subset ID
1	400	1/1/2019	Model A
2	500	1/31/2019	Model A
3	500	5/1/2019	Model A
4	600	5/31/2019	Model A
5	550	6/30/2019	Model A
6	600	7/30/2019	Model A

In the Returns sheet:

- The **Quantity Returned** column indicates the number of units that were returned.
- The **Date of Return** column indicates the date the units were returned.
- The **Date In-Service** column indicates how long the units were in service before they were 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	1/31/2019	1/1/2019	Model A
2	11	4/1/2019	1/31/2019	Model A
3	7	7/22/2019	5/1/2019	Model A
4	8	8/27/2019	5/31/2019	Model A
5	12	12/27/2019	5/31/2019	Model A
6	3	12/29/2019	6/30/2019	Model A

In the Future Sales sheet:

- The **Quantity In-Service** column is for recording the projected number of units to be sold. This information is used in the forecast analysis.
- The **Date In-Service** column is for logging the anticipated manufacturing/sales/shipment period.

	Quantity In-Service	Date In-Service	Subset ID
1	500	8/1/2019	Model A
2	500	9/1/2019	Model A
3	500	10/24/2019	Model A
4			
5			
6			

End of Observation Period

In the dates-of-failure format, the **End of Observation Period** setting on the Main page of the control panel is used to convert the sales/return data to failures/suspensions. The 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.

EXAMPLE OF A DATES OF FAILURE WARRANTY ANALYSIS

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	1/1/2018	Design A
2	350	3/2/2018	Design A
3	450	4/1/2018	Design A
4	300	6/30/2018	Design A
5	200	8/29/2018	Design A
6	350	10/28/2018	Design A
7	400	1/1/2018	Design B
8	500	1/31/2018	Design B
9	500	5/1/2018	Design B
10	600	5/31/2018	Design B
11	550	6/30/2018	Design B
12	600	7/30/2018	Design B
13	800	9/28/2018	Design B

The following data sheet shows the returns data.

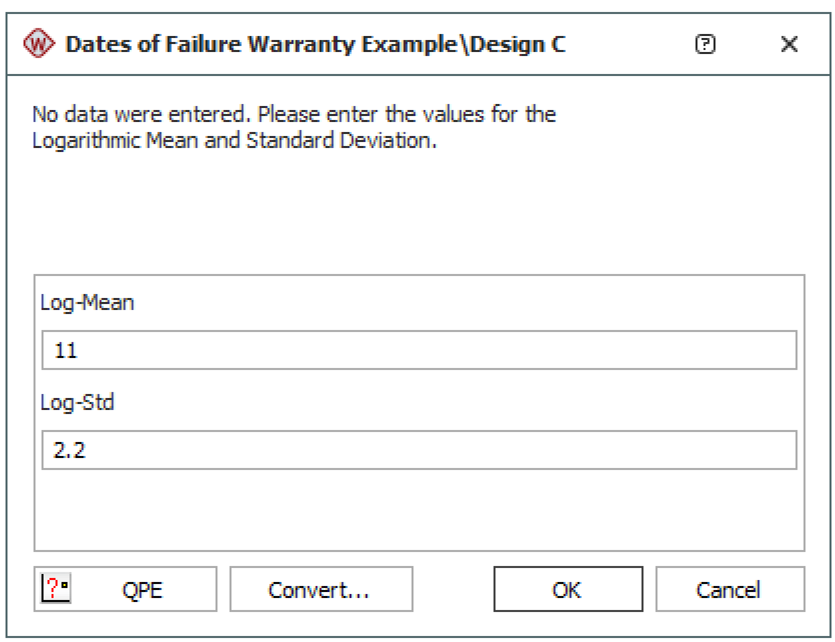
	Quantity Returned	Date of Return	Date In-Service	Subset ID
1	15	1/18/2018	1/1/2018	Design A
2	23	1/26/2018	1/1/2018	Design A
3	16	3/17/2018	1/1/2018	Design A
4	18	5/31/2018	3/2/2018	Design A
5	19	5/31/2018	3/2/2018	Design A
6	20	6/20/2018	3/2/2018	Design A
7	21	7/30/2018	4/1/2018	Design A
8	18	12/27/2018	6/30/2018	Design A
9	19	1/11/2019	8/29/2018	Design A
10	11	1/27/2019	10/28/2018	Design A
11	12	1/31/2018	1/1/2018	Design B
12	11	4/1/2018	1/31/2018	Design B
13	7	7/22/2018	5/1/2018	Design B
14	8	8/27/2018	5/31/2018	Design B
15	12	12/27/2018	5/31/2018	Design B
16	13	1/26/2019	6/30/2018	Design B
17	12	1/26/2019	7/30/2018	Design B
18	14	1/11/2019	9/28/2018	Design B

The future shipments are expected to be as follows.

	Quantity In-Service	Date In-Service	Subset ID
1	400	2/1/2019	Design B
2	500	4/1/2019	Design B
3	550	6/1/2019	Design B
4	2200	8/1/2019	Design C
5	2200	9/1/2019	Design C
6	2200	10/1/2019	Design C

Analyze the Data for Each Product Version

1. The information in the Returns sheet is current as of January 31, 2019. Therefore, on the Main page of the control panel, set the **End of Observation Period** date to **1/31/2019** 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. 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.



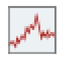
Note: Because the unit of time in the dates of failure format is in days, the parameters that are assigned to a data set must also be in terms of days.

4. Click **OK**. The results show that for Design A, the parameters are: log-mean = 8.4394 and log-std = 2.2311; for Design B, the parameters are: log-mean = 10.0772 and log-std = 2.2973.

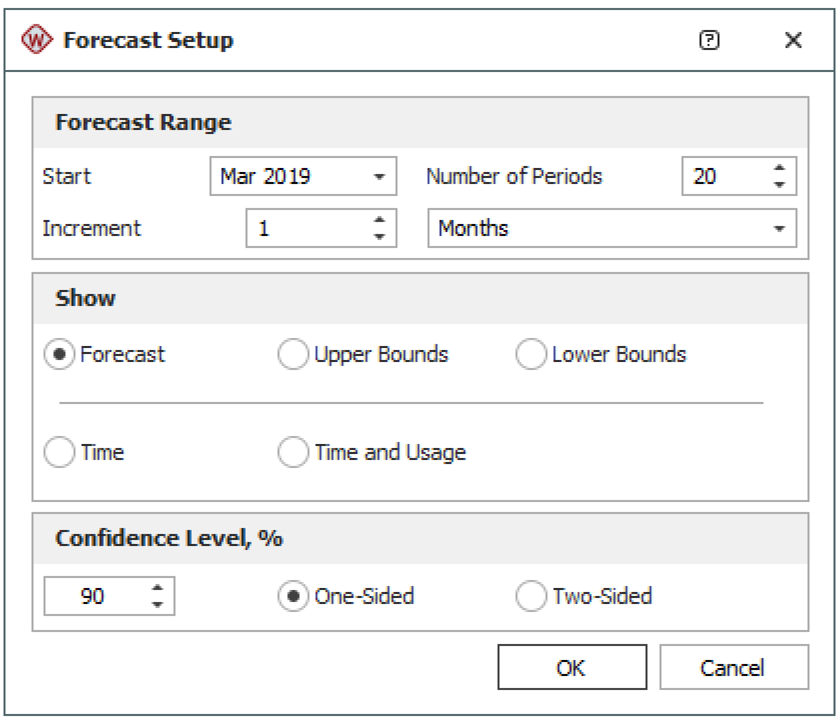
Generate a Warranty Forecast

The next step is to generate a warranty forecast to predict the expected number of product returns.

5. Choose **Warranty > Tools > Forecast** or click the icon on the control panel.



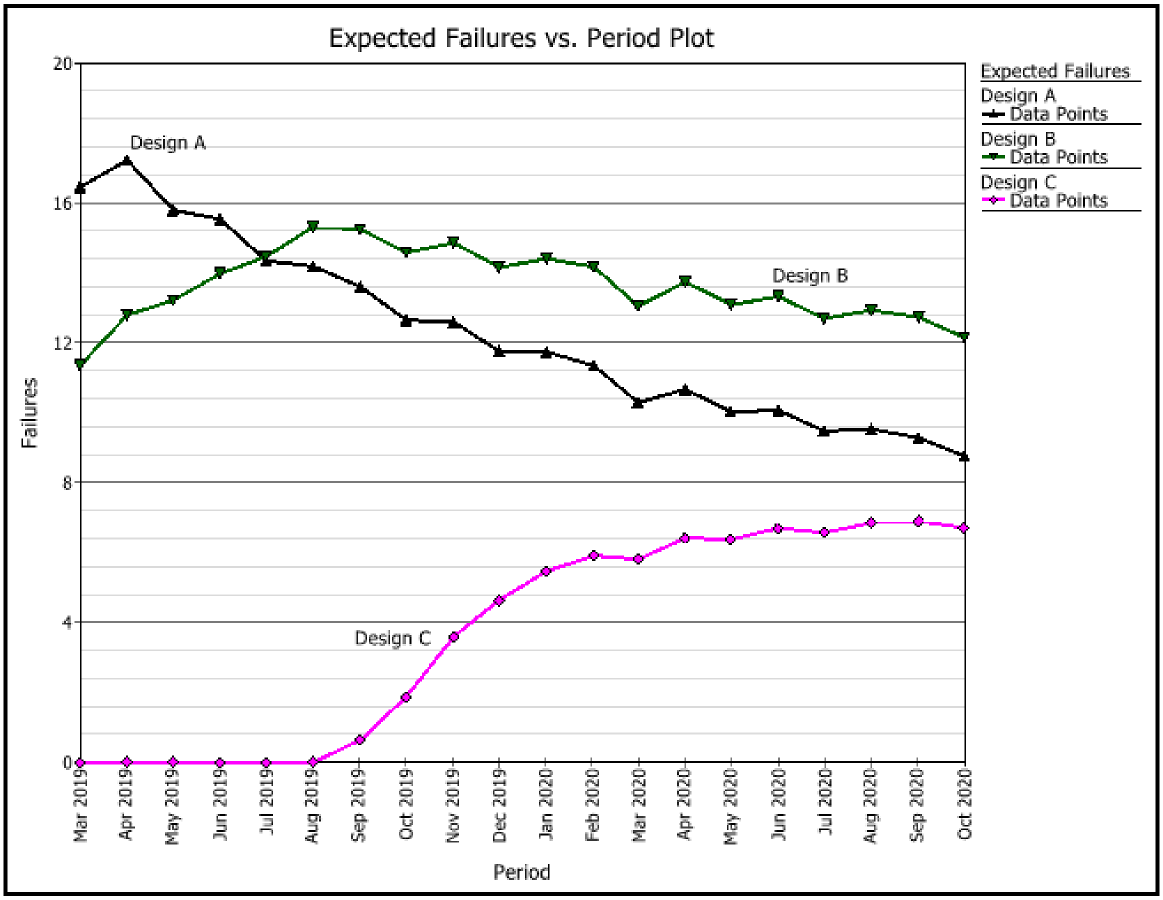
6. In the Forecast Setup window, enter a Start date of **3/1/2019** and obtain forecasts for **20** periods, as shown next. Set the increment to once per month, and then click **OK**.



7. 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 1/1/2018 and that 1 of those sold units is expected to fail between February 1, 2018 (the day after the End of Observation Period date) and March 1, 2018.

	Quantity In-Service	Date In-Service	Subset ID	Mar 01, 2019	Apr 01, 2019	May 01, 2019	Jun 01, 2019
1	200	1/1/2018	Design A	1	1	1	1
2	350	3/2/2018	Design A	2	3	2	2
3	450	4/1/2018	Design A	4	4	4	4
4	300	6/30/2018	Design A	3	3	3	3
5	200	8/29/2018	Design A	2	2	2	2
6	350	10/28/2018	Design A	4	4	4	4
7	400	1/1/2018	Design B	1	1	1	1
8	500	1/31/2018	Design B	1	1	1	1
9	500	5/1/2018	Design B	1	1	1	1
10	600	5/31/2018	Design B	2	2	2	2
11	550	6/30/2018	Design B	2	2	2	2
12	600	7/30/2018	Design B	2	2	2	2
13	800	9/28/2018	Design B	2	2	2	2
14	400	2/1/2019	Design B	1	1	1	1
15	500	4/1/2019	Design B			1	1
16	550	6/1/2019	Design B				
17	2200	8/1/2019	Design C				
18	2200	9/1/2019	Design C				
19	2200	10/1/2019	Design C				
Total				**28**	**30**	**29**	**30**

8. 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.



Usage Format

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.

What's Changed? Starting in Version 2019, when forecasting warranty returns, you can predict the number of failures based on the time and usage as well as the time. For example, for a vehicle with a 3-year/30,000 mile warranty, you can forecast the total number of warranty returns based on both values.

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. 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 units you selected, and the new unit will be displayed next to the results in the control panel.

Data Sheets

The usage format consists of three data sheets: Sales, Returns and 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.

In the Sales sheet:

- 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.

	Quantity In-Service	Date In-Service	Subset ID
1	100	1/1/2019	Washing Machine A
2	150	2/1/2019	Washing Machine A
3	200	3/1/2019	Washing Machine A
4	250	4/1/2019	Washing Machine A
5	250	5/1/2019	Washing Machine A
6	250	6/1/2019	Washing Machine A

In the Returns sheet:

- The **Quantity Returned** column is for entering the number of units returned.
- The **Usage at Return Date** column indicates the amount usage the units accumulated before they were returned. In the following example, the usage unit is in terms of cycles.
- The **Date-In-Service** column indicates how long the units were in service before they were 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	1/1/2019	Washing Machine A
2	6	1842	1/1/2019	Washing Machine A
3	5	1623	2/1/2019	Washing Machine A
4	17	2010	3/1/2019	Washing Machine A
5	12	1324	4/1/2019	Washing Machine A
6	3	984	5/1/2019	Washing Machine A

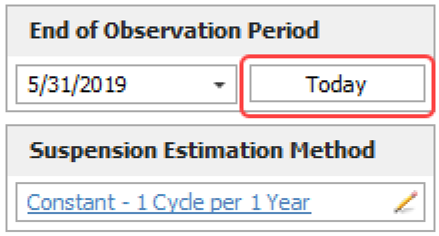
In the Future Sales sheet:

- The **Quantity In-Service** column is for recording the projected number of units to be sold. This information is used in the forecast analysis.
- The **Date In-Service** column is for logging the anticipated manufacturing/sales/shipment period.

	Quantity In-Service	Date In-Service	Subset ID
1	200	8/1/2019	Washing Machine A
2	200	9/1/2019	Washing Machine A
3	200	10/1/2019	Washing Machine A
4			
5			
6			

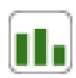
Other Analysis Settings

- The **End of Observation Period** setting on the Main page of the control panel is used to convert the sales/return data to failures/suspensions. The 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. Click **Today** if you wish to set the end date to the current date, as shown next.



- The **Suspension Estimation Method** setting on the Main page of the control panel allows you to specify how Weibull++ will estimate the amount of usage for units still in the field (suspensions). See Estimating Usage.

  To view or change the settings, click the  icon or click the **Suspensions** page icon at the bottom of the control panel.

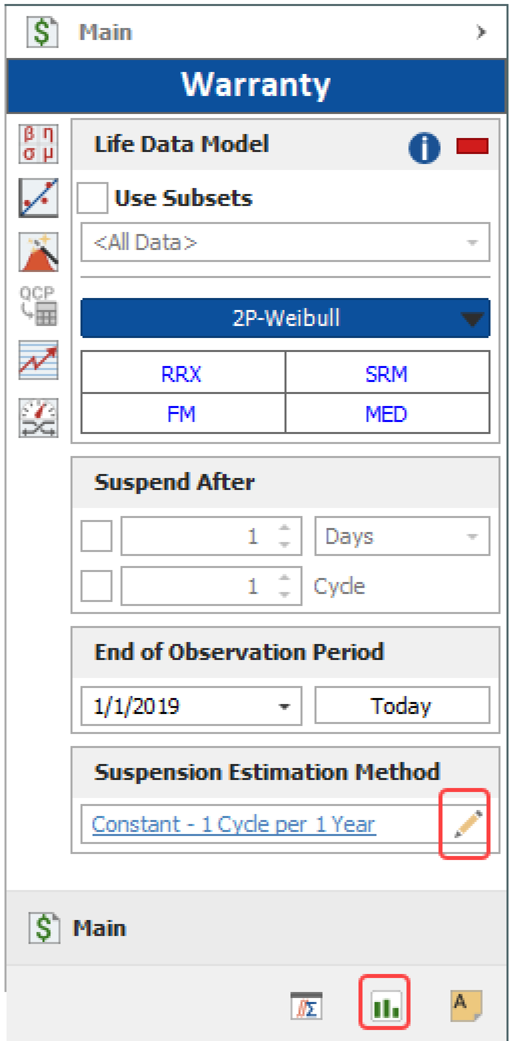


ESTIMATING USAGE

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:

- 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.).
- 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.

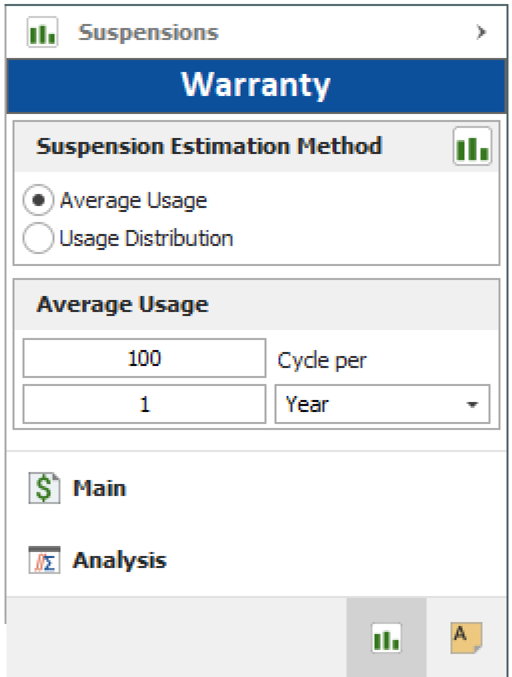
To select a method, click the link in the **Suspension Estimation Method** area of the Main page or click the **Suspensions** page icon at the bottom of the control panel, as shown next.



Both methods are described next.

To perform the 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 unit conversion factors. 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.

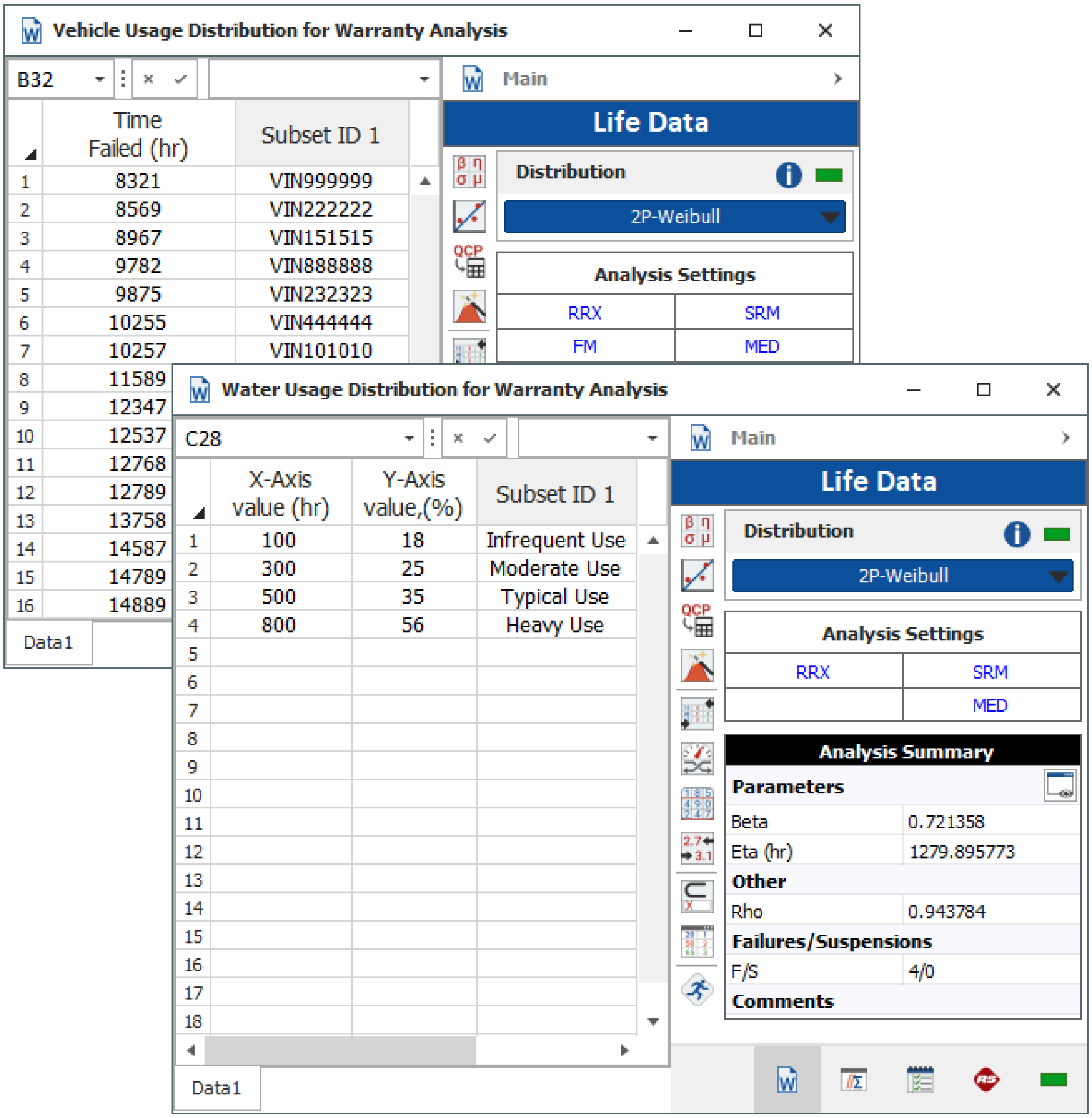
If you enter July 1st for the End of Observation Period date and 1000 miles/per year for the Average Usage, the software will calculate the suspensions from the January 1st sales group as follows (where the daily rate has been rounded to 4 decimal places for the sake of simplicity, and therefore the 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.

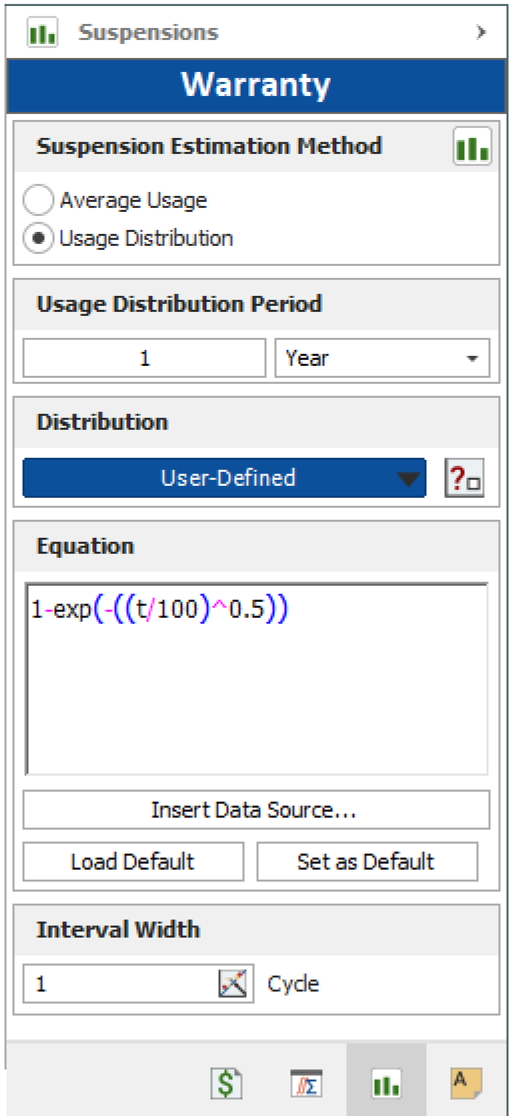
To perform the estimates based on a usage distribution

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++ life data 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++ life data 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 the estimated values of the distribution parameters are not available, you can use the Quick Parameter Estimator (QPE) to solve for them.
- 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 life data 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 life data 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.

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.

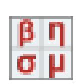
Estimating the Interval Width

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 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 pointer 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.

Figure 1 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).

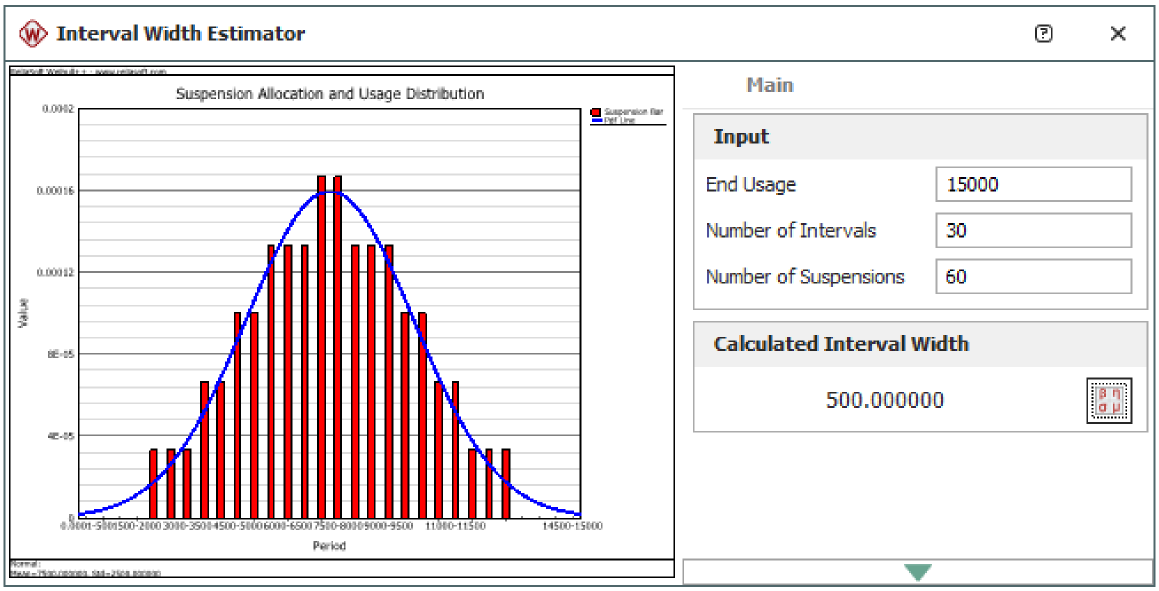


Figure 1: Interval width = 500.

Figure 2 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.

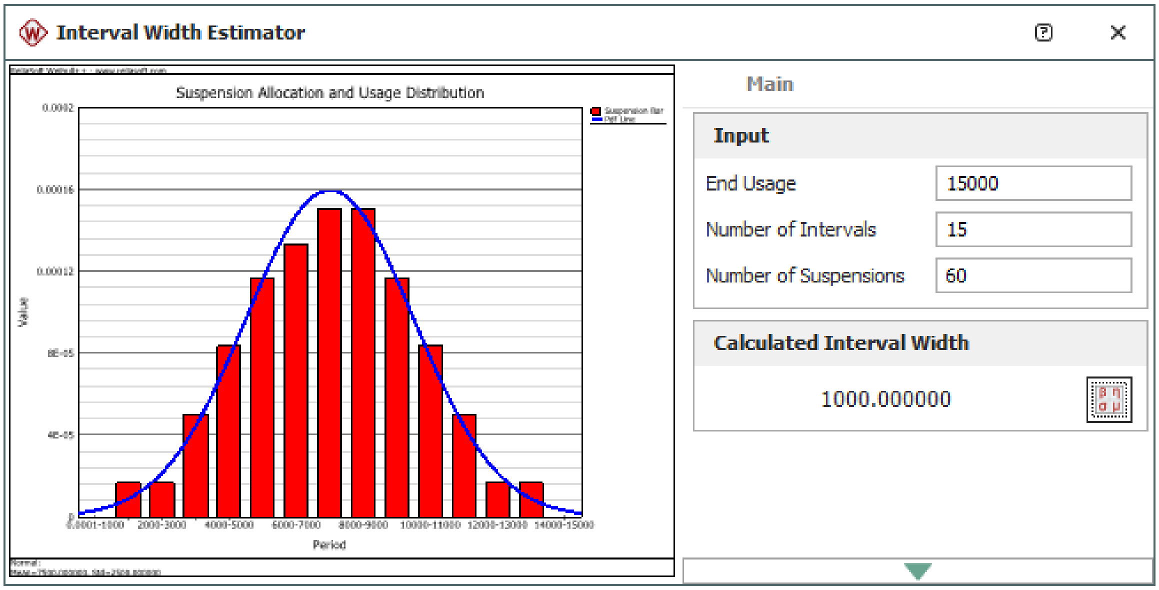


Figure 2: Interval width = 1,000.

Figure 3 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).

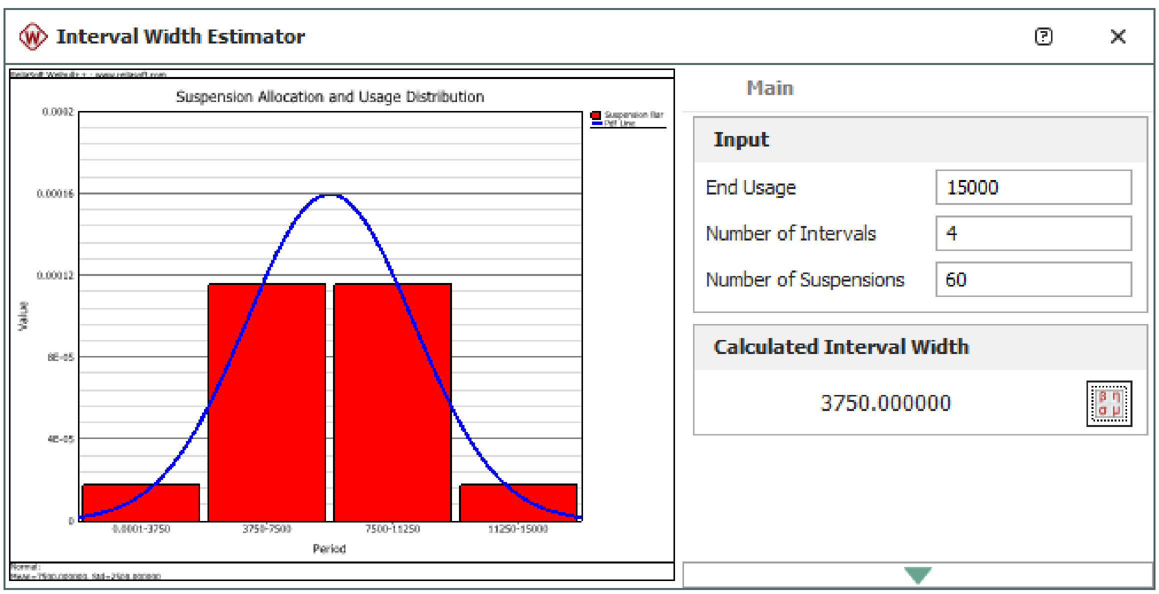


Figure 3: Interval width = 3,750.

EXAMPLE OF A USAGE FORMAT WARRANTY ANALYSIS

The data set used in this example is available in the example database installed with the Weibull++ software (called "Weibull21_Examples.rsgz21"). 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 "Warranty - Usage Format."

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, 2018.

Sales Month	Units Sold	Quantity Returned	Usage at Time of Return (miles)
12/1/2017	9	1	9072
1/1/2018	13	1	9743
2/1/2018	15	1	6857
3/1/2018	20	1	7651
4/1/2018	15	0	-
5/1/2018	25	1	5083
		1	5990
		1	7432
		1	8739
6/1/2018	19	1	3158
7/1/2018	16	1	1136

8/1/2018	20	1	4646
9/1/2018	19	1	3965
10/1/2018	25	1	3117
11/1/2018	30	1	3250

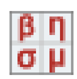
The following pictures show the sales and return data in the usage warranty analysis folio.

	Quantity In-Service	Date In-Service	Subset ID
1	9	Dec-17	
2	13	Jan-18	
3	15	Feb-18	
4	20	Mar-18	
5	15	Apr-18	
6	25	May-18	
7	19	Jun-18	
8	16	Jul-18	
9	20	Aug-18	
10	19	Sep-18	
11	25	Oct-18	
12	30	Nov-18	

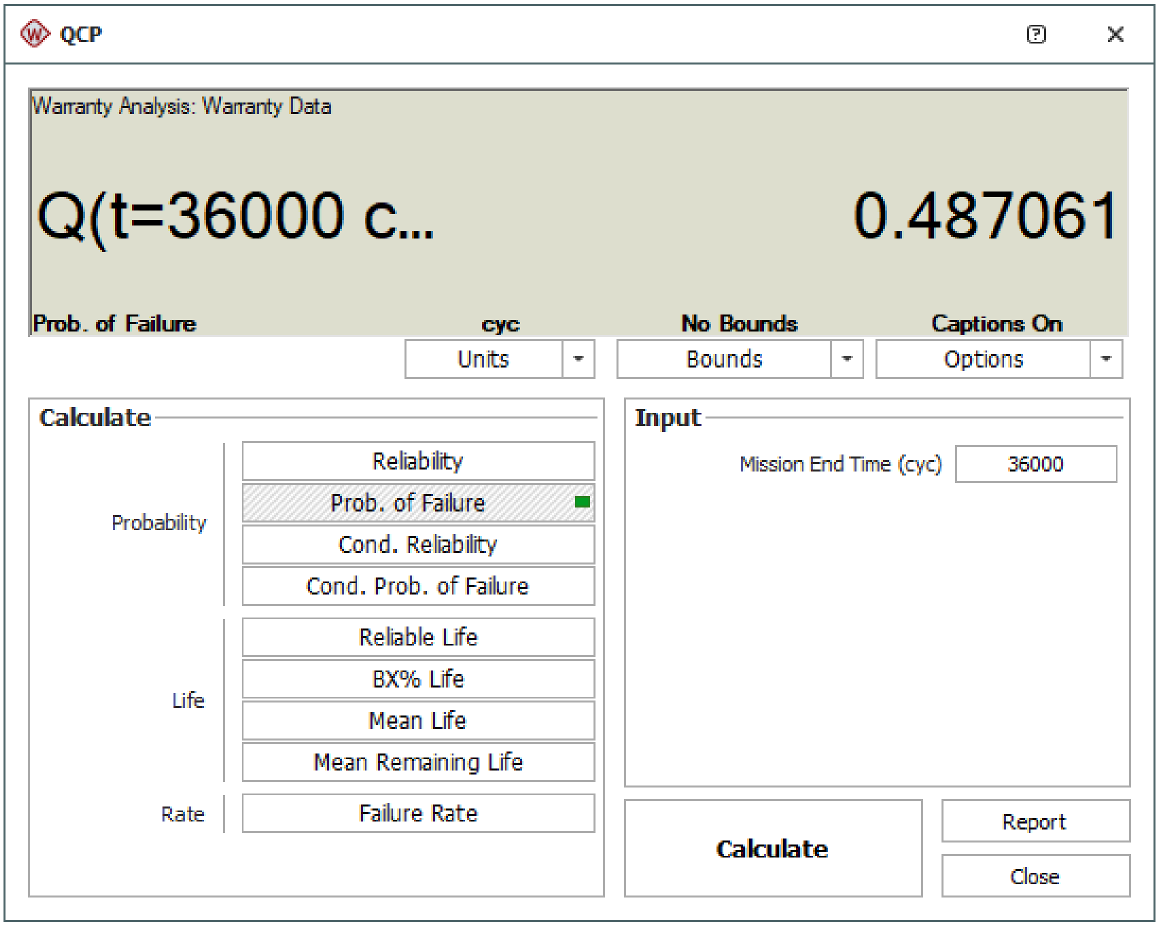
	Quantity Returned	Cycle Usage at Return Date	Date In-Service	Subset ID
1	1	9072	Dec-17	
2	1	9743	Jan-18	
3	1	6857	Feb-18	
4	1	7651	Mar-18	
5	1	5083	May-18	
6	1	5990	May-18	
7	1	7432	May-18	
8	1	8739	May-18	
9	1	3158	Jun-18	
10	1	1136	Jul-18	
11	1	4646	Aug-18	
12	1	3965	Sep-18	
13	1	3117	Oct-18	
14	1	3250	Nov-18	

Analyze the Data Set

1. On the Main page of the control panel, set the **End of Observation Period** date to **12/1/2018**.
2. 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**
3. 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.
4. 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.
5. 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.
6. Click **Calculate** to obtain the result. The result shows that 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.



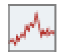
Forecasting Warranty Returns

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.

What's Changed? Starting in Version 2019, for folios that use the usage format, you can predict the number of failures based on the time and usage as well as the time. For example, for a vehicle with a 3-year/30,000 mile warranty, you can show the total number of warranty returns based on both values.

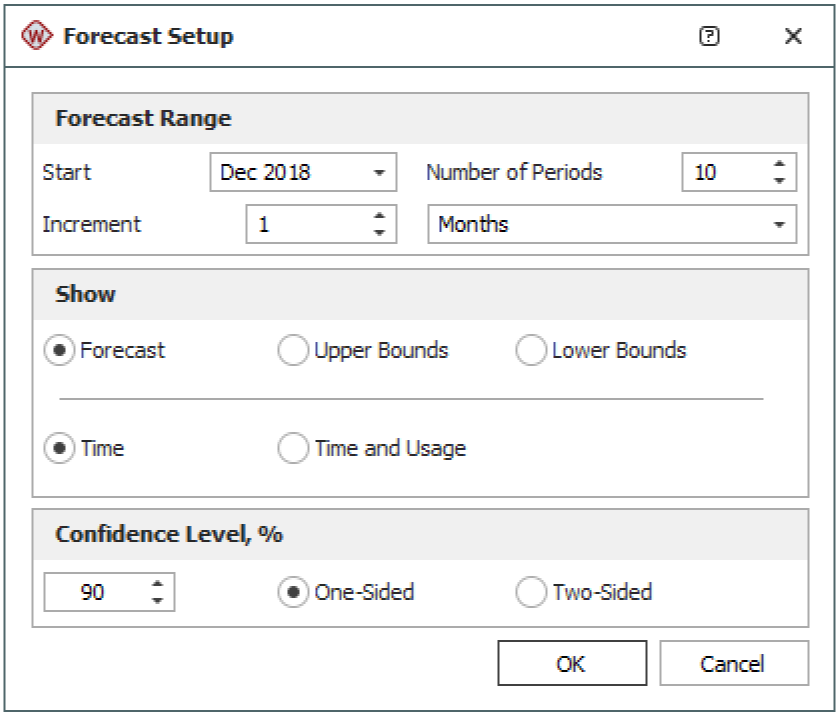
Forecast Setup

To generate a forecast, choose **Warranty > Tools > Forecast** or click the icon on the control panel.



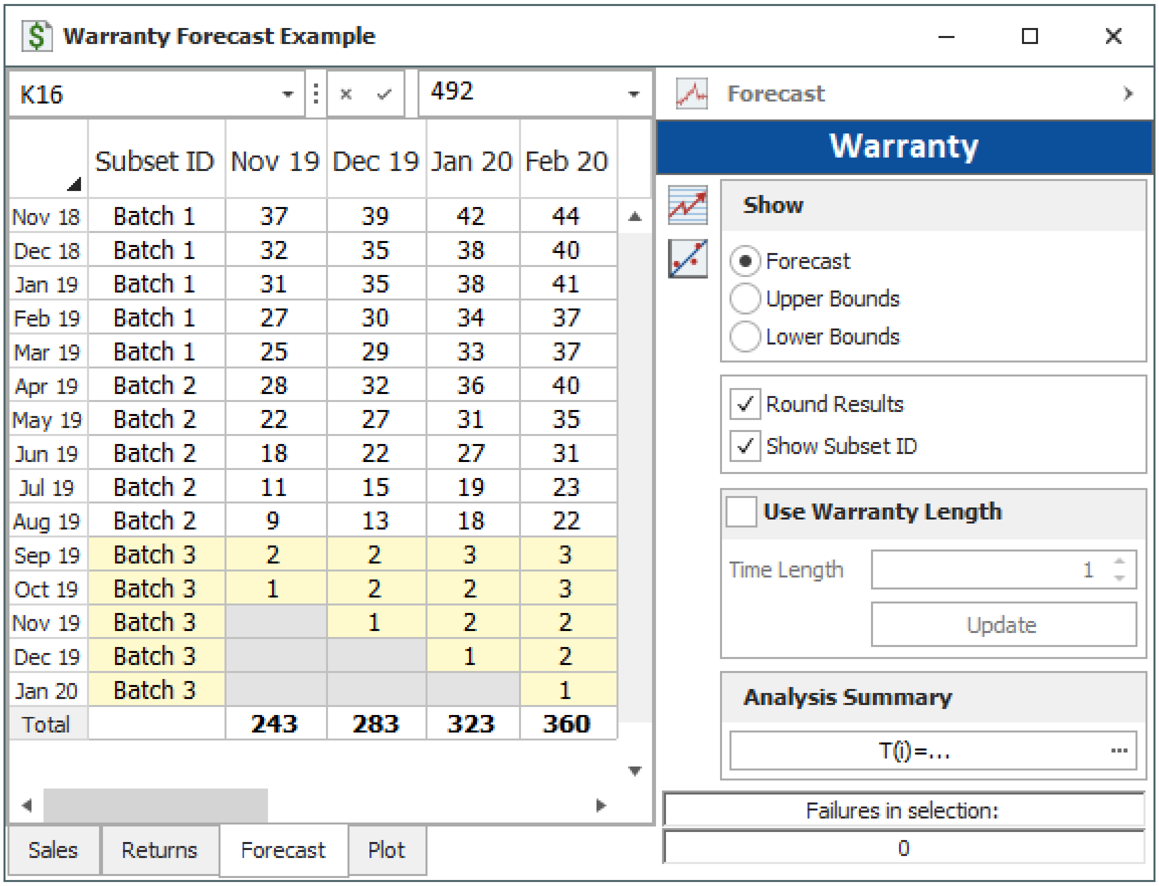
Then in the Forecast Setup window:

1. Enter the **Start** date for the forecast, the number of forecasts (**Number of Periods** field) and the length of each forecast (**Increment** field). In the example shown below, the time unit is in terms of years; therefore, an increment of 1 would mean that 1 forecast period is equivalent to 1 year, while an increment of 2 would mean that 1 forecast period is equivalent to 2 years. Note that:
    - 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.
2. 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.
3. For a usage format folio, select whether you want to generate the returns based on the **Time** only or on the combined **Time and Usage** estimates.
4. Finally, enter the desired confidence level and select whether to calculate **One-Sided** or **Two-Sided** confidence bounds.



Forecast Sheet

The example shown next is a Forecast sheet for the Nevada chart format. The column headings represent the warranty periods and the row headings 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:

1. 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.

2. **Round Results** rounds the estimates to the nearest integer. Note that the column totals are calculated based on the non-rounded results.

3. **Show Subset ID** adds a column in the Forecast sheet that displays the corresponding subset ID of the data in the Forecast sheet.

4. The **Use Warranty Length** check box limits 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.

For the usage format, if selected in the Forecast Setup window, you can also specify the maximum **Usage Length** value. Limiting the usage length allows you to disregard the number of returns that had more usage than the warranty allows. For example, a usage length of 30,000 miles would mean that any units that fail after having been operated for 30,000 miles are out of warranty and are therefore not counted in the total number of returns.

In the forecast sheet, the values shown include the number of failures for both time and usage.

Enter the length of the warranty period in the **Time Length** field and, if applicable, the **Usage Length** fields and click **Update**. These options use the same units you selected for the period increments in the Forecast Setup window and the usage units used in the Returns data sheet.

Warranty Analysis Plots

The features on the plot sheet of warranty analysis folios 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 is available only when you've generated forecast data. It shows the expected number of failures over a period of time. 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 percentage of failures for each time period, with respect to the number of units that are still out in the field operating at the beginning of each period.

  Note: If you select both the **Cumulative Failures** and the **Failures in Percents** options, then the plot will show the percentage of parts failed relative to the number of items still in the field at the time the forecasting began.

  - **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.

Manage Warranty Units in Weibull++

You can specify which of the configurable "database time units" are equivalent to the built-in (not configurable) "warranty time units" used in some warranty folio formats. To set the equivalencies, choose **File > Manage Database > Unit Settings**, and then click the **Warranty Units** tab. (In secure databases, this is available only if the user has the "Manage other database settings" permission.)

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 "database time units" for all time inputs and calculated results in the folio.

What does this mean?

For each ReliaSoft database, you can configure which units will be available for inputs and results, and you can define conversion factors that allow the software to automatically convert data from one unit to another (see Unit Settings).

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++ life data folio), the software must apply the equivalent "database time units" so that a) the results will be consistent with other analyses in the same database and b) any tools that utilize unit conversions (such as the QCP) will be able to perform the conversions properly.

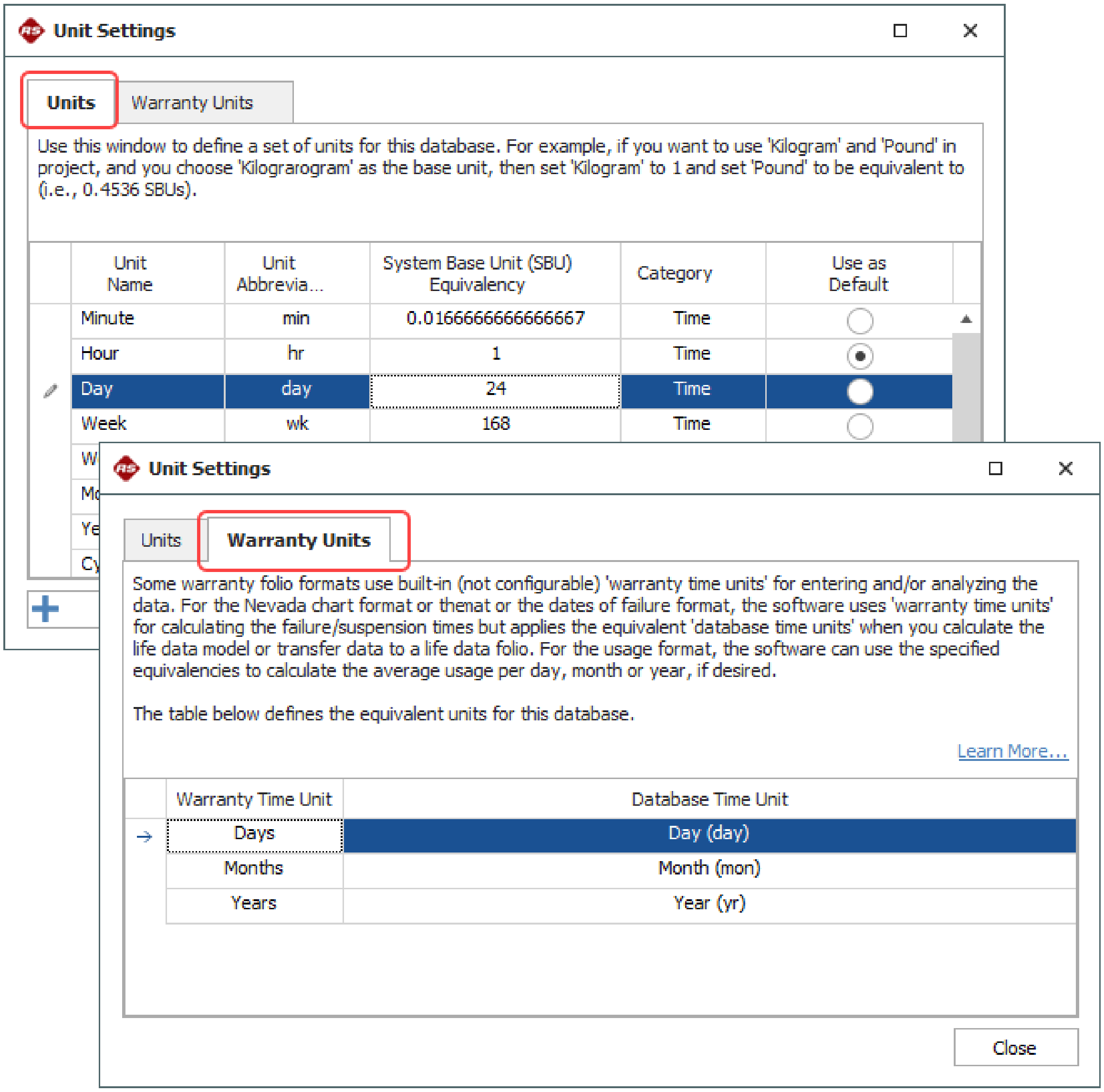
For example, in the following picture, the failure/suspension data in the Results window (which is displayed when you click the **Detailed Summary** icon in the **Analysis Summary** area of the control panel) has been calculated in terms of the "warranty time units" that are appropriate for the sales/returns data entered into the folio (calendar months), while the life data model parameters and QCP results are reported in the equivalent "database time units" (abbreviated as "Mon" under the default settings).

7	Results		
8	Distribution:	Weibull-2P	
9	Analysis:	RRX	
10	CB Method:	FM	
11	Ranking:	MED	
12	Beta	2.055328	
13	Eta (Mon)	38.947464	
14	LK Value	-21735.07082	
15	Rho	0.999518	
16	Fail \ Susp	3901 \ 56272	
17	LOCAL VAR/COV MATRIX	Var-Beta=0.000638	CV Eta Beta=-0.012949
18		CV Eta Beta=-0.012949	Var-Eta=0.352518
19	Num In State	F/S	End Time
20	1483	S	0
21	28	F	1
22	3951	S	1
23	107	F	2
24	1753	S	2
25	167	F	3
26	4837	S	3
27	218	F	4
28	3694	S	4
29	254	F	5

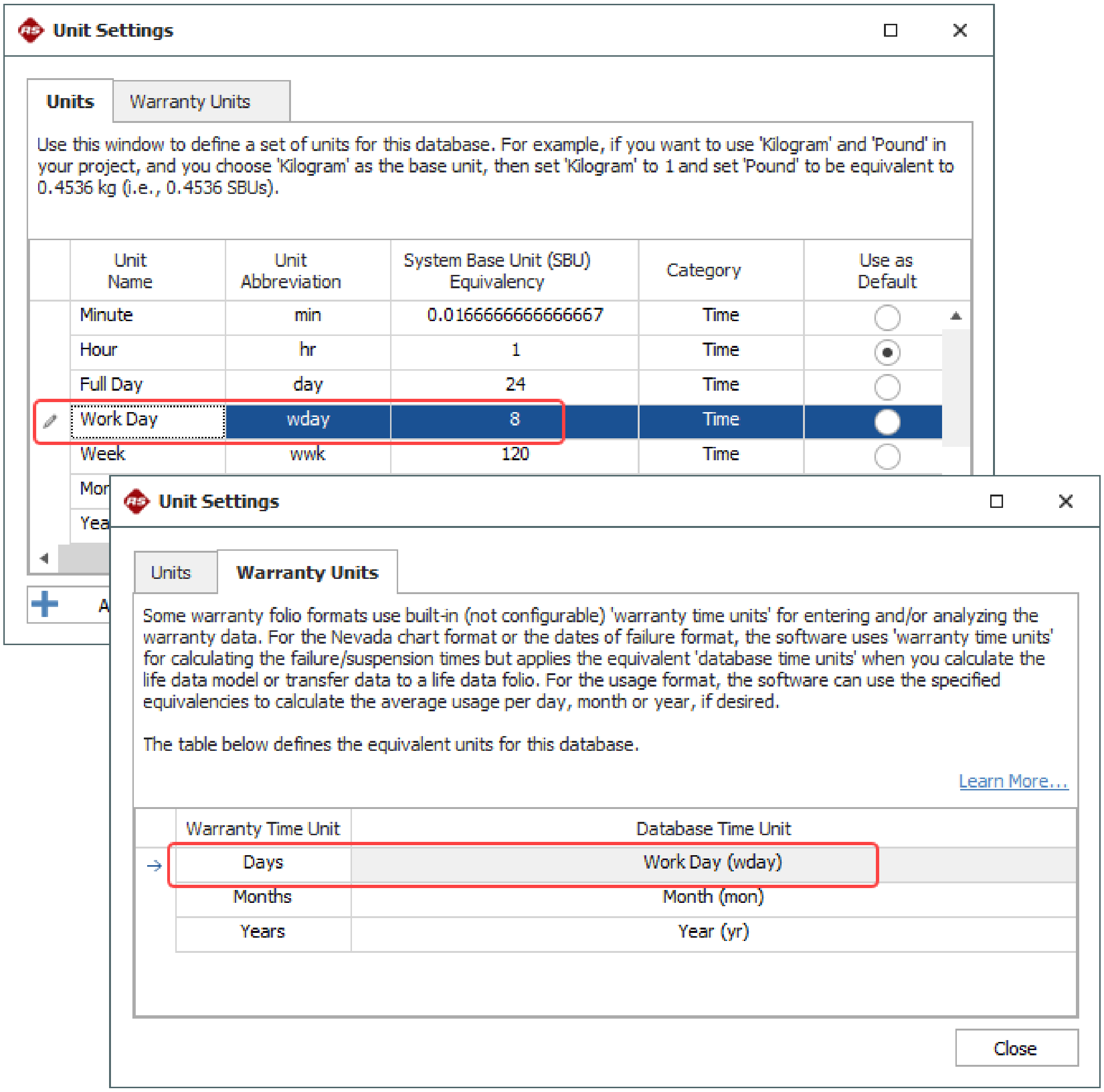
Database time units

Warranty time units

By default, each new database is created with "database 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.

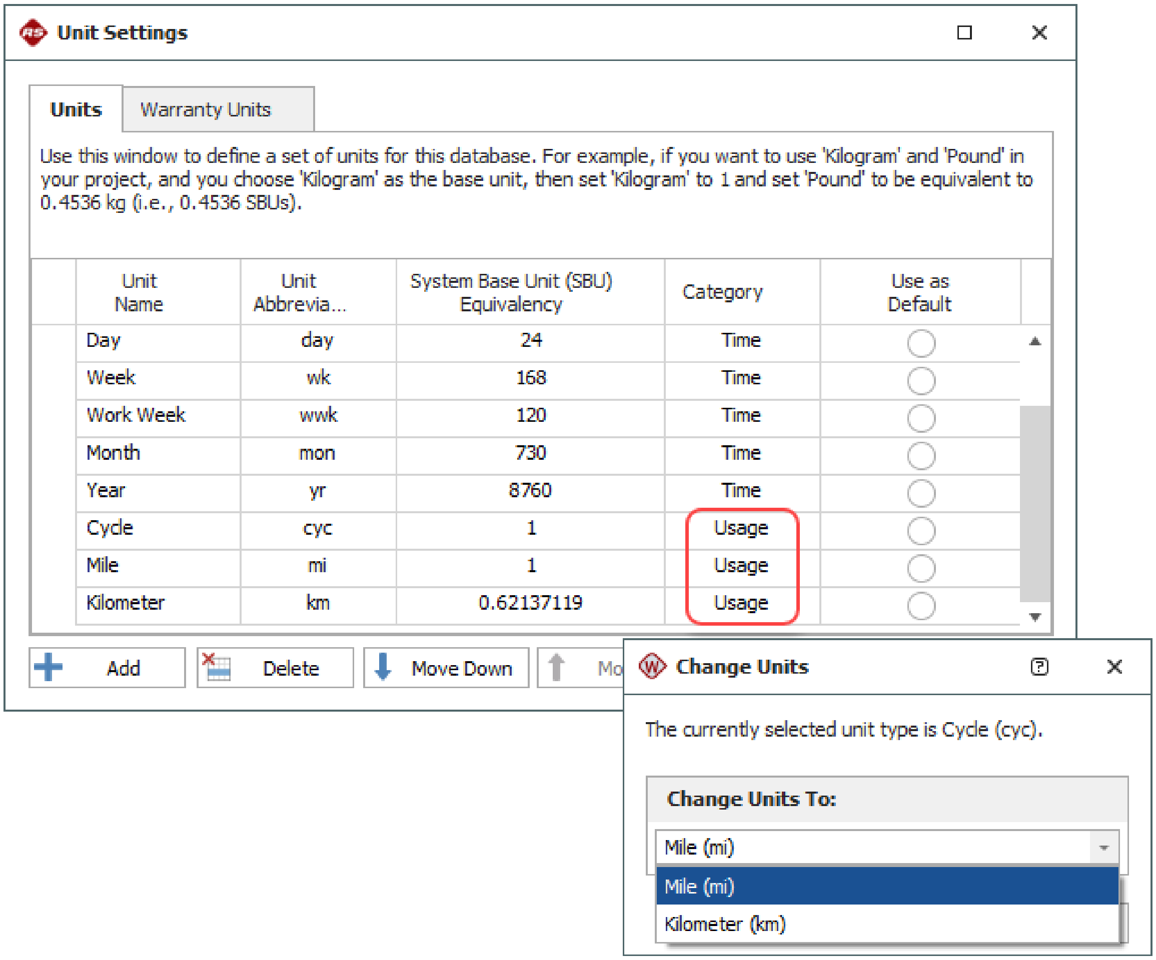


Remember that the failure/suspension data generated by the Nevada chart and dates of failure warranty folios do not get *converted* based on this mapping. The software simply assigns an appropriate "database time unit" to the times that were calculated in "warranty time units."

How are usage format folios affected?

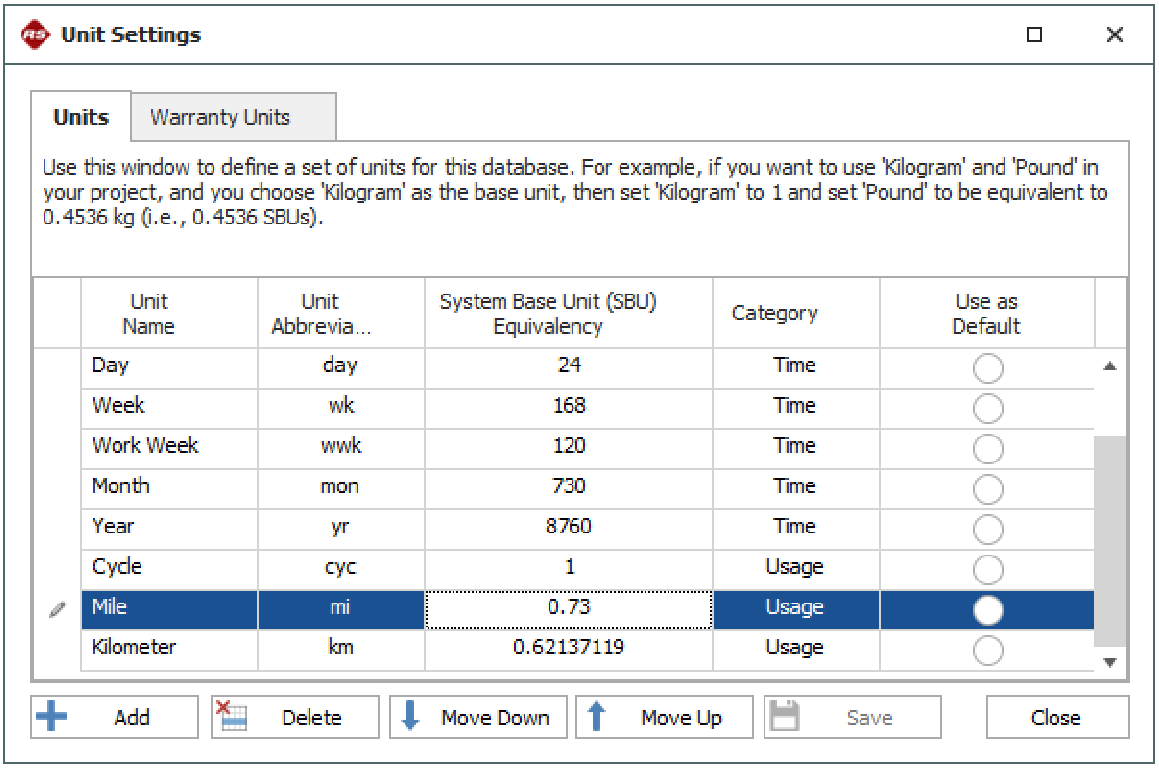
In a usage format warranty folio, you enter the amount of usage (rather than operating time) for the failed units and the software must estimate the amount of usage accumulated by the units that have not failed (suspensions).

- 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 "database 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 Unit Settings window. The default settings are shown next.

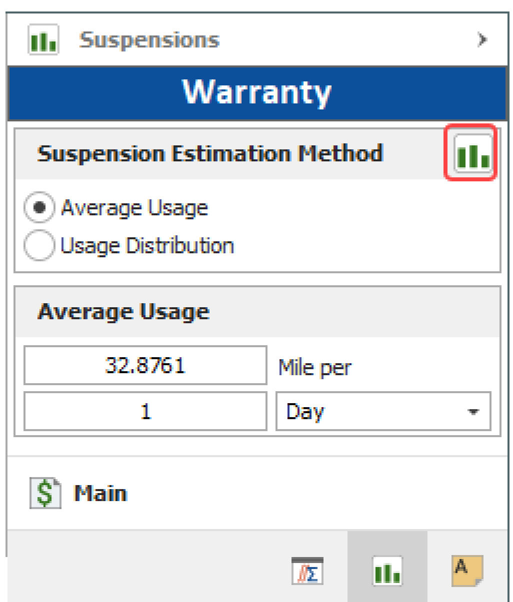


- On the **Suspensions** page of the control panel, you must specify the amount of usage that units typically accumulate per "Day," "Month" or "Year." This refers to the built-in (not configurable) "warranty time units." If you have selected to specify the Average Usage (rather than a Usage Distribution), 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 "database time units."

  For example, suppose that the warranty data has been entered in miles and the "Unit Settings" 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 "database 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

Repairable Systems Analysis Folios

Recurrent Event Data Analysis (RDA) Folios

Recurrent Event Data Analysis

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.

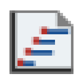
The ReliaWiki resource portal has more information on recurrent event data analysis at http://www.reliawiki.org/index.php/Recurrent Event Data Analysis.

Non-Parametric RDA Folio

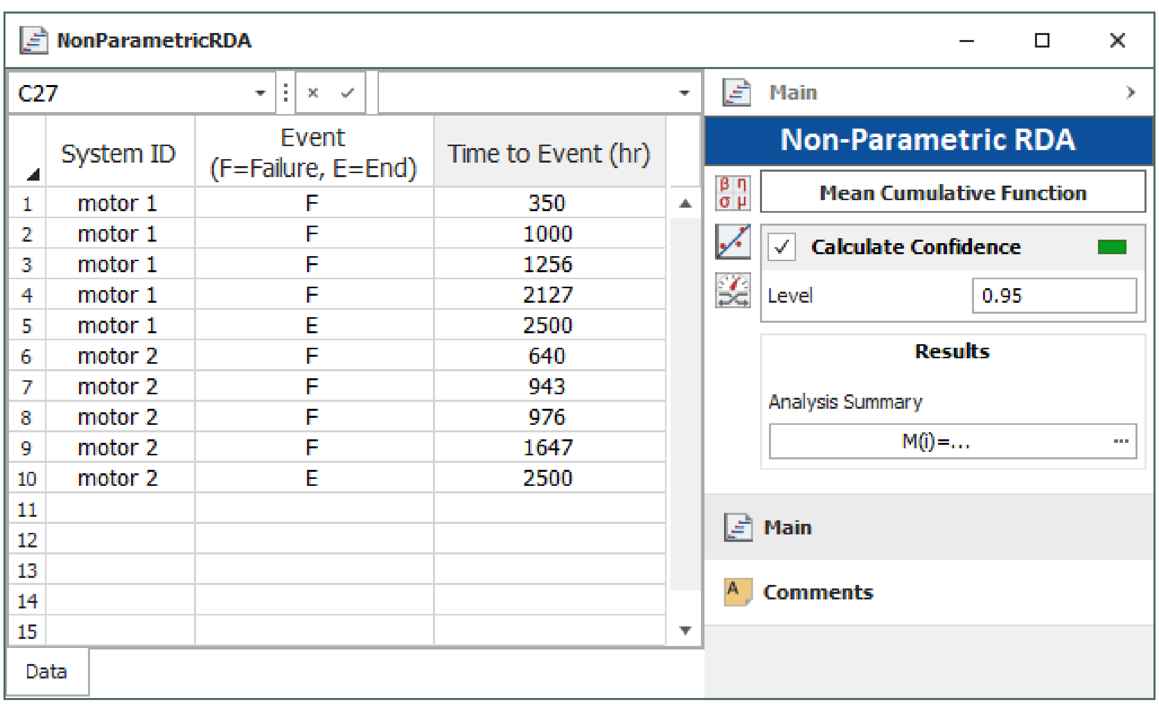
The non-parametric RDA folio is based on the mean cumulative function (MCF). This analysis provides a plot of the MCF to illustrate the average number of recurring failures of a system, or a group of systems, over a given period of time (or distance, cycles, etc.). The MCF can be used to evaluate whether 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. The ReliaWiki resource portal has more information on the MCF at: http://www.reliawiki.org/index.php/Non-Parametric_Recurrent_Event_Data_Analysis.

NON-PARAMETRIC RDA DATA SHEET

To create a non-parametric RDA folio, choose **Home > Insert > Non-Parametric RDA**, or right-click the **Repairable Systems** folder in the current project explorer and choose **Add Non-Parametric RDA** on the shortcut menu.



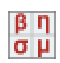
The following picture shows an example of the non-parametric RDA folio data sheet.



- 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 motors.
- 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.
- 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.

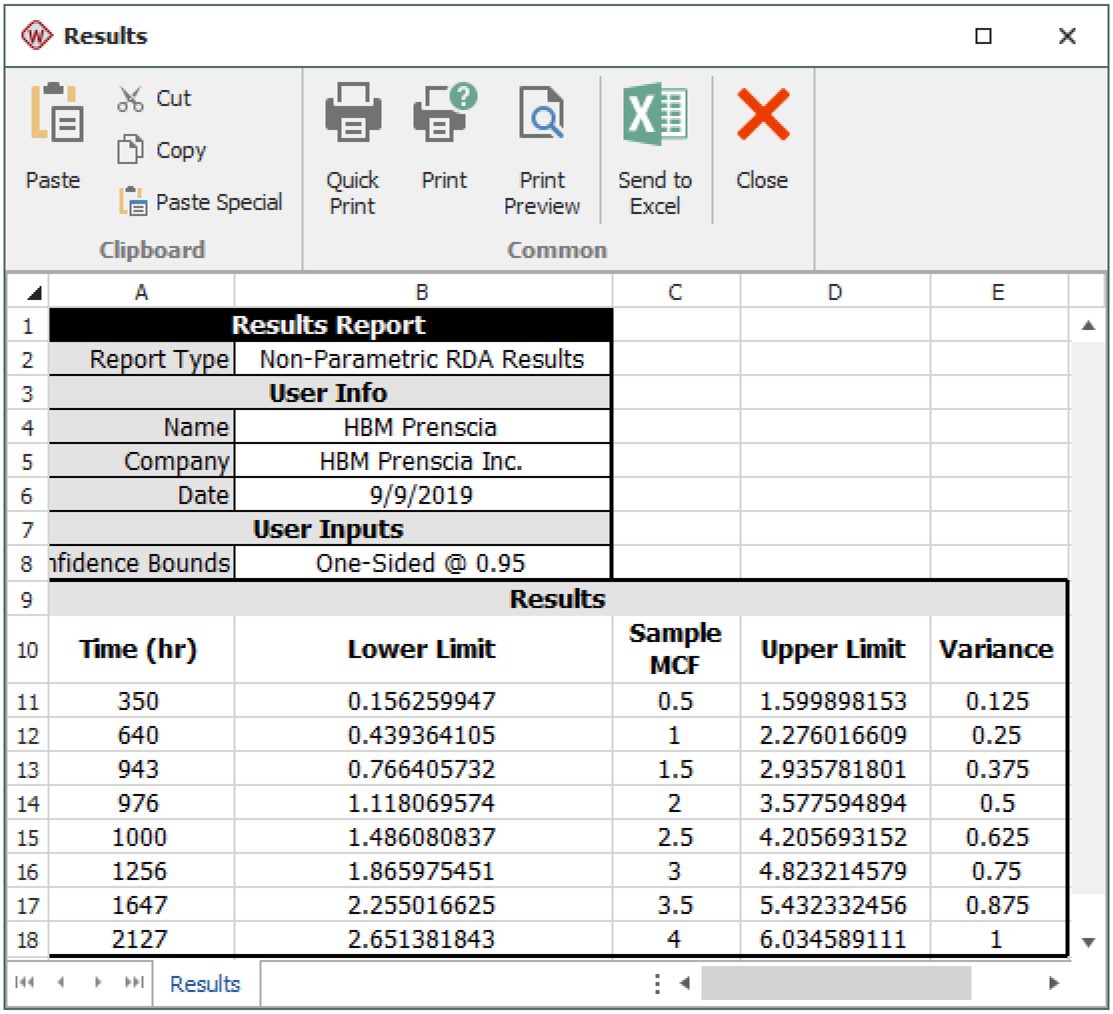
EVALUATING RESULTS

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.

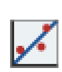


Results Report				
Report Type	Non-Parametric RDA Results			
User Info				
Name	HBM Prenscia			
Company	HBM Prenscia Inc.			
Date	9/9/2019			
User Inputs				
ıfidence Bounds	One-Sided @ 0.95			
Results				
Time (hr)	**Lower Limit**	**Sample MCF**	**Upper Limit**	**Variance**
350	0.156259947	0.5	1.599898153	0.125
640	0.439364105	1	2.276016609	0.25
943	0.766405732	1.5	2.935781801	0.375
976	1.118069574	2	3.577594894	0.5
1000	1.486080837	2.5	4.205693152	0.625
1256	1.865975451	3	4.823214579	0.75
1647	2.255016625	3.5	5.432332456	0.875
2127	2.651381843	4	6.034589111	1

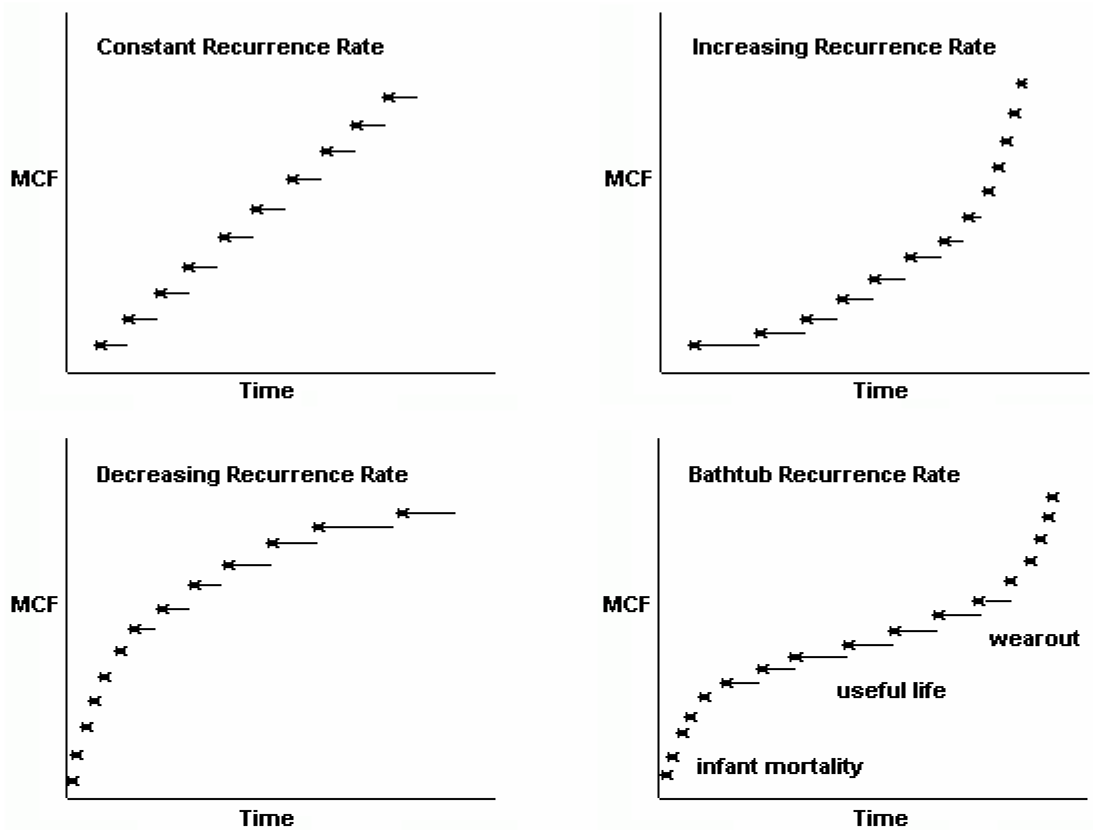
The **Time** column displays the system ages, sorted in chronological order, that were entered for the failure events. The **Sample MCF** column shows the average number of failures at each particular age.

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.



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.

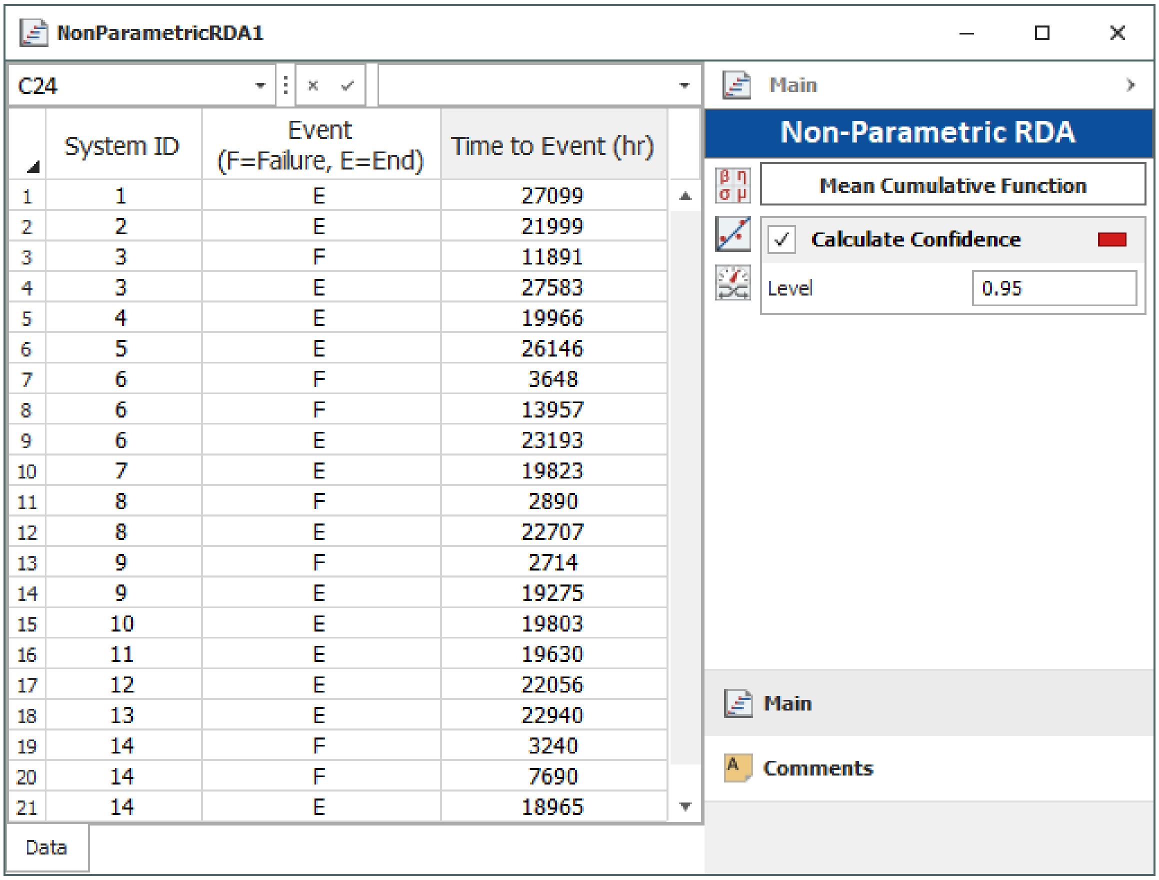
NON-PARAMETRIC RDA EXAMPLE

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 pre-production road test and tracked the transmission repairs in a sample of 14 cars. The following table shows the data. (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.)

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

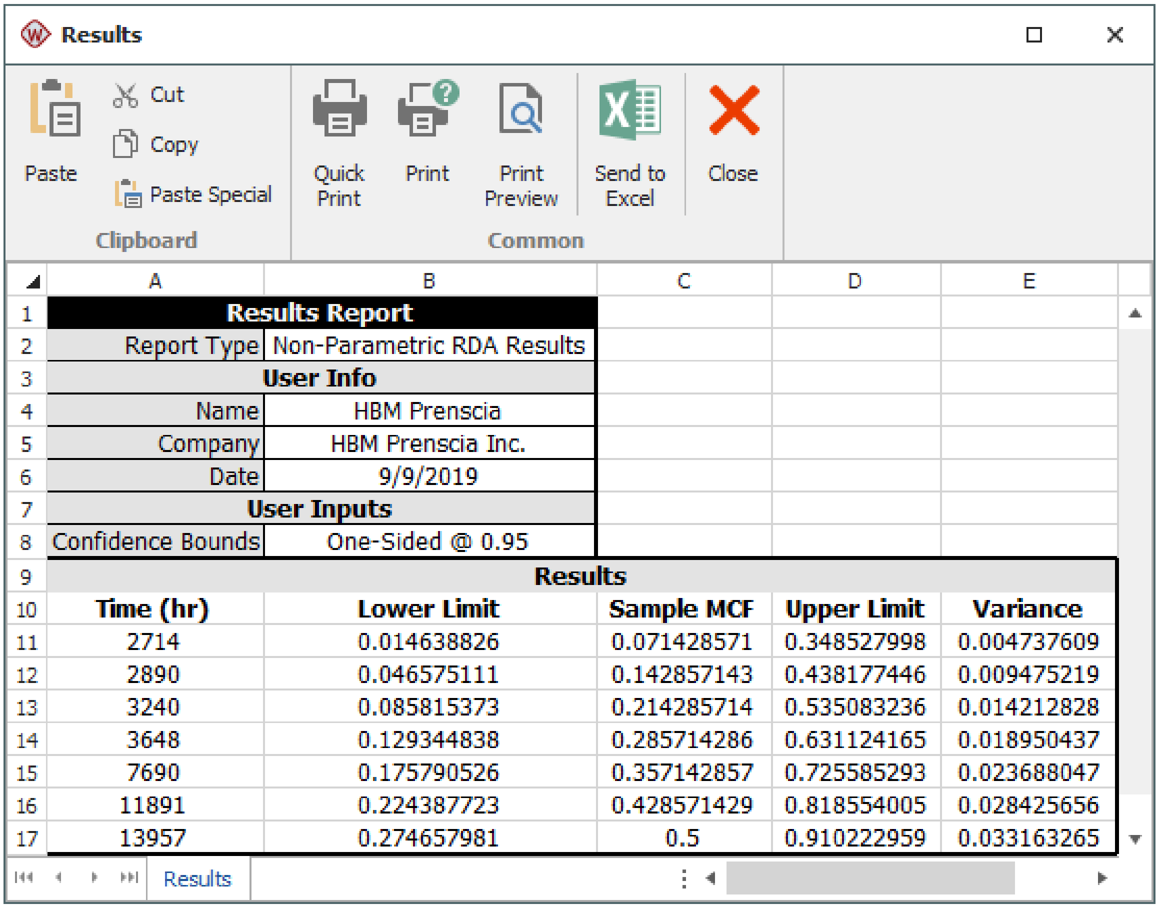
Analyze the Data Set

1. Enter the given data into a non-parametric RDA folio data sheet, as shown next.



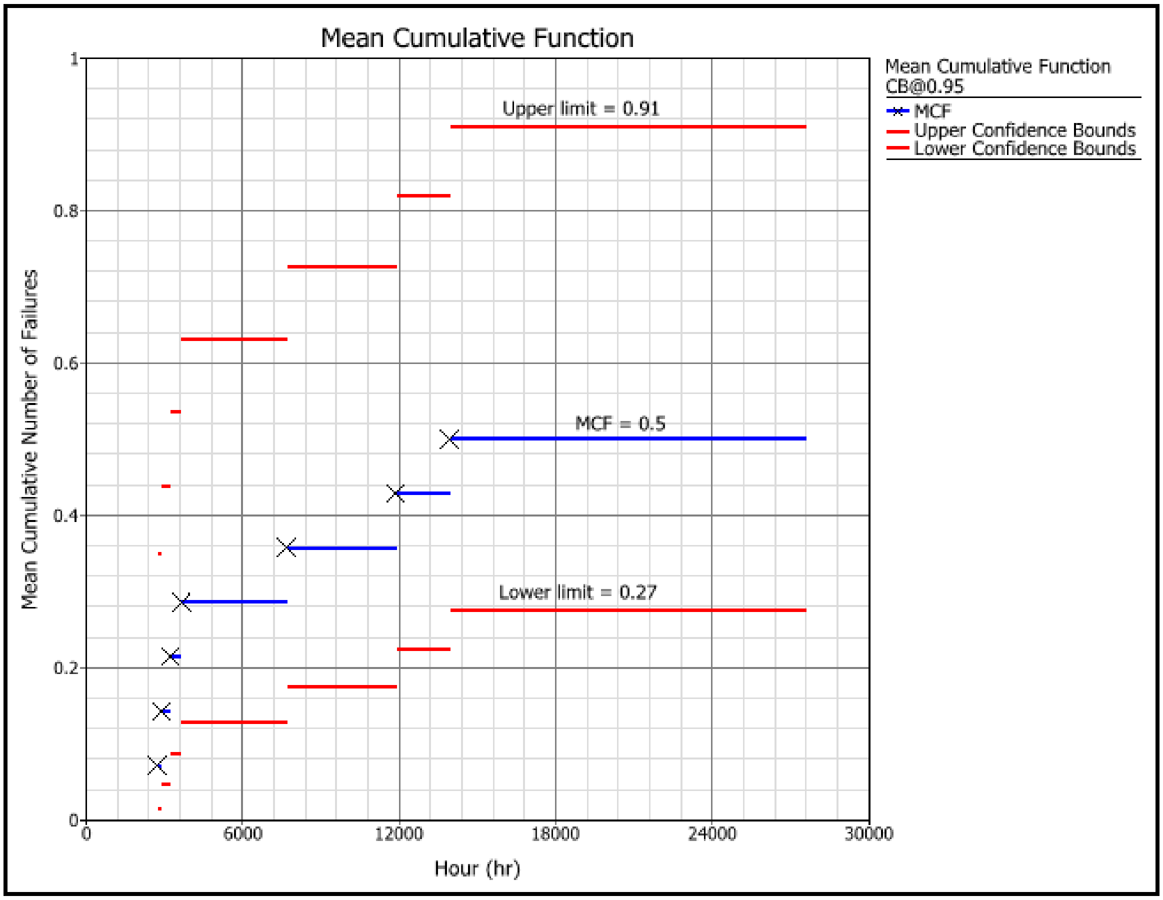
2. Select to calculate the confidence bounds, enter a confidence level of **0.95** and click **Calculate.**

3. Click the **Show Results (...)** button to open the Results window. 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.



Results Report				
Report Type	Non-Parametric RDA Results			
User Info				
Name	HBM Prenscia			
Company	HBM Prenscia Inc.			
Date	9/9/2019			
User Inputs				
Confidence Bounds	One-Sided @ 0.95			
Results				
Time (hr)	**Lower Limit**	**Sample MCF**	**Upper Limit**	**Variance**
2714	0.014638826	0.071428571	0.348527998	0.004737609
2890	0.046575111	0.142857143	0.438177446	0.009475219
3240	0.085815373	0.214285714	0.535083236	0.014212828
3648	0.129344838	0.285714286	0.631124165	0.018950437
7690	0.175790526	0.357142857	0.725585293	0.023688047
11891	0.224387723	0.428571429	0.818554005	0.028425656
13957	0.274657981	0.5	0.910222959	0.033163265

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 Xs 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 estimate 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 Xs 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.

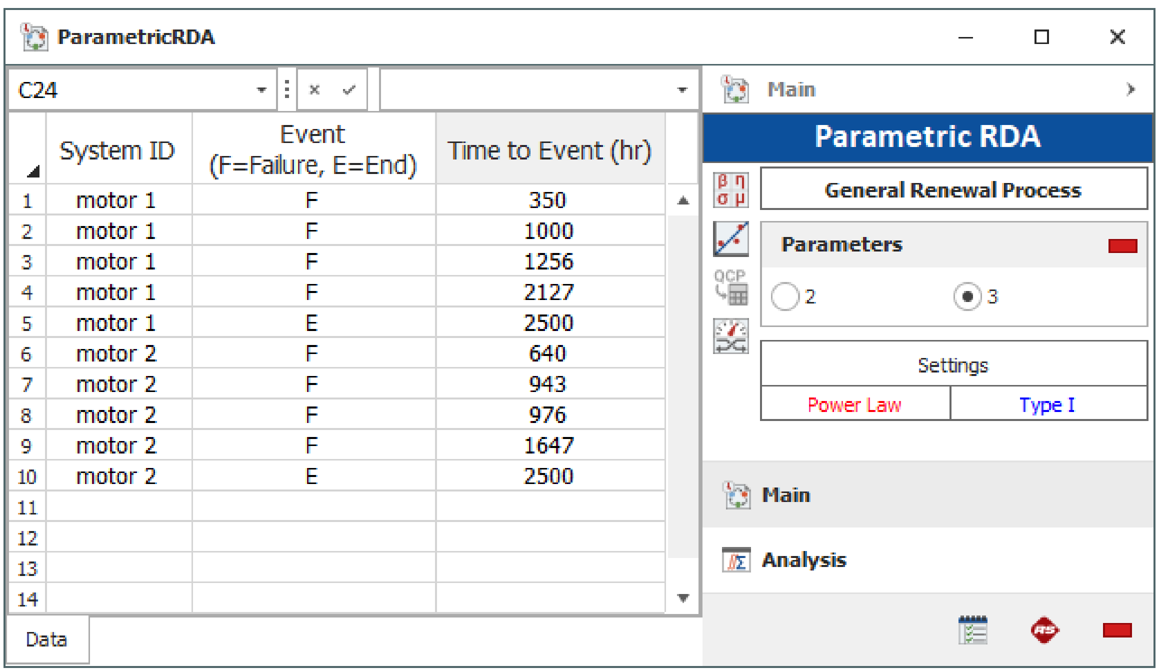
Parametric RDA Folio

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. The ReliaWiki resource portal has more information on the GRP model at: http://www.reliawiki.org/index.php/Parametric_Recurrent_Event_Data_Analysis.

PARAMETRIC RDA DATA SHEET

To create a parametric RDA folio, choose **Home > Insert > Parametric RDA**, or right-click the **Repairable Systems** 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 **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 motors.
- 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 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.

PARAMETRIC RDA FOLIO CONTROL PANEL SETTINGS

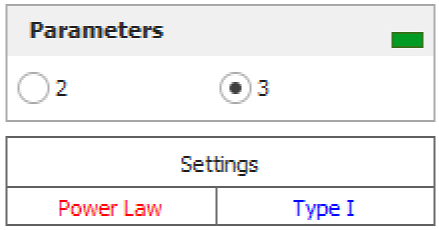
The parametric RDA folio control panel allows you to configure the analysis settings for the data sheet and view/access the results. This topic focuses on the Main page and Analysis page of the parametric RDA folio control panel, which contain most of the tools you will need to perform the analysis. For more information about the control panel in general, see Control Panels.

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 **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 select3parameters, 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.

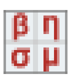


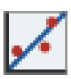
- The **Analysis Summary** area displays the calculated results. The following example shows the calculated values of the parameters and the likelihood (LK) value of the GRP model for a particular data set. You can click the **Detailed Summary** icon to open the Results window, which shows the results in a worksheet that you can copy or print.

Analysis Summary	
Parameters	
Beta	0.827554
Lambda	0.038864
q	0.275771
LK Value	-123.596377

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. 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). 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 values in the current data sheet.

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:

  - **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.

  The ReliaWiki resource portal explains the mathematical concept behind these 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.)

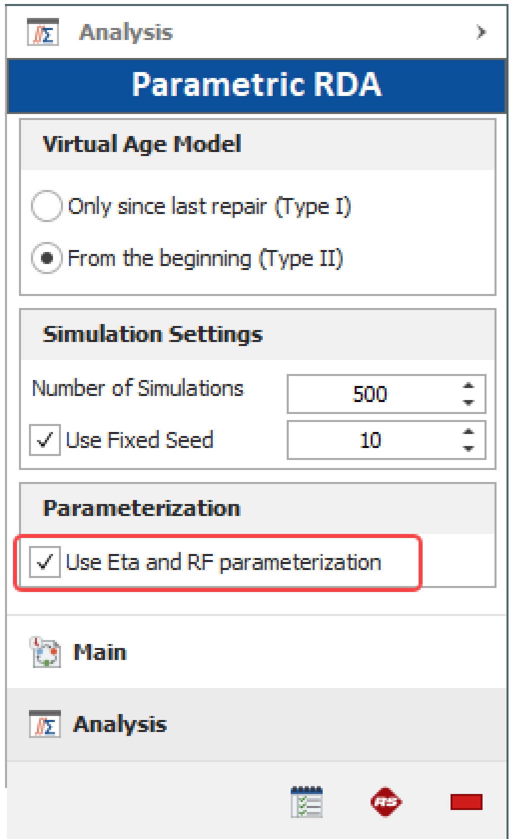
  Since it may not be feasible to classify real-world repair situations into one of these two abstract categories, the virtual age type is usually selected based on what provides a better statistical fit for the data. The model fit can be assessed by using the likelihood (LK) value that is displayed in the **Analysis Summary** area on the control panel, where the higher value indicates the better fit.

- 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.

  - The **Number of Simulations** specifies the number of data points to generate in the simulation.

  - The **Use a Fixed Seed** checkbox 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). To do this, select the check box in the Parameterization area, as shown next.



The relationship between the power law parameter, lambda (λ), and the Weibull parameter, eta (η), is 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:

$$RF = 1 - q$$

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*.

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 Quick Calculation Pad (QCP).

Failure Intensity

Cumulative Failure Intensity

Calculates the average rate of occurrence of failures from time = 0 up to a specified end time. Enter the end time 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++ life data 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%.

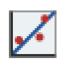
Conditional Probability of Failure

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%.

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 following is a description of the different types of plots that can be created in a parametric RDA folio. (For general information on working with plots, see Plot Utilities.)

- **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.

PARAMETRIC RDA EXAMPLE

*The data set used in this example is available in the example database installed with the Weibull++ software (called "Weibull21_Examples.rsgz21"). 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 "Parametric RDA - Airplane AC Repair."

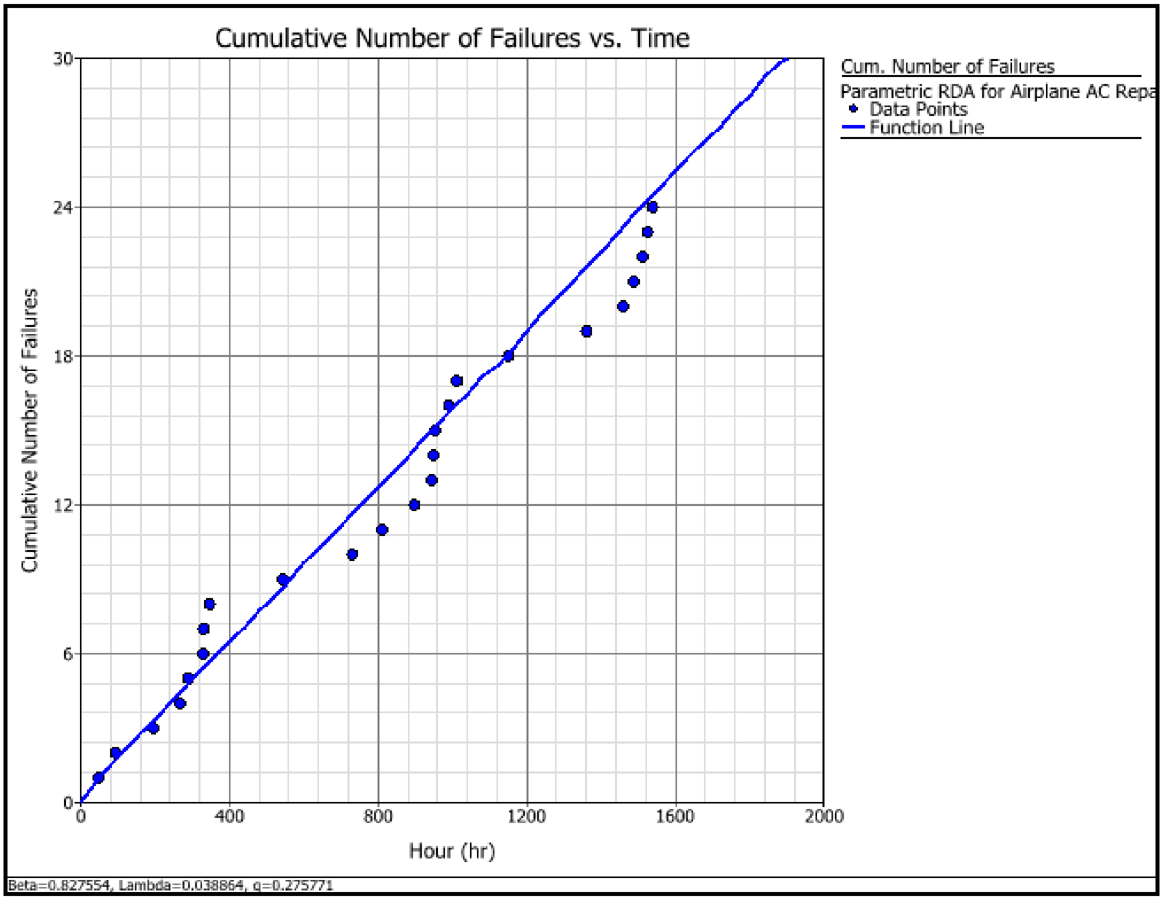
An analyst recorded the successive failures of the air-condition unit of an aircraft. The objective is to obtain the expected number of failures that may occur by 1,800 hours. The following data sheet shows the failure logs of the unit between 0 to 1,539 hours of observation. The observation period ended 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

Analyze the Data Set

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.

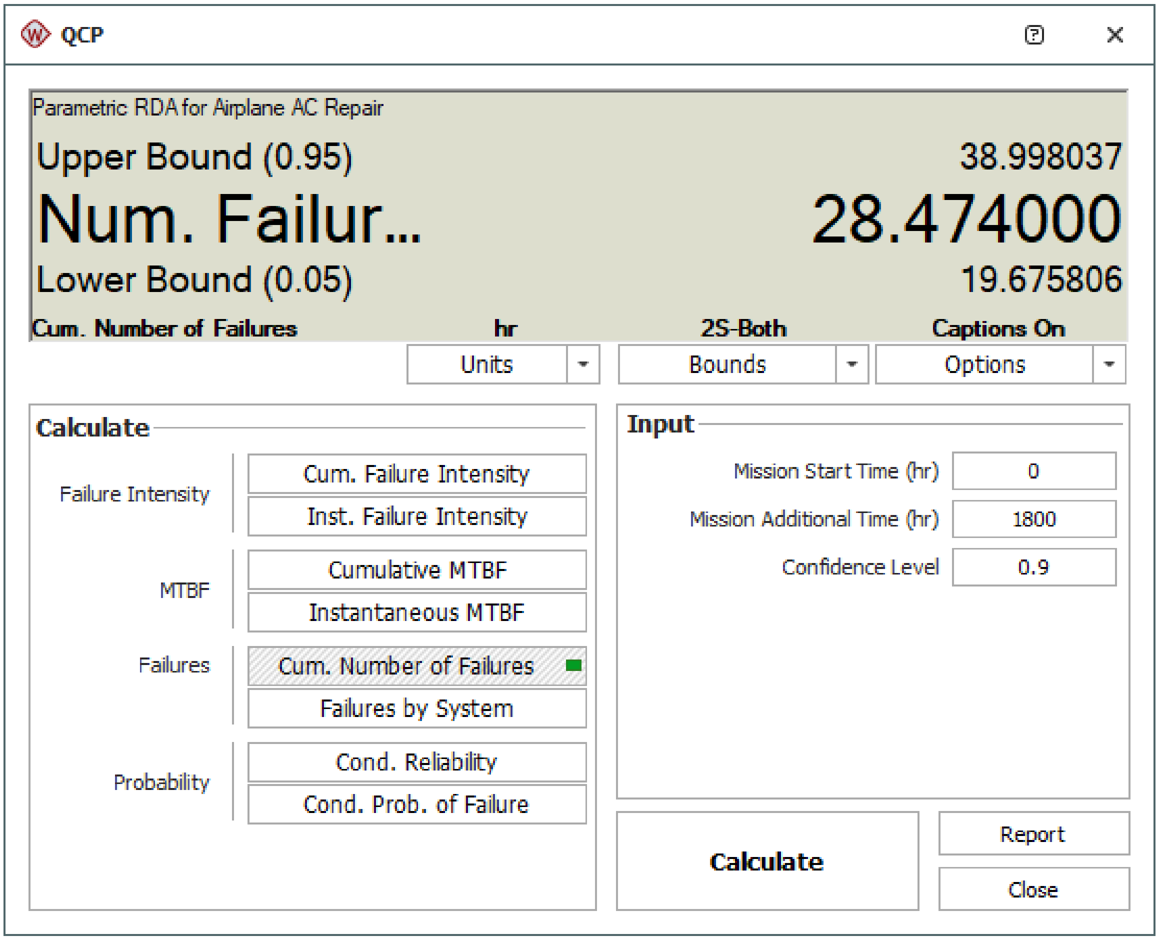
2. Click the **Plot** icon to visualize how the number of failures increases over time, as shown next.



3. To estimate 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**

4. Click **Calculate**. The results show that 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.



Event Log Data Folio

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.

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.

Event Log Folio Data Sheet

To create an event log folio, choose **Home > Insert > Event Log**, or right-click the **Repairable Systems** 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-01-17	11:00:00 AM	Mar-05-17	9:00:00 AM	Electric Saw	Motor				Elecrical Short
2	M-1	F	Apr-01-17	2:30:00 PM	Apr-03-17	11:00:00 AM	Conveyor	Belt				
3	M-1	E	May-03-17	9:00:00 AM	May-06-17	12:00:00 PM	Lubrication	Lubricaton				
4	M-1	F	Aug-03-17	3:30:00 PM	Aug-04-17	9:00:00 AM	Electric Saw	Blade				Worn
5	M-1	E	Sep-01-17	4:10:00 PM	Sep-02-17	4:00:00 PM	Inspections	Inspections				
6	M-1	F	Dec-01-17	10:00:00 AM	Dec-02-17	3:00:00 PM	Conveyor	Wheels				Bearing Failure
7	M-1	F	Jan-04-18	12:30:00 PM	Jan-07-18	7:20:00 AM	Hydraulic Arm	Hydraulic Lines				
8	M-1	F	Feb-01-18	4:00:00 PM	Feb-02-18	7:00:00 AM	Hydraulic Arm	Piston				
9	M-1	F	Mar-07-18	4:15:00 PM	Mar-09-18	9:00:00 AM	Conveyor	Belt				Unstable
10	M-1	F	Apr-05-18	4:30:00 PM	Apr-06-18	4:45:00 AM	Electric Saw	Blade				Crack

- 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 obtain 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.

If your system does not operate on a 24/7 basis, you may want to define the shift schedule of the system so that you obtain accurate calculations for the failure and repair times. See Set Shift Pattern.

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, Other page and Comments page.

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.

- 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 Set Shift Pattern.)

- The **Results** area provides the 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 on 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.

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.

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++ life data 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.

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.

- 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.

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.

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**.

CONTROL PANEL COMMENTS PAGE

The Comments page allows you to enter notes or other text that will be saved with the folio.

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, 2018 at 12 PM and stopped on March 18, 2018 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.

The objective is to analyze each component and obtain the failure and repair distributions for all recorded events.

	Event Type	System Failed		System Restored		Component
		Date	Time	Date	Time	
1	Failure	Jan-02-18	4:00 PM	Jan-02-18	7:49 PM	Pump
2	Maintenance	Jan-09-18	8:30 AM	Jan-09-18	10:43 AM	Motor
3	Failure	Jan-10-18	9:13 AM	Jan-10-18	7:48 PM	Motor

4	Inspec-tion	Jan-12-18	3:26 PM	Jan-12-18	6:46 PM	Pump
5	Failure	Jan-13-18	4:56 PM	Jan-13-18	5:21 PM	Pump
6	Main-tenance	Jan-15-18	1:16 PM	Jan-15-18	4:39 PM	Motor
7	Failure	Jan-20-18	1:38 PM	Jan-21-18	7:15 PM	Pump
8	Failure	Jan-25-18	10:32 AM	Jan 27-18	10:47 PM	Motor
9	Cleaning	Jan-28-18	11:31 AM	Jan-28-18	12:00 PM	Pump
10	Failure	Feb-02-18	2:38 PM	Feb-02-18	7:11 PM	Motor
11	Inspec-tion	Feb-08-18	3:51 PM	Feb-08-18	8:22 PM	Pump
12	Failure	Feb-12-18	4:42 PM	Feb-13-18	9:59 AM	Pump
13	Cleaning	Feb-17-18	2:47 PM	Feb-17-18	7:13 PM	Motor
14	Main-tenance	Feb-25-18	4:31 PM	Feb-25-18	5:00 PM	Pump
15	Failure	Feb-28-18	9:00 AM	Feb-28-18	3:10 PM	Motor
16	Failure	Mar-01-18	10:16 AM	Mar-01-18	10:43 AM	Pump
17	Inspec-tion	Mar-02-18	3:14 PM	Mar-02-18	9:11 PM	Pump

18	Failure	Mar-12-18	8:46 AM	Mar-12-18	9:20 PM	Motor
19	Failure	Mar-13-18	4:45 PM	Mar-13-18	5:13 PM	Pump
20	Main-tenance	Mar-15-18	9:36 AM	Mar-15-18	10:02 PM	Motor

SELECT THE ANALYSIS SETTINGS

The first step is to set up the event log folio for data entry:

- On the Other page of the control panel:
  - Select **Hours** for the time units.
  - Select your preferred format for the date and time.
  - Select the **Use OTSF column** check box and clear the check box for the **Use System column**.
- 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.

Next, enter the given data into the folio:

- 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**.

	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)
1		F	1/02/18	4:00:00 PM	1/02/18	7:49:00 PM	Pump				N
2		E	1/09/18	8:30:00 AM	1/09/18	10:43:00 AM	Motor				Y
3		F	1/10/18	9:13:00 AM	1/10/18	7:48:00 PM	Motor				Y
4		E	1/12/18	3:26:00 PM	1/12/18	6:46:00 PM	Pump				N
5		F	1/13/18	4:56:00 PM	1/13/18	5:21:00 PM	Pump				N
6		E	1/15/18	1:16:00 PM	1/15/18	4:39:00 PM	Motor				Y
7		F	1/20/18	1:38:00 PM	1/21/18	7:15:00 PM	Pump				N
8		F	1/25/18	10:32:00 AM	1/27/18	10:47:00 PM	Motor				Y
9		E	1/28/18	11:31:00 AM	1/28/18	12:00:00 PM	Pump				N
10		F	2/02/18	2:38:00 PM	2/02/18	7:11:00 PM	Motor				Y
11		E	2/08/18	3:51:00 PM	2/08/18	8:22:00 PM	Pump				N
12		F	2/12/18	4:42:00 PM	2/13/18	9:59:00 AM	Pump				N
13		E	2/17/18	2:47:00 PM	2/17/18	7:13:00 PM	Motor				Y
14		E	2/25/18	4:31:00 PM	2/25/18	5:00:00 PM	Pump				N
15		F	2/28/18	9:00:00 AM	2/28/18	3:10:00 PM	Motor				Y
16		F	3/01/18	10:16:00 AM	3/01/18	10:43:00 AM	Pump				N
17		E	3/02/18	3:14:00 PM	3/02/18	9:11:00 PM	Pump				N
18		F	3/12/18	8:46:00 AM	3/12/18	9:20:00 PM	Motor				Y
19		F	3/13/18	4:45:00 PM	3/13/18	5:13:00 PM	Pump				N
20		E	3/15/18	9:36:00 AM	3/15/18	10:02:00 PM	Motor				Y

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 (January 1, 2018 at 12 PM), as well as the date and time the system monitoring ended (March 18, 2018 at 1 PM). Clear the check box for **System is new on start date**.

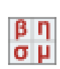
Enter the shift schedule of the system by choosing **Event Log > Actions and Settings > Set Shift Pattern** or by clicking the icon on the control panel.

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

ANALYZE THE DATA SET

- Calculate the results by choosing **Event Log > Analysis > Calculate** or by clicking the icon on the control panel.



- To view the failure and repair distributions, click the **Show Analysis Summary (...)** button on the control panel. The following report shows the results. In this example, the first table shows the failure and repair distributions for each component due to each type of event. The second table shows the total uptime and downtime for each component due to each type of event.

Results							
			Failure Results			Repair Results	
ID	Analysis ID*	Distribution	Parameter 1	Parameter 2	Distribution	Parameter 1	Parameter 2
Motor	E	Weibull 2-RRX	1.211553	257.502537	Exponential 1-RRX	0.156434	
Pump	E	Weibull 2-RRX	1.883864	131.512949	Exponential 1-RRX	0.308864	
Motor	F	Lognormal 2-RRX	4.764942	0.705949	Lognormal 2-RRX	2.464623	1.080161
Pump	F	Weibull 2-RRX	2.367426	126.539065	Lognormal 2-RRX	0.856889	2.040006
Level	ID	Uptime	Downtime				
1	E-Motor	669.783333	22.466667				
1	E-Pump	598.9	14.766667				
1	F-Motor	635.983333	94.116667				
1	F-Pump	598.9	52.05				

TRANSFER THE RESULTS TO A LIFE DATA FOLIO

You can perform further analysis by transferring the failure/repair data from the event log folio to a Weibull++ life data folio.

Close the Results window and 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++ life data 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.

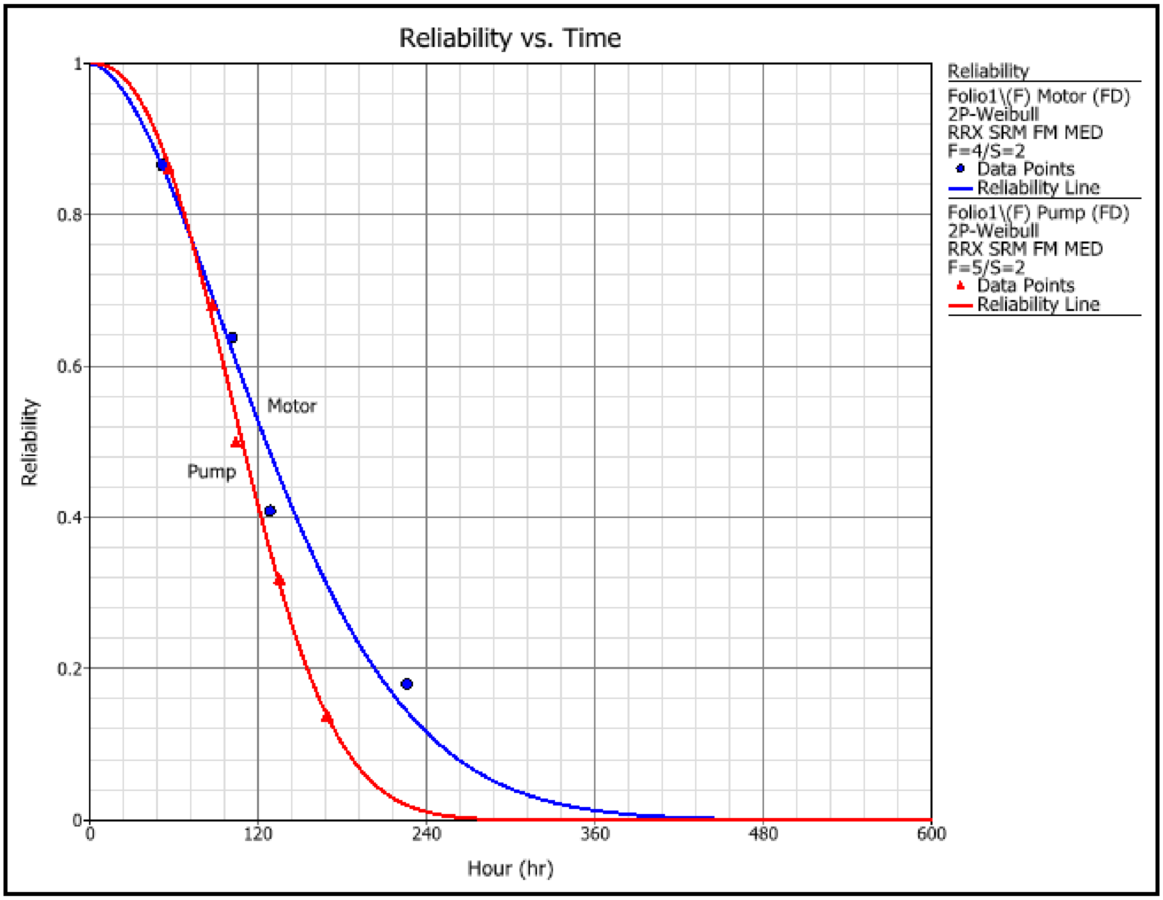
C13

	State F or S	Time to F or S (hr)	Subset ID 1
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) | (F) Pump (RD) | (E) Motor (FD) | (E) Motor (RD) | (F) Motor (FD) | (F) Motor (RD) | (E) Pump (FD) | (E) Pump (RD)

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.



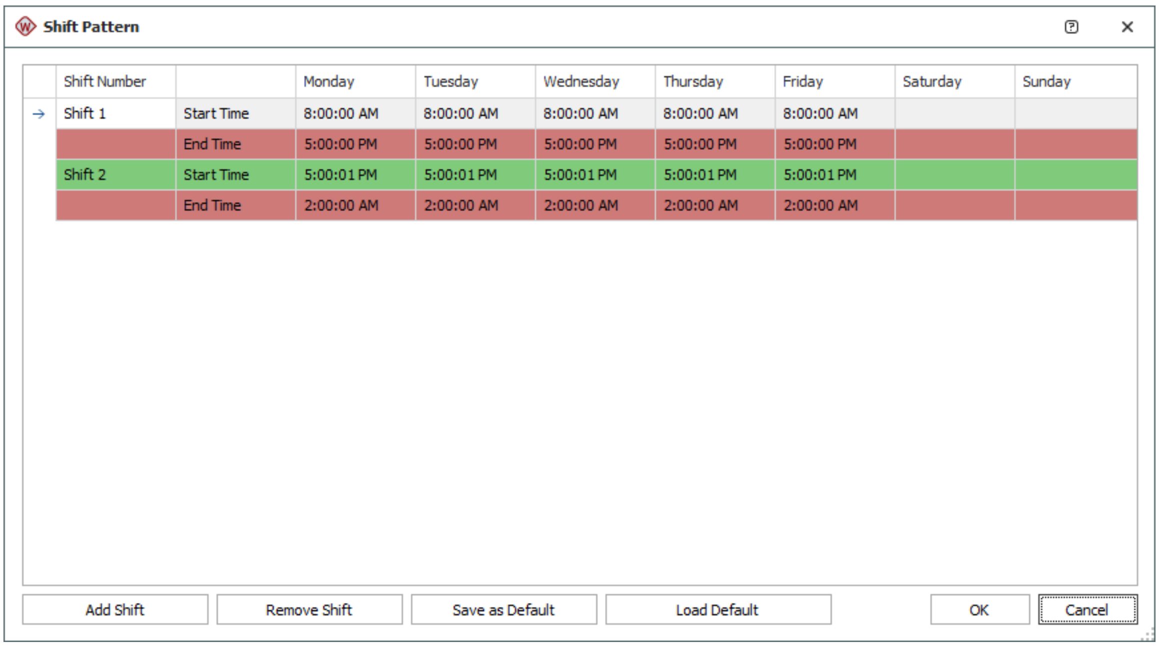
Folio Tools

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 a piece 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.

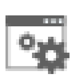


The buttons at the lower left side of the window allow you to add or remove shifts. To save the current shift pattern as the default for new event log folios, click **Save as Default**. To load the default shift pattern settings for any existing event log folio, click **Load Defaults**.

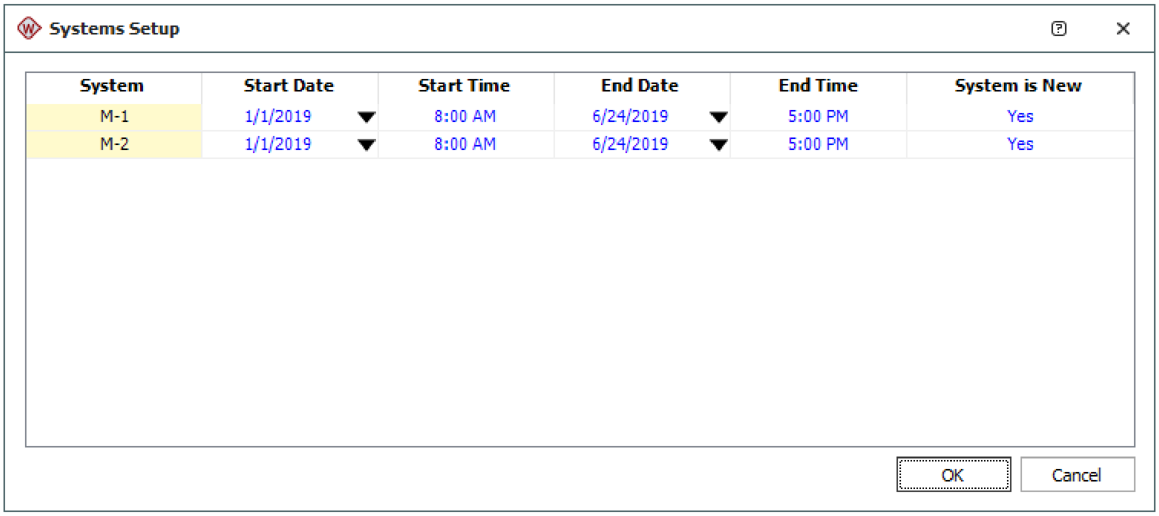
SYSTEMS SETUP

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.

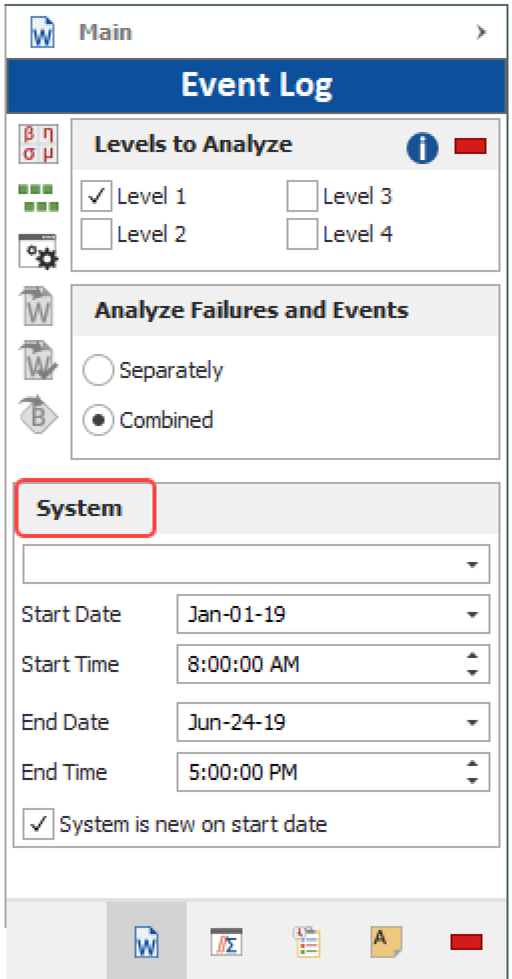


The following picture shows an example of a setup for two machines, M-1 and M-2.



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.

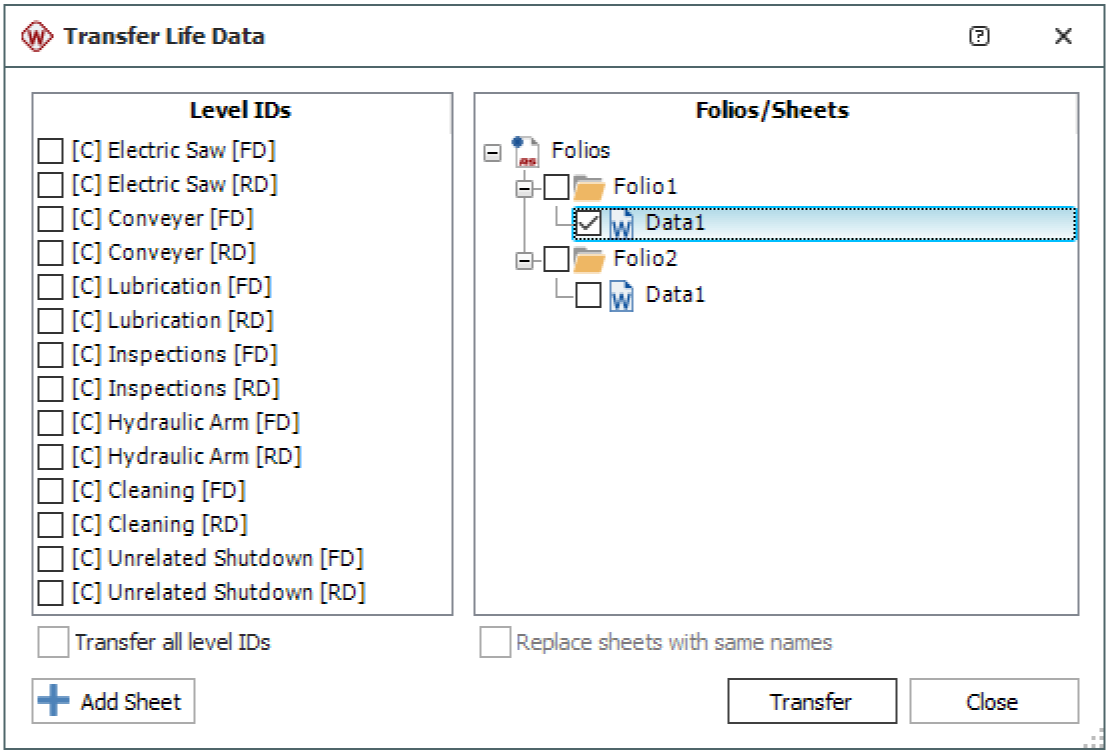
Note that the information you provide in the Systems Setup window will be automatically copied to the **System** area on the Main page of the control panel and vice versa.



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++ life data 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 single 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** heading at the top of the panel and click **Add Folio**.
3. Click **Transfer** to transfer the data.
4. Repeat the steps above for each data set you wish to transfer.

To transfer all data sets at once:

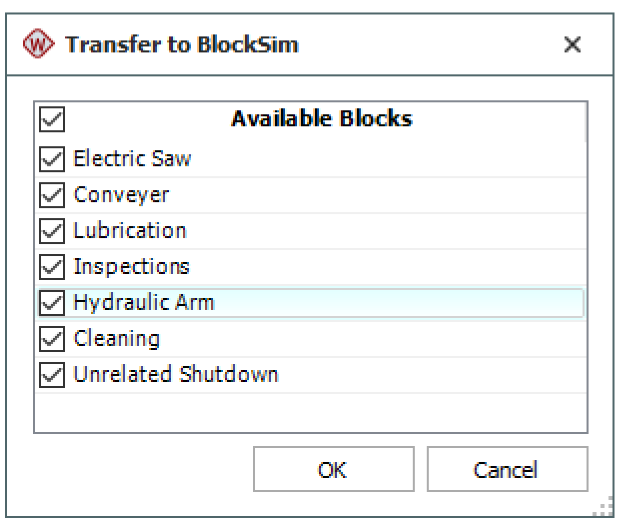
1. Select the **Transfer all level IDs** check box.
2. On the right panel, select the check box for the folio where the data will be transferred. Note that the utility will create new data sheets in the folio for each data set. If the folio already contains data sheets that have the same name as the data to be transferred, you can choose to overwrite the data in those sheets by selecting the **Replace sheets with same names** check box.If you want to create a new folio instead, click **Add Folio**.
3. Click **Transfer** to transfer the data.

TRANSFER TO BLOCKSIM

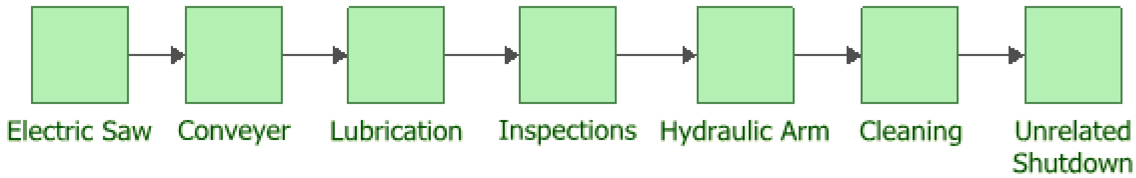
If you have ReliaSoft's system availability and maintainability analysis software, BlockSim, activated on your computer, you can export the analysis results as blocks in a new BlockSim simulation diagram. Each block represents a 1st level component, where the block's reliability model is based on the component's failure distribution and the corrective task is based on the component's repair distribution. (See Universal Reliability Definition (URD).)

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

Life-Stress Data Folio

IN THIS CHAPTER

For life-stress data 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 Accelerated Life Testing (ALTA) life-stress data 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. 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.

Note: Life-stress data folios are available only if supported by your license. For more information, contact us.

Life-Stress Data Folio Setup

For life-stress data only.

The Accelerated Life Testing (ALTA) life-stress data folio contains the tools for performing an accelerated life testing data analysis.

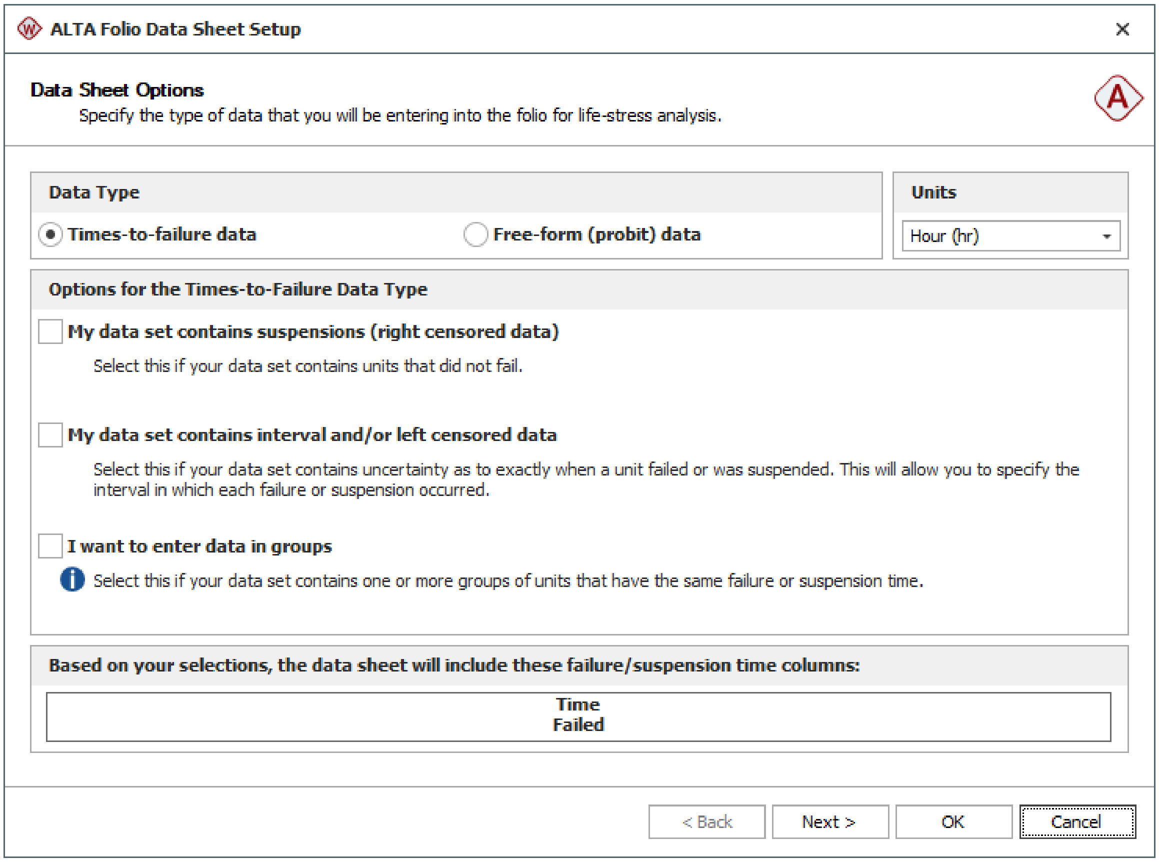
To add an life-stress data folio to a project, choose **Home > Insert > Life-Stress Data**, or right-click the **Life-Stress Data** folder in the current project explorer and choose **Add Life-Stress Data** on the shortcut menu.

A setup window will prompt you to select a data type to use in the folio, the stresses to use and a unit appropriate for your data (the unit may be measured in mileage, distance, weight, etc.; see Unit Settings).

There are two types of data available: 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. (See Accelerated Life Testing Data Types.)

Any of the times-to-failure data types can also be entered as grouped data. The ReliaWiki resources portal has more information on grouped data analysis at: http://www.reliawiki.org/index.php/Grouped_Data_Parameter_Estimation.

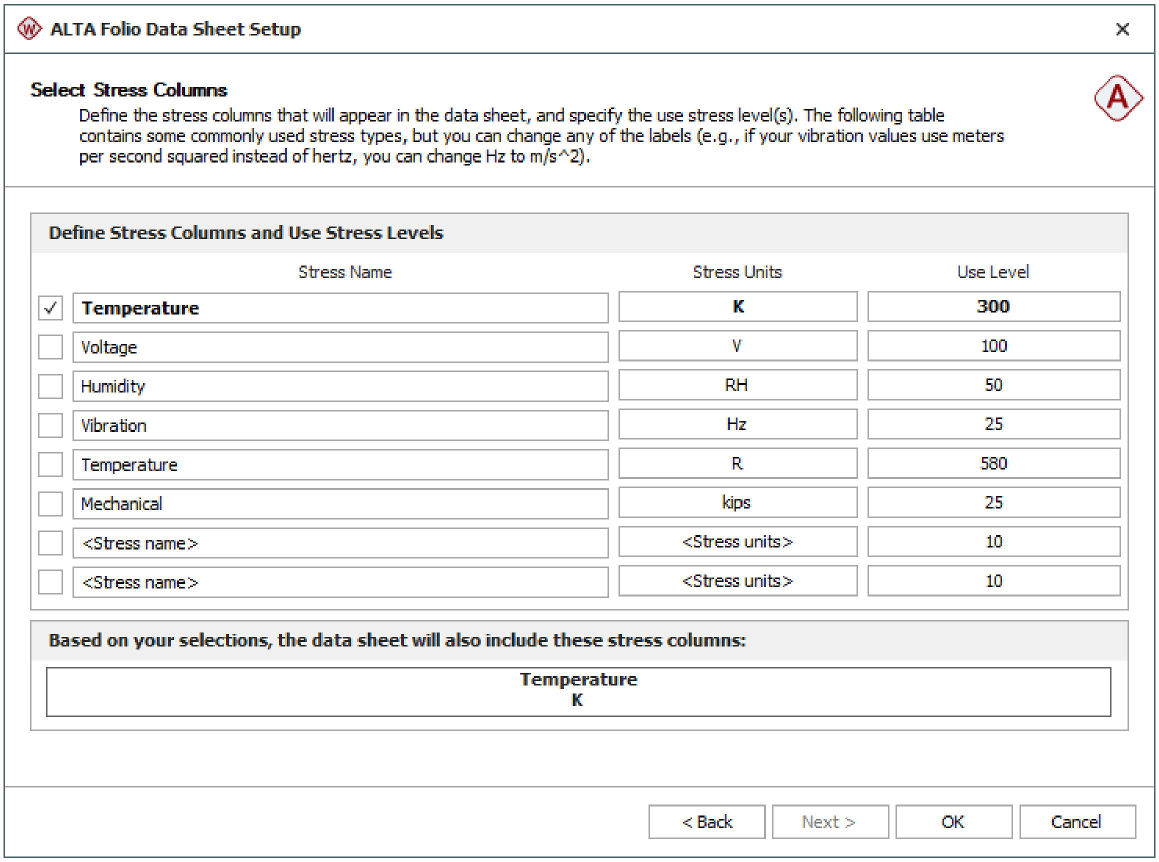
The first page of the Data Sheet Setup window is shown next.



Note that when you select a data type, the preview table at the bottom of the setup window will show you the names of the data entry columns that will appear in the data sheet.

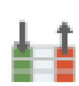
If you are creating a new folio, you will also need to specify the time units appropriate for your data. Units may be measured in time, distance, weight, etc. The appropriate columns in the data sheet will be automatically configured for the units you select. If you later wish to change the time units of an existing data sheet, click the **Change Units** icon on the Main page of the control panel of the data sheet.

If you wish to use the same stress column and stress level settings that were used when you last set up an Accelerated Life Testing life-stress data 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 **Stress Name** or **Stress Units**. These fields are merely for adding text to the headings 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 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.

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 Celsius or Fahrenheit to absolute units, choose **Life-Stress Data > Options > Convert Stress Values**.

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.

Accelerated Life Testing Data Types

For life-stress data only.

For Accelerated Life Testing analysis, there are two types of life-stress 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.

In a life-stress data folio, the data type you select determines the data entry columns that appear in the data sheet. The following sections describe each data type and demonstrate how to enter data in a life-stress data folio data sheet.

Tip: You can change the data entry columns for an existing data sheet anytime by choosing **Life-Stress Data > Format and View > Alter Data Type**.

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 6 units at two stress levels 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 life-stress data folio data sheet where all the units in the sample failed. The exact time of each failure is recorded.

	Time Failed (hr)	Temperature K ✓
1	248	406
2	456	406
3	528	406
4	164	416
5	176	416
6	289	416

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 in 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)	Temperature K ✓
1	2	24	408
2	10	48	408
3	1	96	408
4	1	100	393
5	2	120	393
6	5	150	393

Times-to-Failure with Suspensions (Right Censored Data)

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

Times-to-Failure with Interval and Left Censored Data

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

Free-Form (Probit) Data

Starting in Version 2019, 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 items that failed after X number of hours at a given stress.

To use this data type, select the **Free-form (probit) data** check box in 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) at specified stress levels of a product.

	X-Axis value (hr)	Y-Axis value,(%)	Temperature K
1	100	28.9	393
2	200	31.38	393
3	300	49.99	393
4	400	15.04	408
5	500	20.48	408
6	600	25.25	408

Life-Stress Data Folio Control Panel

For life-stress data only.

The Accelerated Life Testing (ALTA) life-stress data 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 topic focuses on the Main page of the life-stress data folio 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 Control Panels.

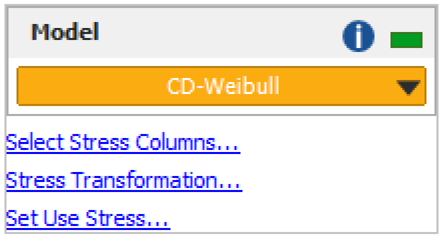
Control Panel Main Page

The Main page of the life-stress data folio control panel includes the following settings:

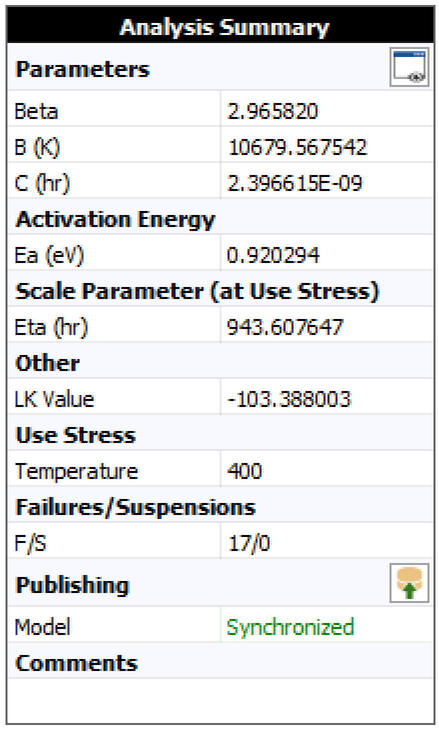
- The **Model** drop-down list allows you to select the life-stress relationship and life distribution that will be used to extrapolate failure behavior at normal operating conditions. The selected life distribution is fitted to the data obtained at each accelerated stress level used in the test, and then the selected life-stress relationship is used to extrapolate the probability density function (*pdf*) at the use stress level from the *pdf*s at the accelerated levels.There are five life-stress relationships to choose from in Accelerated Life Testing Standard and four additional relationships in Accelerated Life Testing PRO. For both versions of Accelerated Life Testing, there are three life distributions to choose from. The choice of the life-stress relationship depends on various factors, including the types and number of stresses used in the test. 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.

Tip: If you are not certain which life distribution would provide the best fit for your data, you can use the Accelerated Life Testing Distribution Wizard to guide you through the selection process.

- Life distributions: 2-parameter Weibull, lognormal (not available with the proportional hazards relationship) and 1-parameter exponential.
- Life-stress relationships in Accelerated Life Testing Standard:
  - **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).
- Additional life-stress relationships in Accelerated Life Testing PRO:
  - **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. This model is not available for free-form data.
- Directly underneath the Model drop-down list are options for selecting the stress columns and, when applicable, the stress transformations that will be used in the analysis, and defining the use stress level for every stress that is selected to be used in the analysis.



- The **Analysis Summary** area displays the calculated parameter values and other values for your selected model. As an example, the following picture shows the analysis results for a particular data set.



Depending on your application settings, additional information may also be displayed:

- The **Ea** (activation energy) value is displayed when temperature-related models, such as Arrhenius, Eyring, generalized Eyring, temperature-humidity or temperature-nonthermal, are used. It represents the effect of temperature; a large value 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. 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 use stress level of every stress that was used in the analysis.
- The number of failures and suspensions in the data set.
- The model status (as shown in the previous picture) is displayed if the result of the analysis is associated with a published model. The status "Synchronized" indicates that the published model reflects the latest results from the analysis. If the 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." (See Publishing Models.)

Additional Results

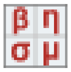
The following tools may also be used in conjunction with your own engineering knowledge about the model in order to further evaluate the fit of the model to the data set.

- 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.
- 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.
- **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.

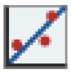
  You can visualize the confidence bounds by using plots, or you can view the calculated values of the confidence bounds by using the Quick Calculation Pad.

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.

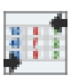


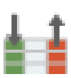
Calculate estimates the parameters of the chosen model, based on the current data set and the specified analysis settings. This tool is also available by choosing **Life-Stress Data > Analysis > Calculate**.


Plot creates a new sheet in the folio that provides a choice of applicable plot types. For life-stress data 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 **Life-Stress Data > Analysis > Plot**.

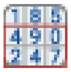
QCP opens the Accelerated Life Testing Quick Calculation Pad, which allows you to calculate results, such as the mean life and the probability of failure, based on the currently active data sheet. This tool is also available by choosing **Life-Stress Data > Analysis > Quick Calculation Pad**.


Distribution Wizard opens the Accelerated Life Testing Distribution Wizard, which helps you select the life distribution that will best fit your data. This tool is also available by choosing **Life-Stress Data > Analysis > Distribution Wizard**.


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 **Life-Stress Data > Format and View > Alter Data Type**.


Add or Remove Columns allows you to add/remove columns for stress levels and unit IDs.


Change Units opens the Change Units window, which allows you to change the units for the values in the current data sheet.

 **Auto Group Data** opens the Auto Group Data window, which allows you to group together identical data points. Grouping data significantly speeds up calculations when you have a very large data set. This tool is also available by choosing **Life-Stress Data > Format and View > Auto Group Data**.

 **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.

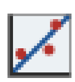
 **Likelihood Ratio Test** opens the Likelihood Ratio Test tool, which allows you to assess the assumption of a common shape parameter across all the stress levels used in the test. This assumption underlies the use of any model that includes the Weibull or lognormal life distribution. You can also open this tool by choosing **Life-Stress Data > Options > Likelihood Ratio Test**.

 **Transfer Data to Life Data Folio** allows you to transfer data from the life-stress data folio to a life data folio.

Accelerated Life Testing Plots

For life-stress data only.

To create a plot in a life-stress data folio, choose **Life-Stress Data > Analysis > Plot** or click the icon on the Main page of the control panel.



Tip: You can add additional plot sheets to the folio by choosing **Life-Stress Data > Folio Sheets > Insert Additional Plot**. The additional sheets can function as overlay plots to display results from multiple data sheets in the current folio on a single plot.

The following is a description of the different types of plots that can be created in a life-stress data folio. (For general information on working with plots, see Plot Utilities.)

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 **Mean Remaining Life** plot show the expected survival time of a product given the current age of the product.
- The **Stress Profile** plot is available only when the analysis uses stress profiles. The plot shows how the stress values for each profile changed over time. The plot also shows the data points from the test and the stress level at which they were obtained. If the test includes multiple stresses, click **Set Use Stress** to select which use stress values to show in the plot.

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 *pdf*s, 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 *pdf*s 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 Specifying Life Lines.) The life-stress relationship is linearized whenever possible.

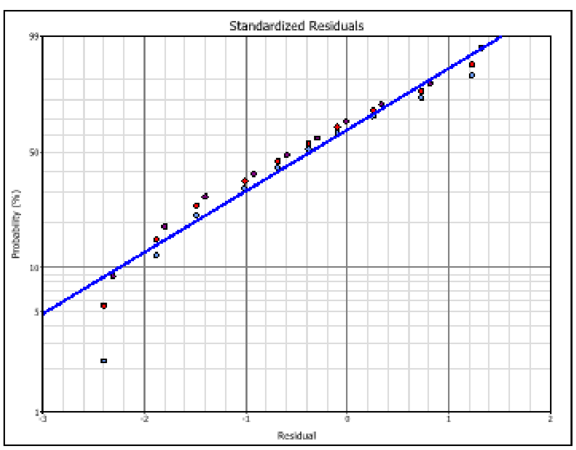
- The **Std vs. Stress** plot shows the standard deviation of failure time as a function of stress level, thus providing information about the spread of data at every given stress level on the x-axis.

- 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.

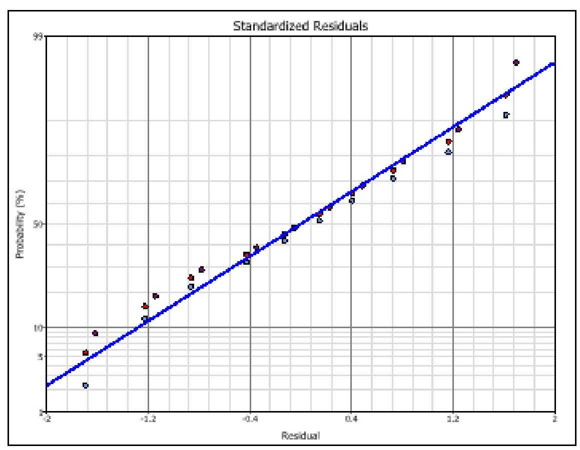
The following plots are residual plots. In these plots, a residual value for each data point is displayed. As a result, these plots are useful for assessing model assumptions, revealing inadequacies in the model and revealing any extreme observations. The ReliaWiki resource portal has more information 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.

Specifying Life Lines

For life-stress data only.

You can specify additional life lines to be plotted on life vs. stress plots in Accelerated Life Testing 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.

Select Stress Columns

For life-stress data only.

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 Main page 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.

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.

Tip: You can also choose which stress columns to include in the analysis by selecting the check boxes in the column headings of the data sheet.

Stress Transformation

For life-stress data only.

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 is selected to be used in the analysis. To access this window, click **Stress Transformation** directly underneath the **Model** area of the Main page of the control panel.

The **Transformation** drop-down list provides three options for how the stress will be transformed within the model:

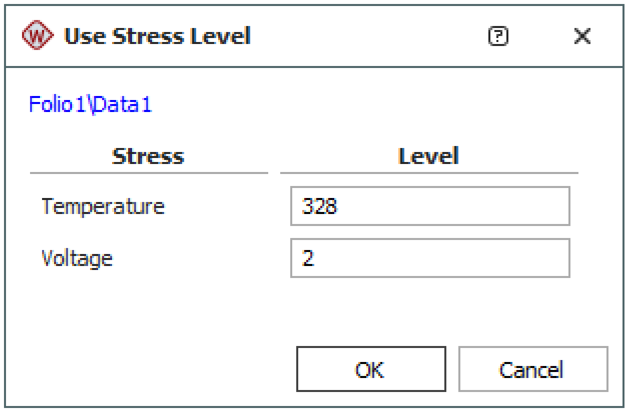
- Use the **None X=V** transformation for stress types associated with the exponential life-stress relationship (LSR). This transformation is commonly used for indicator variables (e.g., 0 = on/off and 1 = continuous operation).
- Use the **Reciprocal X=1/V** transformation for stress types associated 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.

Use Stress Level

For life-stress data only.

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 life-stress data folio. To access the window, click **Set Use Stress** directly underneath the **Model** area of the Main page 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 profiles to describe time-dependent stress conditions that the product will experience during normal use.

QCP Calculations for Accelerated Life Testing

For life-stress data only.

Accelerated Life Testing 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 Quick Calculation Pad (QCP).

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 duration 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 duration of the interval in the **Mission Additional Time** field.

For example, a product may have a 10% probability of failure for 3 years if 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.

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 life-stress data 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 Accelerated Life Testing life-stress data 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.

For more complex repairable system analyses, see ReliaSoft's BlockSim.

Mean Remaining Life

Calculates the expected remaining life given that the product, component or system has survived to time t at the stress level specified in the **Stress** field. Enter the time at which you wish to calculate the mean remaining life in the **Mission End Time** field.

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 **Accelerated 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.

Single-Stress Example

For life-stress data only.

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

Create a Life-Stress Data Folio

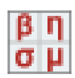
1. Choose **Home > Insert > Life-Stress Data**. On the first page of the setup window, make sure only the **My data set contains suspensions** check box is selected. Then, select **Hour (hr)** from the **Units** drop-down list and click **Next**.
2. On the second page of the setup window, select **Temperature** as the stress type. Set the use level temperature to **323** with **K** as the unit. Click **OK** to create the folio.
3. Enter the given data in the 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.

Analyze the Data Set

4. On the Main page of the folio's control panel, select the **Arrhenius-Weibull** model and click the **Calculate** icon. The parameters of the model will be automatically computed and displayed in the **Analysis Summary** area of the control panel.

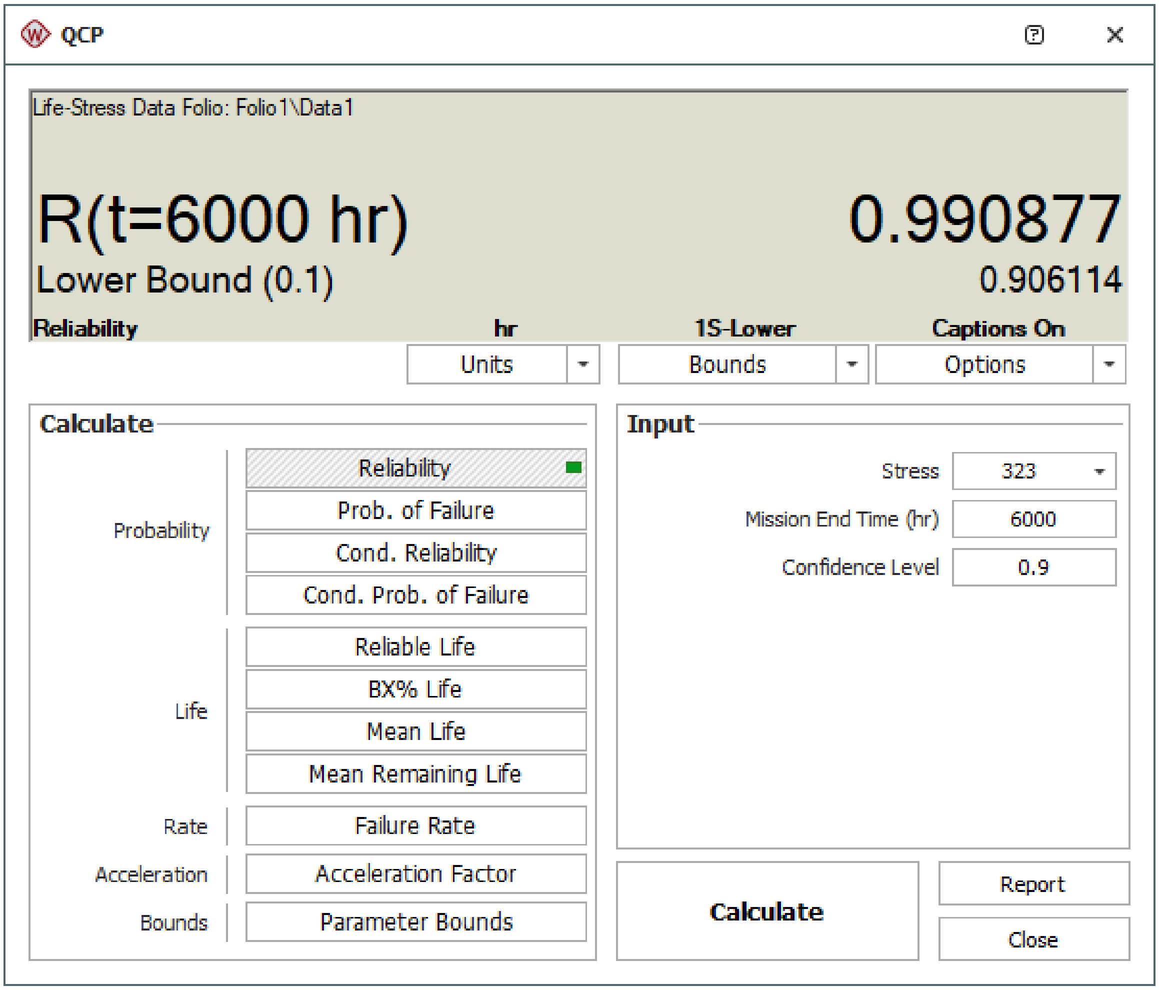


5. To estimate the reliability of the device at 6,000 hours, click the **Quick Calculation Pad** icon on the control panel.

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**

7. 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.



Two-Stress Example

For life-stress data only.

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

Create a Life-Stress Data Folio

1. Choose **Home > Insert > Life-Stress Data**. 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. Select **Hour (hr)** from the **Units** drop-down list and click **Next**.

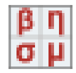
2. On the second page of the setup window, select **Temperature** and **Voltage** as the stress types. Set the use level temperature to **328** with **K** as the unit, and set the use level voltage to **2** with **V** as the unit. Click **OK** to create the folio.

3. In the data sheet of your new folio, select the check boxes inside the **Temperature** and **Voltage** column headings. Both stresses will now be used in the calculation for the selected model.

4. Enter the given data into the data sheet.

Analyze the Data Set

5. 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) and click the **Calculate** icon. The parameters of the model will be automatically computed and displayed in the **Analysis Summary** area of the control panel.

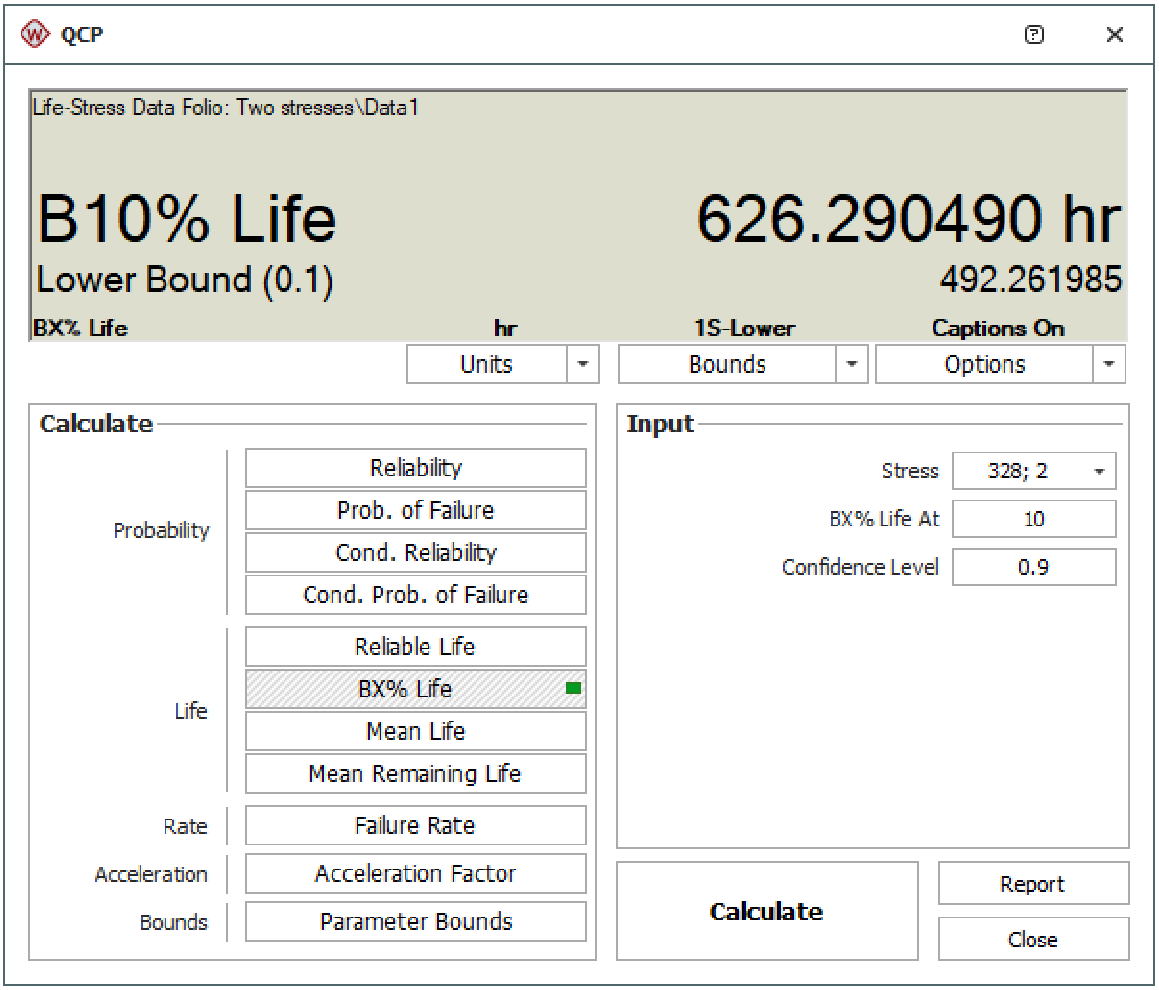


6. To obtain the two-sided 90% confidence bounds on the product's B10 life, click the **Quick Calculation Pad** icon on the control panel.

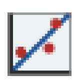
7. In the QCP, choose to calculate the **BX% Life** with **Two-Sided Bounds**. Select **Hour** for the time units and then make the following inputs:

   - Stress = **328; 2**
   - BX% Life At = **10**
   - Confidence Level = **0.9**

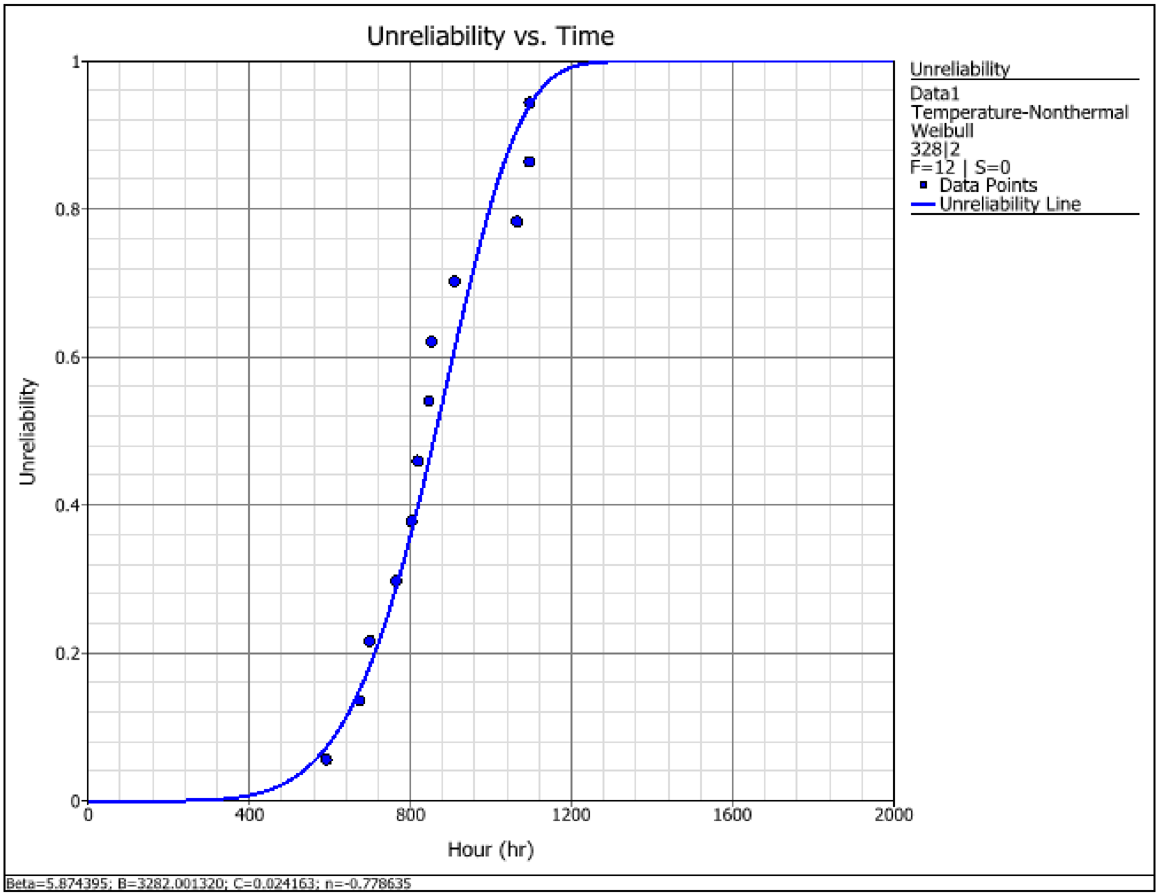
8. Click **Calculate** to display the results, as shown next. The results show, with 90% confidence, that the time at which the product will have an unreliability of 10% is between 459.7681 and 853.1253 hours.



9. To view a plot of the result, click the **Plot** icon on the Main page of the control panel.



10. On the Plot page, select **Unreliability vs. Time** from the **Plot Type** drop-down list.



Time-Dependent Stress Profiles

For life-stress data only.

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.

You can create profiles, which are resources that allow you to represent how stress levels will vary with time. Profiles are designed to be used with the cumulative damage model, which is available only in Accelerated Life Testing PRO. When you select a cumulative damage model in the life-stress data folio, all of the profiles validated in your project become available to describe use level stress conditions and stress conditions during testing.

Note: If you are an Accelerated Life Testing Standard user who needs to use time-dependent stress profiles in your analysis, please contact ReliaSoft for information about upgrading to Accelerated Life Testing PRO.

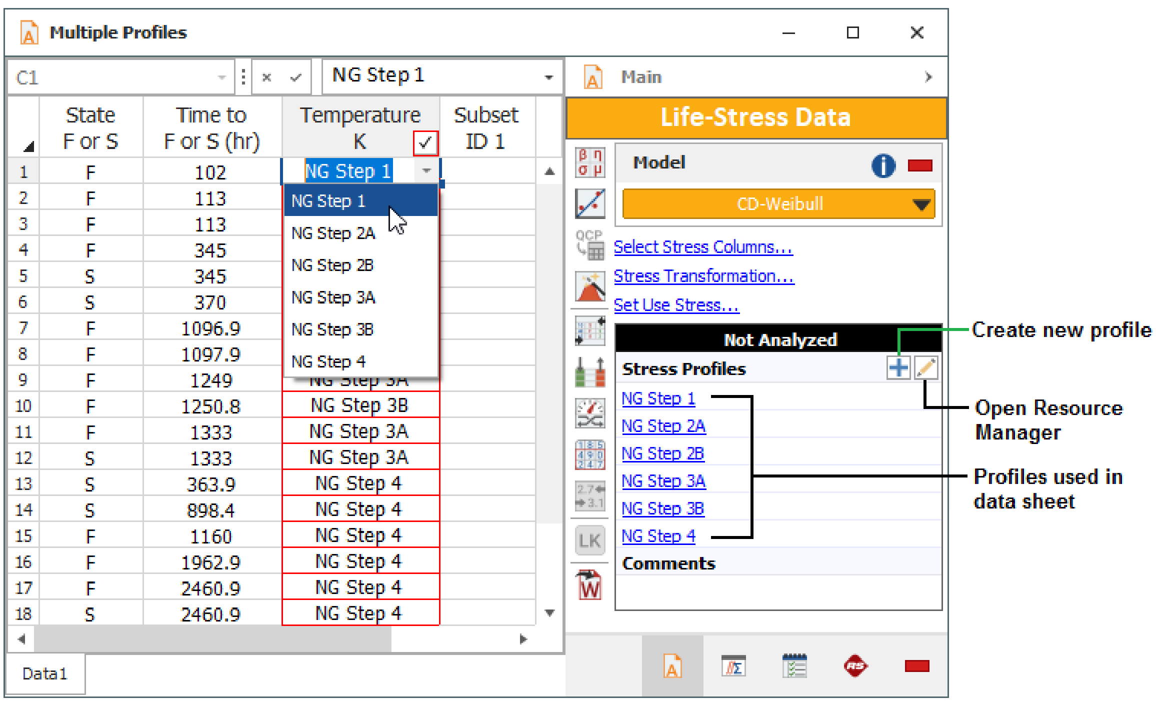
Stress Profiles in the Life-Stress Data Folio

There are two ways you can use profiles in a life-stress data folio. Note that:

- Because stress and time units are not defined in a profile, it is important to apply profiles only to folios that are intended to use the same stress and time units.
- If you update a profile that is associated with a life-stress data folio, the calculations in the folio will not be updated automatically. To update the calculations, click the **Calculate** icon on the folio's control panel.

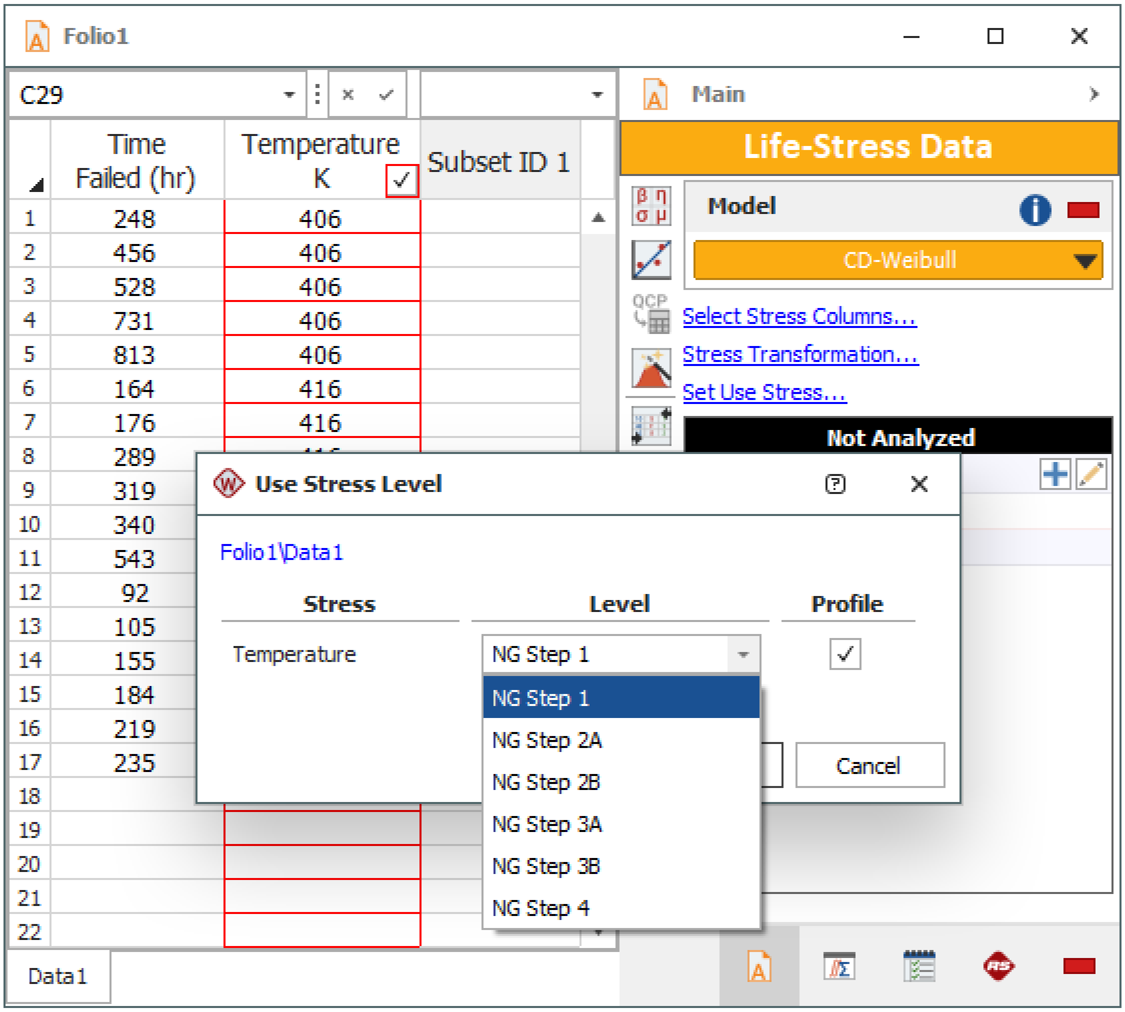
Use Profiles to Describe Stress Conditions

Select the cumulative damage life-stress model on the folio's control panel. To assign a stress profile to a data point, double-click a cell in the stress column, then open the drop-down list to see all validated stress profiles in the current project. The control panel will show a list of stress profiles used in the data sheet.



Use Profiles to Describe the Use Stress Condition

Select the cumulative damage life-stress model on the control panel, then click the **Set Use Stress** link. In the Use Stress Level window, select the **Profile** check box for all applicable stresses. A drop-down list containing all the validated stress profiles in the project appears. Note that when you use a stress profile in this way, some plots (e.g., the use level probability plot) will not be available.



Stress Profile Example

For life-stress data only.

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-dependent 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-dependent
310	Time-dependent
330	Time-dependent
352	Time-dependent
360	Time-dependent
366	Time-dependent
371	Time-dependent
374	Time-dependent
378	Time-dependent
381	Time-dependent
385	Time-dependent

Create a Life-Stress Data Folio and Stress Profile

1. Choose **Home > Insert > Life-Stress Data**. 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. Click **OK** to close the setup window and create the new life-stress data folio.

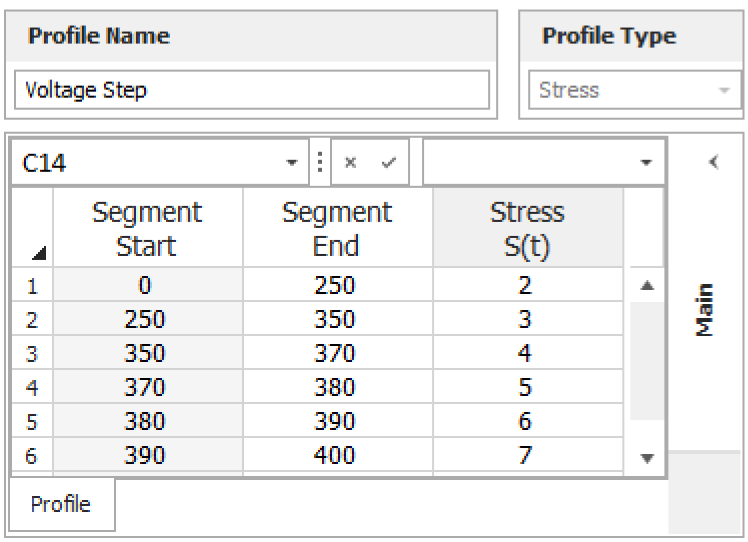
3. Enter the failure times into the data sheet.

4. On the control panel of the data sheet, select **CD-Weibull**. Click **Stress Transformation** and apply the **Logarithmic** transformation to the stress.

5. To create a stress profile:

   a. Click the icon on the control panel.

   b. In the Profile window, rename the profile to "Voltage Step." Enter the 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

   c. On the control panel, select **Continue with last value**. Click the **Validate Stress Profile** icon to validate the information in your new profile. Finally, click the **Plot** icon to view a stress vs. time plot of the profile.

   d. Click **OK** to save the profile with the project and close the window.

6. To assign the stress profile to a failure time, double-click the corresponding cell in the Voltage column of the data sheet, then click the drop-down arrow and choose **Voltage Step** from the

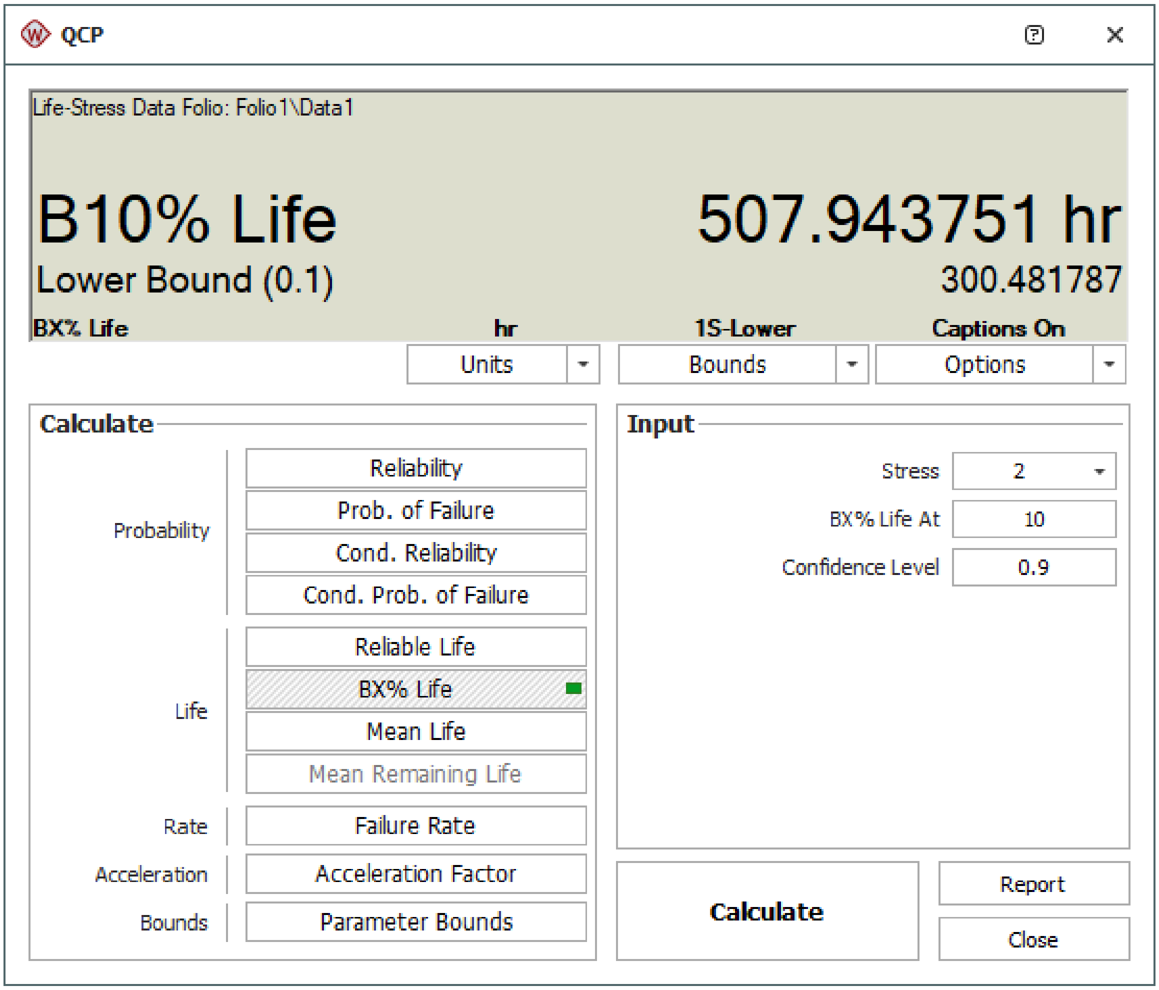
list. After you have filled out the data sheet, it should appear as shown next.

	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

Analyze the Data Set

1. Click the **Calculate** icon on the folio's control panel. The parameters of the model will be automatically computed and displayed in the **Analysis Summary** area of the control panel.
2. To estimate the B10 life of the device, choose **Life-Stress Data > Analysis > Quick Calculation Pad** or click the icon on the control panel.

3. 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**
4. Click **Calculate** to display the results. 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.



Tools

Accelerated Life Testing Distribution Wizard

For life-stress data only.

The Distribution Wizard for Accelerated Life Testing life-stress data 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 **Life-Stress Data > 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.

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.

Likelihood Ratio Test

For life-stress data only.

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 Accelerated Life Testing that include the Weibull or lognormal distribution, except the cumulative damage models in Accelerated Life Testing PRO.

To use the Likelihood Ratio Test tool, first calculate the parameters in your life-stress data folio. After the shape parameter is calculated in the folio, open the tool by choosing **Life-Stress Data > Options > Likelihood Ratio Test** or clicking its icon on the Main page of the 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.

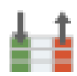
Finally, click **Calculate** to view the results of the test. The **Results** area of the window displays the calculated likelihood ratio test statistic, *T*, and chi-squared value. If *T* is not greater than the chi-squared value, then the estimated shape parameters do not differ significantly at the specified level. A written report of the results also appears in this area.

The Likelihood Values tab 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.

Add or Remove Columns

For life-stress data only.

To add or remove additional stress columns and Subset ID columns to a data sheet, choose **Life-Stress Data > Format and View > Add/Remove Columns**.



- **Add Subset ID Column** adds a new Subset ID column.
- **Add Stress Column** adds a new stress column. You can add up to 8 stress columns to the data sheet.

To remove a stress or subset ID column from the list, clear the check box next to the column's name.

Tip: If you wish to later change the heading of any stress or subset ID column, simply right-click the column heading and choose **Rename Stress/Rename Column**.

Convert Stress Values

For life-stress data only.

In life-stress data folios, you can easily convert all the values in a specified stress column to another value.

First, select the stress column that contains the values you want to convert by clicking the heading of that column or selecting any cell in that column. All of the values in the selected column will be converted. For example, if you select one cell and convert the contained value, all the other values in that column will be converted as well.

Then choose **Life-Stress Data > Options > Convert Stress Values**.

Select how you want to convert the values:

- Multiply or divide the values in the column by a specified value. Use the input field to enter the value that you wish to have each stress value multiplied or divided by.
- Celsius to Kelvin
- Fahrenheit to Kelvin
- Celsius to Rankine
- Fahrenheit to Rankine

If you want to move the original, unconverted values into a subset ID column in the data sheet for future reference, select the **Put current values in Subset ID** check box and then use the drop-down list to choose the subset ID column to move the values into. *If there is already information in the selected 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. The second subset ID column will be available in the drop-down list when you next open the Convert Stress Values window.

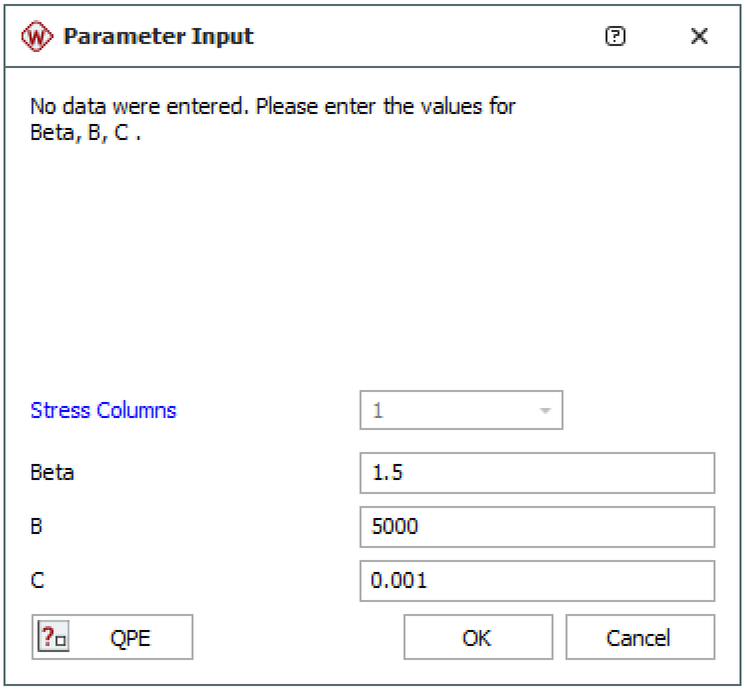
Calculations with Insufficient Data Entered

For life-stress data only.

The Parameter Input window will appear when you attempt to calculate a life-stress data sheet that does not contain enough data to fit a model.

No Data Entered

If you attempt to calculate a data sheet in a life-stress data 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 Accelerated Life Testing 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.

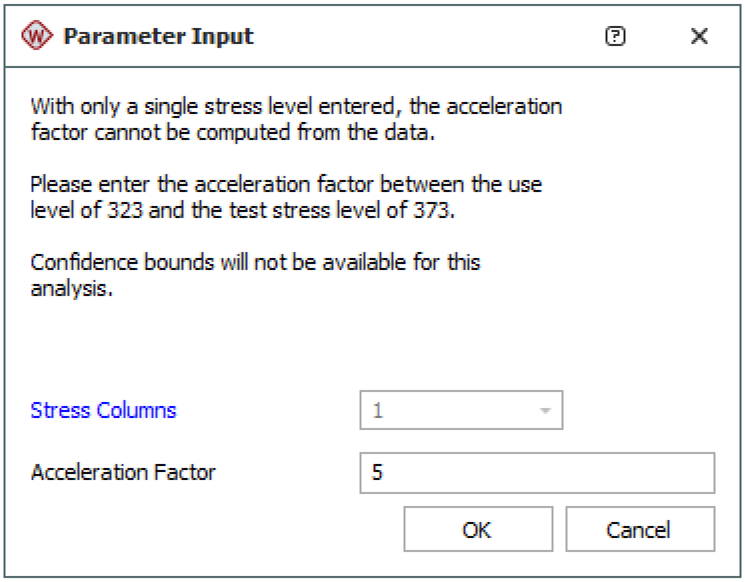


After you enter the parameters and click **OK**, the data sheet will be updated automatically with the specified model. Any plots or QCP results you produce from this data sheet will be based on this model.

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.

Not Enough Stress Levels

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 confidence bounds.

Transfer Data to Life Data Folio

For life-stress data only.

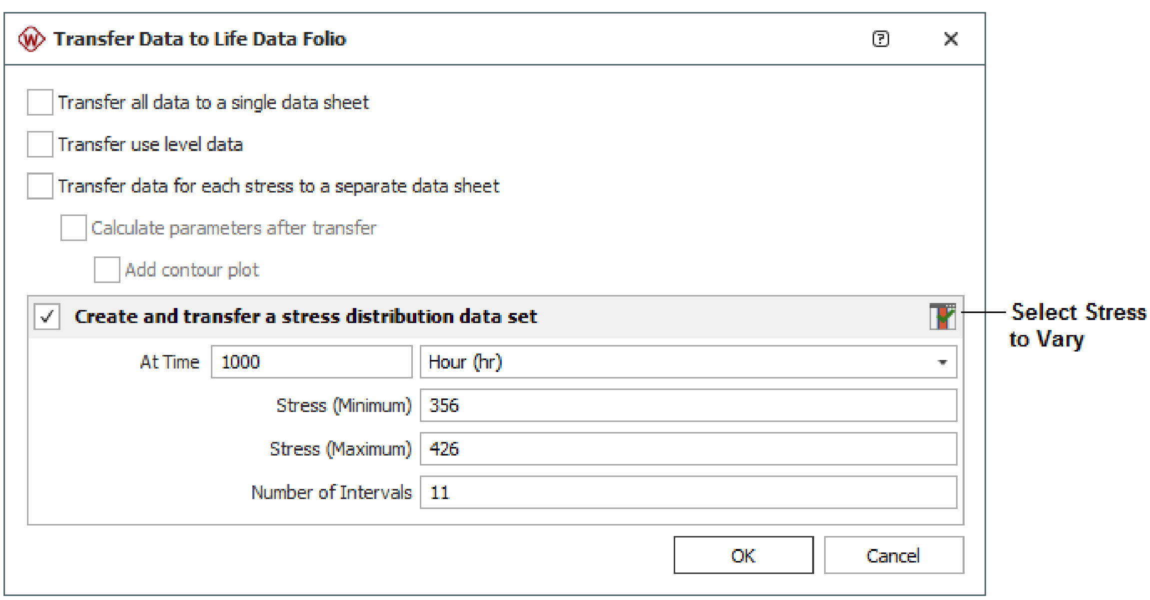
You can transfer data in a life-stress data folio to a life data folio in order to perform life data analysis on your accelerated test data.

To transfer the data, click the **Transfer Data to Life Data Folio** icon on the control panel.

Then select how you want to transfer the data:

- **Transfer all data to a single data sheet** copies the values from the stress columns to the subset ID column of the new life data folio data sheet. In multiple-stress situations, a stress level is a combination of stress values. (If desired, you can later use the Batch Auto Run tool to extract data subsets from the data sheet.)

- **Transfer use level data** creates a free-form data sheet within the new life data folio and copies the probability of failure over time values (Y-Axis Values) at the specified use stress levels (X-Axis values) into it.
- **Transfer data for each stress to a separate data sheet** groups the accelerated data set according to stress level and then places each group into its own data sheet in the new Weibull++ life data folio. In multiple-stress situations, a stress level is a combination of stress values. There are two additional options:
  - **Calculate parameters after transfer** automatically calculates each data sheet using the life distribution from your selected model.
  - **Add contour plot** creates a contour plot that compares the calculated parameters for each data sheet, allowing you to determine, for example, whether the shape parameter estimates for the data obtained at different stress levels differ significantly at a specified confidence level. (This option is not available for free-form data.)
- **Create and transfer a stress distribution data set** calculates the product's unreliability at the specified time and at different stress intervals. The stress intervals 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 life data folio. In multi-stress situations, you can click the **Select Stress to Vary** icon 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.



In the free-form data sheet, the X-axis represents the stress intervals and the Y-axis represents the unreliability (as a percentage) for each interval, as shown in the following 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. If the specified time 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.

	X-Axis value (hr)	Y-Axis value,(%)	Subset ID 1
1	356	31.57032135	
2	363	37.74333041	
3	370	44.40309185	
4	377	51.39444263	
5	384	58.51960148	
6	391	65.55099682	
7	398	72.25104889	
8	405	78.3968227	
9	412	83.80570167	
10	419	88.35707548	
11	426	92.00511787	

Life Data and Life-Stress Data Analysis Folios

IN THIS CHAPTER

Degradation Analysis Folio

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.

Weibull++ offers two types of degradation analysis folios:

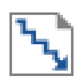
- **Nondestructive degradation analysis** folios are for cases where it is possible to directly measure the degradation of a physical or performance characteristic over time (e.g., depth of tire tread, voltage of battery, propagation of crack size, etc.). In this case, you take multiple measurements per test unit over time. The folio extrapolates the times at which the amount or level of degradation is serious enough that the unit is considered to be failed, and then automatically performs life data analysis on the extrapolated failure times.
- **Destructive degradation analysis** folios are for cases where it might not be possible to measure the degradation without invasive or destructive techniques that would directly affect the subsequent performance of a product (e.g., corrosion in a chemical container, strength measurement of adhesive bond, etc.). Only one measurement is taken per test unit and multiple samples are required at different points in time. Unlike the nondestructive degradation folio, which extrapolates the failures times, the destructive degradation analysis folio fits the measurements directly to the chosen degradation model and life distribution, and then uses the result to predict future failure behavior.

The ReliaWiki resources portal has more information on degradation analysis at: http://www.reliawiki.org/index.php/Degradation_Data_Analysis.

What's Changed? Starting in Version 2019, you can now use the Inspection View with nondestructive folio data sheets.

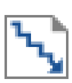
Degradation Analysis Folio Setup

To add a degradation analysis folio, choose **Home > Insert > Degradation**, or right-click the **Degradation** folder in the current project explorer and choose **Add Degradation** on the shortcut menu.



In the setup window, select the type of degradation analysis folio to create. If you have Accelerated Life Testing activated on your computer, all four options will be available. The Accelerated Life Testing degradation analysis folios are similar to life data except that accelerated stress levels are used during testing to further reduce the test time required to obtain the predicted failure times.

Weibull++

 Nondestructive Degradation

 Destructive Degradation

Accelerated Life Testing

 Accelerated Nondestructive Degradation

 Accelerated Destructive Degradation

Degradation Folio Data Sheet

There are two options for displaying data in degradation analysis folio data sheets:

- The Unit View is available for all analysis folio data sheets. This corresponds to the view that is used in earlier versions of the software.
- The Inspection View is available only for nondestructive folio data sheets.

For nondestructive degradation analysis folios, there are two options for the data sheet view: the Unit View (which corresponds to the view used in all older versions of the software) and the Inspection View (new in Version 2019). When you create the folio, you will be able to choose which view to use.

For destructive degradation analysis folios, only the Unit View is available.

Unit View

When using this view, you enter measurements on a separate row for each unit and inspection time:

- **Inspection Time** - the time at which the degradation was measured. Time may also be measured as the amount of usage the unit accumulates or as the factor that degrades the unit. In nondestructive degradation analyses, the level of degradation may be measured either continuously or at predetermined intervals of time.
- **Degradation** - the degradation measurement.
- **Unit ID** - the test unit being measured.
- **Stress** - for Accelerated Life Testing degradation analysis folios, the stress type and stress level.

The following example shows a Weibull++ nondestructive degradation analysis for two automobile tires, A and B. The inspection time is in mileage because of its direct relation to the wear on the tires.

	Inspection Time (mi)	Measurement	Unit ID
1	5000	6.1	A
2	10000	5	A
3	15000	4	A
4	20000	3.2	A
5	5000	5.9	B
6	10000	5.1	B
7	15000	4.3	B
8	20000	3.3	B

The following example shows an accelerated nondestructive degradation analysis for two LED units, A1 and A2. The light intensity is inspected every 50 hours and measured in milliamperes.

	Inspection Time (hr)	Measurement	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

Inspection View

When using this view, you enter measurements for multiple units at each inspection time.

- **Inspection Time** - the time at which the degradation was measured. Time may also be measured as the amount of usage the unit accumulates or as the factor that degrades the unit.
- **Unit ID(s)** - the specific test unit being measured, with separate columns for each test unit. (You can rename a column by right-clicking the column heading and choosing **Rename Column** on the shortcut menu.)

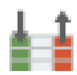
The following example shows a Weibull++ nondestructive degradation analysis for two automobile tires.

	Inspection Time (mi)	A	B
1	5000	6.1	5.9
2	10000	5	5.1
3	15000	4	4.3
4	20000	3.2	3.3

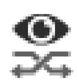
The following example shows a nondestructive degradation analysis for four LED units.

	Inspection Time (hr)	Amps mA ✓	A1	A2	A3	A4
1	100	35	86.6	82.1		
2	200	35	78.7	71.4		
3	300	35	76	65.4		
4	100	40			82.7	79.8
5	200	40			70.3	68.3
6	300	40			64	62.3

You can add additional unit IDs to the data sheet by clicking the **Add/Remove Columns** icon on the Main page of the control panel.



To switch between the views, choose **Degradation > Format and View > [Switch to Unit View/Switch to Inspection View]**.



Conditions

If you are switching from the Inspection View to the Unit View:

- For nondestructive degradation folios with life data analysis:
  - Any unit ID column that does not have a measurement value associated with it will become its own line in the Unit View.
  - Any inspection time that does not have a measurement value associated with it will become its own line in the Unit View.

- For nondestructive degradation folios with Accelerated Life Testing:
  - Any unit ID column that does not have a measurement value associated with it will become its own line in the Unit View.
  - Any unique combination of inspection time and stress column that does not have a measurement value associated with it will become its own line in the Unit View.

Control Panel Settings

The degradation folio control panel contains most of the tools you need to perform a degradation analysis. For more information about control panels in general, see Control Panels.

- **Degradation Model** allows you select the mathematical model that best describes the product's degradation over time. The available degradation models are: linear, exponential, power, logarithmic, Gompertz and Lloyd-Lipow. In Weibull++ nondestructive degradation analyses,you can also choose User-Defined. See User-Defined Degradation Models.

  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.

- **Critical Degradation** is 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.
- **Suspend After** is available for nondestructive degradation analyses only. It 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.

Nondestructive Degradation

- **Life Data Model/ALTA Model** contains the settings for performing life data/life-stress data analysis on the extrapolated failure times. These settings have no effect on the degradation analysis calculations; they affect only the life data/life-stress data analysis calculations.The settings in this area are identical to the settings on the control panel of a life data folio or life-stress data folio except for the **Use extrapolated intervals** check box, which allows you to extrapolate the failure/suspension times as interval censored data. Selecting this option calculates the intervals as bounds around the extrapolated data points.

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. See Publishing Models.

- **Degradation Results** displays the parameters of the chosen degradation model and the extrapolated failure/suspension times. Click the button to view the results in a worksheet.
- **Life Data/ALTA Results** displays the parameters of the life data/life-stress data model. Click the button to view the results in a worksheet.

Destructive Degradation

- **Measurement Distribution** allows you to select a distribution that represents the variability of a degradation measurement at a given time. The maximum likelihood estimation (MLE) method is used to estimate the distribution's parameters, and the Fisher matrix (FM) method is used to solve for the confidence bounds.
- **Analysis Summary** displays the calculated parameters and other values for the selected distribution. Click anywhere within the area to view the results in a worksheet.

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.

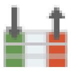
Calculate analyzes the data set. This command is also available by choosing **Degradation > Analysis > Calculate**.

Plot creates a new sheet in the folio that provides a choice of applicable plot types. This tool is also available by choosing **Degradation > Analysis > Plot**.

QCP opens the Quick Calculation Pad, which allows you to calculate life data/life-stress data analysis metrics, identical to the calculations you can perform in a life data/life-stress data 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**.

 **Distribution Wizard** opens the Distribution Wizard, which helps you select the distribution that best fits the data set. This tool is also available by choosing **Degradation > Analysis > Distribution Wizard**.

 **Add/Remove Columns**

- For life data nondestructive folio data sheets, this allows you to add or remove subset ID columns when using the Inspection View.
- For Accelerated Life Testing nondestructive data sheets, this allows you to add or remove columns for stress levels and, when using the Inspection View, add or remove subset ID columns.

This tool is also available by choosing **Degradation > Format and View > Add/Remove Columns**.

 **Change Units** opens the Change Units window, which allows you to change the units for the values in the current data sheet.

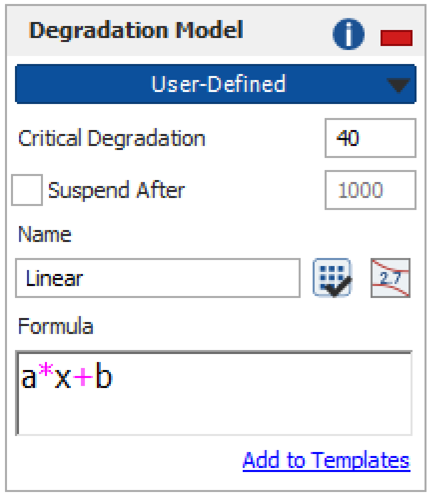
User-Defined Degradation Models

The life data nondestructive degradation analysis folio includes the option to predict failure times based on a user-defined degradation model. To use this option, you must provide the equation that defines the model, as well as specify ranges and initial guesses for the software to use when fitting the model.

Entering User-Defined Equations

To define a new degradation model, choose **User-Defined** from the drop-down list in the **Degradation Model** area. Then enter the model's **Name** and a **Formula** that represents the degradation over time. The formula can include parameters with any name (e.g., *b* or *beta*), but it should include *x* as the random variable for time.

As an example, the linear degradation model is defined next.

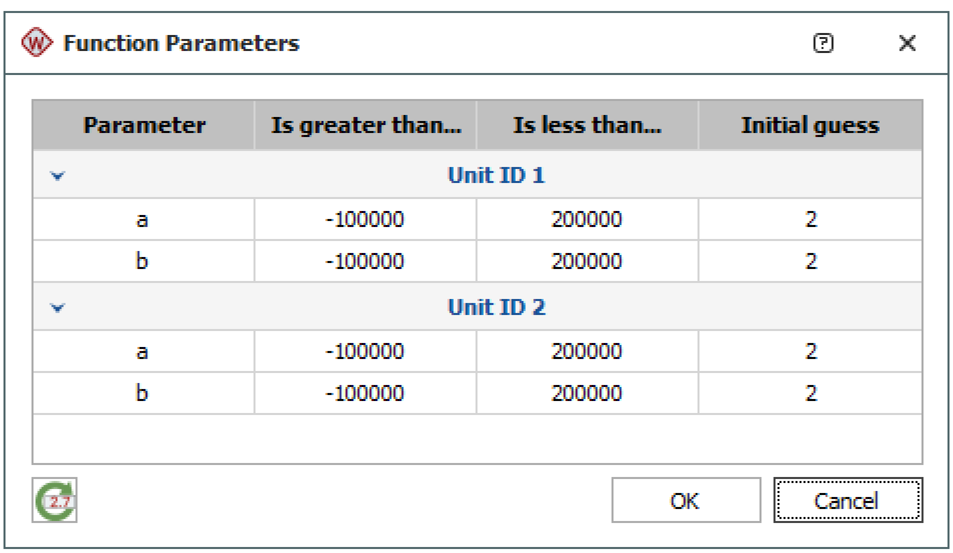


To save the equation in the project for later use, click the **Add to Templates** link.

To select an equation that was previously saved, click the **Select from Templates** icon. 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.

Entering Ranges and Guesses for Parameter Estimation

After specifying the user-defined degradation model, click the **Function Parameters** icon to provide a starting point for estimating the parameters. The following example shows the parameter estimates for each unit ID in a particular data set.



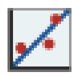
For each Unit ID, enter the following values:

- 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.
- 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.

To reset all your inputs in this window to the defaults, click the **Initialize Parameters** icon .

Degradation Plot Sheet

After you analyze the degradation data, 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.

Nondestructive Degradation Folio Plots

In addition to the standard life data/life-stress 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.

Destructive Degradation Folio Plots

The following plots are available for the destructive degradation folios. The last two degradation plots pertain specifically to the degradation model based on the fitted measurement distribution.

- **Reliability vs. Time**
- **Unreliability vs. Time**
- **Pdf Plot**
- **Failure Rate vs. Time**
- **Degradation vs. Time (Linear)** shows the effect of time on the degradation measurements, using a linear scale for time and degradation. Multiple *pdf*s, each at a different inspection time, are displayed on the plot. The measurements obtained at each time are shown at the base of the associated *pdf*.

  A *degradation line* will be displayed over the *pdf*s, which shows how the measurement distribution's location parameter changes over time. And 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.
- **Degradation vs. Time (Log)** plots the same information as the Degradation vs. Time (Linear) plot, but using logarithmic axes for time and degradation.

Degradation Model Wizard

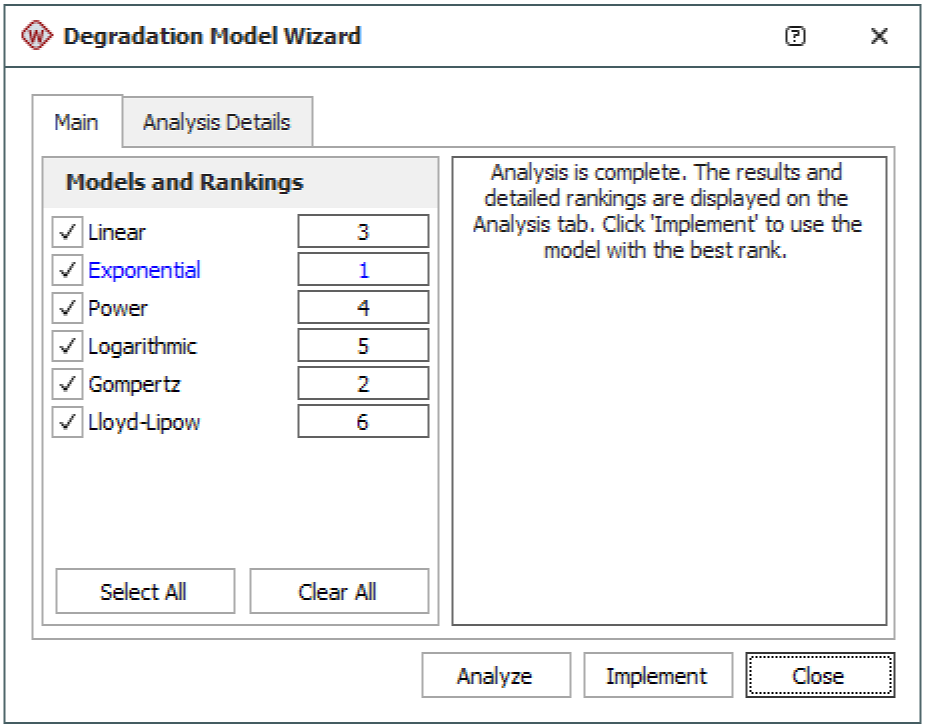
The Degradation Model Wizard performs a goodness of fit test to determine the best degradation model for your data. Note that the degradation model wizard only serves as a guide. You should compare its suggestion with information about the product being modeled before making the final decision.

You can access the wizard by choosing **[Degradation/Destructive Degradation] > Analysis > Model Wizard** or by clicking its icon on the Main page of the control panel.

Evaluating and Using Degradation Models

1. On the Main tab of the Degradation Model wizard, select the models you would like to consider. The available models will vary depending on which degradation analysis folio you are working with.

2. Click **Analyze** to start the evaluation. The results of the evaluation will be presented as in the example shown next.

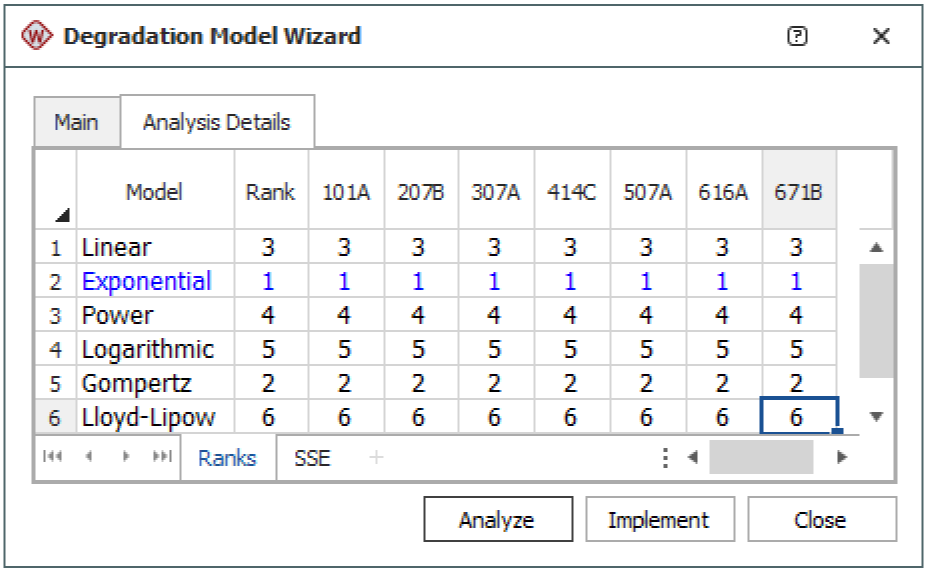


The models will be ranked according to how well they fit the data, with rank 1 being the best fit. In the example shown above, the exponential model is the suggested model for the data set. Click **Implement** to extrapolate failure times using the suggested model.

Viewing the Analysis Details

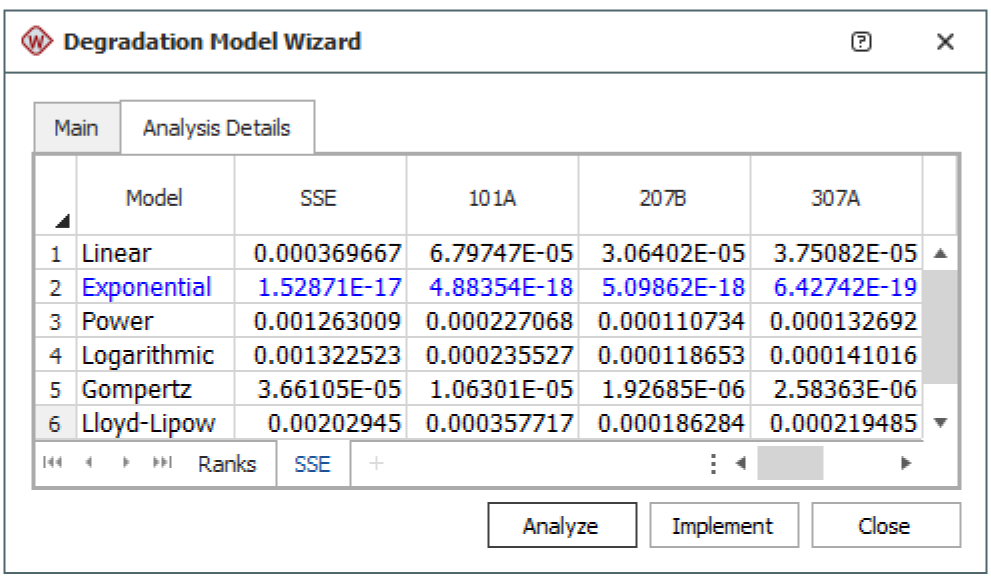
The wizard uses the Sum of Square Error (SSE) to evaluate the fit of the data. The calculations behind the ranking can be viewed on the Analysis Details tab, which contains two data sheets:

- 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 all devices, while the Gompertz model is the second best fit for all devices.



	Model	Rank	101A	207B	307A	414C	507A	616A	671B
1	Linear	3	3	3	3	3	3	3	3
2	Exponential	1	1	1	1	1	1	1	1
3	Power	4	4	4	4	4	4	4	4
4	Logarithmic	5	5	5	5	5	5	5	5
5	Gompertz	2	2	2	2	2	2	2	2
6	Lloyd-Lipow	6	6	6	6	6	6	6	6

- 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	101A	207B	307A
1	Linear	0.000369667	6.79747E-05	3.06402E-05	3.75082E-05
2	Exponential	1.52871E-17	4.88354E-18	5.09862E-18	6.42742E-19
3	Power	0.001263009	0.000227068	0.000110734	0.000132692
4	Logarithmic	0.001322523	0.000235527	0.000118653	0.000141016
5	Gompertz	3.66105E-05	1.06301E-05	1.92685E-06	2.58363E-06
6	Lloyd-Lipow	0.00202945	0.000357717	0.000186284	0.000219485

Life Data Degradation Analysis Example

*The data set used in this example is available in the example database installed with the Weibull++ software (called "Weibull21_Examples.rsgz21"). 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.

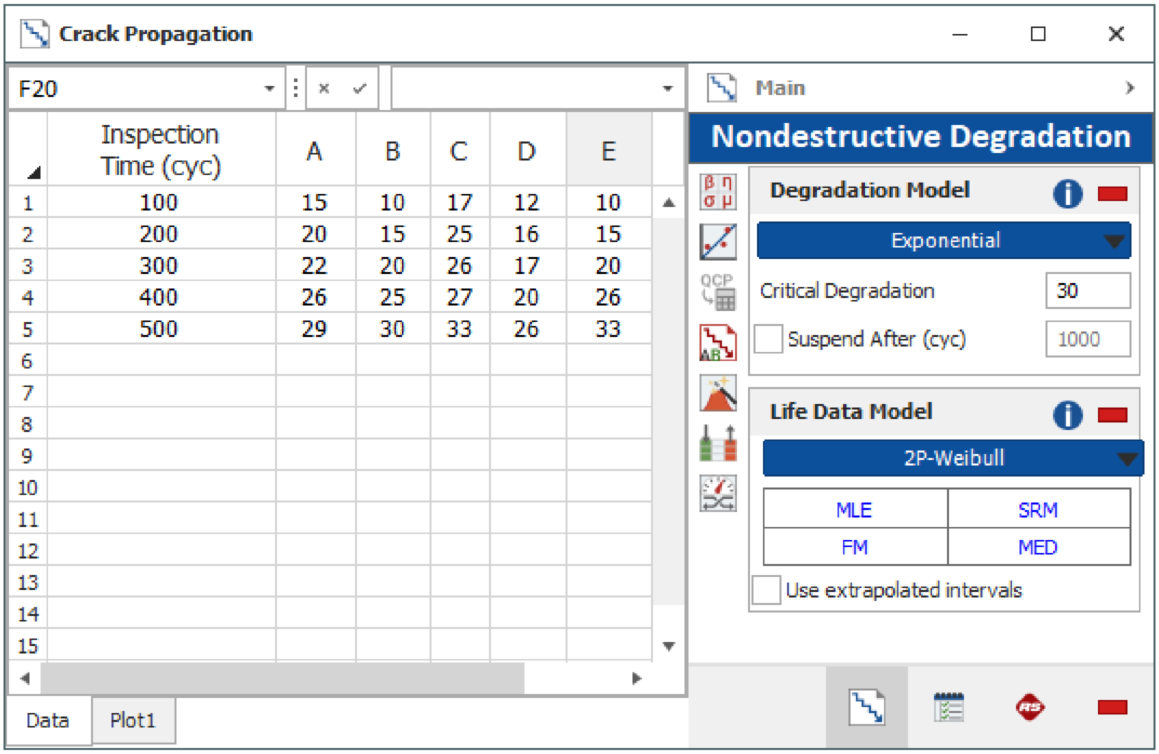
The data sheet in the "Crack Propagation" folio displays the results of the inspections for the five units, A through E.

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.

Analyze the Data Set

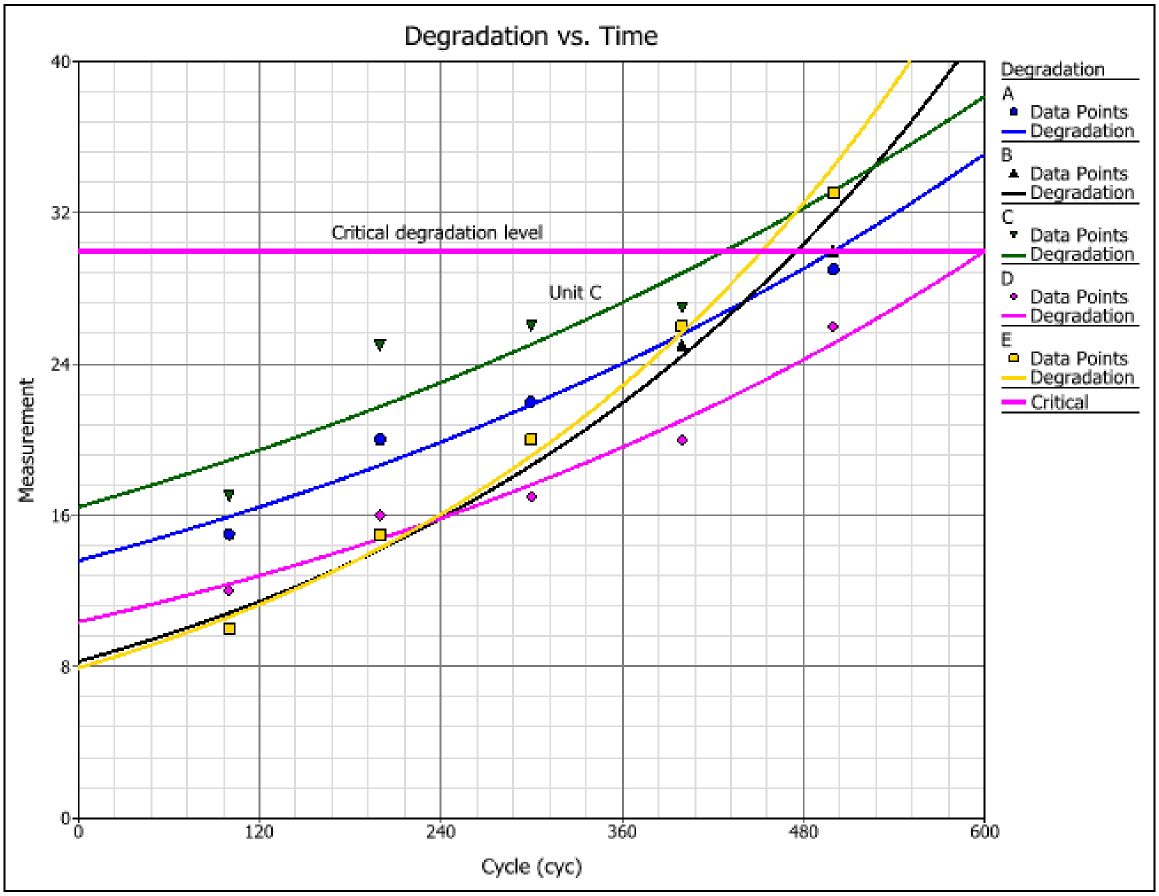
1. The first step is set up the data sheet for the analysis. Click the **Change Units** icon on the Main page of the control panel

2. In the Change Units window, select to change the time units to **Cycles**.
3. On the control panel, select the **Exponential** degradation model and enter a value of **30** for the **Critical Degradation**.
4. For the life data model, select the **2P Weibull** distribution and the **MLE** analysis method. Use the default settings for all other options, as shown next.



5. 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.

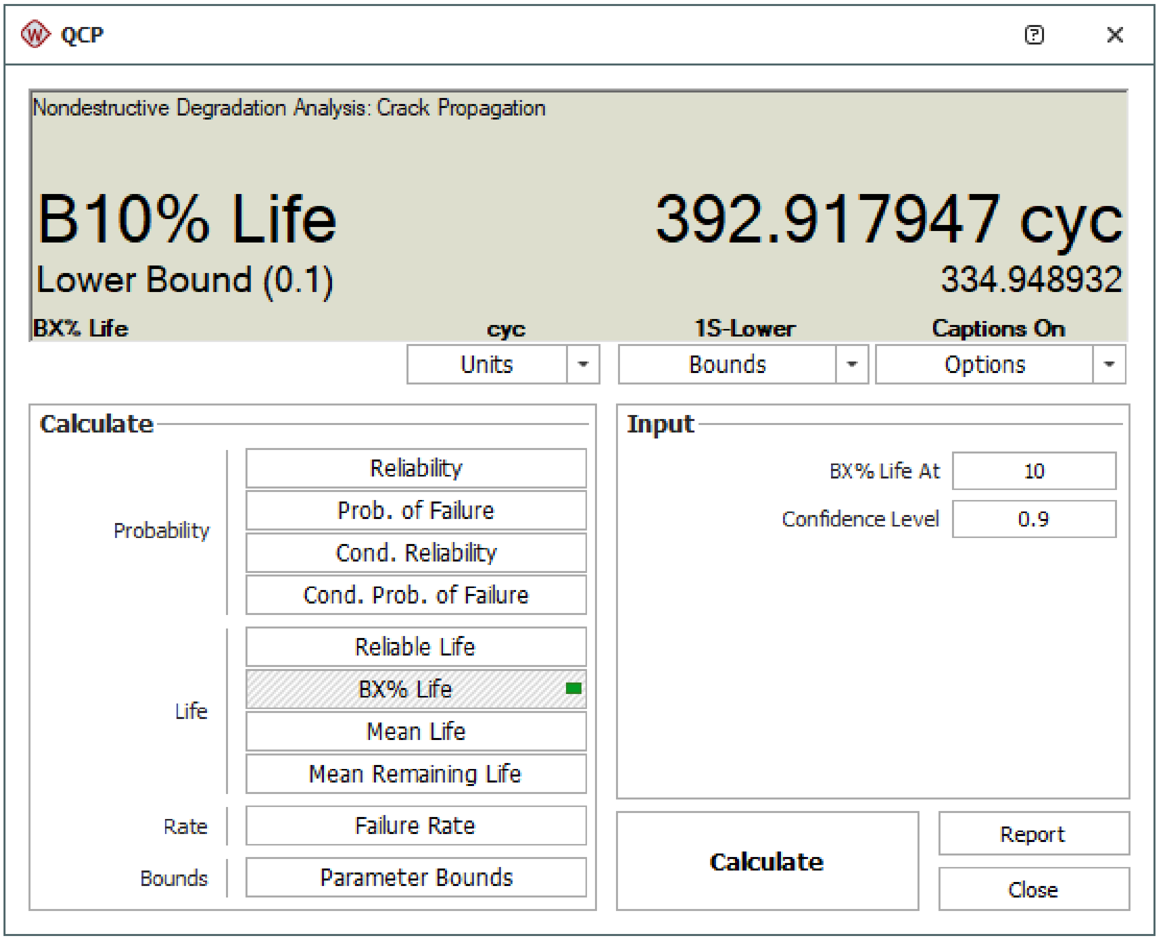
Estimate the B10 Life

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.

7. In the QCP, choose to calculate the **BX% Life** with **lower 1-sided** confidence bounds. Select **Cycles** for the units and then make the following inputs:

- BX% Life At = **10**
- Confidence Level = **0.9**

8. Click **Calculate** nto obtain the results. 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.



Accelerated Life Testing Degradation Analysis Example

For life-stress data only.

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 Accelerated Life Testing 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.

Light Intensity (mcd) of Units Tested at 35 mA

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 40 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

Create an Accelerated Nondestructive Degradation Analysis Folio

1. Choose **Home > Insert > Degradation**. In the Add Degradation window, select the **Accelerated Nondestructive Degradation** option and click **Next**.
2. In 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. Click **OK** to create the folio.

Define Stress Columns and Use Stress Levels			
	Stress Name	Stress Units	Use Level
✓	**Amps**	**mA**	**28**
	Voltage	V	100
	Humidity	RH	50

3. Enter the given data into the data sheet. As an example, the data for units A1 and A2 are entered as shown next.

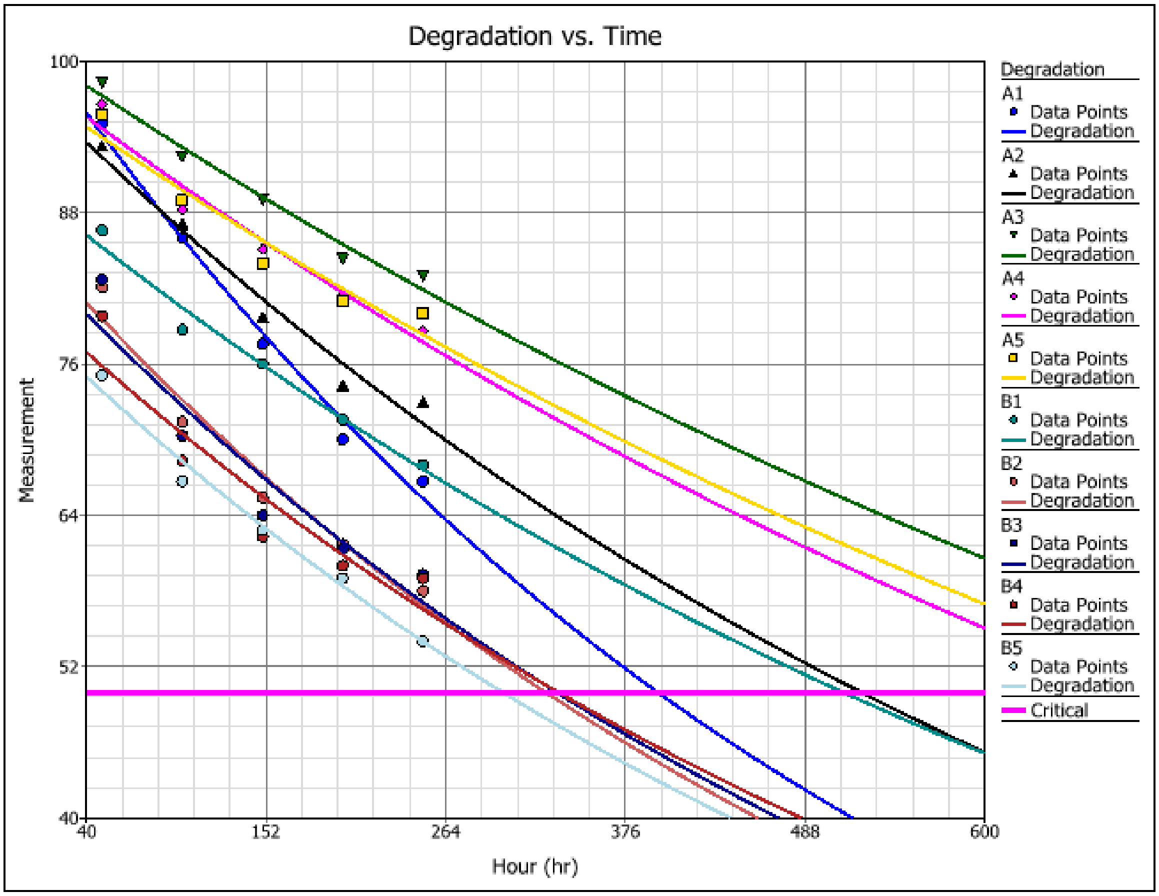
	Inspection Time (hr)	Measurement	Amps mA ✓	Unit ID
1	50	95.1	35	A1
2	100	86	35	A1
3	150	77.6	35	A1
4	200	70	35	A1
5	250	66.7	35	A1
6	50	93.3	35	A2
7	100	87.1	35	A2
8	150	79.7	35	A2

Analyze the Data Set

4. In the **Degradation Model** area of the control panel, 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.

5. 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.



6. 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.

7. To view the results of the accelerated life testing data analysis, close the Results window and click **ALTA Results (...)** on the control panel.

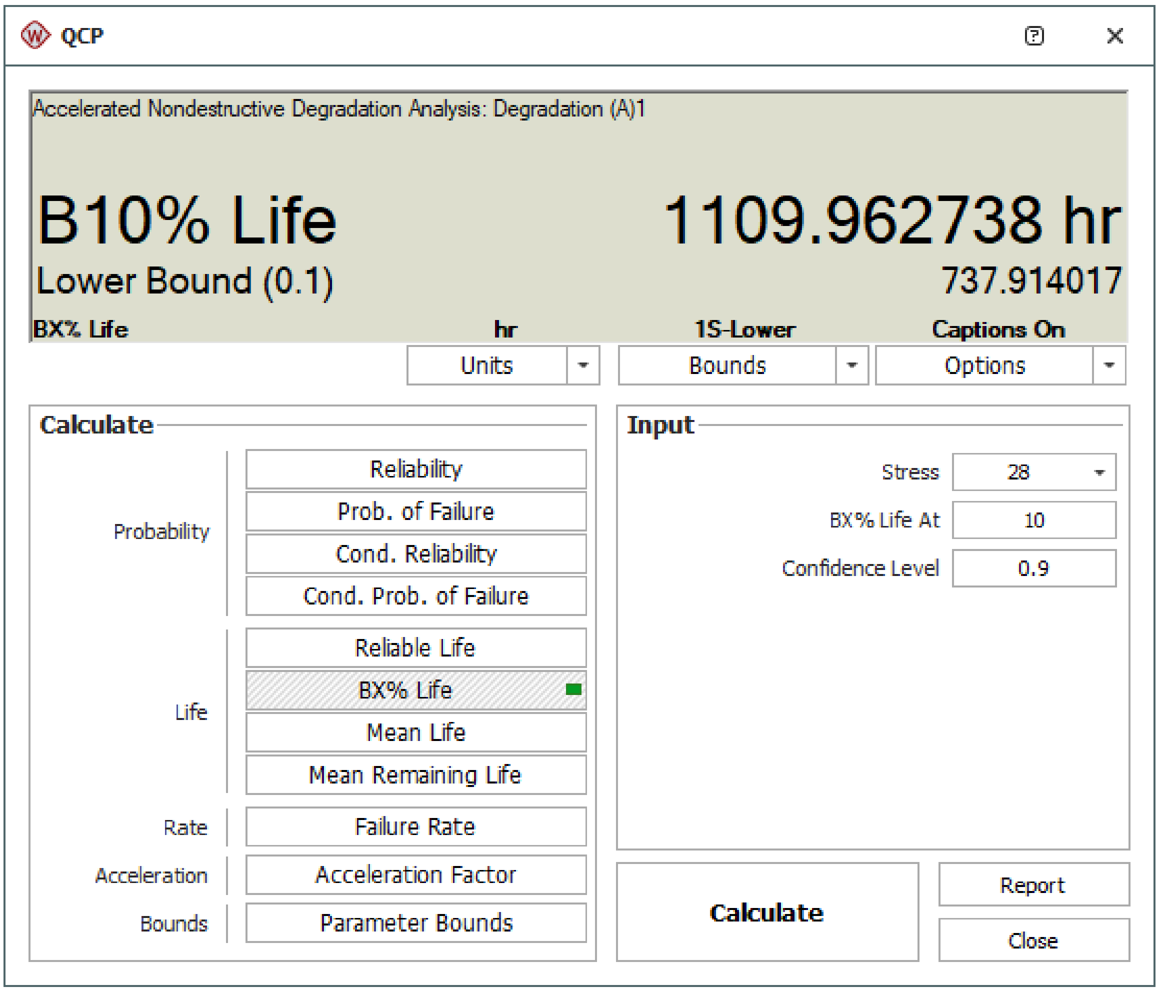
Estimate the B10 Life

8. To estimate the time at which 10% of units are expected to fail under normal conditions, open the QCP by choosing **Life-Stress Data > Analysis > Quick Calculation Pad** or by clicking the icon on the Main page of 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:

   - BX% Life At = **10**
   - Confidence Level = **0.9**

10. Click **Calculate** to obtain the results. 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.



Comparison Folios

Tests of Comparison

Weibull++ provides two tools designed for statistical comparison of data sets:

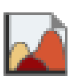
- The **Life Comparison** tool calculates the probability that the times-to-failure of one population will be greater or less than the times-to-failure of the second. This allows you to compare life distributions from two alternate designs, manufacturers, lots, assembly lines, etc. in order determine 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. Both tools are described in the following sections.

Folio Setup

To add a life comparison or stress-strength folio to a project, right-click the **Comparison** folder in the current project explorer and choose the folio type.

 Life Comparison

 Stress-Strength Comparison

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.

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 [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).]:

$$P[t_2 \geq t_1] = \int_0^{\infty} f_1(t) \cdot R_2(t)dt \tag{1}$$

where f(t) is thepdfof data set 1 and

- 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).

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 ≤ X1 ≤ U, and X1 = Stress, X2 = Strength.

For data sets that are calculated in an Accelerated Life Testing life-stress data folio, the comparison is performed at the given use stress levels of both data sets; therefore, Eqn. (1) can also be expressed as:

$$P\left[t_2 \geq t_1\right] = \int_0^\infty f_1\left(t, V_{use1}\right) \cdot R_2\left(t, V_{use2}\right) dt \qquad (5)$$

Tests of Comparison Control Panel

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.

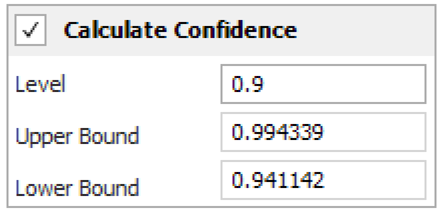
Note: Information from the source data sheets are copied, not linked to the analysis. This means that if changes were made to the data sheets, you must re-select the data sheets to copy the new information to the analysis. To re-select data sheets, click the  icon on the control panel.

CONTROL PANEL ANALYSIS PAGE

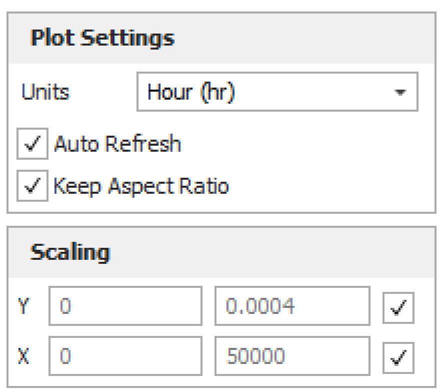
The Analysis page of the control panel contains the following settings:

- The **Results** area shows the calculated probability value. Click the **Show Results (...)** button to display a report of the results.
  - 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.
- The **Calculate Confidence** check box provides the option to compute the two-sided confidence bounds of the probability value. Enter the desired confidence level (in decimal) in the

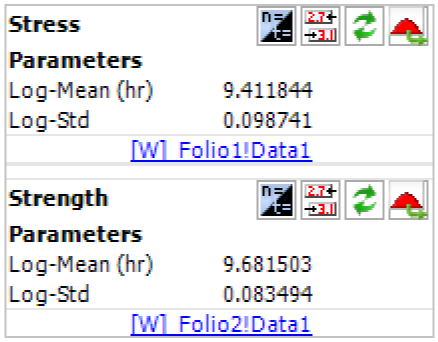
Level field, and then click the **Calculate** icon on the control panel to display the results in the **Upper Bound** and **Lower Bound** fields.



- The following settings affect how the plot is displayed:
  - The **Units** drop-down list allows you to choose the unit for the plot's time scale.
  - The **Auto Refresh** check box automatically refreshes the plot if any of the data sets, inputs or settings are modified.
  - The **Keep Aspect Ratio** check box maintains 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 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 Analysis page of the control panel and in the areas dedicated to the selected data sheets (e.g., the **Stress** area on the control panel, as shown in the previous picture). Use these tools to manage data and experiment with the results of your analysis.

Calculate calculates the probability value.

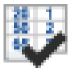
- In the life comparison folio, the value represents the probability that the life of the second data set exceeds the life of the first data set.
- In the stress-strength folio, the value represents the reliability of the product (i.e., the probability that the product can withstand the applied stress). This can also be displayed as the probability of failure if you select the check box on the Setup page of the control panel.

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.

RS Draw launches ReliaSoft Draw, which allows you to annotate your plot and view your plot in greater detail.

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 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.

Reliability Demonstration Test opens the Reliability Demonstration Test window, which 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.

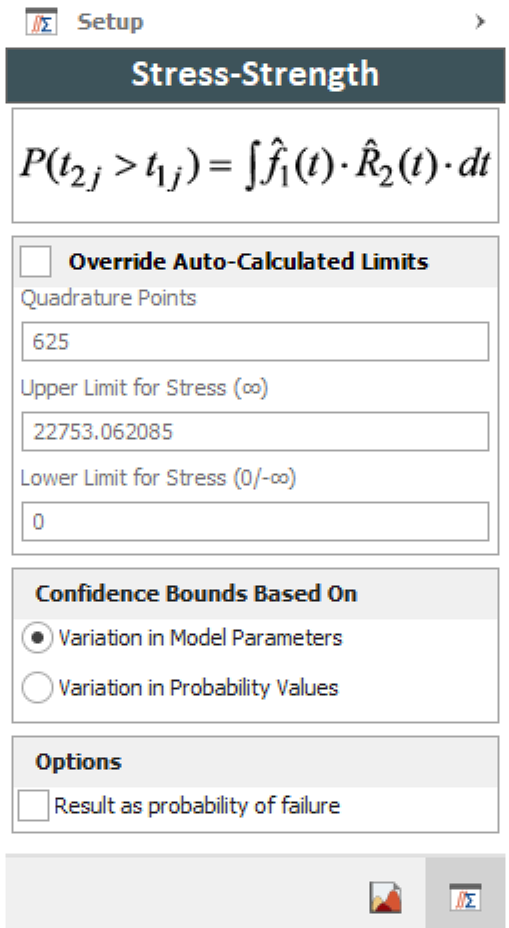
Alter Parameters (also called Alter Parameters w/o Recalculation) allows you to manually alter the value of one (or more) parameters, while keeping the original values of the rest of the parameters and the variance/covariance matrix the same. The LK value, the solution line on plots and all subsequent 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. All subsequent analysis made on the data source, such as calculations made via the QCP and other tools, will be based on the modified parameter values; therefore, the analysis may be appropriate only within the context of your specific scenario.

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 picture shows the Setup page of a stress-strength folio.



- The limits of the integral in the probability equation are from zero to infinity. 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.

  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.

- 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. 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.

  Note that confidence bounds calculations are not possible when one or both data sheets is an Accelerated Life Testing life-stress data 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:
    - 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 Accelerated Life Testing life-stress data 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.
- 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.

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.

Table 1: Times-to-Failure of the Old Product Design

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

Table 2: Times-to-Failure of the New Product Design

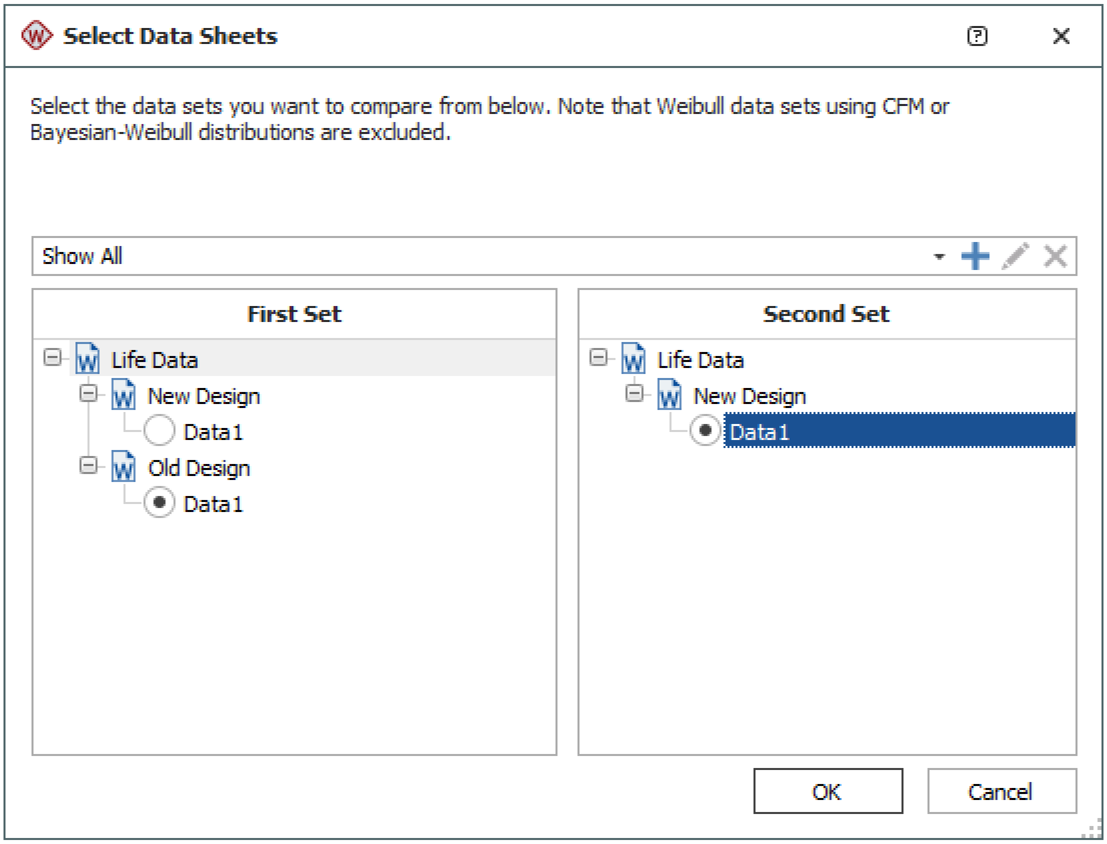
15	32	61	67	75
116	148	178	181	183

CREATE A LIFE COMPARISON FOLIO

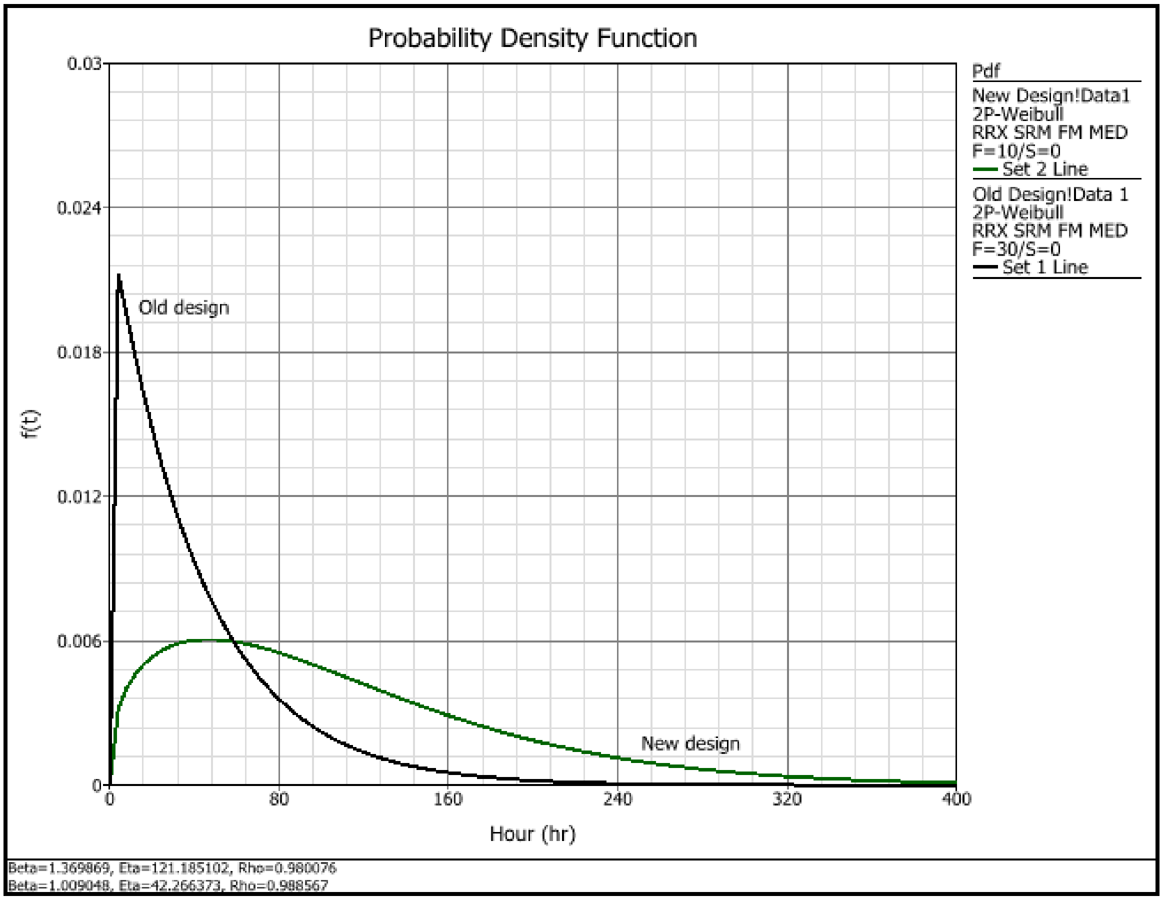
1. Enter the given data into two separate Weibull++ life data folio data sheets. Calculate the parameters for both data sheets by using the **2P-Weibull** distribution and the **RRX** analysis method.

2. Next, add a life comparison folio to the project. On the Home tab, click the **Life Comparison** icon in the Insert gallery.

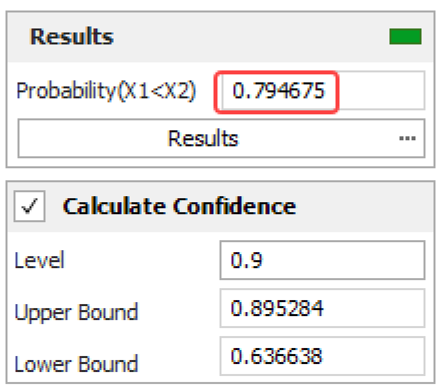
3. Select the data sheets for the two designs, as shown next. Click **OK**.



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.



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 the strength data for 50 specimens.

The objective is to estimate the reliability of the steel rods against fractures.

Table I: 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)

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

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 database 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. 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

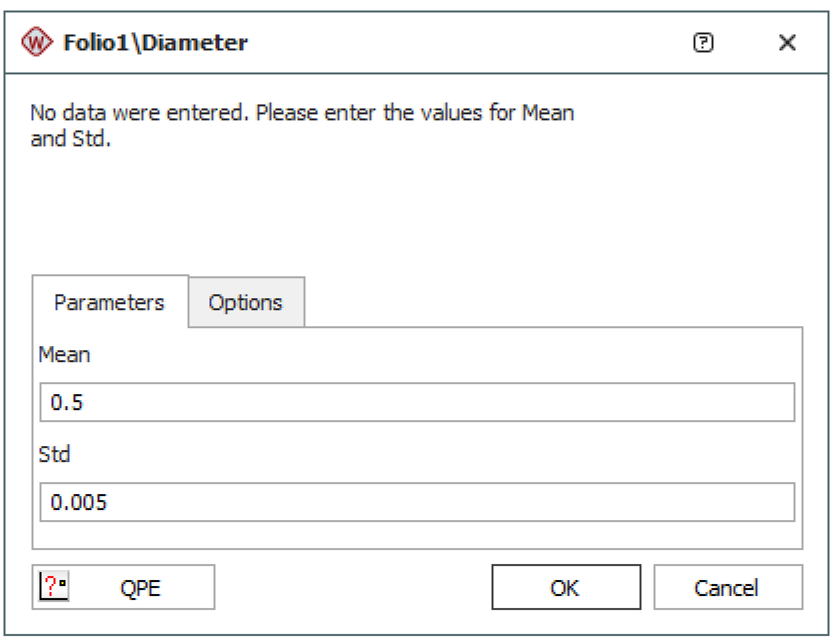
In this example, stress is defined as:

$$stress = \frac{load}{sectional - area} = \frac{F}{\left(\pi d^2\right)/4} \tag{S.1}$$

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.

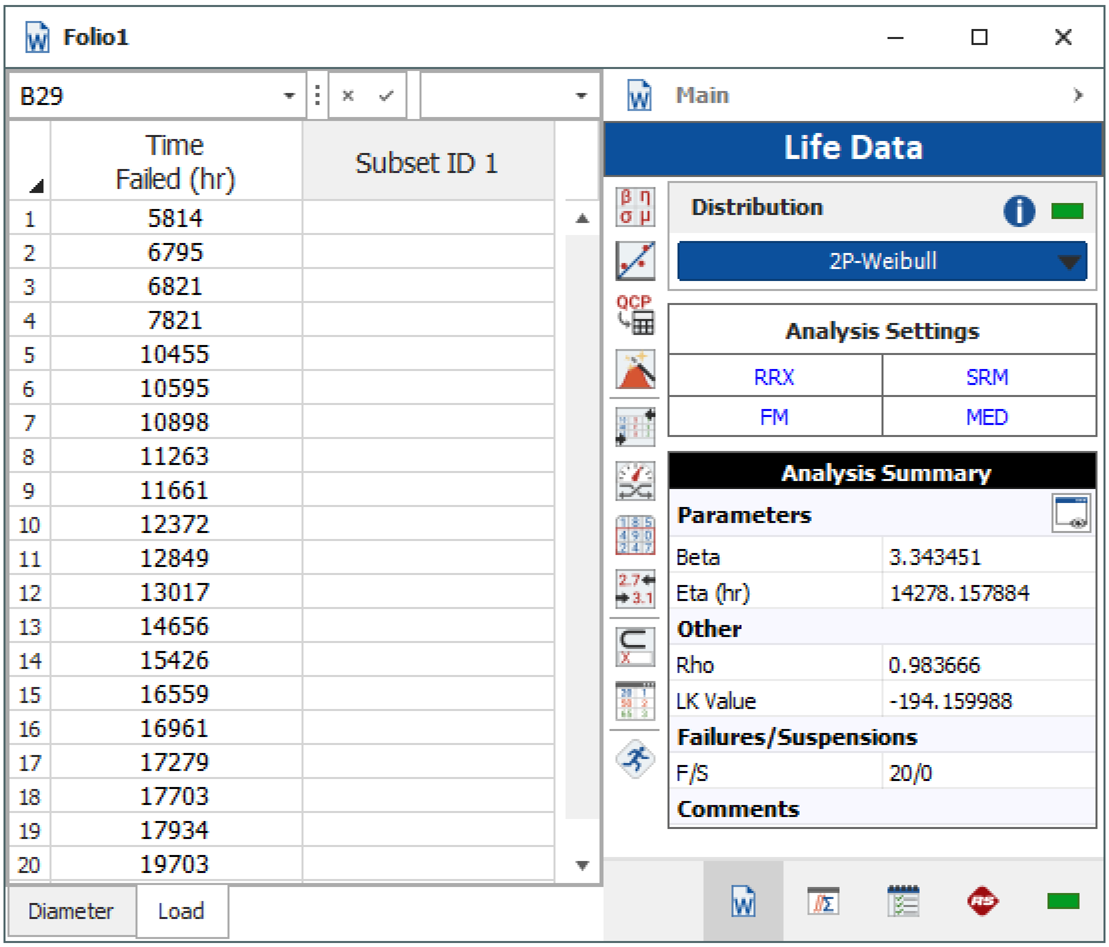
   To use this information, create a new Weibull++ life data folio in the project. Rename the data sheet to "Diameter." On the control panel, choose the **Normal** distribution and the **RRX** parameter estimation method. Click the **Calculate** icon on the Main page of the control panel, then in the input window, enter **0.5** for the mean and **0.005** for the standard deviation, as shown next.



2. 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.

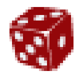
3. For the distribution of the tensile load, we can use the data from Table I. Create a new data sheet in the same folio by right clicking the data sheet tab area (the area at the bottom of the window that shows the name of the data sheets) and choosing **Insert Data Sheet** on the shortcut menu. Rename the data sheet to "Load."

4. 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.

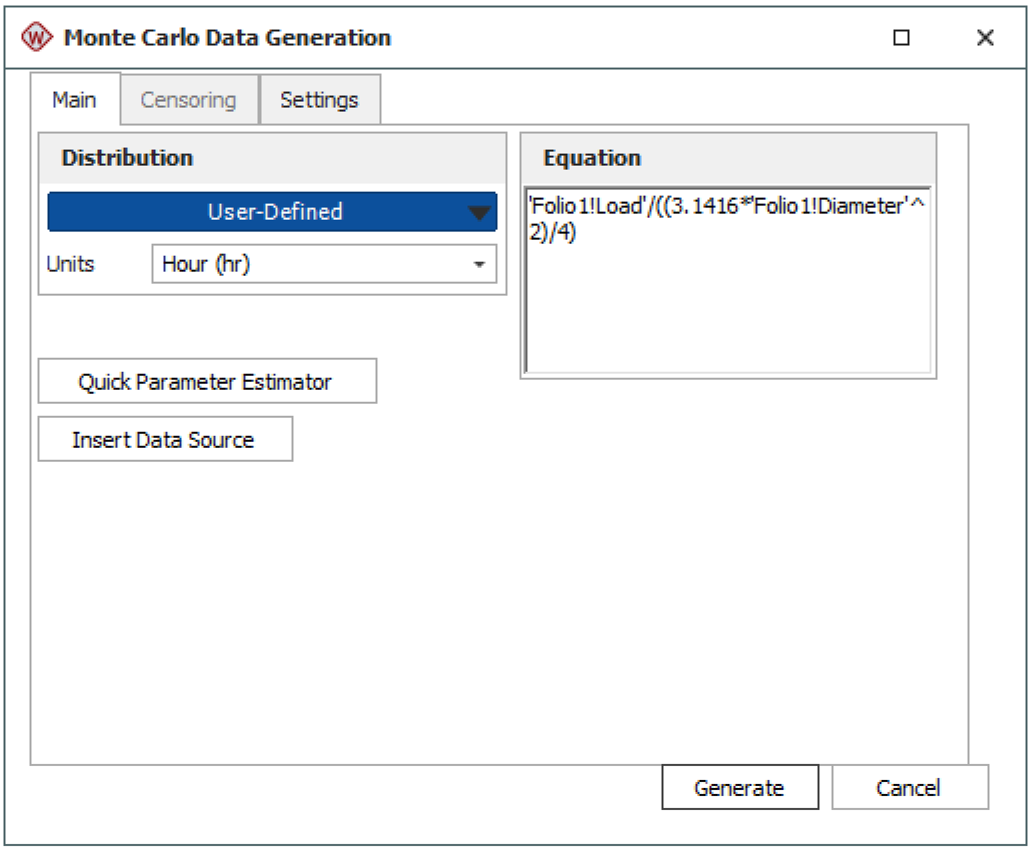


5. Now that the distributions of the variables governing Eqn. (S.1) are available, we can now obtain the stress data set. There are several conventional methods for synthesizing distributions. For this example, we will use a Monte Carlo simulation:

   a. Open the Monte Carlo utility by choosing **Home > Insert > Monte Carlo**. (If you have Accelerated Life Testing or Reliability Growth activated on your computer, you will be given a choice between "Monte Carlo," "Stress-Dependent Monte Carlo" or "Repairable Systems Monte Carlo." Choose **Monte Carlo**.)



b. 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.



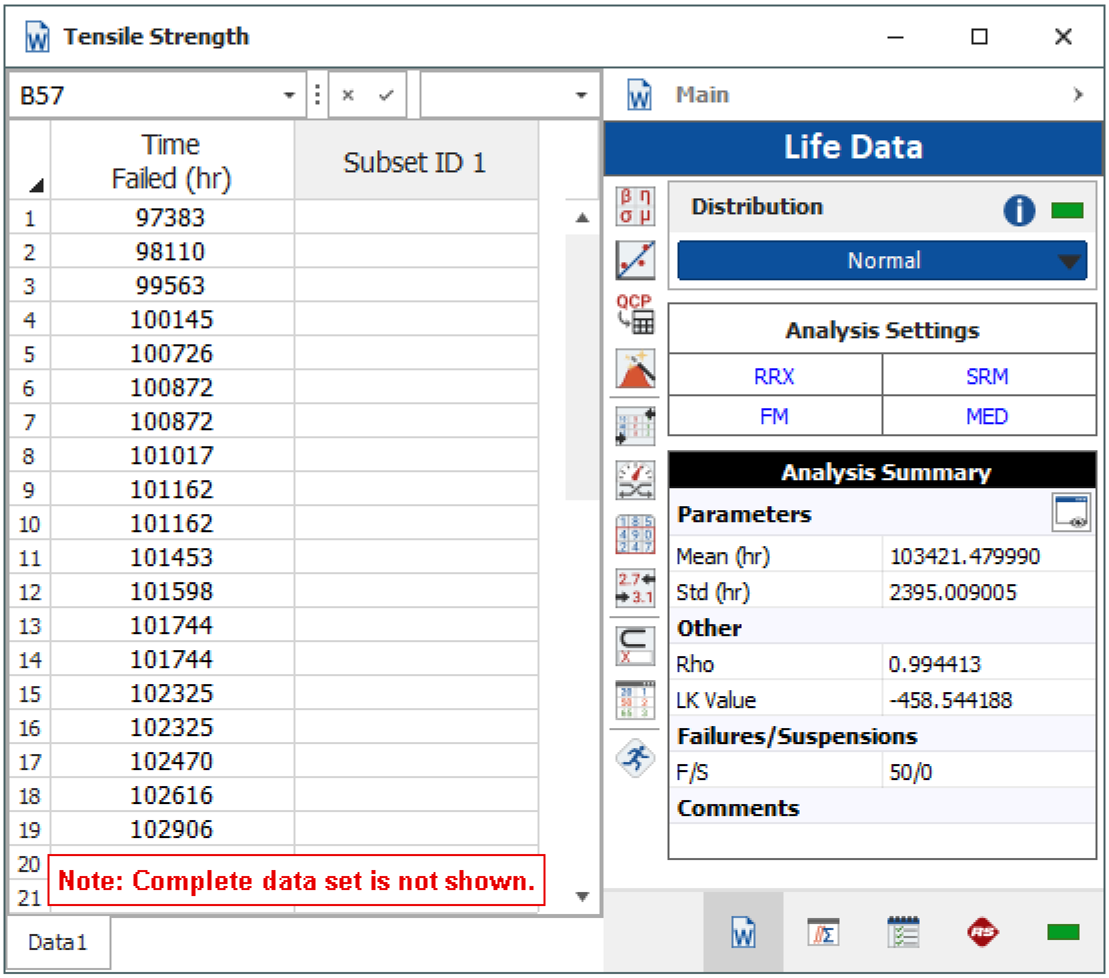
c. Click the **Settings** tab. In the **Data Points** area, enter **1,000** and then click **Generate** to generate the stress data set. This creates a new folio in the project. Rename the new folio to "Stress Values."

6. 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

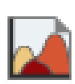
7. To obtain the strength distribution, use the tensile strength data from Table II. Create a new Weibull++ life data folio and rename it to "Tensile Strength."

8. 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.

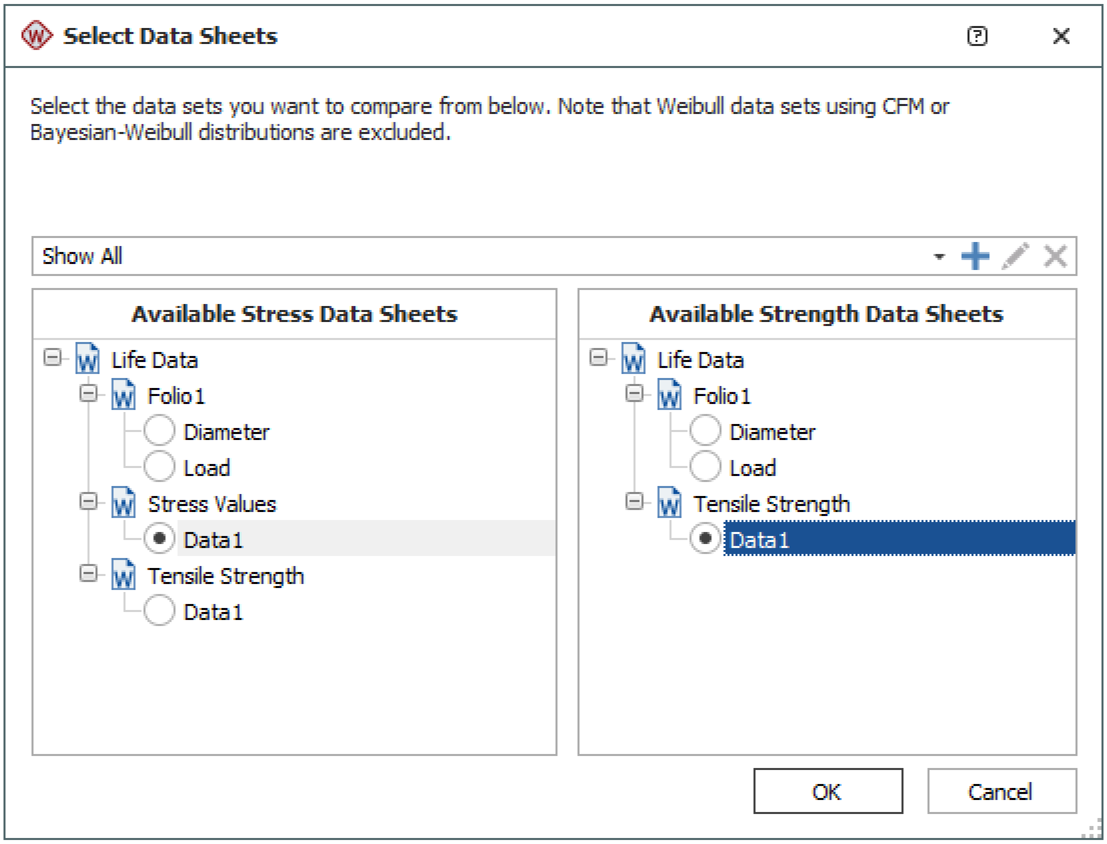


ESTIMATE THE RELIABILITY

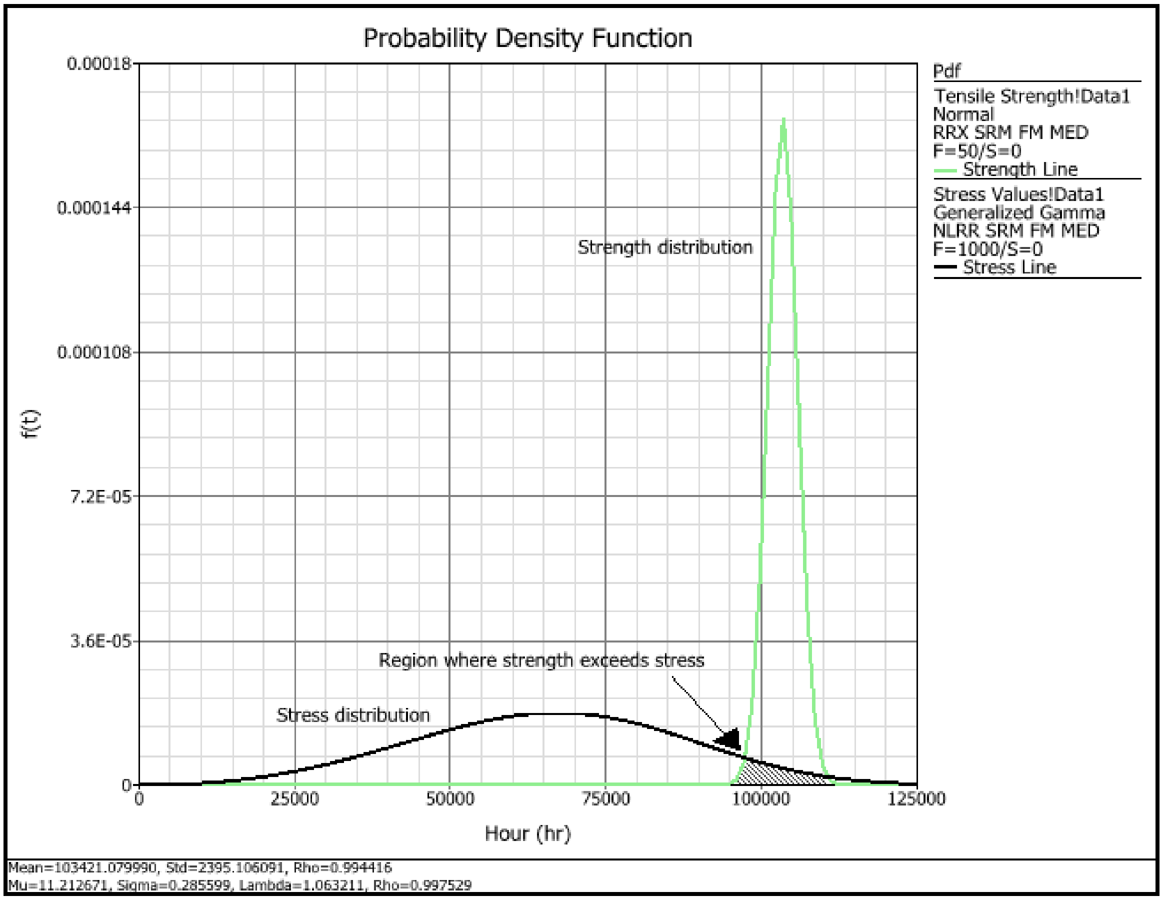
9. Finally, use the stress-strength folio to calculate the reliability of the steel rods. Add the folio to the project by choosing **Home > Insert > Stress-Strength Comparison**.



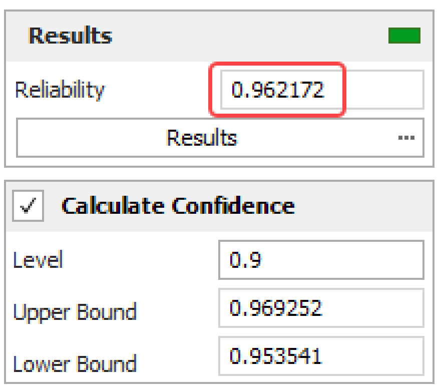
- Select the data sheets containing the stress and strength data, as shown next. Click **OK**.



- 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).



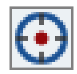
- The result on the control panel indicates that the reliability of the steel rods against fractures is estimated to be 96.2171%, as shown next.



Target Reliability Parameter Estimator

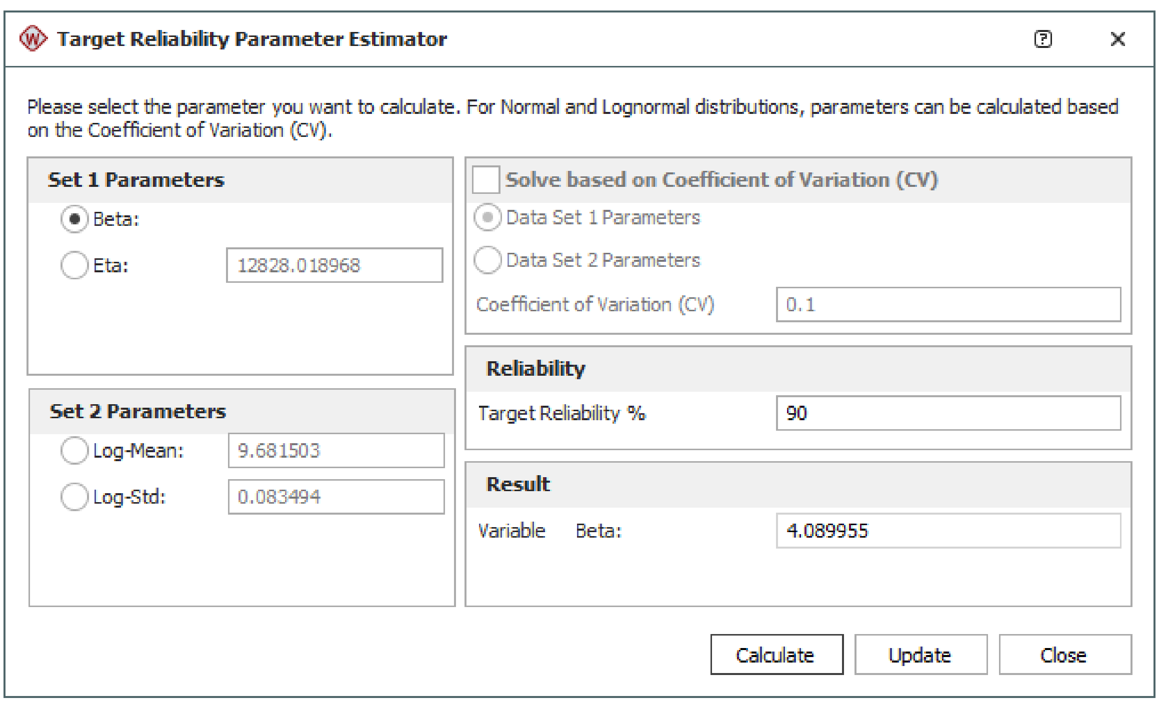
The life comparison and stress-strength tools include the Target Reliability Parameter Estimator, which allows you to explore how you could attempt to change one of the parameters in order to meet a specified target reliability. The plot and reliability estimate will be based on the altered distributions, 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 life data and life-stress 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 life data 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 life data 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 4.0899.



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.

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.

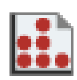
One-Way ANOVA Folio

The one-way ANOVA folio offers a simple way to evaluate the effect of a single factor on a single response. Like a one factor design, it allows you to take a detailed look at the effect of the factor (using up to 255 levels), and it helps you determine whether a change in output is due to a change in the input (level) instead of random error.

The factor in a one-way ANOVA folio is always a qualitative factor. For this reason, predictions cannot be made for factor levels that are not tested, nor can optimization be performed.

Note: In general, one-way ANOVA is a statistical method that is well suited to examining existing data, rather than planning an experiment. For planning an experiment involving a single factor, the one factor design in the design folio is the appropriate tool.

To add a one-way ANOVA folio to your project, choose **Home > Insert > One-Way ANOVA**.



One-Way ANOVA Control Panel

The one-way ANOVA folio control panel allows you to control the settings for analyzing response data and displays the results of the analysis. This topic focuses on the Main page and Analysis page of the one-way ANOVA folio control panel, which contains most of the tools you will need to perform the analysis. For more information about the control panel in general, see Control Panels.

CONTROL PANEL MAIN PAGE

- The **Settings** area is used to view the settings and status of the analysis.

Settings
Risk Level: 0.1
Transformation: Y' = Y
Analyzed
9 Observations

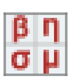
  In this example:

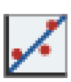
  - The specified risk level is 10%. To configure this setting, click the text. The Analysis Settings page of the control panel will be displayed, allowing you to set the risk level.
  - No transformation is used (i.e., the "transformed" response, *Y′*, is equal to the non-transformed response, *Y*). To configure this setting, you can click the text or the **Select Transformation** icon in the folio tools.
  - "Analyzed" is shown in green, which indicates that the response data has been analyzed using the current settings. Otherwise, this would show "Modified" in red.
  - The last row indicates that the analyzed data consists of 9 observations.

- The **Analysis Summary** area is shown when the current response data has been analyzed using the current settings. This area displays the *p* value for the factor and a statement regarding whether or not the factor levels affect the response. Click the **View Analysis Summary** icon in this area to view the details of the analysis results.

Folio Tools

The folio tools are arranged on the left side of the control panel. Use these tools to configure your analysis and experiment with the analysis results. Depending on the type of design you are working with, the control panel may contain some or all of the following tools:

 **Calculate** analyzes the data for each response that is selected to be included in the analysis. To exclude a response from the analysis, clear the check box in its column heading.

 **Plot** creates a plot based on the analysis. If you click this icon before the current data set has been analyzed, an analysis will be performed automatically.

 **Design - View Analysis Summary** opens a window that contains detailed information about current and past analysis results.

 **Select Transformation** opens a window that allows you to select a transformation to apply to each response.

CONTROL PANEL ANALYSIS SETTINGS PAGE

The Analysis Settings page of the one-way ANOVA folio control panel allows you to set the **Risk Level**, or alpha value. This is a measure of the risk that the analysis results are incorrect (i.e., alpha = 1 - confidence level).

One-Way ANOVA Data Tab

The Data tab in the one-way ANOVA folio offers a column for each level of the factor (up to 256 levels). Enter the response values observed for each level of the factor in the appropriate column. An example is shown next.

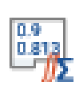
	Operator A	Operator B	Operator C	Operator D
1	59.88	59.87	60.83	61.01
2	60.12	60.32	60.87	60.87
3	60.88	60.42	60.56	60.69
4	60.98	59.99	61	60.53
5	59.9	60.12	60.5	60.63

You can rename columns in the one-way ANOVA folio data tab by right-clicking the column heading and choosing **Rename Column** on the shortcut menu.

One-Way ANOVA Analysis Results

When accessed from a one-way ANOVA folio, the Analysis Summary window will contain detailed information about analysis results, including information that describes how the factor levels affect the response.

If the current response data has been analyzed, you can open the window by clicking the **View Analysis Summary** icon on the control panel.



If the current response data has not been analyzed, the icon will still be available so you can view the folio's analysis history.

Select an item in the Available Report Items panel to display it on the spreadsheet. Each item is described next.

ANALYSIS RESULTS

The **ANOVA table** provides general information about the effects of the factor levels on the selected response.

ANOVA Table Columns

- **Source of Variation** is the source that caused the difference in the observed output values. This can be a factor, factorial interaction, curvature, block, error, etc. If your design includes more than one factor and you have selected to use grouped terms in the analysis (specified on the Analysis Settings page of the control panel), the effects will be grouped by order (i.e., main effects, two-way interactions, etc.). Sources displayed in red are considered to be significant.

- The number of **Degrees of Freedom** for the **Model** is the number of regression coefficients for the effects included in the analysis (e.g., two coefficients might be included in the regression table for a given main effect). The number of degrees of freedom for the **Residual** is the total number of observations minus the number of parameters being estimated.

- **Sum of Squares** is the amount of difference in observed output values caused by this source of variation.

- **Mean Squares** is the average amount of difference caused by this source of variation. This is equal to Sum of Squares/Degrees of Freedom.

- **F Ratio** is the ratio of Mean Squares of this source of variation and Mean Squares of pure error. A large value in this column indicates that the difference in the output caused by this source of variation is greater than the difference caused by noise (i.e., this source affects the output).

- **P Value** (alpha error or type I error) is the probability that an equal amount of variation in the output would be observed in the case that this source does not affect the output. This value is compared to the risk level (alpha) that you specify on the Analysis Settings page of the control panel. If the *p* value is less than alpha, this source of variation is considered to have a significant effect on the output. In this case, the term and its *p* value will be displayed in red.

- The following values are shown underneath the ANOVA table, and they indicate how well the model fits the data:

- **S** is the standard error of the noise. It represents the magnitude of the response variation that is caused by noise. Lower values indicate better fit.
- **R-sq** is the percentage of total difference that is attributable to the factors under consideration. It is equal to Sum of Squares(factor)/Total Sum of Squares. Higher values usually indicate better fit.
- **R-sq(adj)** is an R-sq value that is adjusted for the number of parameters in the model. Higher values indicate better fit.

The **Data Summary table** gives the mean and standard deviation of the output at each level of the factor.

Data Summary Table Columns

- **Factor Level** is the name of the qualitative level.
- **Number in Level** is the number of data points obtained at the factor level.
- **Estimated Mean** is the average of the data points obtained at the level.
- **Standard Deviation** is the standard deviation of the data points obtained at the level.

The **Mean Comparisons table** provides information on comparisons between levels of the factor.

Mean Comparisons Table Columns

- **Contrast** gives the paired comparison of any two levels. Level 1 - Level 2 means the difference between Level 1 and Level 2. Contrasts displayed in red are considered to be significant.
- **Mean Difference** is the mean value of the difference in output between the two levels.
- **Pooled Standard Error** is the standard error of the mean difference in output between the two levels.
- **Low Confidence** and **High Confidence** are the confidence bounds on the mean difference.
- **T Value** is the normalized difference, which is equal to Mean Difference/Pooled Standard Error.
- **P Value** is the probability that an equal amount of variation in the output would be observed in the case that there is no significant difference between the levels. This value is compared to the risk level (alpha) that you specify on the Analysis Settings page of the

control panel. If the *p* value is less than alpha, there is considered to be a significant difference between the levels. In this case, the contrast and its *p* value will be displayed in red.

The **Regression Equation** information is presented using two tables.

Regression Equation Tables

- The **Response** table displays the response that the regression equation applies to and the units of measurement that were entered for the response (if any).
- The **Additional Settings** table shows the transformation and risk level you entered for the response.

One-Way ANOVA Plots

The following plots are available for one-way ANOVA folios. For general information on working with plots, see Plot Utilities.

LEVEL PLOTS

Level plots allow you to compare the visually evaluate the effects of different factor levels on the selected response.

- The **Comparison Chart** shows the standardized difference for each paired comparison of factor levels.
- The **Response vs. Level** plot shows the observed output, or response, as well as the calculated mean output, at each level of the factor.
- The **Level Mean** plot shows the mean output at each level of the factor. The center point of each level line is the calculated mean and the end points represent the high and low confidence bounds on the mean based on the alpha (risk) value specified on the Analysis Settings page of the control panel.
- The **Box Plot** shows the output at each level of the factor. The height of each box represents the spread of output values at that level.
- The **Mean PDFs** plot shows the *pdf* of the mean response at each level of the factor.

RESIDUAL PLOTS

Residuals are the differences between the observed response values and the response values predicted by the model at each combination of factor values. Residual plots help to determine the validity of the model for the currently selected response. When applicable, a residual plot allows the user to select the type of residual to be used:

- **Regular Residual** is the difference between the observed Y and the predicted Y.
- **Standardized Residual** is the regular residual divided by the constant standard deviation.
- **Studentized Residual** is the regular residual divided by an estimate of its standard deviation.
- **External Studentized Residual** is the regular residual divided by an estimate of its standard deviation, where the observation in question is omitted from the estimation.

The plots are described next.

- The **Residual Probability*** plot is the normal probability plot of the residuals. If all points fall on the line, the model fits the data well (i.e., the residuals follow a normal distribution). Some scatter is to be expected, but noticeable patterns may indicate that a transformation should be used for further analysis. Two additional measures of how well the normal distribution fits the data are provided by default in the lower title of this plot. Smaller values for the Anderson-Darling test indicate a better fit. Smaller *p* values indicate a worse fit.
- The **Residual vs. Fitted*** plot shows the residuals plotted against the fitted, or predicted, values of the selected response. If the points are randomly distributed around the "0" line in the plot, the model fits the data well. If a pattern or trend is apparent, it can mean either that the model does not provide a good fit or that Y is not normally distributed, in which case a transformation should be used for further analysis. Points outside the critical value lines, which are calculated based on the specified alpha (risk) value, may be outliers and should be examined to determine the cause of their variation.
- The **Residual Histogram*** is used to demonstrate whether the residual is normally distributed by dividing the residuals into equally spaced groups and plotting the frequency of the groups. The **Residual Histogram Settings** area allows you to:
  - Select **Custom Bins** to specify the number of groups, or bins, into which the residuals will be divided. Otherwise, the software will automatically select a default number of bins based on the number of observations.
  - Select **Superimpose pdf** to display the probability density function line on top of the bins.

DIAGNOSTIC PLOTS

- The **Box-Cox Transformation*** plot can help determine, for the currently selected response and model, what transformation, if any, should be applied. The plot shows the sum of squares of the residuals plotted against lambda. The value of lambda at the minimum point of this curve is considered the "best value" of lambda, and indicates the appropriate transformation, which is also noted by default in the lower title of the plot.

* These plots are available only when there is error in the design, indicated by a positive value for sum of squares for Residual in the ANOVA table of the analysis results.

One-Way ANOVA Example

*The data set used in this example is available in the example database installed with the software (called "Weibull21_DOE_Examples.rsgz21"). 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 example project is "One Factor - Operators in Pulp Mill."

Consider the following experiment, which was performed at a pulp mill. Plant performance is based on pulp brightness, as measured by a reflectance meter. Each of the four shift operators (denoted by A, B, C and D) made five pulp handsheets from unbleached pulp. Reflectance was read for each of the handsheets using a brightness tester. The data set is shown next.

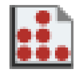
Operator A	Operator B	Operator C	Operator D
59.88	59.87	60.83	61.01
60.12	60.32	60.87	60.87
60.88	60.42	60.56	60.69
60.98	59.99	61	60.53
59.9	60.12	60.5	60.63

A goal of the experiment is to determine whether there are significant differences between the operators in making the handsheets and reading their brightness. If there are such differences, then the operators may not be creating the handsheets or recording their observations in a consistent manner.

CREATING THE FOLIO

To determine whether there are differences between the operators, the experimenters create a one-way ANOVA folio. Then they enter the response values for analysis. The design matrix and the response data are given in the "Operator Study - One-Way ANOVA" folio. The following steps describe how to create this folio on your own.

1. On the Home tab, click the **One-Way ANOVA** icon in the Insert gallery.

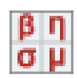


2. Right-click the Level 1 column heading and choose **Rename Column** on the shortcut menu. Change the column name to "Operator A" and click **OK**.
3. Rename the second, third and fourth column to "Operator B," "Operator C" and "Operator D," respectively.

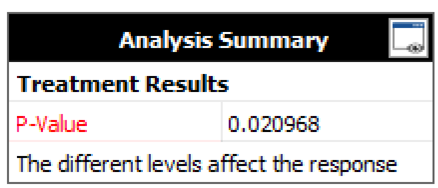
ANALYSIS AND RESULTS

The data set for this example is given in the "Operator Study - One-Way ANOVA" folio of the example project. After you enter the data from the example folio, you can perform the analysis by doing the following:

- Click the **Calculate** icon on the Data tab control panel.



- The result in the **Analysis Summary** area show that the *p* value is 0.0210. In other words, there is a 2.1% chance that the variation would be observed in the case when there is no significant difference between the response levels. Since the *p* value is less than the risk level (10%) entered on the Analysis Settings tab of the control panel, the conclusion is that the different factor levels (i.e., operators) do affect the response.



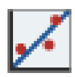
Analysis Summary	
Treatment Results	
P-Value	0.020968
The different levels affect the response	

- To see more detailed comparisons of the responses at different factor levels, click the **View Analysis Summary** icon and view the Mean Comparisons table.

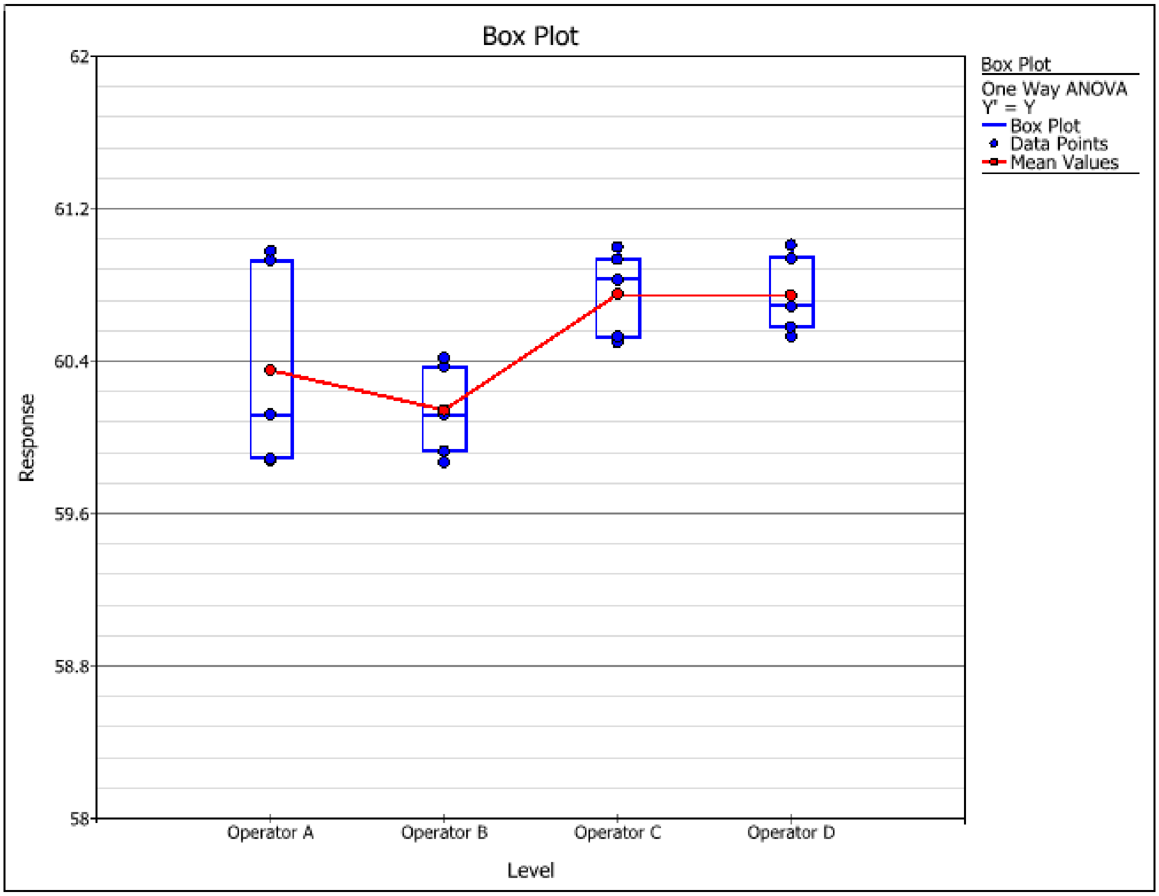
Mean Comparisons						
Contrast	Mean Difference	Pooled Standard Error	Low Confidence	High Confidence	T Value	P Value
Operator A - Operator B	0.208	0.205597	-0.150948	0.566948	1.011689	0.326742
Operator A - Operator C	-0.4	0.205597	-0.758948	-0.041052	-1.94556	0.069494
Operator A - Operator D	-0.394	0.205597	-0.752948	-0.035052	-1.91637	0.073358
Operator B - Operator C	-0.608	0.205597	-0.966948	-0.249052	-2.95725	0.00927
Operator B - Operator D	-0.602	0.205597	-0.960948	-0.243052	-2.92806	0.00985
Operator C - Operator D	0.006	0.205597	-0.352948	0.364948	0.029183	0.977079

The compared factor levels are listed in the Contrast column (e.g., "Operator A - Operator B" is the comparison of those two operators). When a *p* value is red, the compared levels are significantly different. For example, there is a significant difference between Operator A and Operator C, but there is no significant difference between Operator A and Operator B.

- To see a plot that also allows you to compare the different factor levels, click the **Plot** icon.



Then view the box plot. In this plot, you can see that see that operators C and D are nearly the same.



CONCLUSIONS

The results show that there is a significant difference between operators A and C, between A and D, and so forth. As a result, the operators receive further training to insure that they create the hand-sheets and record their observations consistently.

RBDs

Weibull++ Diagrams

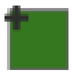
The diagrams in Weibull++ allow you to build reliability block diagrams (RBDs) using blocks that are linked to calculated data sheets from life data and life-stress data folios.

One typical application for this tool is for performing complex failure modes analysis. As explained in the Competing Failure Modes (CFM) Analysis topic, the CFM analysis method that is built-in to the Weibull++ life data 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 (see Using RBDs for Failure Modes Analysis).

Building Diagrams in Weibull++

To add a diagram in Weibull++, choose **Home > Insert > Diagram**, or right-click the **RBDs** folder in the current project explorer and choose **Add Diagram** on the shortcut menu.

To add blocks to a diagram, choose **Diagram > Blocks** and then select the type of block you wish to add.

 **Add Block** adds a standard block that represents a data sheet in a life data or life-stress data folio. 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. You can use any data sheet in any life data or life-stress data folio, as long as it has been calculated and is not using one of the competing failure modes distributions. (A red flag over a block indicates that the associated data sheet in the folio has either been deleted or needs to be calculated.)

By default, standard blocks are named after the data sheets they represent. If you want to use a different name, right-click the block and choose **Rename Block**.

 **Add Node** adds a node block that functions like a switch that diagram paths move through. A node block models 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.

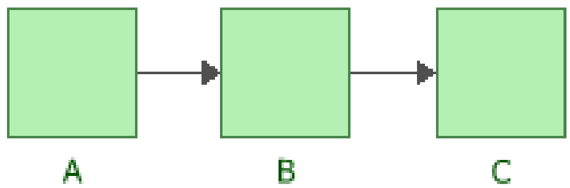
Double-click the node block to specify the number of paths that must reach the node in order for the system to succeed.

 **Add Junction** adds a junction block, which is a non-failing block. 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 Diagram Analysis Constraints). 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).

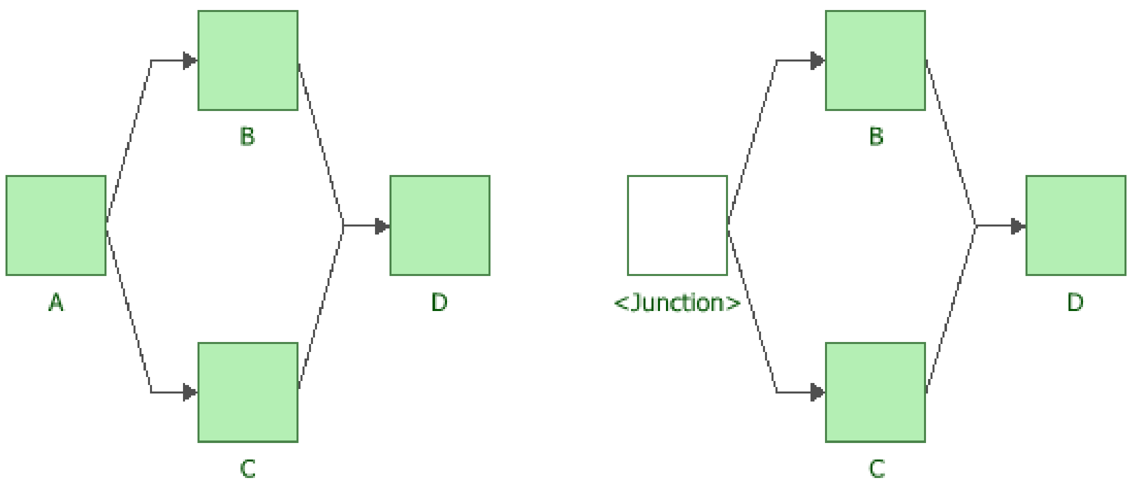
Once you have added blocks to the diagram, you can then use connectors to connect the blocks and build the RBD. From the RBD, you can calculate the exact system reliability equation for the system, and obtain plots and results based on that equation.

Examples of RBD Configurations

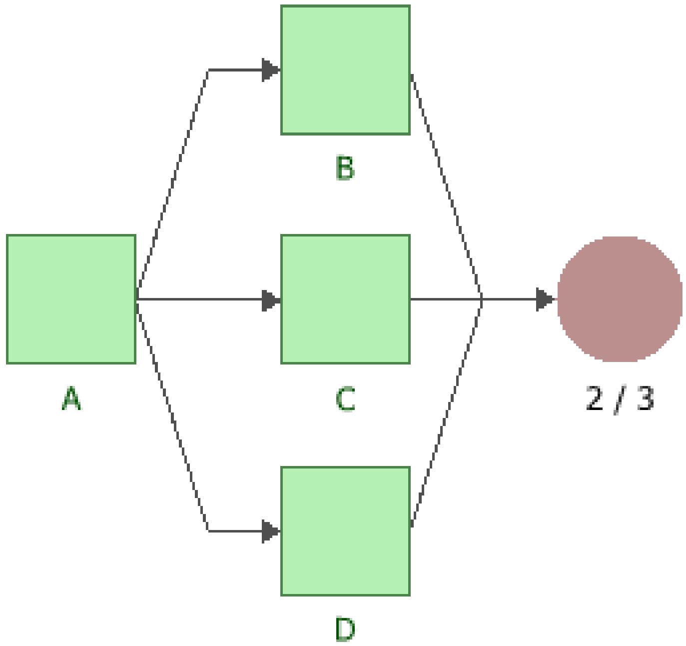
In a basic ***series configuration***, all items must work for the system to work. The following example shows standard blocks connected in series. The system fails if Block A, Block B or Block C fails.



In a basic ***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. (The second RBD shows an example of a junction block used as a starting block.)



The following example shows a ***k-out-of-n configuration*** where 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 following example shows a complex diagram with a node configuration. If more than three of the five blocks (blocks C, E, F, G and H) leading to the node block fail, the system will fail.

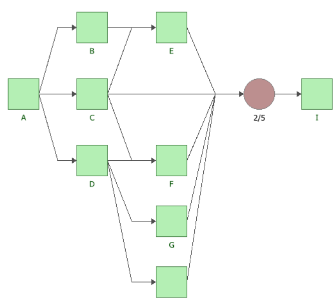
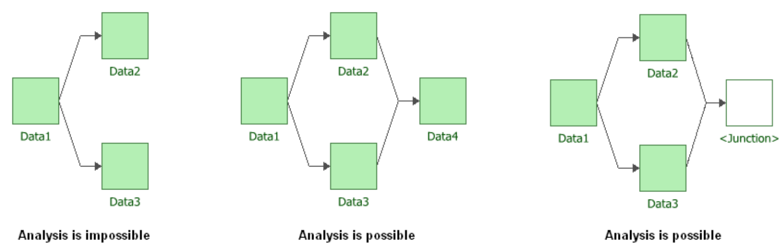


Diagram Analysis Constraints

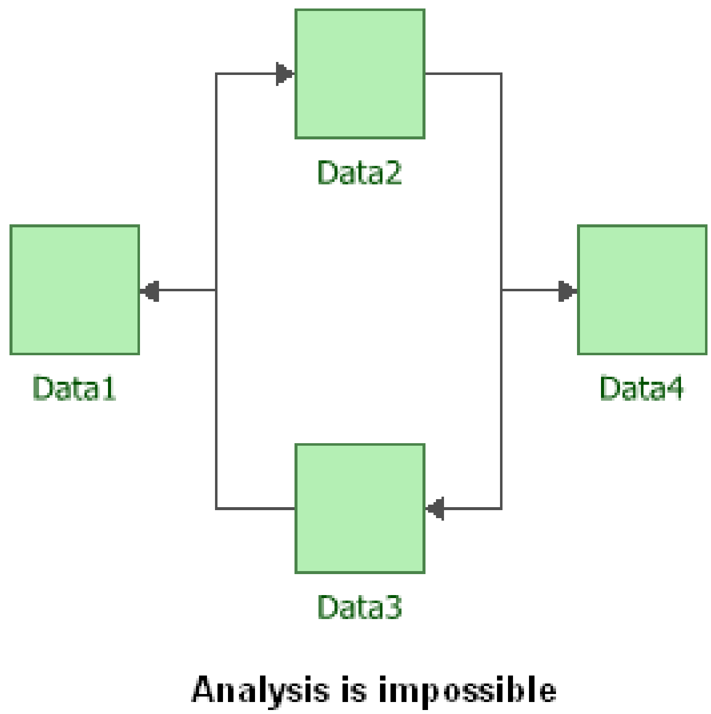
In order to permit analysis, all RBDs created in the software must conform to the following constraints:

- 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. There must be a clear path towards the end of the RBD and the RBD cannot loop back on itself.



Connecting Blocks

Connectors represent the flow of the process or system being diagrammed, and the lines contain information about the relationship between the specific source and destination blocks that they connect. In process flow simulation (PFS) diagrams, connectors are considered to represent pipes. Each pipe carries a specific type of throughput flow, and there may be multiple flow types in a diagram.

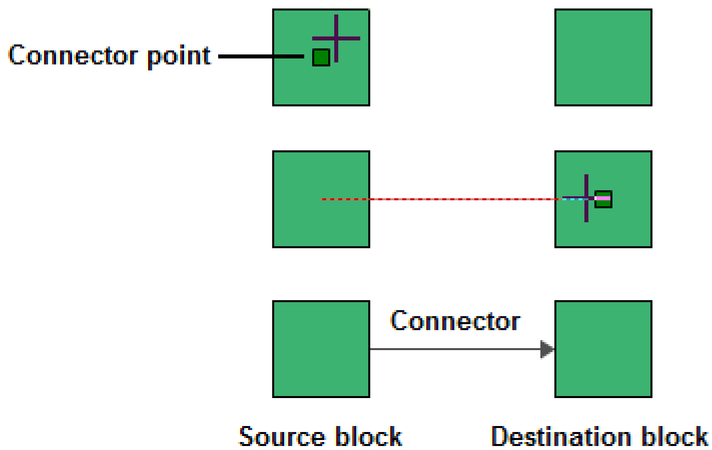
ADDING CONNECTORS

To add a connector between blocks, you can:

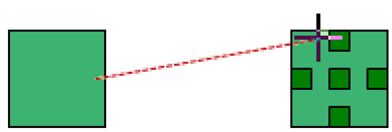
- Right-click the diagram (not blocks) and choose **Connect Blocks** or choose the command in the ribbon.
- Hold down the **ALT** key.

In PFS diagrams, use the down arrow on the **Connect Blocks** ribbon command to specify which flow type you are currently adding before using either of these methods.

In both methods, the pointer will change to display small crosshairs. Click the source block, hold down the left mouse button and drag a line from the source block to the destination block. When the crosshairs are located over the connection point box, release the mouse button to create a connector.



Note that existing connectors can be dragged to different destination blocks, if needed. Also note that the curved, custom curve and custom angled connector types give you the option to connect to the top or bottom of a block, if desired. When this option is available, the blocks will show additional connection point boxes when you create or move a connector.



In BlockSim RBDs, you can add multiple connectors at once by selecting either multiple source blocks or multiple destination blocks and then adding a connector. By default, you will see a message allowing you to choose to add the connections for all of the blocks in the selection. If desired, you can select to stop showing the message in the future and to always perform the action you select going forward. This can be changed in the future by reselecting the **Ask for confirmation when connecting multiple blocks** option on the Other Settings page of the BlockSim Application Setup. Note that if you clear this option manually within the Application Setup, the default behavior (adding the connection only to the block you actually placed the connector on) will be used.

Starting in Version 20.0.2, you can also connect multiple blocks in fault trees. This adheres to the same constraints that apply to all fault trees; please refer to the topics on each block type (listed here) and to Fault Tree Analysis Constraints for details. No error message is shown for invalid connections when connecting multiple blocks, but only valid connections will be added. The Application Setup option mentioned also above applies in this case.

In fault trees, you will normally connect multiple source blocks to one destination block. If you attempt to reverse this, the software will connect the blocks in the direction that makes logical sense (in much the same way that when you connect a single OR gate to a single OR gate, the gate on top will always be the destination).

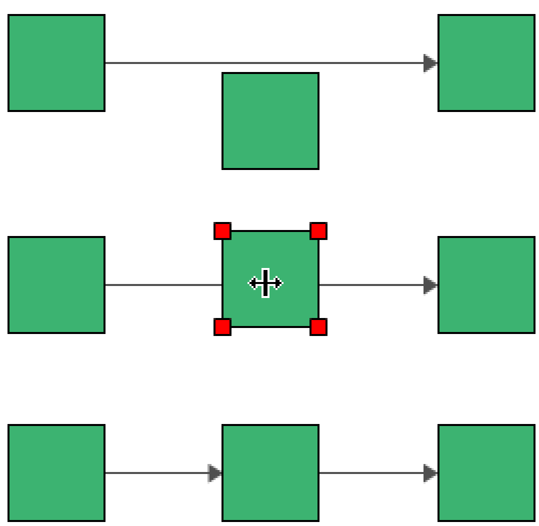
STOP ADDING CONNECTORS

To stop adding connectors and return the pointer to its normal mode, release the **ALT** key or click inside the diagram.

Tip: If you do not click the diagram or clear the **Connect Blocks** option to return the pointer to its normal mode, you will not be able to perform certain other activities, such as moving or deleting blocks. If you are experiencing difficulties with the application, make sure that the pointer is in its normal mode.

INSERTING BLOCKS/SPLITTING CONNECTORS

RBDs, PFS diagrams, Markov diagrams and event analysis flowcharts allow you to insert a block into the flow of the diagram without having to delete and add connectors. Simply drag an unconnected block onto an existing connector. When the pointer changes to a double-headed arrow, drop the block to split the existing connector and insert the block.



CONNECTOR TYPES

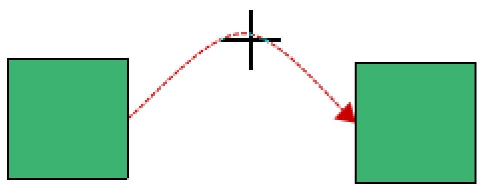
There are several types of connector that can be used in a diagram. The simplest is a straight line but there are options for adding line bends to create curved or angled lines. For details about available connector styles and defining preferences, see Connector Style Settings.

Using Curved, Custom Curve or Angle Connectors

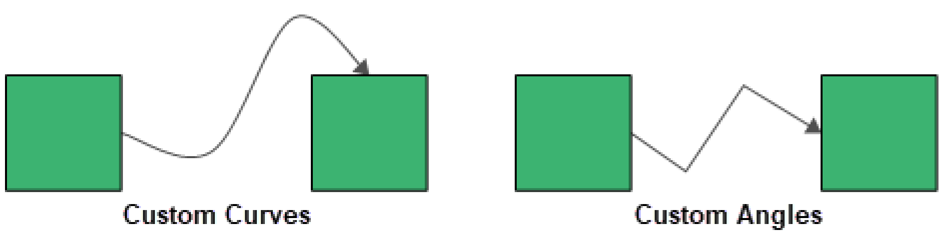
When using curved, custom curve or custom angle connectors, you can choose the location where they connect to the blocks (top, middle or bottom) and add one or more bends to the line. These options can be used separately or in combination.

To add or modify line bends:

- **Curved connectors** have a single bend point in the center of the connector. To change the angle, click and drag the point to the desired location.



- For **custom curve and custom angle connectors**, add line bend(s) by clicking anywhere on the connector and dragging the selected point to the desired location.



These custom bends can be removed by right-clicking the connector and choosing **Remove all Bend Points** or, to remove a single bend point, by right-clicking the bend point and choosing **Remove Bend Point**.

Arranging and Resizing Blocks

There are a number of commands that can be used to arrange and resize the blocks in a diagram. They can be used separately or in combination.

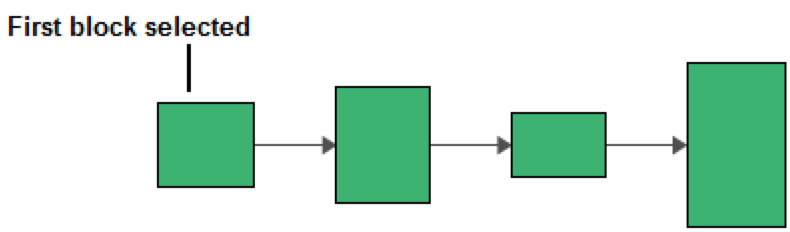
- To select multiple blocks, select the first block and then hold down the **CTRL** key while clicking each of the other blocks.

- If you apply a command and don’t like the result, you can use **CTRL+Z** to return to the original layout.

ALIGN BLOCKS

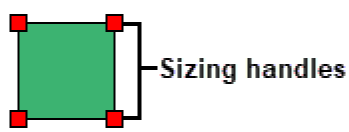
To align the blocks (align by center, top, bottom, left, etc.), select the block you want to use as the reference alignment position then select the other blocks. Right-click inside one of the blocks and choose **Align Blocks** then the desired alignment style.

The selected blocks will be aligned with the first block selected. For example, if you choose **Align Middles**, the blocks will be aligned to the vertical center of the first selected block, as shown next.



RESIZE

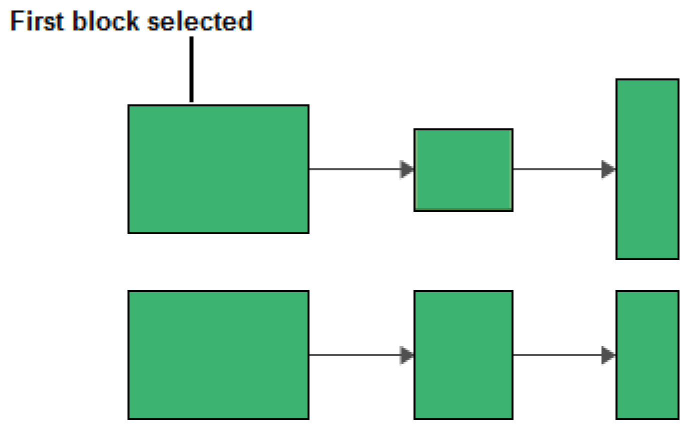
Select the block then click one of the sizing handles and drag it to the desired size.



MAKE SAME SIZE

To make the blocks the same size, select the block that has the desired dimension(s) then select the other blocks that you want to resize. Right-click inside one of the blocks and choose **Make Same Size** then choose to match the width and/or height of the first selected block.

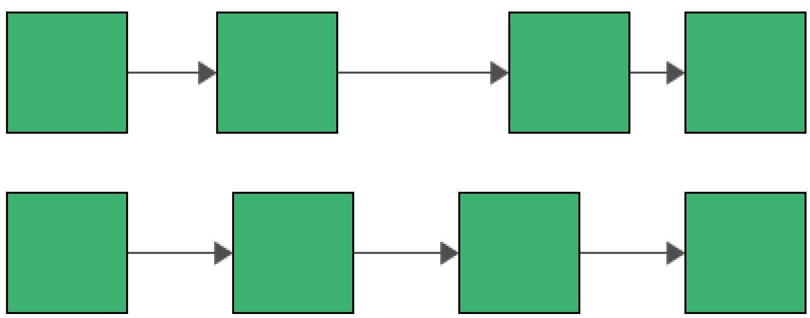
For example, if you choose **Height**, the blocks will be resized to the same height as the first selected block.



MAKE SPACING EQUAL

To adjust the spacing between blocks, select the blocks that will be affected (the selection order does not matter). Right-click inside one of the blocks and choose **Make Spacing Equal** then the desired direction.

The selected blocks will be moved so that they are equally spaced over the original size of horizontal and/or vertical area.



AUTO ARRANGE

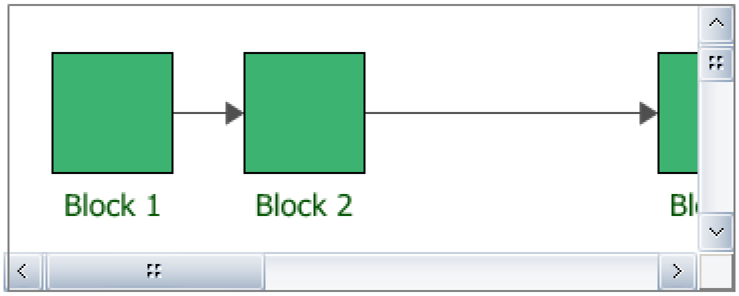
To automatically arrange the blocks in the diagram so that all blocks are placed in logical order depending on their connections, right-click the diagram (not blocks) and choose **Auto Arrange**.

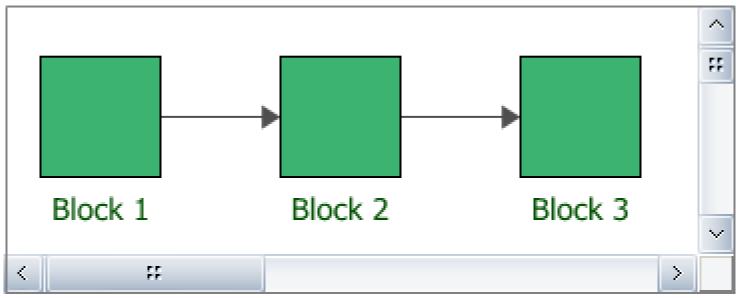
Note that for Markov diagrams in BlockSim, you also need to select the starting block that is going to be the top left block in the diagram.

The space between blocks depends on the settings specified in the Background and Grid page of the Diagram Style window.

AUTO FIT WIDTH

In some BlockSim diagrams, you can also choose **Auto Fit Width** to automatically arrange the blocks in the diagram to fit evenly spaced within the width of the diagram window.





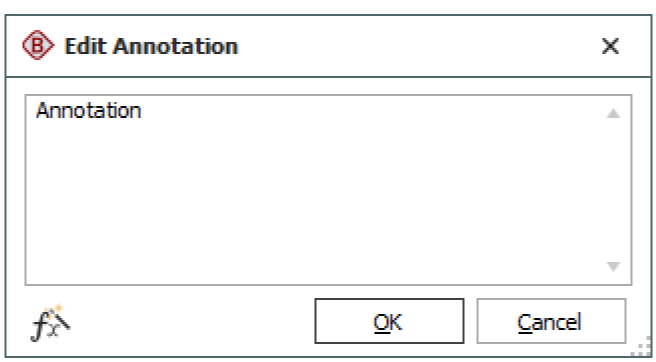
Adding Annotations

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, right-click the diagram (not blocks) and choose **Add Annotation**.

The annotation block will appear in the diagram.

By default, the text "Annotation" will appear in the block. You can change this text in the Edit Annotation window, which is accessed by double-clicking the annotation block.



In BlockSim diagrams, you can click the **Function Wizard** icon to insert a function that provides information about the diagram.

If you are working with an analytical diagram, you can insert:

- General Functions, which return basic information about the database and project, such as the project name or the current date.
- Data Source Functions, which return information based on the model used with the specified data source, such as the acceleration factor or the reliability at given time.

If you are working with a simulation diagram, you can insert:

- General Functions, which return basic information about the database and project, such as the project name or the current date.
- Simulation Functions, which return information about the simulation results and settings.

Note that if you want to add additional functions to the same annotation box, click the **Function Wizard** icon, again.

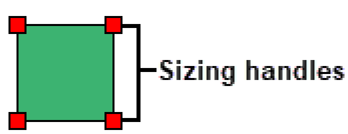
Keyboard and Mouse Combinations

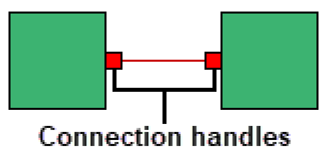
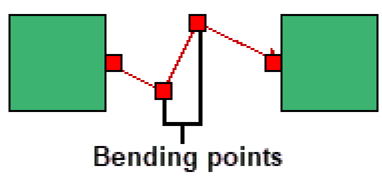
This topic describes the various keyboard and mouse key combinations that you can use while working with diagrams.

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. Note that while you can include Standby Containers and Load Sharing Containers in a selection, any changes you make, such as resizing the containers, do not apply to the blocks or connectors in the containers. If you start a selection inside a container, that selection can include only those blocks and connectors within that container.

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.

Resize block(s) by 1-unit increments	Select the block(s) then press **SHIFT** and the desired arrow key. The up and down arrow 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 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, refer to the Connector Style Settings topic.	 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**.

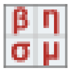
Scroll horizontally to the far right	Press **END**.
Scroll to the upper left corner of the diagram	Press **CTRL+HOME** or **CTRL+PAGE UP**.
Scroll to the lower right corner of the diagram	Press **CTRL+END** or **CTRL+PAGE DOWN**.
Scroll by the size of the visible area of the diagram	With nothing selected, press **CTRL** + the desired arrow key. For vertical scrolling, you can also press **PAGE UP** and **PAGE DOWN**.
Move the diagram in any direction (pan)	Press **SHIFT**, click the diagram and drag. This is like dragging the diagram sheet itself (e.g., to see objects lower in the diagram, you would drag the diagram upward).
Other	
Zoom in or out	Press **CTRL** and rotate the mouse wheel.
Exit the Format Painter or Connect Blocks modes	Press **ESC**.
Create a copy of a block and move the copy to the desired location	Press **CTRL+SHIFT** and click the block then drag to the desired location, either within the diagram or in another diagram of the same type.
Add blocks to the mirror group currently shown in the Mirror Group Manager window	Press **CTRL+SHIFT** and drag the block from the diagram into the Mirrored Blocks area of the Mirror Group Manager.

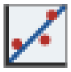
Analyzing Diagrams

Weibull++ Diagram Control Panel

The diagram control panel in Weibull++ contains the following options.

MAIN PAGE


Analyze analyzes the relationships of the components in the diagram and puts the reliability-wise configuration into a mathematical formula.

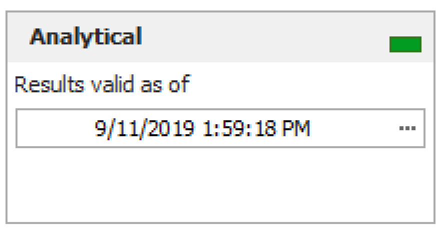

Plot creates a plot based on the analysis results. Clicking the **Plot** icon before the diagram has been analyzed will automatically perform the analysis and then plot the data. (See Plot Utilities.)


QCP opens the Quick Calculation Pad (QCP), which allows you to obtain reliability results based on the algebraic solution for the diagram.


Transfer to BlockSim is available only if BlockSim is activated on your computer. It exports the blocks into a new BlockSim simulation diagram. All the blocks must be associated with a published model before they can be exported.

- The **Analytical** area displays the status and results of the analysis. If the light is green, as shown below, 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.



- The **Information** area displays information for the diagram (if a block is not currently selected) or for the selected block.

RBD	
Number of Blocks	7
Created	
HBM Prenscia	
Date	7/17/2019
Time	11:04 AM
Modified	
HBM Prenscia	
Date	9/11/2019
Time	2:00 PM
Comments	

Standard Block	
Data Sheet	Data1
Failure Distribution	2P-Weibull

DATA SHEETS PAGE

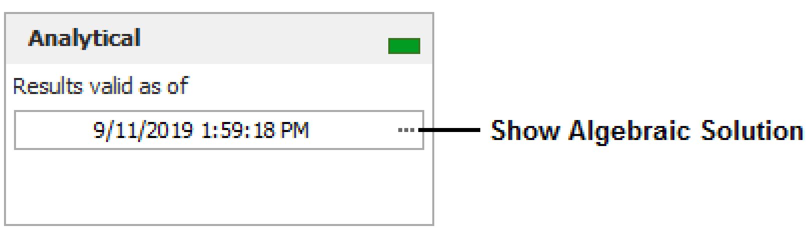
The Data Sheets page contains a list of all of the data sheets in the project that are available for use in the diagram.

You can drag a data sheet into the diagram to create a standard block. You can use any data sheet in any life data or life-stress data folio, as long as it has been calculated and is not using one of the competing failure modes distributions.

Equation Viewer

In Weibull++ diagrams, the Equation Viewer shows the algebraic equation for the system reliability, *pdf* and failure rate as a function of the block reliabilities.

To open the Equation Viewer, click the **Show Algebraic Solution (...)** icon on the control panel.



There are two ways to view the system reliability equation: as a symbolic equation or as the complete equation. With either view, you can:

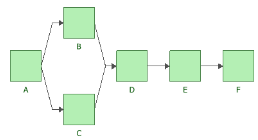
- Select **Show Legend** to display the failure distribution for each block in the diagram.
- Select **Show Colors** to use color-coding in the window to make the equations easier to view.

If you have made changes to these settings, click **Refresh** to update the contents of the window.

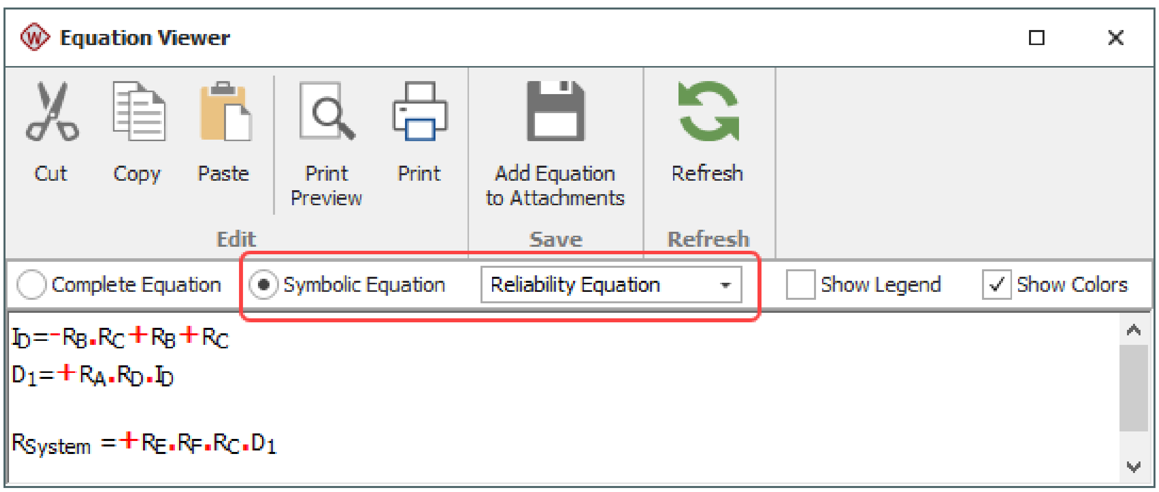
You can click **Add Equation to Attachments** to save the equation to the project attachments as an embedded *.rtf file.

SYMBOLIC EQUATION VIEW

The symbolic equation view uses tokens to represent portions of the equation. You can 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:

$$R_{System} = +R_E \cdot R_F \cdot (+R_A \cdot R_D \cdot I_D) \quad (4)$$

Now Eqn. (4) contains the token *ID*, which is defined in Eqn. (1). Substituting this into Eqn. (4) yields:

$$R_{System} = +R_E \cdot R_F \cdot (+R_A \cdot R_D \cdot (-R_B \cdot R_C + R_B + R_C)) \quad (5)$$

COMPLETE EQUATION VIEW

When you use the complete equation view, the software automatically performs all token substitutions. 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.

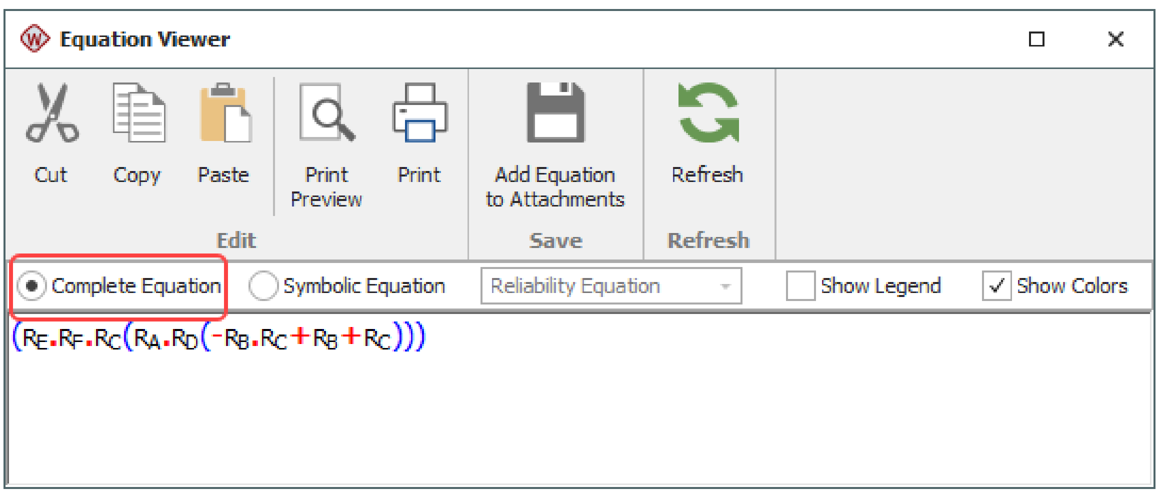


Diagram Skins and Appearance Settings

Most of the appearance settings in a diagram can be customized to meet your particular preferences.

Diagram Skin – sets (or resets) all of the settings in the Diagram Style for a particular diagram.

Diagram Style – contains all of the style settings for the current diagram (e.g., background color and grid style) as well as default settings for the blocks and connectors within this diagram.

Block Style and **Connector Style** – allow you to override the default settings for specific blocks or connectors, if desired.

For consistency and ease of use, the simplest option is to choose a preferred skin and set it as the default for your computer. Each new diagram you create will have the preferred settings applied automatically. If desired, you also have the option to change settings for a particular diagram, and/or for particular blocks or connectors.

Tip: If you use the Block Style or Connector Style windows to modify the settings for specific blocks or connectors, those settings will no longer be linked to the default settings in the Diagram Style and they won't be updated if you change the skin. If you want the block/connector settings to be determined by the diagram style/skin, make sure they are set to "Diagram Default."

Using Diagram Skins

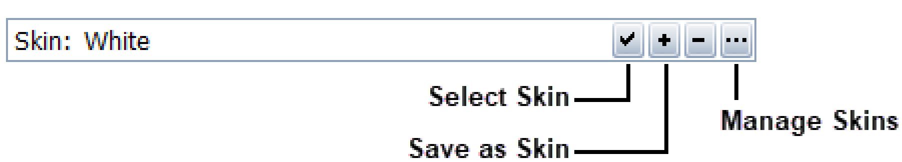
A diagram skin provides a quick and easy way to set (or reset) all of the settings in the Diagram Style window for a particular diagram.

Diagram skins are universal across all ReliaSoft applications. This means that a single skin can contain the appearance settings for all types of diagrams in BlockSim, Weibull++ and XFMEA/RCM++. However, you will only be able to see/modify the settings for the type of diagram that you are currently working with.

The universal skins are stored with a *.dskin extension at C:\Users\Public\Public Documents\ReliaSoft\Skins. If you have already customized skins from a prior version, see Converting Version 9 Skins.

In order to create, modify or manage the skins on your computer, you must first open a diagram then right-click and choose **Diagram Style**.

The commands that you will use are located in the lower-left corner of the window.



What's Changed? Starting in Version 18, you cannot modify the skins that are installed with the software. Any changes that you make must be saved as new skins with their own names.

SELECT SKIN

To apply an existing skin to the current diagram:

1. Click the **Select Skin** (✓) icon.
2. Choose a skin from the list and click **OK**. The name of the selected skin will be displayed in Skin field.
3. Click **OK** to close the Diagram Style window and apply the relevant settings to the diagram.

If you don’t like the result, you can undo the action by pressing **CTRL+Z**, or simply return to the Diagram Style window and select a different skin.

MODIFYING OR CREATING A SKIN

1. Open a diagram that has the settings you want to reuse or create the diagram. Open the Diagram Style window.
2. Make your changes. For details on the settings that can be changed, see the Diagram Style Settings.
3. Click the **Save as Skin** (+) icon. Enter a name for the skin and click **OK**. If you specify a name that already exists, you’ll be asked to confirm that you want to replace the settings for this particular diagram type.

   Note: Skin names cannot contain the following characters: < > | / \ ; ? : * or quotes. The names are not case sensitive.

Skins are stored with a *.dskin extension in the Skins folder (C:\Users\Public\Public Documents\ReliaSoft\Skins).

SAVE AS SKIN

To use the settings in the current diagram to create a new skin, or to update an existing skin:

1. Click the **Save as Skin** (+) icon.
2. Enter a name for the skin and click **OK**. If you specify a name that already exists, you'll be asked to confirm that you want to replace the settings in the existing skin for this particular diagram type. (The settings that are specific to other diagram types will remain unchanged.)

MANAGE SKINS

To view or manage the skins that are available to use on your computer, click the **Manage Skins** (**…**) icon. From here you can:

- Click **Import** to browse for *.dskin files in another location (e.g., skins that were created by another user or on a different computer) and copy them into the designated folder for your computer.

> ***Tip:*** You can also copy/paste *.dskin files directly into the C:\Users\Public\Public Documents\ReliaSoft\Skins folder. However, they won't appear in the skins list until you restart the application.

- Select a skin and click **Rename**, then specify a new name. This will change the name of the *.dskin file in the designated folder for your computer.
- Select a skin and click **Delete**. This will remove the *.dskin file from the designated folder on your computer and cannot be undone.
- Click **Set Defaults** and then specify which skin will be used by default for each new diagram you create.

Converting Version 9 Skins

Starting in Version 10, diagram skins are universal to all ReliaSoft applications that use diagrams. This means that a single skin can contain the appearance settings for all types of diagrams in BlockSim (and, for versions prior to 2020, RENO), Weibull++ and XFMEA/RCM++.

Many of the predefined skins installed with Version 9 applications have already been merged and converted into universal skins that are ready to use in any diagram type in later versions. For example, the new White.dskin replaces the old RBDWhite.dskinrbd, FTDWhite.dskinftd, etc.

If you created your own customized skins in Version 9, this topic explains how to convert them for use in later versions.

PREPARING THE VERSION 9 SKINS FOLDERS

The conversion utility considers the old Version 9 skins that are stored in the following locations:

- C:\Users\Public\Public Documents\ReliaSoft\Skins\BlockSim9
- C:\Users\Public\Public Documents\ReliaSoft\Skins\Xfmea9
- C:\Users\Public\Public Documents\ReliaSoft\Skins\RCM9
- C:\Users\Public\Public Documents\ReliaSoft\Skins\Weibull9

Before running the utility, you can:

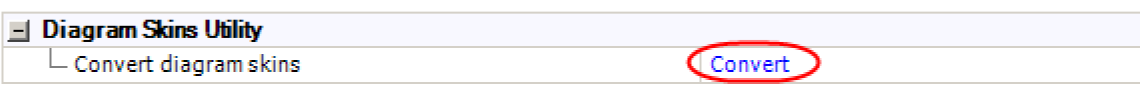
1. Delete any Version 9 skins that you don't wish to use or convert. (This is not required but it may help to avoid confusion when you run the conversion utility.)
2. For any Version 9 skins that you wish to combine into the same universal skin, rename them to the same filename (while still retaining the different file extensions).

   For example, if you want to combine RBDCustom.dskinrbd and FTDCustom.dskinftd into the same universal skin, you could rename the files to Custom.dskinrbd and Custom.dskinftd.

RUNNING THE CONVERSION UTILITY

After you have prepared the Version 9 skins folders as described above:

1. Choose **File > Application Setup**.
2. On the Common Settings > Other page, click **Convert**.



3. The Convert Skins window will display all of the unique filenames that remain in the Version 9 skins folders. (If you want to combine multiple skins into the same new universal skin, make sure you have already renamed the files as described above.) Select the skins you want to use in later versions and click **Convert**.
4. Close the application so the skin(s) will be added to the Skins folder.

The new file(s) are stored at C:\Users\Public\Public Documents\ReliaSoft\Skins with a *.dskin extension and "_V9" appended to the name.

A NOTE ABOUT DIAGRAM BACKGROUNDS

Universal skins contain appearance settings for diagram background and grid, and connector and block handles that are applied to all diagram types. When you create a universal skin from individual skins with different background/grid/handle settings, only one set of these appearance settings will be stored in the skin. The settings that are saved into the universal skin are from the last skin that was merged. If these are not the settings you desire, you can use the Diagram Style window to change them.

Diagram Style Settings

In the Diagram Style window, you can define diagram-level appearance settings, and default settings for the blocks and connectors within the diagram. These settings can be saved in a diagram skin and reused with other diagrams.

To open the Diagram Style window, right-click the diagram (not blocks) and choose **Diagram Style**.

This window consists of multiple pages. The number and type of pages depend on the type of diagram that you are working with. Settings that can be defined are divided into two groups: Diagram and Defaults.

- At the diagram level, you can define the appearance of the diagram sheet and those aspects of the appearance of diagram components (i.e., blocks and connectors) that cannot be configured at the individual component level.
- You can define the default appearance settings for diagram components (i.e., connectors, blocks and annotations). See Defaults for Blocks and Connectors.

BACKGROUND AND GRID

You can customize the background and grid appearance of the diagram.

- **Grid Style**: Whether to display grid lines and, if so, the style of the lines (e.g., solid, dash, etc.).
- **Grid Spacing**: The size (width and height) of the grid squares, in inches.
- **Shadow**: The size and color of the shadow "cast" by blocks in the diagram. Note that this applies to all block types, but the shadow will be shown only if the **Show Shadow** check box is selected on the Defaults > General page for that block type.
- **Colors**: The colors used for the background, grid lines and page separator lines.
- **Auto Arrange**: The horizontal and vertical spacing between blocks, in inches, that will be used if you choose the Auto Arrange ribbon command.

  For fault trees in BlockSim, you can also select the **Always Auto Arrange** check box to turn on Auto Arrange by default.

- **Block Caption ID**: Whether to display the block ID, which is an internal ID number assigned to each block when it is created. You can choose not to display the ID or to display it in decimal or hexadecimal format. If it is displayed, it will appear directly below the block name (i.e., its location, alignment and font will be dictated by the properties that are specified for the block caption).

CONNECTOR HANDLES AND INDICATORS

You can customize the following diagram-level connector properties. These settings apply to all connectors of all types within the diagram and cannot be overridden at the connector level.

- If you select **Show Connectors Over Blocks**, any connectors that cross blocks will be shown over (i.e., in front of) those blocks. Otherwise, the blocks will be displayed on top.
- Where connectors attach to blocks if the blocks are not in direct horizontal or vertical alignment. If the **Automatic Connection Preference** field is set to **Left-Right**, the connector will attach to the sides of the blocks; if it is set to **Top-Bottom**, the connector will attach to the top and bottom of the blocks. If **No Preference** is chosen, the connector will attach to the closest edges.
- The **colors** used for:
  - All connector handles and selected connectors.
  - Connectors to/from highlighted blocks (i.e., blocks that are part of a multiple selection).
  - Connectors leading to or from a selected block. This can be helpful in visualizing how a block is connected within a complex diagram.
- The **size of the arrow head**, in inches.
- The **thickness** of connectors in a traced path.

BLOCK HANDLES AND GENERAL INDICATORS

You can customize the following diagram-level block properties. These settings apply to all blocks of all types within the diagram and cannot be overridden at the block level.

- The properties for a **highlighted block**, including:
  - The **colors** used for the handles that appear when you are resizing a block or connecting blocks.
  - The colors used for the caption, border and fill (i.e., the area background).
  - The fill style (i.e., how the color is applied to the area).

- **Flag settings**:

  In BlockSim, can be used to indicate anything you want. For example, each flag could represent a level of analysis that has been completed. These flags are not used by BlockSim for analyses or simulations.

  In Weibull++, the block flags are used to mark blocks that represent a data sheet that has been changed and has not been calculated or a data sheet that has since been deleted.

  - Block flag size: Block flags, which are added by selecting one or more blocks, right-clicking one of the blocks and choosing **Set Block Flag**.
  - Block flag borders: The style (e.g., solid, dash, etc.), color and thickness of the border for block flags.

- **Cross out lines for blocks**: In BlockSim diagrams, the style, color and thickness of the line that is used to cross out blocks that are set as failed. Blocks can be set as failed by selecting the **Set block as failed** option in the Block Properties window. You can also set this option by right-clicking the block and choosing **Set Block as Failed**.

BLOCK CORNER INDICATORS (BLOCKSIM ONLY)

The Block Corner Indicators pages allow you to customize the appearance of status indicators that are displayed at the corners of blocks. Such indicators include the colored shapes that are used to indicate certain block and text labels that may appear independently or in conjunction with a shape. The available Block Corner Indicators pages will vary depending on the type of diagram that you are working with.

DEFAULTS FOR BLOCKS AND CONNECTORS

Default appearance settings can be defined for diagram components (i.e., connectors, blocks and annotations. A diagram skin can hold the appearance settings for all diagram types (e.g., RBD, fault tree, process flow, etc.); however, you can only modify the settings for the diagram type you are working with. To define these settings, see:

- Block style settings
- Connector style settings

Block Style Settings

The block style settings can be defined at the diagram level in the Diagram Style window or at the block level in the Block Style window. The block-specific settings defined in the Block Style window have priority over the default settings defined in the Diagram Style window for the diagram you are working with. For details on how the diagram-level and connector-level settings relate to each other, see Appearance Settings.

Diagram level: To open the Diagram Style window, right-click the diagram (not blocks) and choose **Diagram Style**.

Block level: To open the Block Style window, select one or more blocks, right-click one of the blocks then choose **Block Style**. Note that if you are using this window, you must clear the various **Use Default** check boxes in order to modify some settings.

GENERAL PROPERTIES

Block Properties

- **Shape and shadow**: The options available in the **Shape** drop-down list depend on the block type. You can change the shape of standard blocks in RBDs, events in fault trees and all blocks in process flow and cause/effect diagrams; all other block types have permanently assigned shapes. You can add a shadow to any available shape.
- **Size**: Unlike the other options on this page, changes to this setting made at the diagram level will apply only to blocks subsequently added to the diagram.
- **Color and style**: The background color is the underlying color applied to the block; the fill color is applied over the background color in the pattern specified by the fill style.

Caption Properties

The caption text comes from a block name in BlockSim, folio name in Weibull++ or property name in XFMEA/RCM++.

- **Location and Horizontal Alignment**: Specify where the caption will be displayed (e.g., under the block, inside the block, etc.) and whether it will be left aligned, center aligned or right aligned.
  - If you choose to place the caption to the left, right or inside the block, you can also specify the **Vertical Alignment** (i.e., at the top, center or bottom of the block).
  - If you choose to hide the caption, the Alignment fields will not be available.
- The **width** and **height** of the area used for the text. You can enter a fixed value for either or both of these. If **Block Width** is selected in the **Width** field, the caption text will wrap, if necessary, up to the vertical limit set in the **Height** field. If **Auto Height** is selected in the

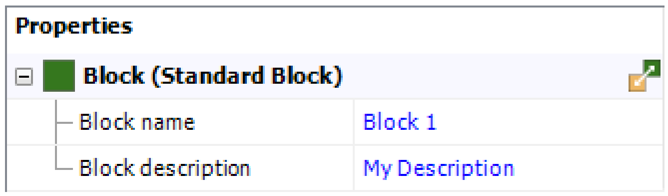
Height field, no limit is placed on the height of the text area.

- The **color** and **font** used for the text.

DESCRIPTION BOX PROPERTIES

These properties are available for BlockSim blocks only.

The description box displays the text entered in the Description field of the Block Properties window.



Description Box Properties

- Box **width** and **height**: You can enter a fixed value for either or both of these or you can set them to be the same as the settings used for the caption area.
  - If **Block Width** is selected in the Width field, the description text will wrap, if necessary, up to the vertical limit set in the Height field.
  - If **Auto Height** is selected in the Height field, no limit is placed on the height of the text area.
- Box **color** and **style**: The background color is the underlying color applied to the description box; the fill color is applied over the background color in the pattern specified by the fill style.

Description Text Properties

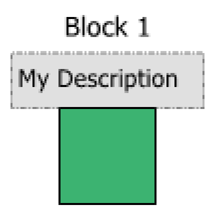
- Text **Location** and **Horizontal Alignment** allow you to specify where the description area will be displayed (i.e., above the block, inside the block or hidden) and whether it will be left aligned, center aligned or right aligned.
  - If you choose to place the caption inside the block, you can also specify the **Vertical Alignment** (i.e., at the top, center or bottom of the block).
  - If you choose to hide the description area, the Alignment fields will not be available.

  Note that if you place the description inside the block, only the description text will be shown. The settings for the surrounding area (i.e., background color, fill color and border properties) will not be used. The properties for the background color fill color and fill style of the block will be used instead.

- The **color** and **font** used for the text.

Using the Same Location Settings for Captions and Descriptions

Block captions and descriptions both have the option to place the text above or inside the block (and other locations, for captions). If you choose the same location for both, the description text will display under the caption text.



IMAGES

To load an image:

1. Clear **Use default image**, if applicable.
2. Click **Load**, and browse for and select the desired image file.
3. Use the **Position** drop-down list to specify how the image will be placed on the block (e.g., stretched to fit inside the block's current dimensions, centered, etc.). If you select stretched, you can also define the margins you want.
4. Select the **Transparency Mask Color** check box to choose a color that will be shown as transparent when the image is displayed on the block.

   Note: If you choose to use an image that is larger than 800x600 pixels, you will be given the option to down sample the image in order to reduce its size and alleviate possible delays related to database size.

5. Click the **Clear** button to remove the loaded image and use no image at all.

RETURNING BLOCK-LEVEL SETTINGS TO DIAGRAM DEFAULTS

To use the diagram style settings again, in the Block Style window, choose **Diagram Default** from the drop-down lists and select the **Use Default** check boxes.

Connector Style Settings

The connector style settings can be defined at the diagram level in the Diagram Style window or the connector level in the Connector Style window. The connector-specific settings defined in the Connector Style window have priority over the default settings defined in the Diagram Style window for the diagram you are working with. For details on how the diagram-level and connector-level settings relate to each other, see Diagram Skins and Appearance Settings.

Note: In BlockSim process flow simulation (PFS) diagrams, connectors are considered to represent pipes. Each pipe carries a specific type of throughput flow, and there may be multiple flow types in a diagram. For these diagrams, flow types (connector styles) can also be defined at the diagram level via the Flow Types window (right-click the diagram and choose **Flow Types**). This window works similarly to the Diagram Style window, and connector-specific settings also have priority over settings defined in the Flow Types window.

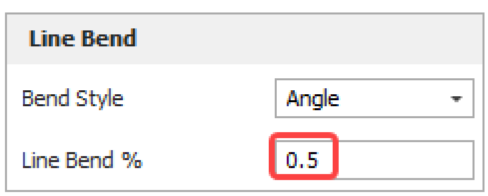
Diagram level: To open the Diagram Style window, right-click the diagram (not blocks) and choose **Diagram Style**.

Connector level: To open the Connector Style window, select one or more connectors, right-click one of the connectors then choose **Connector Style**.

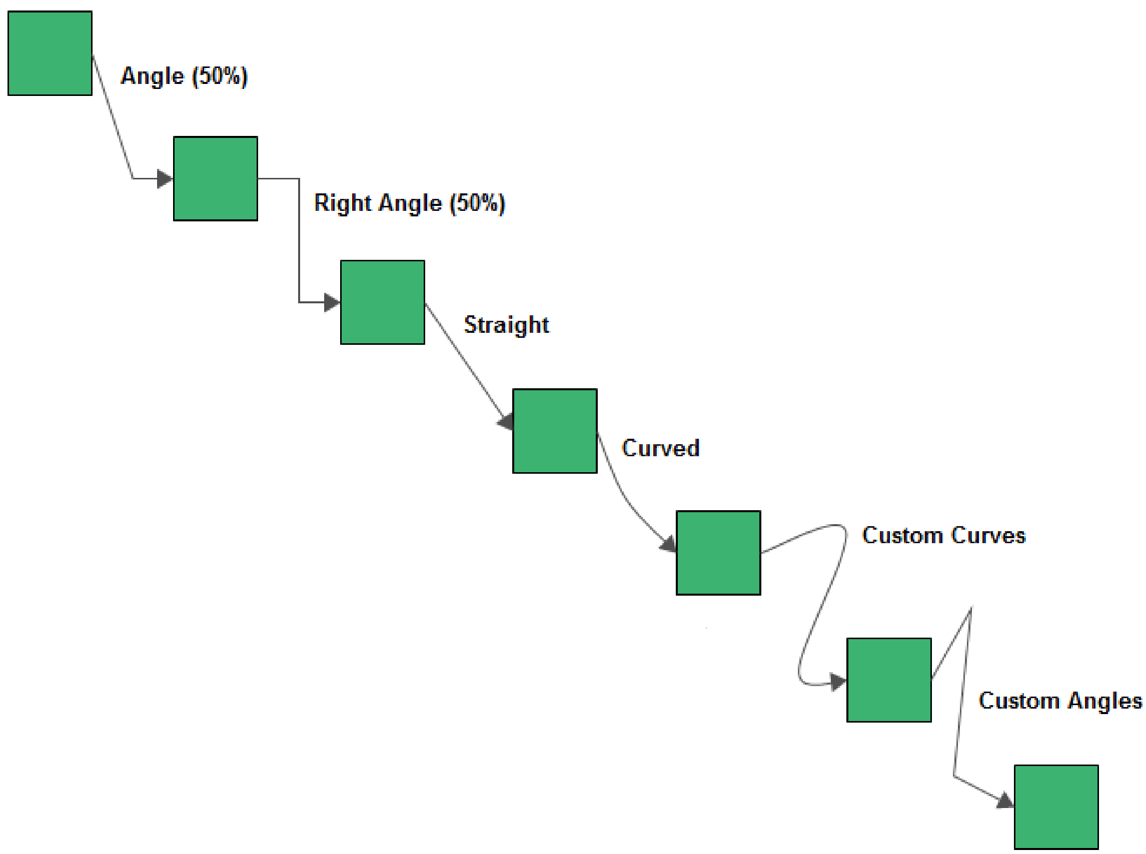
Note that if you are using the this window, you must clear the **Use Default** check boxes in order to modify some settings.

LINE SETTINGS

- **Line Settings**: Specify the connector's line style (e.g., solid, dash, etc.), thickness and color.
- **Line Bend and Arrows**: Use the **Line Bend** drop-down list to specify the kind of bend that you want in the lines. Note that if you select **Angle** or **Right Angle**, the **Line Bend %** field will be enabled. This field allows you to specify the point, in percent, at which the line will bend. For example, if you enter **0.5** (50%), an angle (bend) will appear in the line at the horizontal halfway point.

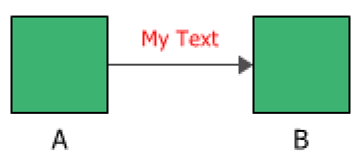


The available bend styles are shown next.



CAPTION SETTINGS

Specify the location, text color and font used for the caption. The caption text for a connector is entered in the Connector Style window.



Note that in BlockSim, you cannot enter caption text for connectors used in phase diagrams, event analysis flowcharts and Markov diagrams.

FLOW TYPE

When you are working with a BlockSim PFS diagram, the Diagram Style window and the Flow Types window also require you to specify a name (for future reference) and an abbreviation (used in the diagram, plots and results) for each flow type.

RETURNING CONNECTOR-LEVEL SETTINGS TO DIAGRAM DEFAULTS

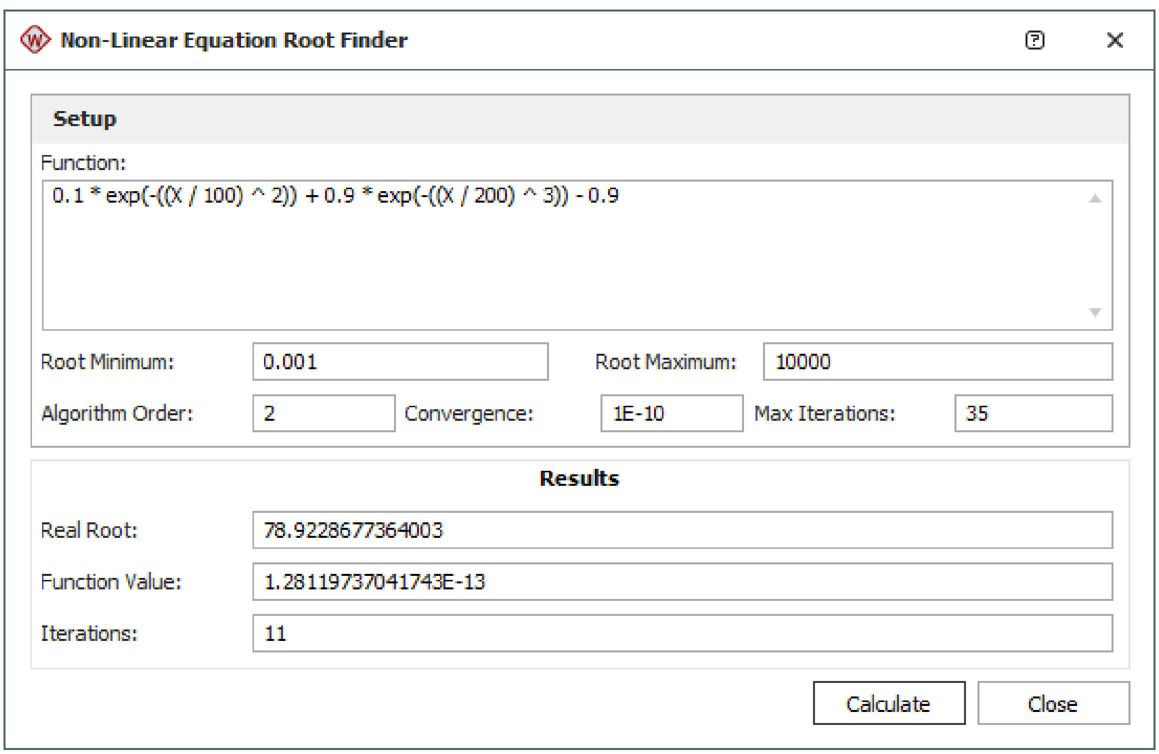
To use the diagram style settings again, in the Connector Style window, choose **Diagram Default** from the drop-down lists and select the **Use Default** check boxes.

Non-Linear Equation Root Finder

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 > ReliaSoft > Quick 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:

1. Enter the user-defined equation in the **Function** field. Use the variable *x* for the unknown variable.
2. In the **Root Minimum** and **Root Maximum** fields, enter the initial guess estimates of the root.
3. 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.
4. In the **Convergence** field, set a tolerance value at which the root of the equation should converge (i.e., $f(x) = 0$).
5. 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.
6. 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.

Analysis Tools

IN THIS CHAPTER

Batch Auto Run

The Batch Auto Run utility allows you to extract data subsets from an existing data set in a life data folio or a reliability growth data folio. The options depend on the folio type.

Life Data Folios

For life data folios, 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.

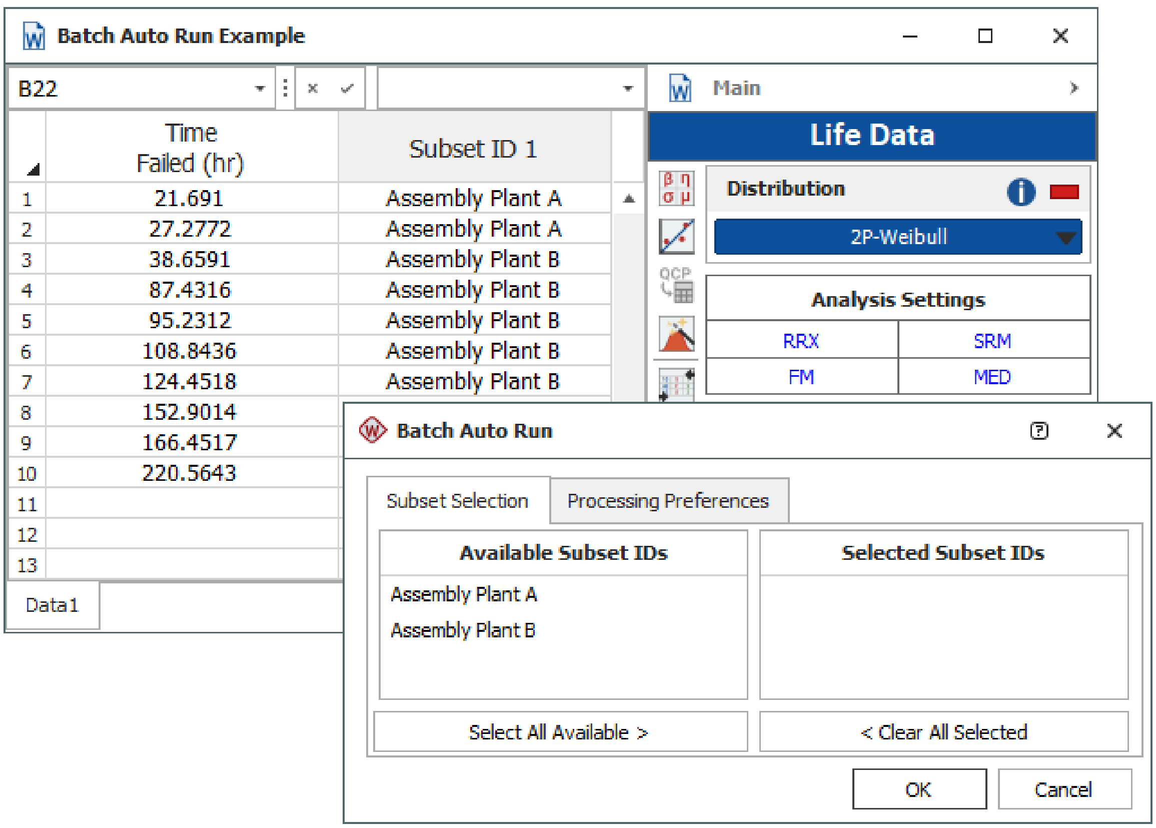
Batch Auto Run Setup

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, as shown in the following example.

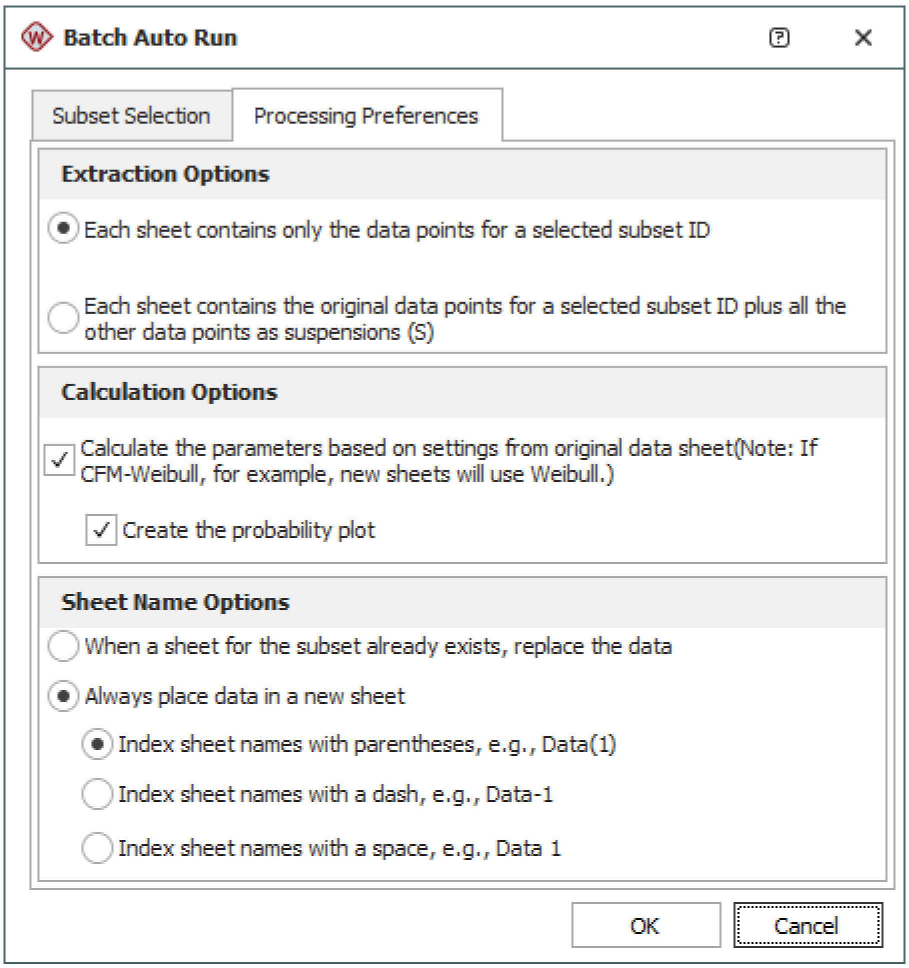


Double-click an available subset ID to include it in the batch auto run process or drag the subset ID to the **Selected Subset IDs** column. If you have more than one subset ID column in your data sheet, a drop-down list will be available to allow you to choose which column to use in the analysis.

You can also click the **Select All Available** button to automatically include all subset IDs in the column 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++ life data folio.



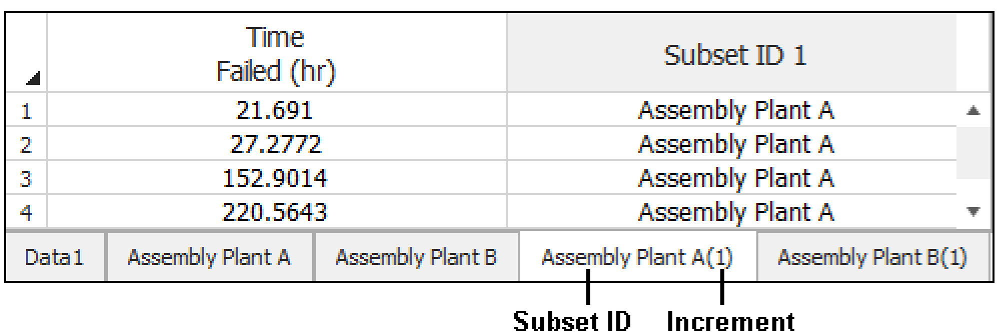
- **Extraction Options** set how the data of the selected subset IDs will be extracted. The first option simply extracts the data points for each subset ID into separate data sheets so that you can analyze the data separately.

  The second option extracts 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 Competing Failure Modes (CFM) Analysis.

- **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** specifies where 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 parentheses, a dash or a space. The following example shows the increment number separated by parentheses.



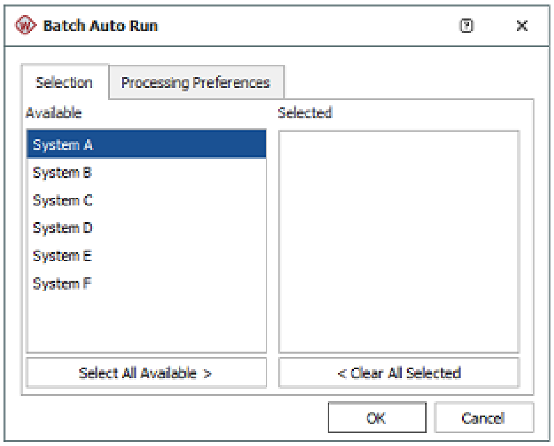
Reliability Growth Data (RGA) Folios

For fielded data or with some of the multiple systems data types (Concurrent Operating Times, with Dates and with Event Codes), you can use Batch Auto Run to extract all the data that relates to a particular system ID.

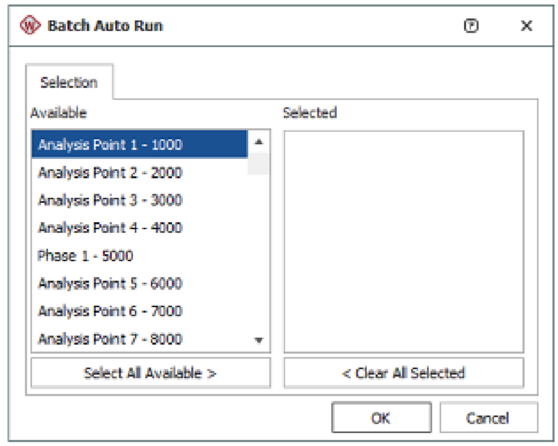
For multi-phase data, you can use Batch Auto Run to extract all the events (i.e., failures, time of implemented fixes, etc.) that occurred within a particular phase or analysis point.

To use Batch Auto Run, choose **Growth Data** > **Options** > **Batch Auto Run** or click the icon on the Main page of the control panel.

You will be presented with a list of all the available subsets in your data sheet, similar to the examples shown next.



System IDs



Analysis Points and Phases
(shows the time when each point/phase ended)

Double-click an available subset to include it in the batch auto run process or drag the subset to the **Selected** column. You can also click the **Select All Available** button to automatically include all subsets in the process.

If you are extracting data based on the system ID, you can choose to extract the selected subsets into individual data sheets by clicking the **Processing Preferences** tab and selecting the **Place each system in an individual sheet** check box. Otherwise, clear the option if you prefer to have the selected subsets appear all together in one data sheet.

Tip: If you wish to extract the first times-to-failure of all unique BC modes or unique BD modes in a data set, use the Mode Processing window.

Alter Parameters

The Alter Parameters feature is available for life data folios, life-stress data folios and reliability growth data folios. It enables you to experiment with possible alternative scenarios by altering the estimated parameters of the fitted model.

Life Data and Life-Stress Data Folios

For life data and life-stress data folios, after you calculate the parameters for your data set, you can access the tool by choosing **[Life Data/Life-Stress Data] > 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++ life data 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, 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.

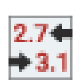
For both options, the LK values, solution line on plots and all subsequent analyses will be based on the modified set of parameters. To go back to the true parameters, click the **Calculate** icon on the folio's control panel.

Reliability Growth Data (RGA) Folios

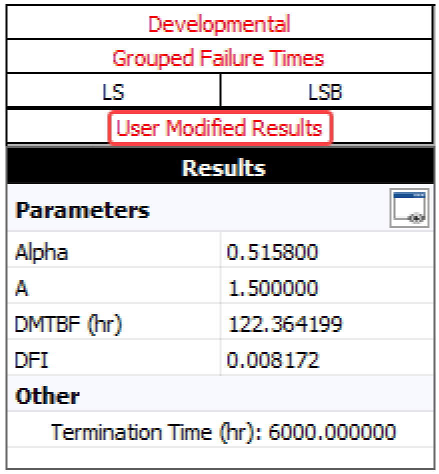
For reliability growth data folios, the feature is available for any data set calculated using any model except the Crow Extended and Crow Extended - Continuous Evaluation. Furthermore, for data sets calculated with the Crow-AMSAA (NHPP) model, this functionality is not available for Change of Slope calculations.

This feature assumes that the likelihood function and the Fisher Matrix (evaluated at the original parameter estimates) remain the same. In plots, the position of the data points remain the same but the solution line and all subsequent analyses made via the Quick Calculation Pad and other tools are based on the modified parameter values. Therefore, the analysis may be appropriate only within the context of your specific "what-if" analysis.

To use this feature, choose **Growth Data > Options > Alter Parameters** or click the icon on the Main page of the control panel.

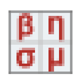


The Alter Parameters window appears. Type the new value for the parameter(s) you wish to change. The new value of the parameters will appear in the **Results** area of the growth data folio control panel with the label "User Modified Results"displaying in the analysis settings area, as shown in the following example.

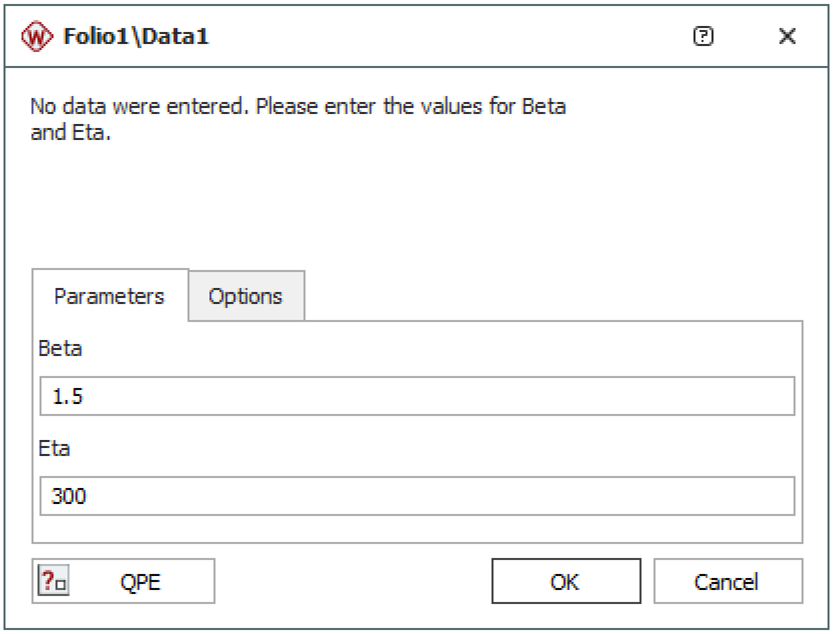


Calculations with No Data Entered

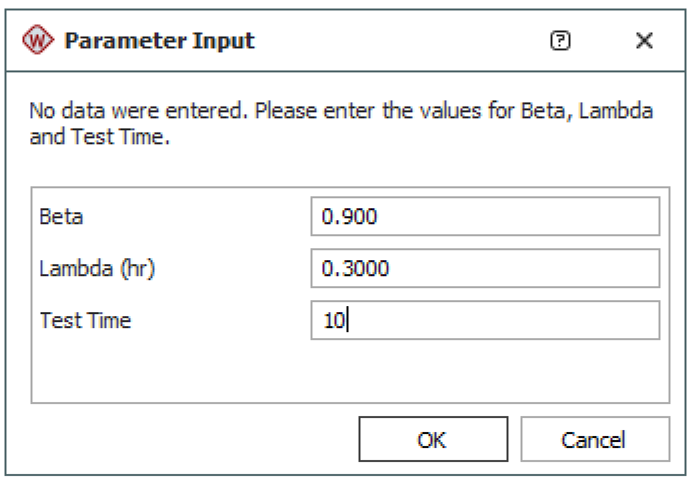
Weibull++ gives you the flexibility to generate plots and calculate metrics for reliability models without entering the data that the reliability model is based upon. To do this, create a life data folio data sheet or a growth data folio sheet and, without entering any data, choose a distribution and 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 a 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.



Options for a life data folio

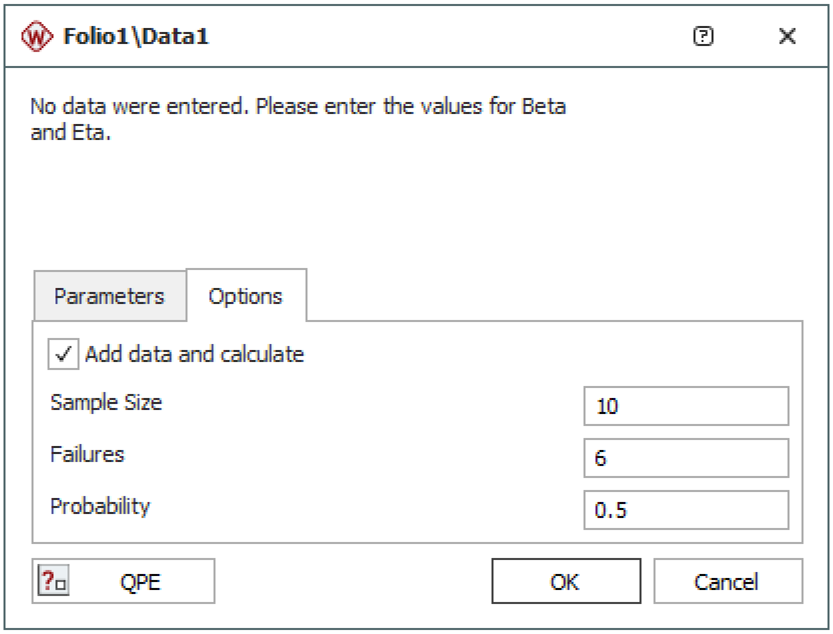


Options for a reliability growth data folio

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.

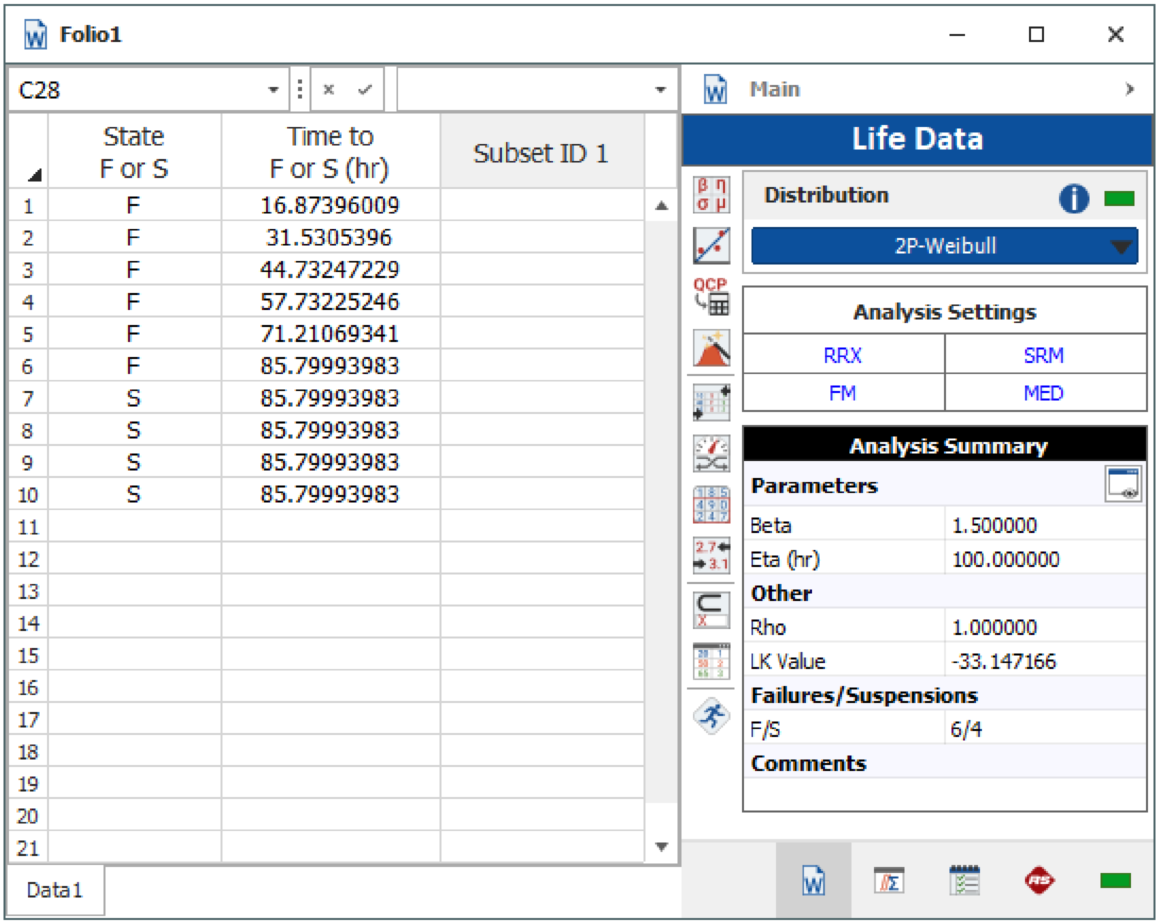
Generating a Data Set Based on the Specified Reliability Model

For life data folios, you can generate a data set based on a 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 *ith* failure will occur after the generated failure time *Ti*. 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 (T1) 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.

Create Random Data

The Create Random Data tool allows you to create randomly generated data for an analysis. This tool is available for life data folios, life-stress data folios, DOE design folios and reliability growth data folios.

To use the tool, select the range of cells for which you would like to fill with random data, then right-click any of the selected cells and choose **Random Data** on the shortcut menu. This launches the Create Random Data window, which provides the following options:

- **Columns to Fill** and **Rows to Fill** areas, select which columns/rows in the data sheet to populate with random data. If you had selected a range of cells before accessing the tool, the fields will be populated automatically.

- **Generated Data** area, choose the method for generating the random data:
  - **Random data** allows you to specify a range of numerical values from which the tool will randomly select data points. The range is specified using a **starting point** and a **variability** value (in decimals) that describe how far the values can vary from the starting point. For example, if you specified a starting point of **100** and a percent variability of **0.2**, the data points would be randomly selected from a set of values that ranges from 80 to 120.

    If you clear the check boxes, the tool will use a randomly selected starting point and/or variability for each column of data.
  - **Random values in a range** allows you to specify a minimum and maximum range of numerical values from which the tool will randomly select data points.
  - **Random points from a set of values** allows you to manually define a set of values from which the randomly selected data points will be chosen. These can be quantitative or qualitative values, each separated by a comma (with or without a space). For example, if you are randomly entering values for a column that only accepts the values "F" and "S," you would enter **F, S** in this field.

When you select one of the first two options in the **Generated Data** area, the **Decimal places** field will be enabled, which allows you to specify the number of decimal places that will be used for each randomly generated data point.

Note: Generating large numbers of stress levels in Accelerated Life Testing may affect software performance.

Auto Group Data

The Auto Group Data tool is available for life data folios, life-stress data folios and for reliability growth data folios.

Life Data and Life-Stress Data Folios

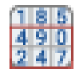
The Auto Group Data tool allows you to automatically group together identical data points. This helps to speed up calculations when you have a large data set.

- For life data folios, two data points are considered identical if their failure/suspension times AND the value/text in their Subset IDs columns are identical.

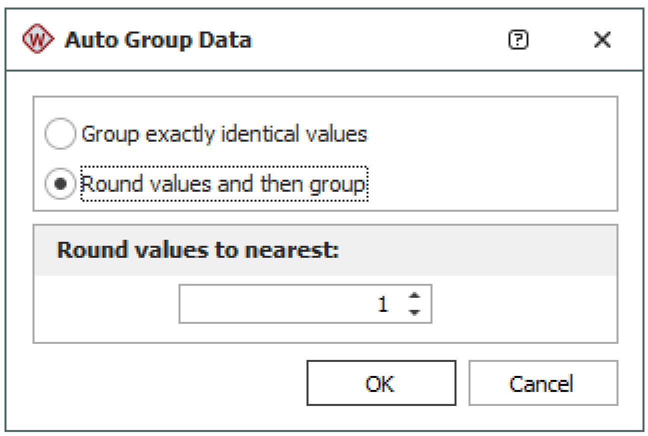
- For life-stress data folios, two data points are considered identical if their failure/suspension times AND stress levels AND the value/text in their Subset IDs columns are identical.

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.

To group your data, click the **Auto Group Data** icon on the Main page of the control panel.



You can choose to group exactly identical values, or to first round the failure/suspension times to the nearest whole number or multiple of 10, and then group the identical data.



Reliability Growth Data (RGA) Folios

The Auto Group Data tool is available for data types where the exact failure times have been recorded (i.e., Failure Times and Multi-Phase Failure Times) and for the Fleet data type.

Times-to-Failure and Multi-Phase Data

For the Failure Times and Multi-Phase Failure Times data types, you might use the Auto Group Data tool to smooth out the analysis of a data set that failed the goodness-of-fit test. Grouping the data can also be used to organize the failure times into specified intervals that show the trend in the data rather than the individual failures.

To group data, choose **Growth Data > Options > Auto Group Data** or click the icon on the Main page of the control panel.

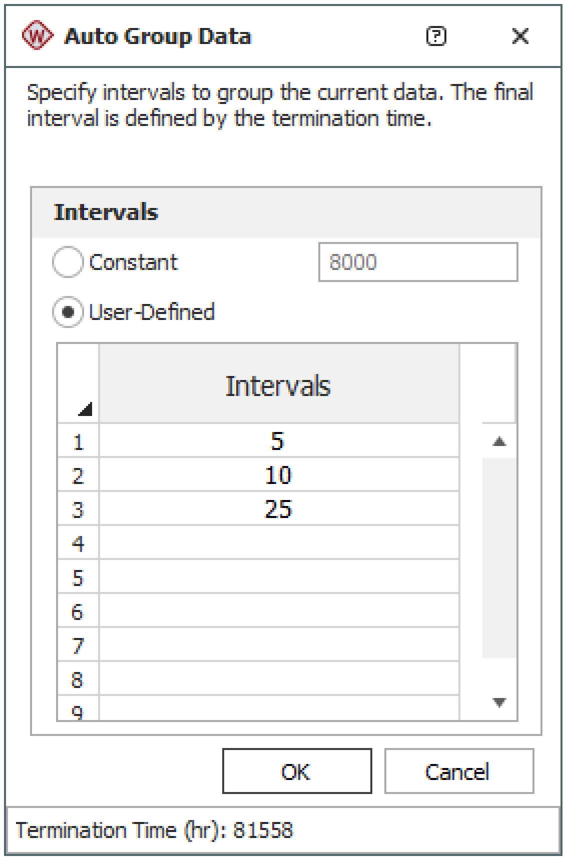
A new data sheet using the corresponding grouped data type (i.e., either Grouped Failure Times or Multi-Phase Grouped Failure Times) will be added to the same folio and populated with the grouped data.

Fleet Data

For the Fleet data type, the Auto Group Data tool appears automatically when you calculate the data sheet. In this case, the analysis stacks the failure times of each system in the data set into a cumulative timeline, and then converts it to grouped data based on the intervals that you specify. The parameters of the Crow-AMSAA (NHPP) or Crow Extended models are then calculated from the grouped data.

Grouping the Data

The following picture shows the Auto Group Data window. You can choose to specify intervals that are all the same width (constant) or intervals of different widths (user-defined). In both cases, the final interval is defined by the test termination time. The intervals must be representative of the data. The appropriate intervals will depend upon your knowledge and assumptions about the systems under test.



- Select **Constant** if you want to specify the same width for all intervals. For example, suppose the failure times are 10, 20 and 30 hours, and the test terminates at 40 hours. If you group the data by a constant width of 20 hours, the grouping will show two failures between 0 to 20 hours and one failure between 20 to 40 hours.
- Select **User Defined** if you want to use intervals of different widths in order to group the data by some qualitative characteristic or other meaningful interpretation of the data. Use the spreadsheet below the option to specify the end time of each interval. The values in the spreadsheet must increase in each subsequent row, but the intervals do not have to be of equal length. For the same example discussed above (failure times at 10, 20 and 30 hours and the test terminates at 40 hours), if you specify the intervals as 5 and 10, the grouping will show no failures between 0 to 5 hours, one failure between 5 to 10 hours and two failures between 10 to 40 hours.

Reliability Growth Analysis (RGA) Folios

IN THIS CHAPTER

Reliability Growth Data Folio

For reliability growth data analysis only.

The Reliability Growth data folio (RGA) is the most commonly used folio for reliability growth analysis. Depending on the data type selected, the folio can be used for several different types of analysis. This chapter describes general information about growth data folios that apply to most analyses.

Note: Reliability Growth data folios are available only if supported by your license. For more information, contact us.

For more focused information about performing a particular type of analysis, you can go directly to the topic of interest, which provides information about the applicable data types and models, analysis assumptions, available results, application examples, etc.

- **Traditional Reliability Growth Analysis**: This type of analysis is used for data from in-house reliability growth testing that was conducted during the developmental stages for a product. The analysis assumes that all *fixes* (i.e., permanent design improvements) are applied immediately after failure and before testing resumes, and that a reliability growth model can be fitted to the data in order to track how the reliability changes over time. The metrics of interest may include the reliability, MTBF, failure intensity, expected number of failures for a given time, and the amount of testing that will be required to demonstrate a specified reliability. Depending on the data type, the following statistical models can be used in the analysis: Crow-AMSAA (NHPP), Standard Gompertz, Modified Gompertz, Lloyd-Lipow, Duane or Logistic.
- **Reliability Growth Projections, Planning and Management**: This type of analysis focuses on how the reliability growth management strategy (i.e., which modes are fixed and when) affects the reliability growth potential of the product. Instead of assuming that all fixes are applied immediately after failure and before the observational period resumes, you can use classifications to account for different fix strategies employed for different failure modes.
  - The Crow Extended model is designed for a single test phase, and it classifies failure modes as A = no fix, BC = fixed at some time during the test or BD = fix delayed until after the end of the test.
  - The Crow Extended – Continuous Evaluation model can be used for single or multiple test phases, and it classifies failure modes as A = no fix, BC = fixed immediately before testing resumes or BD = fixed at some point after testing resumes (i.e., later in the same test phase, between test phases, in a subsequent test phase or after all test phases).
  - The growth planning folio and multi-phase plot can be used to develop an effective reliability growth test plan, and visualize the test results across multiple phases.
- **Repairable Systems Analysis**: This type of analysis is used for data from repairable systems operating in the field under typical customer usage conditions. Such data might be obtained from a warranty system, repair depot, operational testing, etc. You can use the power law or Crow-AMSAA (NHPP) models for repairable system analysis based on the assumption of minimal repair (i.e., the system is "as bad as old" after each repair) to calculate metrics such as the

expected number of failures, rate of wearout or the optimum time to replace or overhaul a system to minimize life cycle costs.

Creating a Growth Data Folio

For reliability growth data analysis only.

To add a growth data folio to a project, choose **Home > Insert > Growth Data** or right-click the **Growth Data** folder in the current project explorer and choose **Add Growth Data**.

A setup window will prompt you to select a data type to use with the folio. The data type you select determines the data entry columns that appear in the data sheet and the type of analysis you can perform (see Reliability Growth Data Types). Click **Next** to go to the next step to configure your data sheet; otherwise, click **OK** to immediately create the folio.

DATA SHEET CONFIGURATION OPTIONS

The configuration options that are available in the setup window will depend on your chosen data type:

- The **Units** drop-down list is available for all data types except the discrete and Multi-Phase Mixed data types. Use this list to select the units appropriate for your data. Units may be measured in mileage, 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 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 units of the existing data. Authorized users can define the units that will be available for use in any project within the database and set up the conversion factors.

- The **Use advanced systems view** check box is available for fielded/developmental multiple systems data types. (See Normal and Advanced Systems View.)

- The **Number of systems** field is available for fielded/developmental multiple systems data types. If you are working with the Multiple Systems - Known Operating Times data type, this field allows you to specify the number of "Time System" columns that will appear in the new data sheet. For all other multiple systems data types, this field allows you to specify the number of systems that will appear in the navigation panel of the Advanced Systems View (see Normal and Advanced Systems View).

- The **Number of comment columns** field is available for all data types. Use this field to specify the number of "Comments" columns that will appear in the new data sheet. The information in those columns will not affect the calculations in the folio.

Reliability Growth Data Types

For reliability growth data analysis only.

There are many different ways that data for reliability growth analysis (RGA) may be collected. The type of data you collect determines which types of analyses you can perform and which statistical models you can use. This chapter describes all of the different types of data supported by the software. For more focused information about performing a particular type of analysis, you can go directly to the topic of interest. (See Traditional Reliability Growth Analysis; Reliability Growth Projections, Planning and Management (Crow Extended Model); or Repairable Systems Analysis.)

There are two general types of data that can be analyzed with the software: developmental and fielded data. *Developmental data* are obtained from reliability growth testing conducted in-house, during the stages of development for a particular product. *Fielded data*, on the other hand, are obtained from repairable systems operating in the field under typical customer usage conditions. To provide a reference of all your options, the following table shows the three different (but related) types of analysis you can perform in Weibull++, the statistical models that are available for each, and the data types that can be used.

				Analysis Methods										
				Traditional Reliability Growth Analysis						Growth Planning and Management		Fielded Systems Analysis		
			Models	Crow-AMSAA (NHPP)	Standard Gompertz	Lloyd-Lipow	Modified Gompertz	Duane	Logistic	Crow Extended	Crow Extended - Continuous Evaluation	Crow-AMSAA (NHPP)	Power Law	Crow Extended
			Data Sheets											
Data Types	Developmental Data	Times-to-Failure	Failure Times	✓				✓		✓				
			Grouped Failure Times	✓				✓		✓				
			Multiple Systems - Known Operating Times	✓				✓		✓				
			Multiple Systems - Concurrent Operating	✓				✓		✓				
			Multiple Systems with Dates	✓				✓		✓				
			Multiple Systems with Event Codes							✓				
		Discrete	Sequential	✓	✓	✓	✓	✓	✓					
			Sequential with Mode		✓	✓	✓		✓					
			Grouped per Configuration	✓	✓	✓	✓	✓	✓					
			Mixed Data	✓						✓				
		Multi-Phase	Multi-Phase Failure Times								✓			
			Multi-Phase Grouped Failure Times								✓			
			Multi-Phase Mixed Data								✓			
		Reliability			✓	✓	✓		✓					
	Fielded Data		Repairable Systems							✓			✓	✓
			Fleet							✓		✓		✓

Times-to-Failure Data

For reliability growth data analysis only.

The times-to-failure data types are used for analyzing failure times recorded from in-house reliability growth testing. Different data sheets are available depending on how the testing was conducted and what information is available:

- If you will be analyzing failure times from a single system (or the combined times from multiple systems), there is a choice of two data sheets depending on whether you will enter exact failure times or grouped (interval) data:
  - Failure Times
  - Grouped Failure Times
- If you will be analyzing failure times from multiple identical systems (where the data from each system is identified by the specific System ID), there is a choice of four data sheets depending on how the operating times of the systems are determined. (See Times-to-Failure Data from Multiple Systems for more information about these data sheets.)
  - Multiple Systems - Known Operating Times
  - Multiple Systems - Concurrent Operating Times
  - Multiple Systems with Dates
  - Multiple Systems with Event Codes

If you will assume that all fixes are applied immediately after failure and before testing resumes (traditional reliability growth analysis), then you can use the **Crow-AMSAA (NHPP)** or **Duane** models. If you want to account for different fix strategies used for different failure modes (growth projections analysis), choose the **Crow Extended** model.

When you select the Crow Extended model, the **Classification** and **Mode** columns will be inserted into the data sheet. You can also manually insert or remove these columns by choosing **Growth Data > Format and View > [Insert Columns/Delete Columns] > Projections**. The Crow Extended model and mode classifications are discussed in Failure Mode Classifications.

By default, all data sheets include a **Comments** column for logging any pertinent information about each row of data. You can add a second comments column or delete the columns by choosing **Growth Data > Format and View > [Insert Columns/Delete Columns] > Comments**. The information in these columns does not affect the calculations in the folio.

See also Minimum Data Requirements for Times-to-Failure Data.

FAILURE TIMES

The Failure Times data type is used for situations where the exact failure times have been recorded. This data type can be used to analyze the data from a single system or for the combined times from multiple systems. For example, if three identical systems are tested and one fails at 10 hours, the recorded failure time is 30 hours.

The failure times are recorded in the **Time to Event** column in the data sheet. The times can be cumulative (where each row shows the total amount of test time when the failure occurred) or non-cumulative (where each row shows the incremental test time from when the last failure occurred). (See Cumulative vs. Non-Cumulative Data.)

The following example shows a data set in which the times are cumulative. The first failure (row 1) occurred at 10 hours. The second failure (row 2) occurred 5 hours later at 15 hours of total test time. The rest of the data can be read in a similar manner.

	Time to Event (hr)	Comments
1	10	
2	15	
3	20	
4	25	
5	30	
6	35	

Failure Times Data Sheet for Traditional Reliability Growth Analysis (Cumulative)

The next example shows the same data set, but where the times are non-cumulative.

	Time to Event (hr)	Comments
1	10	
2	5	
3	5	
4	5	
5	5	
6	5	

Failure Times Data Sheet for Traditional Reliability Growth Analysis (Non-Cumulative)

When you use the Crow-AMSAA (NHPP) or Duane model, the assumption is that fixes are applied when failures are found; therefore, each row in the data sheet will represent a different design configuration. Alternatively, when you use the Crow Extended model, you assume that different fix strategies are used for different failure modes (i.e., A = no fix, BC = fixed during test or BD = delayed fix). You will be required to identify and classify the failure mode responsible for each failure, as well as specify the effectiveness factor for each delayed fix (see Failure Mode Classifications).

Tip: If your data set failed the goodness-of-fit test, you can smooth out the analysis by organizing the data into groups and transferring them into a Grouped Failure Times data sheet. To do this, choose **Growth Data > Options > Auto Group Data**. For more information on how to use the tool, see Auto Group Data.

GROUPED FAILURE TIMES

The Grouped Failure Times data type is used for cases where the failures occurred within specified time intervals, but the exact times are not known (i.e., interval data). This data type can be used to analyze the data from a single system or for the combined times from multiple systems (e.g., if three identical systems are tested and one fails between 10 and 20 hours, the recorded interval is 30 to 60 hours).

This data sheet can also be used for situations when the exact failure times are available, but it is useful to group the data into intervals for analysis purposes. For example, you could use this data sheet to smooth out the analysis of a Failure Times data set that failed the goodness-of-fit test. (See Auto Group Data.)

The **Interval End Time** column is for recording the times that an interval ended. The times are cumulative (where each row shows the total amount of test time accumulated by the end of each interval). The **Failures in Interval** column is for recording the number of failures that occurred in each interval. The failure numbers are non-cumulative (where each row shows the number of failures that occurred in that interval only). (See Cumulative vs. Non-Cumulative Data.)

When you use the Crow-AMSAA (NHPP) or Duane model, the assumption is that fixes are applied at the end of each interval; therefore, each row in the data sheet will represent a different design configuration. The following example shows a data set where five identical systems are inspected every week. The systems accumulate a total of 250 hours of test time each week. In the first week (row 1), the total number of system failures that occurred in that week is 3. In the second week (row 2), the total number of system failures that occurred in that week is 8. The rest of the data sheet can be read in a similar manner.

	Failures in Interval	Interval End Time (hr)	Comments
1	3	250	
2	8	500	
3	6	750	
4	2	1000	
5	10	1250	
6	5	1500	

Grouped Failure Times Data Sheet for Traditional Reliability Growth Analysis

When you use the Crow Extended model, you assume that different fix strategies are used for different failure modes (i.e., A = no fix, BC = fixed during test or BD = delayed fix). You will be required to identify and classify the failure mode responsible for each failure, as well as specify the effectiveness factor for each delayed fix, if any (see Failure Mode Classifications).

For example, the following data set shows that in the first week (rows 1 to 2), the total number of system failures that occurred in that week is 3. Two of these failures are identified as BC100 (row 1) and one failure is identified as BD1 (row 2). In the second week (rows 3 to 8), the total number of system failures that occurred in that week is 8, and the failures are identified as BD1, BC150, BC4, BC60 (with 3 occurrences), BD23, BC3 and A75.

	Failures in Interval	Interval End Time (hr)	Classification	Mode	Comments
1	2	250	BC	100	
2	1	250	BD	1	
3	1	500	BC	150	
4	1	500	BC	40	
5	3	500	BC	60	
6	1	500	BD	23	
7	1	500	BC	3	
8	1	500	A	75	

Grouped Failure Times Data Sheet with the Crow Extended Model.

Times-to-Failure Data from Multiple Systems

For reliability growth data analysis only.

The multiple systems data types are for analyzing failure times from multiple identical systems (where the data set from each system is identified by the specific System ID). In multiple systems analysis, Weibull++ combines the operating hours of the systems to create a single *equivalent system*, which allows you to evaluate all the failures and fixes that occurred during testing. The parameters of the equivalent system, along with the results of the goodness-of-fit tests for that system, are calculated automatically when you analyze the data sheet. Any plots generated for the data set and analyses via the Quick Calculation Pad will be also based on the equivalent system.

There is a choice of four data sheets depending on how the operating times of the systems are determined:

- Multiple Systems - Known Operating Times
- Multiple Systems - Concurrent Operating Times
- Multiple Systems with Dates
- Multiple Systems with Event Codes

In all these data sheets, the operating time of the failed system is recorded for each failure. The following table summarizes how these data sheets obtain or estimate the operating times of the other non-failed systems.

Data Type	Operating Times of Non-Failed Systems	All systems must operate at same rate?	All systems must start the test at the same time?
Known Operating Times	User provides the exact operating times for both failed system and all other non-failed systems.	No	
Concurrent Operating Times	Weibull++ uses the failure time of the failed system as the operating times of the non-failed systems.	Yes	
Multiple Systems with Dates	Weibull++ uses calendar dates to estimate the operating times of non-failed systems based on their average daily usage rate for the relevant time period.	No	
Multiple Systems with Event Codes	*Same as "Concurrent Operating Times."*	Yes	

If you will assume that when failures are found the same set of fixes is applied to all of the systems at the same time (traditional reliability growth analysis), then you can use the **Crow-AMSAA (NHPP)** or **Duane** models. Once the fix has been implemented for all systems, the test is resumed; therefore, each row in the data sheet will represent a different design configuration.

If you want to account for different fix strategies used for different failure modes (growth projections analysis), choose the **Crow Extended** model. You will be required to identify and classify the failure mode responsible for each failure (i.e., A = no fix, BC = fixed during test or BD = delayed fix), as well as specify the effectiveness factor for each delayed fix.

When you select the Crow Extended model, the **Classification** and **Mode** columns will be inserted into the data sheet. You can also manually insert or remove these columns by choosing **Growth Data > Format and View > [Insert Columns/Delete Columns] > Projections**. The Crow Extended model and mode classifications are discussed in Failure Mode Classifications.

By default, all data sheets include a **Comments** column for logging any pertinent information about each row of data. You can add a second comments column or delete the columns by choosing **Growth Data > Format and View > [Insert Columns/Delete Columns] > Comments**. The information in these columns does not affect the calculations in the folio.

See also Minimum Data Requirements for Times-to-Failure Data.

MULTIPLE SYSTEMS - KNOWN OPERATING TIMES

The Known Operating Times data type is used for situations when multiple identical systems are tested and the exact operating times for all systems are known. When a failure occurs in any of the systems, the exact operating times for all systems (both failed and non-failed) are recorded. The analysis assumes that the fixes are applied to all of the systems at the same time so they continue to have the same configuration.

The **Time System [x]** columns are for recording the failure times/operating times of the units. The times can be cumulative (where each row shows the total amount of time when the failure occurred) or non-cumulative (where each row shows the incremental time from when the last failure occurred). (See Cumulative vs. Non-Cumulative Data.) By default, the data sheet is configured for two systems. To insert or remove additional columns, choose **Growth Data > Format and View > [Insert Columns/Delete Columns] > System/Unit**.

The **Failed System ID** column is for recording the ID number of the system that failed. It must be represented by a positive integer.

The following data sheet shows an example in which the times are cumulative. The first failure occurred for System 1 (indicated by a 1 in the first column for row 1). At that point, the operating times for both systems were recorded — 7 hours for System 1 and 10 hours for System 2 — and the same fix was applied on both systems. The next failure occurred for System 2 (indicated by a 2 in the first column for row 2). Once again, the operating times for both systems were recorded — 15 hours for System 1 (8 since the last event) and 17 hours for System 2 (7 since the last event). The rest of the data can be interpreted in a similar manner.

	Failed System ID	Time System 1 (hr)	Time System 2 (hr)	Comments
1	1	7	10	
2	2	15	17	
3	1	20	23	
4	2	25	28	
5	1	29	32	
6	2	39	45	

Multiple Systems - Known Operating Times Data Sheet for Traditional Reliability Growth Analysis (Cumulative)

The next example shows the same data set, but the times are non-cumulative.

	Failed System ID	Time System 1 (hr)	Time System 2 (hr)	Comments
1	1	7	10	
2	2	8	7	
3	1	5	6	
4	2	5	5	
5	1	4	4	
6	2	10	13	

Multiple Systems - Known Operating Times Data Sheet for Traditional Reliability Growth Analysis (Non-Cumulative)

MULTIPLE SYSTEMS - CONCURRENT OPERATING TIMES

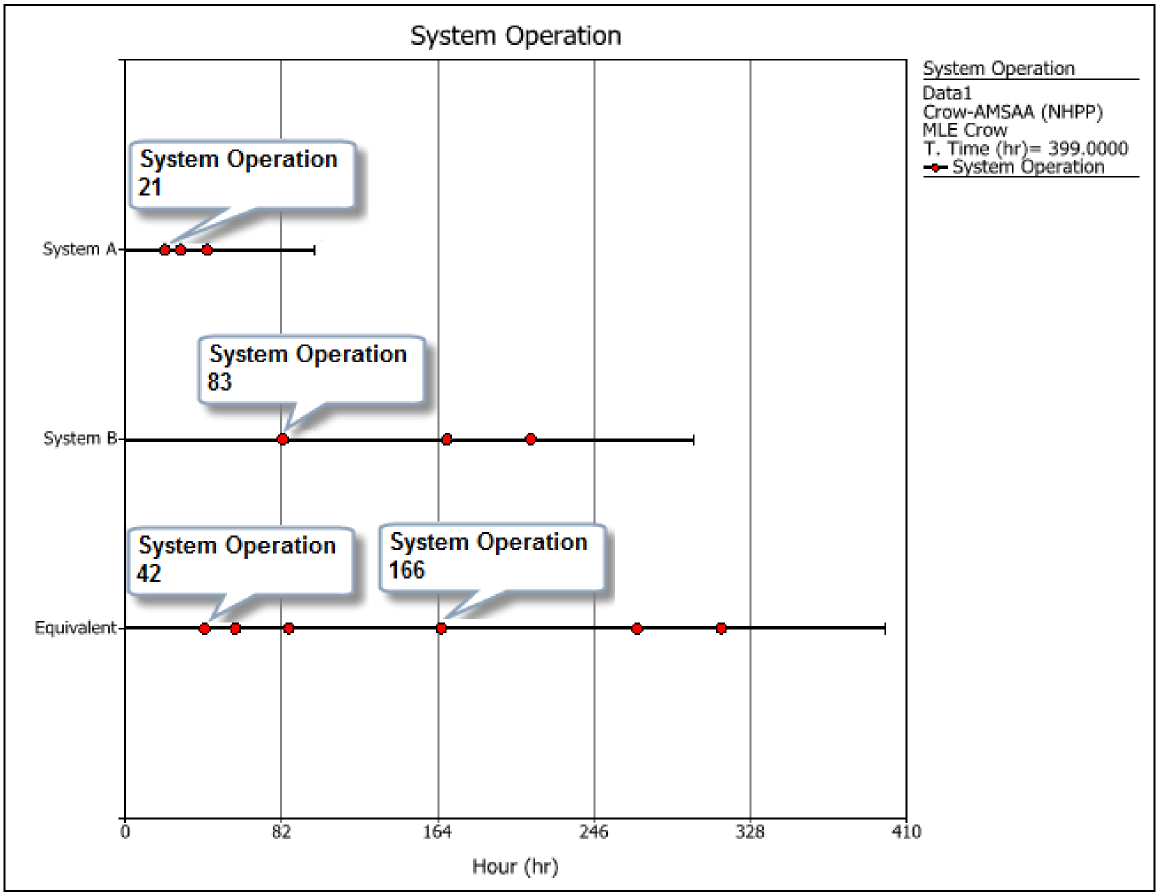
The Concurrent Operating Times data type is used for situations when multiple identical systems are tested but a system's exact operating time can be known only when it fails. The analysis assumes that the systems all began with the same configuration, they operate concurrently, they accumulate usage at the same rate and they receive fixes at the same time. Therefore, when a failure occurs in any of the systems, the exact operating time for the failed system is recorded and this time is also used as the operating times for the other non-failed systems.

With this data type, you can enter the data in the Normal View or Advanced Systems View. The following example shows the Normal View. The **System ID** column is for recording the ID of the system that the data point relates to. The **Event** column specifies whether the row represents the start time (S), failure time (F) or end time (E) of the system. The **Time to Event** column is for recording the total cumulative test time when the specified event occurred.

	System ID	Event	Time to Event (hr)	Comments
1	System A	S	0	Start
2	System A	F	21	
3	System A	F	29	
4	System A	F	43	
5	System A	E	100	End
6	System B	S	0	Start
7	System B	F	83	
8	System B	F	169	
9	System B	F	213	
10	System B	E	299	End

Multiple Systems - Concurrent Operating Times Data Sheet (Normal View)
for Traditional Reliability Growth Analysis

The following plot shows the failure times of both systems in the example, along with the timeline for their *equivalent system*. When the first failure occurred at 21 hours (System A), the total operating time for the test (represented by the equivalent system) is 42 hours because there are two systems running concurrently. In other words, the analysis multiplies the failure time of the failed system by the total number of systems in the test to obtain the equivalent operating time. Similarly, the first failure of System B (at 83 hours) is shown in the equivalent system as occurring at 166 hours.



MULTIPLE SYSTEMS WITH DATES

Like the Concurrent Operating Times data type, the Multiple Systems with Dates data type is used for situations when multiple identical systems are tested but a system's exact operating time can be known only when it fails. However, this data type can be used when the systems did not all begin with the same configuration and/or they are not operated concurrently (although it still assumes that the fixes are applied to all of the systems at the same time). When a failure occurs in any of the systems, the exact operating time for the failed system is recorded and the software estimates the operating times for the other non-failed systems using the exact calendar dates recorded for all events. Specifically, the software uses the dates to calculate the average daily usage rate of each non-failed system over the relevant time period.

The data sheet is the same as the Concurrent Operating Times data sheet but with a **Date** column for recording the calendar date of the events. You can enter the data in the Normal View or Advanced Systems View. The following example shows a data set in the Normal View.

	System ID	Event	Time to Event (hr)	Date	Comments
1	System A	S	0	1/1/2019	Start
2	System A	F	43	1/6/2019	
3	System A	F	66	1/18/2019	
4	System A	F	115	1/31/2019	
5	System A	E	159	2/3/2019	End
6	System B	S	0	1/5/2019	Start
7	System B	F	83	1/10/2019	
8	System B	F	169	1/15/2019	
9	System B	F	213	2/1/2019	
10	System B	E	299	2/3/2019	End

Multiple Systems with Dates Data Sheet (Normal View)
for Traditional Reliability Growth Analysis

The following plot shows the failure times of both systems in the example, along with the timeline for their *equivalent system*. For example, the plot shows that when System A failed on 1/18/2019, it had accumulated 66 hours of operating time, but we don't know the exact operating time for System B at that point.

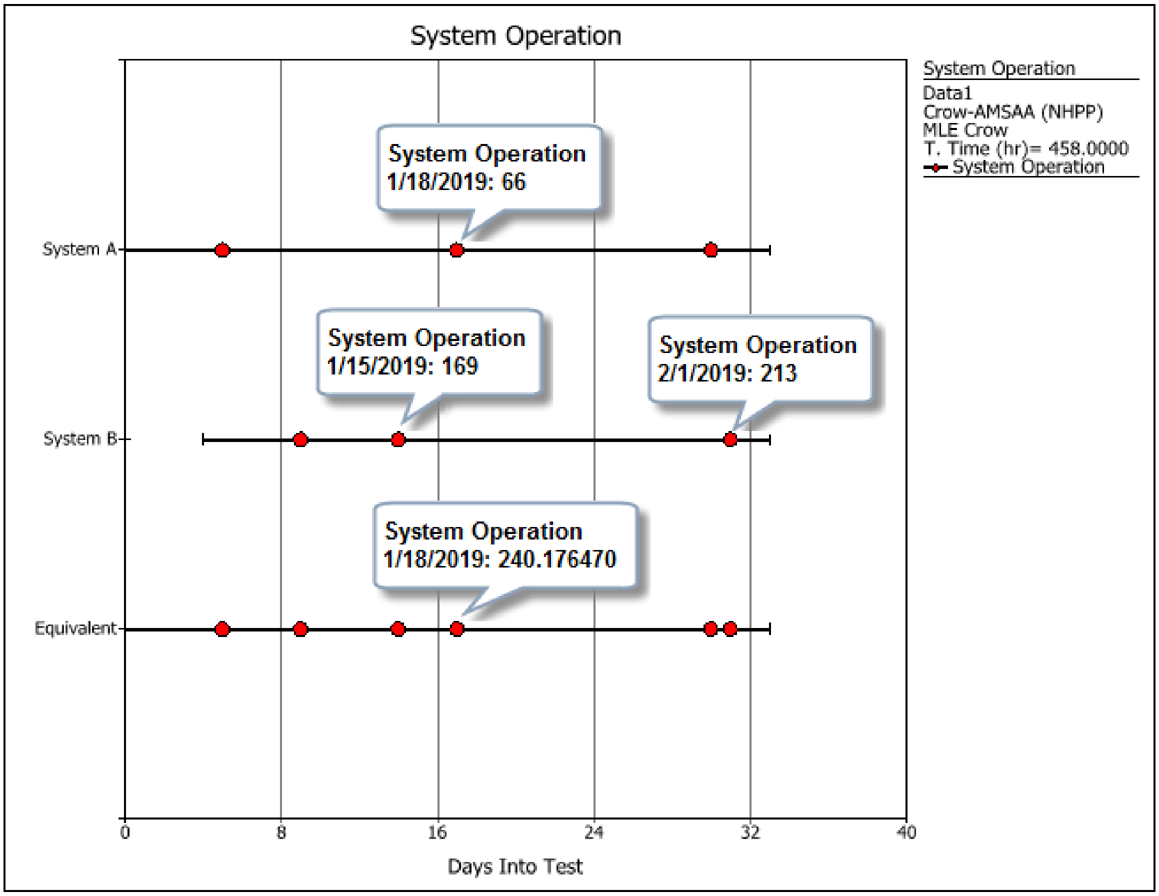
We do know that when System B failed on 1/15/2019 it had accumulated 169 hours of test time and then when it failed again on 2/1/2019, it had accumulated 213 hours. The total accumulated test time of System B between these two periods is 213-169=44 hours.

If we divide this result by the number of days (17), we obtain the average daily usage rate of System B during that period (2.588235 hours/day).

This can then be used to estimate the number of hours System B accumulated in the 2 days between the first failure of System B on 1/15/2019 and the first failure of System A on 1/18/2019 (2 x 2.5882 = 5.176470 hours).

Therefore, the analysis assumes that the operating time of System B was 174.176470 hours when System A failed on 1/18/2019.

For the equivalent system, the estimated operating time on 1/18/2019 is the summation of the observed operating time for System A and the estimated operating time for System B, which is 240.176470 hours.



MULTIPLE SYSTEMS WITH EVENT CODES

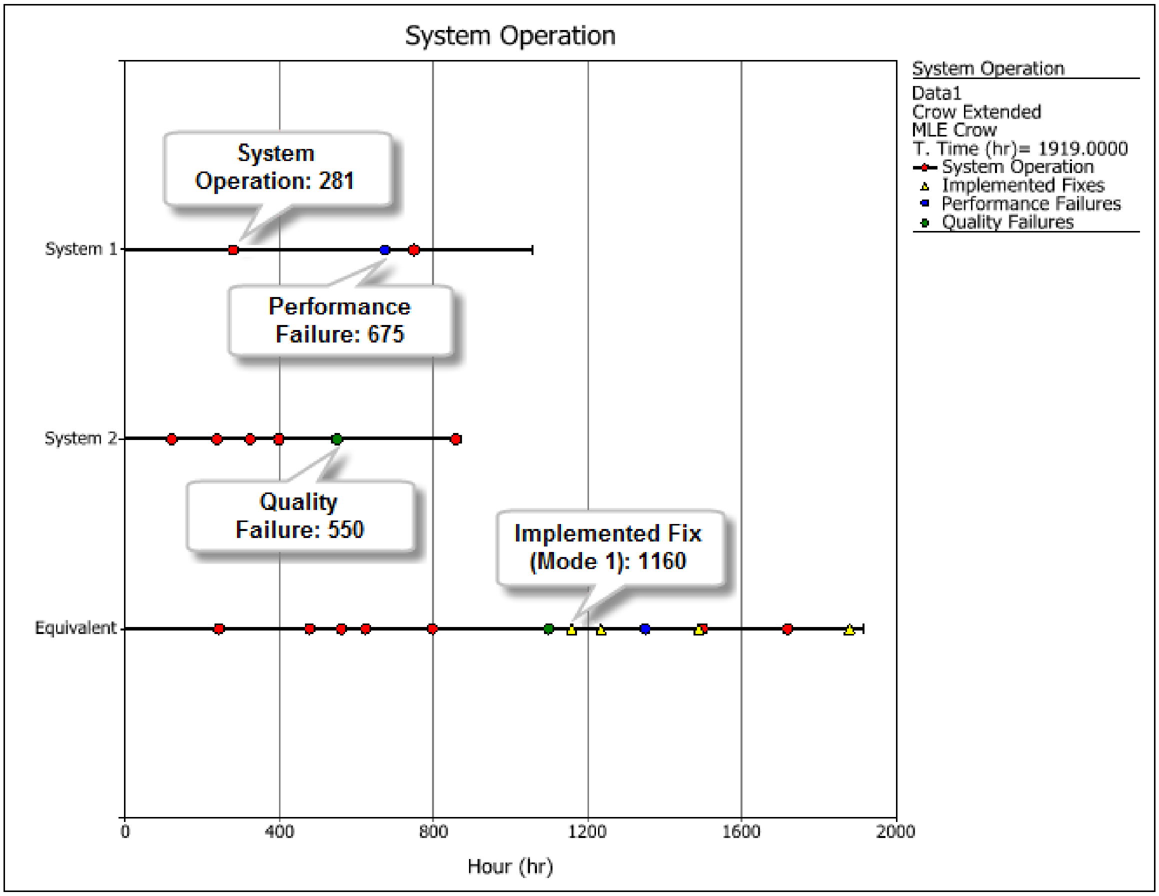
The Multiple Systems with Event Codes data type is similar to the Concurrent Operating Times data type except that it always requires you to identify and classify the failure mode responsible for each failure (see Failure Mode Classifications). It also allows you to use codes to identify other types of events besides failures (e.g., the time when a fix was implemented for a particular failure mode, performance or quality issues that can be included or excluded from the analysis, etc.). This data type can be used when fixes are not implemented simultaneously for all systems. The Crow Extended model is used for the analysis.

With this data type, you can enter the data in the Normal View or Advanced Systems View. The following example shows the Normal View. The **Classification** and **Mode** columns are for identifying and classifying the failure mode responsible for each failure. The **Event** column specifies the type of event the data point represents. See Event Codes for Crow Extended for information on all five possible event types. In the following data sheet, S = start time, F = failure time, I = the time when the fix for a particular failure mode was implemented, P = performance failure, Q = quality failure and E = end time.

	System ID	Event	Time to Event (hr)	Classification	Mode	Comments
1	System 1	S	0			Start
2	System 1	F	281	BC	1	
3	System 1	I	300	BC	1	
4	System 1	I	375	BC	36	
5	System 1	P	675	BD	42	
6	System 1	F	750	BC	25	
7	System 1	I	1020	BC	25	
8	System 1	E	1059			End
9	System 2	S	0			Start
10	System 2	F	122	BD	5	
11	System 2	F	240	BC	36	
12	System 2	F	325	BC	1	
13	System 2	F	399	BC	17	
14	System 2	I	430	BC	17	
15	System 2	Q	550	BD	40	
16	System 2	F	860	BD	22	
17	System 2	E	860			End

Multiple Systems with Event Codes Data Sheet (Normal View)
for Reliability Growth Projections Analysis

In this case, the process of combining the data set for the equivalent system is the same as described above for the Concurrent Operating Times data sheet, but with the addition of taking into account the time of implemented fixes across different systems. The implemented fix time is obtained by computing for the total time that the system spent in the same design configuration for a particular mode before the implemented fix took place. For these plots, the different types of events are shown in different colors: failure (red circle), implemented fix (yellow triangle), performance failure (blue circle) and quality failure (green diamond).



The ReliaWiki resource portal provides an example that demonstrates how the software builds the equivalent system for this data type at http://www.reliawiki.org/index.php/Equivalent_System_Example.

Discrete Data

For reliability growth data analysis only.

Discrete data are obtained from one-shot devices with only two possible outcomes from the test: success or failure. An example is a missile that gets fired once and it either succeeds or fails. This type of data is also referred to as *success/failure* or *attribute* data. Different data sheets are available depending on how the testing was conducted and what information is available. This also determines what kind of analysis can be performed.

With the first three discrete data types, you can use the Crow-AMSAA (NHPP), Standard Gompertz, Modified Gompertz, Lloyd-Lipow, Duane or Logistic models to track how the reliability of the system changes over time (traditional reliability growth analysis):

- Sequential
- Sequential with Mode
- Grouped per Configuration

The fourth discrete data type can be used for either traditional reliability growth analysis or projections (growth projections analysis):

- Mixed Data

For the Mixed data type, if you will assume that all fixes are applied immediately after failure and before testing resumes (traditional reliability growth analysis), you can use the **Crow-AMSAA (NHPP)** model. If you want to account for different fix strategies used for different failure modes (growth projections analysis), choose the **Crow Extended** model.

When you select the Crow Extended model, the **Classification** and **Mode** columns will be inserted into the data sheet. You can also manually insert or remove these columns by choosing **Growth Data > Format and View > [Insert Columns/Delete Columns] > Projections**. The Crow Extended model and mode classifications are discussed in Failure Mode Classifications.

By default, all data sheets include a **Comments** column for logging any pertinent information about each row of data. You can add a second comments column or delete the columns by choosing **Growth Data > Format and View > [Insert Columns/Delete Columns] > Comments**. The information in these columns does not affect the calculations in the folio.

See also Minimum Data Requirements Discrete Data.

SEQUENTIAL

The Sequential data type is used for one-shot devices where a single trial is performed for each system configuration and improvements are made before the next trial. Only the outcome (success/failure) is recorded for each trial.

The device is inspected after each trial, and then fixes are applied before the next trial begins; therefore, each row number in the data sheet represents a particular design configuration. In the following example, the device succeeded (S) for the first configuration (row 1), but failed (F) for the second configuration (row 2). The rest of the data can be read in a similar manner.

	Success/Failure	Comments
1	S	
2	F	
3	F	
4	F	
5	S	
6	F	

SEQUENTIAL WITH MODE

The Sequential with Mode data type is used for one-shot devices where a single trial is performed for each system configuration and improvements are made before the next trial occurs. In addition to the outcome (success/failure) for each trial, the specific failure modes are also recorded.

The data sheet is the same as the Sequential data sheet, but with the addition of a **Failure Mode** column for recording the mode responsible for each failure. This allows you to track the possible recurrence of a failure mode after the fixes have been applied. If the mode does not appear again in the later trials, then the probability of failure is reduced. This is known as *failure discounting*. If you do not enter any failure modes, then no failures are discounted in the analysis (and the results will be the same as in the Sequential data type if you use the same growth model). The following figure shows an example of the data sheet.

	Success/Failure	Failure Mode	Comments
1	S		
2	F	mode a	
3	F	mode b	
4	F	mode c	
5	S		
6	F	mode a	

GROUPED PER CONFIGURATION

The Grouped per Configuration data type is used for one-shot devices where multiple trials are performed for each design configuration (e.g., testing 5 missiles of the same design equals 5 trials). Improvements are made after all the devices within the group have been tested. Therefore, each row in the data sheet represents a particular design configuration. For each stage of testing, both the number of trials and the number of failures are recorded (e.g., 3 failures out of 10 trials for configuration A, 2 failures out of 10 trials for configuration B, etc.).

The **Number of Trials** column is for recording the number of devices that were tested for each design configuration. The **Number of Failures** column is for recording the number of failed units in the specified configuration. The values in these columns can be cumulative (where each row shows the total number of trials and failures since the beginning of the test) or non-cumulative (where each row shows the incremental number of trials and failures from the last configuration). (See Cumulative vs. Non-Cumulative Data.)

The following example shows a data set where the values are cumulative. In the first configuration (row 1), there were 5 failures out of 10 trials. In the second configuration (row 2), there were 3 more failures (for a total of 8) out of 8 more trials (for a total of 18). The rest of the data can be read in a similar manner.

	Number of Trials	Number of Failures	Comments
1	10	5	
2	18	8	
3	27	11	
4	36	13	
5	46	15	
6	56	16	

Grouped per Configuration Data Sheet (Cumulative)

The next example shows the same data set, but the values are non-cumulative.

	Number of Trials	Number of Failures	Comments
1	10	5	
2	8	3	
3	9	3	
4	9	2	
5	10	2	
6	10	1	

Grouped per Configuration Data Sheet (Non-Cumulative)

MIXED DATA

The Mixed data type is used for one-shot devices where some test stages (i.e., design configurations) may have only one trial while other stages may have multiple trials. For each stage of testing, both the number of trials and the number of failures are recorded (e.g., 1 failure out of 1 trial for configuration A, 2 failures out of 5 trials for configuration B, etc.). This data type may be used in cases when you have a different number of samples available from each design configuration. For example, you might test one device initially then later start testing more samples of each design.

The **Failures in Interval** column is for recording the number of failures in a stage. The **Cumulative Trials** column is for recording the total number of trials that were performed since the beginning of the test.

When you use the Crow-AMSAA (NHPP) model for this data type, the assumption is that fixes are applied at the end of each interval; therefore, each row in the data sheet will represent a different design configuration, where each configuration can have any number of trials. For example, the following data set shows that in the first stage (row 1), 4 units were tested and 3 failed. In the second stage (row 2), 1 more unit was tested (for a total of 5) and it did not fail. In the third stage (row 3), 4 more units were tested (for a total of 9) and all 4 failed. The rest of the data can be read in a similar manner.

	Failures in Interval	Cumulative Trials	Comments
1	3	4	
2	0	5	
3	4	9	
4	1	12	
5	0	13	
6	1	15	

Mixed Data Sheet with the Crow-AMSAA (NHPP) Model

When you use the Crow Extended model, you assume that some of the fixes may be delayed until the end of the test and that some of the failure modes may not be fixed (i.e., A = no fix or BD = delayed fix). This is useful for situations when you need to perform the test in stages due to logistic reasons (e.g., cannot launch 10 missiles at the same time), or the purpose of the test is only to uncover failure modes. You will be required to identify and classify the failure mode responsible for each failure, and specify the effectiveness factor for each delayed fix, if any (see Failure Mode Classifications).

For example, the following data set shows that in the first stage (rows 1 to 3), 4 units were tested and 3 failed. The failures are identified as BD125 (row 1), A10 (row 2) and BC230 (row 3). In the second stage (row 4), 1 more unit was tested (for a total of 5) and it did not fail. The rest of the data can be read in a similar manner.

	Failures in Interval	Cumulative Trials	Classification	Mode	Comments
1	1	4	BD	125	
2	1	4	A	10	
3	1	4	BC	230	
4	0	5			
5	1	9	BC	320	
6	1	9	BC	140	

Mixed Data Sheet with the Crow-Extended Model

Tip: If you will assume that some fixes are implemented at the end of an interval while some are implemented at the end of the observed test time, then the Multi-Phase Mixed data type may be more appropriate.

Multi-Phase Data

For reliability growth data analysis only.

Multi-phase data types are designed for practical testing situations where failures can be corrected at the time of failure, delayed until a later time during the current phase, fixed during another phase or fixed at the end of a phase. This data type implements the Crow Extended - Continuous Evaluation model.

When used in conjunction with the growth planning folio, these data sheets allow you to compare the reliability performance of the system at each phase against the goals that were developed in the reliability growth plan.

Different data sheets are available depending on how the testing was conducted and what information is available:

- Multi-Phase Failure Times
- Multi-Phase Grouped Failure Times
- Multi-Phase Mixed Data

By default, all data sheets include a **Comments** column for logging any pertinent information about each row of data. You can add a second comments column or delete the columns by choosing **Growth Data > Format and View > [Insert Columns/Delete Columns] > Comments**. The information in these columns does not affect the calculations in the folio.

See also Minimum Data Requirements for Multi-Phase Data.

MULTI-PHASE FAILURE TIMES

The Multi-Phase Failure Times data type is used for multi-phase data analysis with the Crow Extended - Continuous Evaluation model when the exact time is recorded for each event. It is the same as the Failure Times data type, but with the addition of the **Event**, **Classification** and **Mode** columns for recording the information required to use the continuous evaluation model.

The following example shows the failure times of a single system, entered cumulatively. The **Event** column specifies the type of event the data point represents. See Event Codes for Multi-Phase Data for information on all seven possible event types. In the following data sheet, F = failure time, AP = analysis point (i.e., a designated time at which calculations will be performed and results obtained), and I = the time when the fix for a particular failure mode was implemented. The **Classification** and **Mode** columns are for identifying and classifying the failure mode that is responsible for the failure (for F, P or Q events) or the mode that received a fix (for I events).

	Event	Time to Event (hr)	Classification	Mode	Comments
1	F	110	BC	100	
2	F	224	BD	5004	
3	F	333	BD	5002	
4	AP	555			
5	F	645	BC	101	
6	I	700	BD	5002	

MULTI-PHASE GROUPED FAILURE TIMES

The Multi-Phase Grouped Failure Times data type is used for multi-phase data analysis with the Crow Extended - Continuous Evaluation model when the time of each event falls somewhere within a specified interval. It is the same as the Grouped Failure Times data type, but with the addition of the **Event**, **Classification** and **Mode** columns for recording the information required to use the continuous evaluation model.

The following example shows the total test time at the end of each interval and the number of failures that occurred in each interval. The **Event** column specifies the type of event the data point represents. See Event Codes for Multi-Phase Data for information on all seven possible event types. In the following data sheet, F = failure time, Q = quality-related issue and P = performance-related issue. The **Classification** and **Mode** columns are for identifying and classifying the failure mode that is responsible for the failure (for F, P or Q events) or the mode that received a fix (for I events).

	Event	Failures in Interval	Interval End Time (hr)	Classification	Mode	Comments
1	F	1	1000	BD	2	
2	Q	3	1000	A	76	
3	F	6	1000	BD	8	
4	F	2	1000	BC	1	
5	F	1	1000	BD	7	
6	P	2	1000	BC	125	

MULTI-PHASE MIXED DATA

The Multi-Phase Mixed data type is used for multi-phase data analysis with the Crow Extended - Continuous Evaluation model for one-shot devices, and where some test stages (i.e., design configurations) may have only one trial and other stages may have multiple trials. For each stage of testing, both the number of trials and the number of failures are recorded (e.g., 1 failure out of 1 trial for configuration A, 2 failures out of 5 trials for configuration B, etc.). This data sheet is the same as Discrete Mixed data type, but with the addition of the **Event**, **Classification** and **Mode** columns for recording the information required to use the continuous evaluation model.

The following example shows the number of trials and failures in each test interval. The **Event** column specifies the type of event the data point represents. See Event Codes for Multi-Phase Data for information on all seven possible event types. In the following data sheet, F = failure time and PH = end of phase. The **Classification** and **Mode** columns are for identifying and classifying the failure mode that is responsible for the failure (for F, P or Q events) or the mode that received a fix (for I events).

	Event	Failures in Interval	Cumulative Trials	Classification	Mode	Comments
1	F	1	4	BD	1	
2	F	1	4	BD	2	
3	F	1	4	A	200	
4	F	0	5			
5	F	1	6	BD	3	

Reliability Data

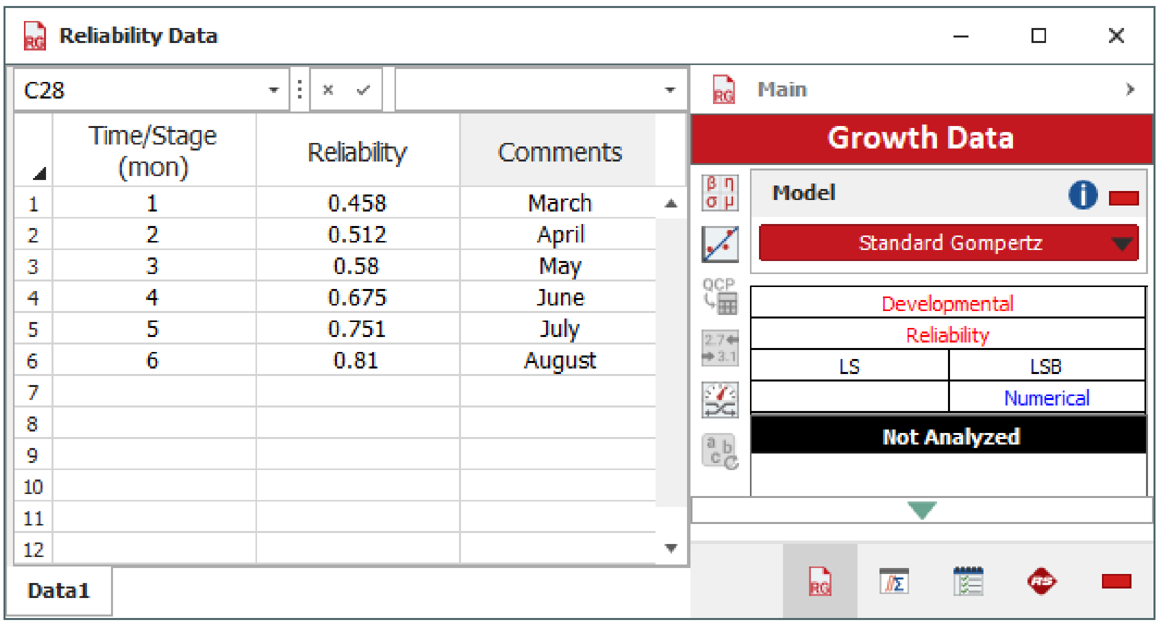
For reliability growth data analysis only.

The Reliability data type is used for modeling the relationship between time and reliability. You can use the Standard Gompertz, Modified Gompertz, Lloyd-Lipow or Logistic models to track how the reliability changes over time.

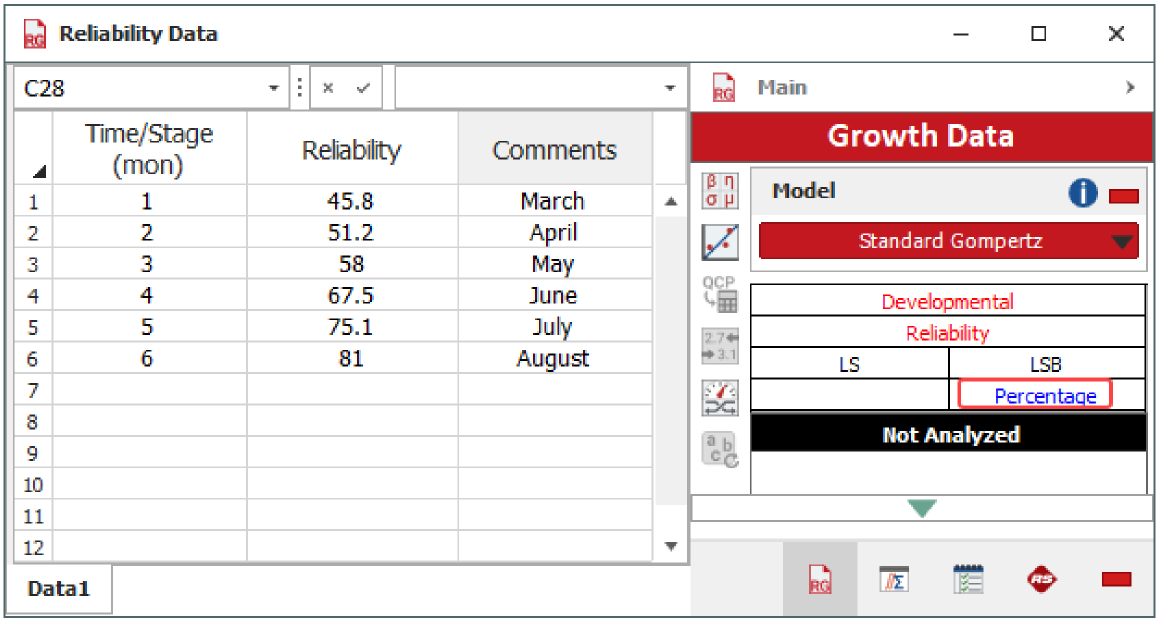
The **Reliability** column is for recording the reliability values. The reliability can be computed by dividing the number of systems still operating by the total number of systems, or by performing life data analysis (e.g., Weibull analysis) or other related methods. The **Time/Stage** column is for recording the cumulative test "time" when the reliability was computed. This can be measured in terms of test stages, design configurations, development time, and the like.

In this data sheet, reliability values are entered as decimals; however, you can configure the data sheet to accept percent values. You can change the setting by selecting or clearing the **Use percents (not decimals) for reliability** check box on the RGA Growth Data Folios page of the Application Setup.

In the following example, the "Numerical" label on the control panel indicates that the reliability values are decimal values.



The next example shows the "Percentage" label on the control panel and the same data set as above, but entered as percent values.



By default, all data sheets include a **Comments** column for logging any pertinent information about each row of data. You can add a second comments column or delete the columns by choosing **Growth Data > Format and View > [Insert Columns/Delete Columns] > Comments**. The information in these columns does not affect the calculations in the folio.

See also Minimum Data Requirements for Reliability Data.

Fielded Data

For reliability growth data analysis only.

The fielded data types are used for analyzing data from repairable systems operating in the field under typical customer usage conditions. Such data might be obtained from a warranty system, repair depot, operational testing, etc.

Although fielded systems analysis applies some of the same statistical models that are used to analyze data from developmental testing, it should be noted that there is a difference in the underlying analysis assumptions. For developmental testing data, the analysis assumes that permanent *fixes* are being applied to improve the inherent reliability of the design; whereas for fielded system data the analysis assumes that systems may fail and be *repaired* many times, and that each system will always be "as bad as old" after a repair (i.e., the concept of *minimal repair*). The fielded systems analysis assumes that permanent design improvements, if any, are delayed until a later time.

Tip: If you assume that each repair partially renews the system (i.e., "better than old but worse than new") or restores the system to "as good as new," then the General Renewal Process (GRP) may be a more appropriate model. This model is available using a Parametric RDA folio.

There is a choice of two data sheets for analyzing data from fielded systems. In general:

- Use the Repairable data type for analyzing the individual failure times for multiple repairable systems operating in the field. The analysis models the number of individual system failures vs. system time.
- Use the Fleet data type for analyzing the failure times for multiple repairable systems from a fleet (rather than individual system) perspective. The analysis groups the data and models the number of fleet failures vs. fleet time.

These data sheets can be used for two kinds of analysis. In general:

- If you wish to perform *repairable systems analysis* based on the assumption of minimal repair (i.e., the system is "as bad as old" after each repair), use the **Power Law** model in a Repairable data sheet. If the power law model does not provide a good fit, you can transfer the data to a Fleet data sheet and use the **Crow-AMSAA (NHPP)** instead.
- If you wish to estimate the jump in reliability that can be expected from rolling out the same set of permanent design fixes to all systems in the field, use the **Crow Extended** model.
  - In a **Repairable** data sheet, the Crow Extended analysis method is intended to be used specifically for "Operational Testing" situations when there is careful control of the test conditions and the failure intensity observed in the field is constant (i.e., beta = 1). Only two of the three failure mode classifications are applicable for such scenarios (A = no fix or BD = delayed fix).
  - In a **Fleet** data sheet, the Crow Extended analysis method does not require the assumption that the failure intensity observed in the field is constant (i.e., beta does not have to equal 1). All three of the failure mode classifications are available (A = no fix, BC = fixed during test and BD = delayed fix).

Note: Weibull++ displays a warning in the growth data folio when the beta = 1 hypothesis is invalid (i.e., when the 90% two-sided confidence bounds on beta do not include 1). You can choose whether to display this warning by using the **Beta = 1 hypothesis is Invalid** option on the RGA Growth Data Folios page of the Application Setup.]

When you select the Crow Extended model, the Classification and Mode columns will be inserted into the data sheet. (See Failure Mode Classifications.) When this model is applied to a Repairable data sheet, the analysis assumes that all permanent design fixes are delayed until a later time and therefore only two of the three failure mode classifications are available: A = no fix and BD = delayed fix.

By default, all data sheets include a **Comments** column for logging any pertinent information about each row of data. You can add a second comments column or delete the columns by choosing **Growth Data > Format and View > [Insert Columns/Delete Columns] > Comments**. The information in these columns does not affect the calculations in the folio.

(See Minimum Data Requirements for Fielded Data.)

REPAIRABLE SYSTEMS

The Repairable data type is used for analyzing the individual failure times for multiple repairable systems operating in the field. It models the number of individual system failures vs. the system time.

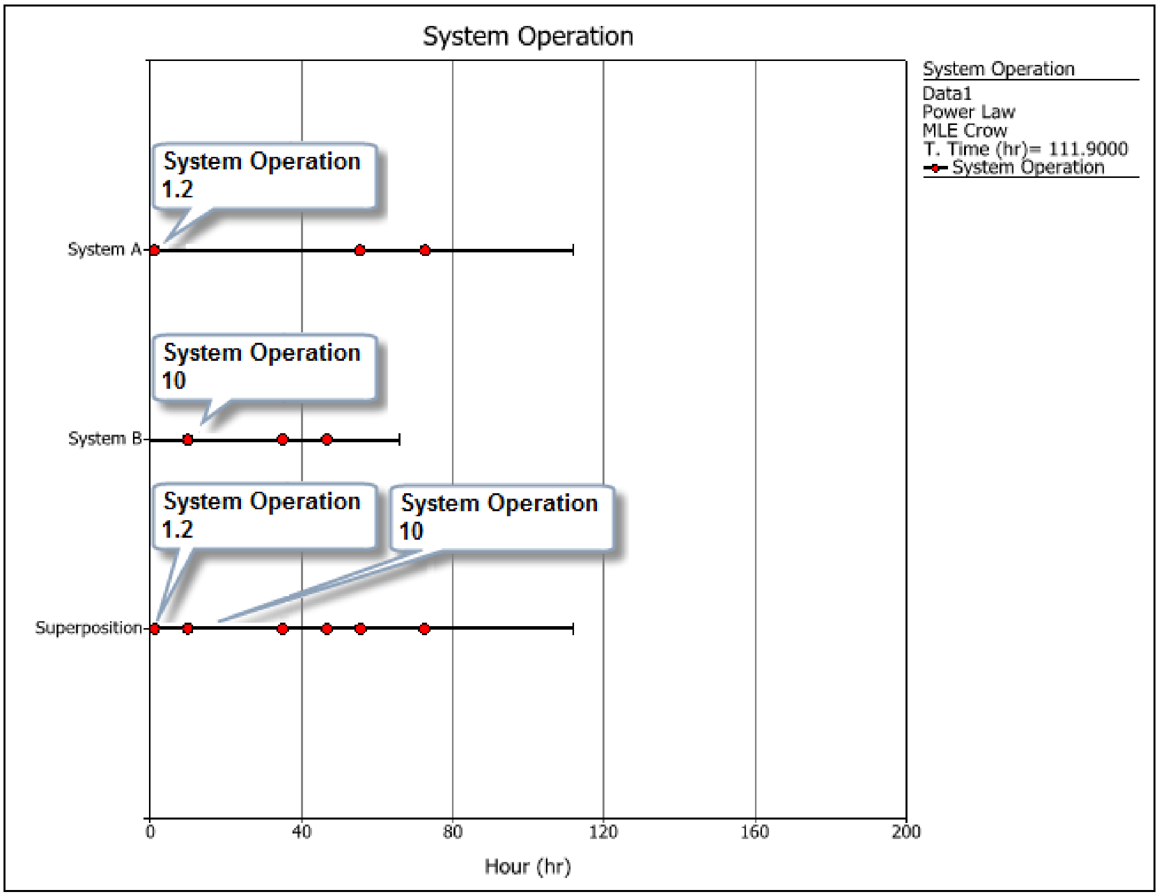
You can enter the data in the Normal View or Advanced Systems View. The following example shows the Normal View. The **System ID** column is for recording the ID of the system. The **Event** column specifies whether the row represents the start time (S), failure time (F) or end time (E) of the system. The **Time to Event** column is for recording the total operating time when the specified event occurred.

	System ID	Event	Time to Event (hr)	Comments
1	System A	S	0	Start
2	System A	F	1.2	
3	System A	F	55.6	
4	System A	F	72.7	
5	System A	E	111.9	End
6	System B	S	0	Start
7	System B	F	10	
8	System B	F	35	
9	System B	F	46.8	
10	System B	E	65.9	End

Repairable Systems Data Sheet (Normal View) for Fielded Systems Analysis

In repairable systems analysis, Weibull++ places the failure times of the systems on a single timeline to create a *superposition system*, which shows the order in which all the failures in the data set occurred. For example, the following plot shows the failure times of both systems in the example, along with the timeline for their superposition system. The timeline shows that the first failure occurred at 1.2 hours (System A) and that the second failure occurred at 10 hours (System B). The termination time in the superposition system is equal to the age of the oldest system.

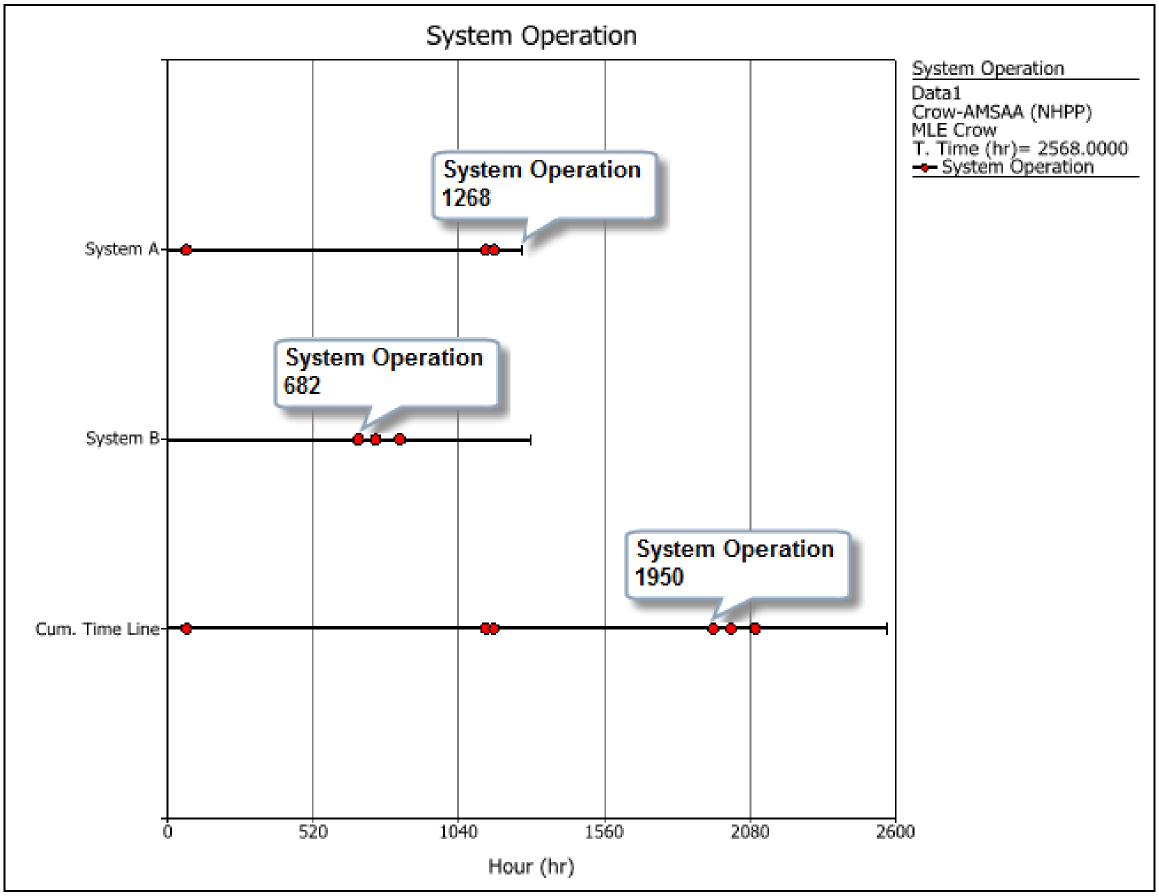
The parameters of the superposition system, along with the results of the goodness-of-fit tests for that system, are calculated automatically when you analyze the data sheet. Any plots and QCP results will be also based on the superposition system.



FLEET

The Fleet data type is used for analyzing the failure times for multiple repairable systems from a fleet (rather than individual system) perspective. It groups the data and models the number of fleet failures vs. the total operating times of all systems in the fleet.

Data entry in the Fleet data sheet is the same as described above for the Repairable Systems data sheet. However, in fleet data analysis, the failure times of each system are stacked into a cumulative timeline. This allows Weibull++ to model the failures in the population over the cumulative fleet operating time. The termination time in the cumulative timeline is equal to the sum of the ages of all the systems in the analysis. For example, the following plot shows that System A operated for a total of 1,268 hours. The first failure of System B (at 682 hours) is added to System A's end time to obtain a failure time of 1,950 hours for the cumulative timeline.



As the example demonstrates, the data sets of all the systems are stacked one after another based on the order in which the systems were entered into the data sheet; if we had entered the data set of System B first, then the first failure time of System A would have been added to the end time of System B. The ReliaWiki resource portal has more information about how the order of the systems may affect the fleet analysis at http://www.reliawiki.org/index.php/Fleet_Data_Analysis.

Tip: For any given observation period, the Fleet data sheet requires a complete record of the failure times of all the systems in the fleet. If your data set is limited to a record of the total amount of operating time for the fleet per observation interval (where the intervals may be measured in weeks, months, etc.) and the total number of system failures in those intervals, then you can use the Grouped Failure Times data type instead.

Cumulative vs. Non-Cumulative Data

For reliability growth data analysis only.

In Weibull++, some data sheets may be configured to have the data input as cumulative or non-cumulative. The following data sheets have this option:

- Times-to-failure data:
    - Failure Times
    - Multiple Systems - Known Operating Times
- Discrete data:
    - Grouped per Configuration
- Multi-phase data:
    - Failure Times

If the data sheet is configured for *cumulative* data entry, each failure time represents the total amount of operating time until the failure. If the data sheet is configured for *non-cumulative* data entry, each failure time represents the incremental amount of operating time since the last failure.

For example, suppose a system fails at 10 hours and then fails again 10 hours later (at 20 hours of test total time) and then fails again 10 hours later (at 30 hours of total test time). If the data sheet is configured for cumulative data entry, the times are 10, 20 and 30. If the data sheet is configured for non-cumulative data entry, the times are 10, 10 and 10, as shown next.

Time to Event (hr)
10
20
30

Cumulative Failure Times

Time to Event (hr)
10
10
10

Non-Cumulative Failure Times

When applicable, this option also applies to the data entry for number of trials and number of failures. For example, in the following data sheets (Discrete Data: Grouped per Configuration), the left picture shows the cumulative data entry and the right picture shows the non-cumulative.

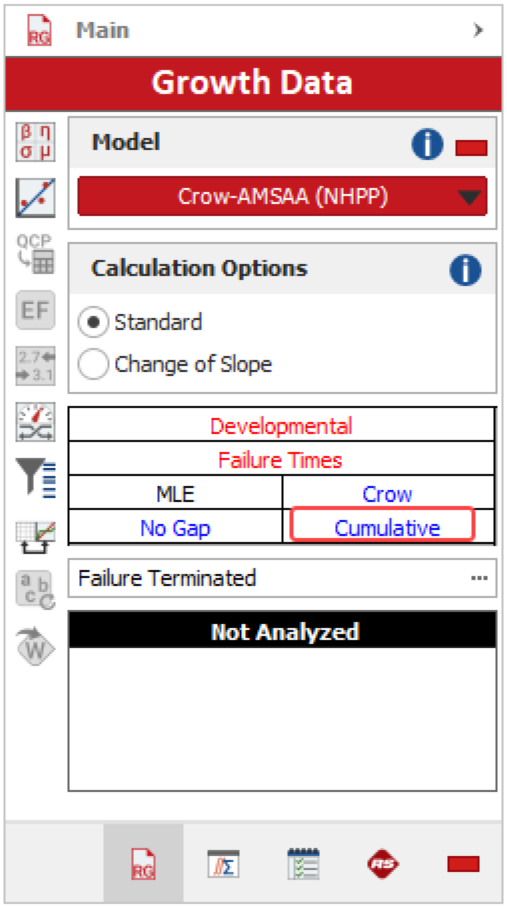
Number of Trials	Number of Failures
10	5
18	8
27	11
36	13
46	15
56	16

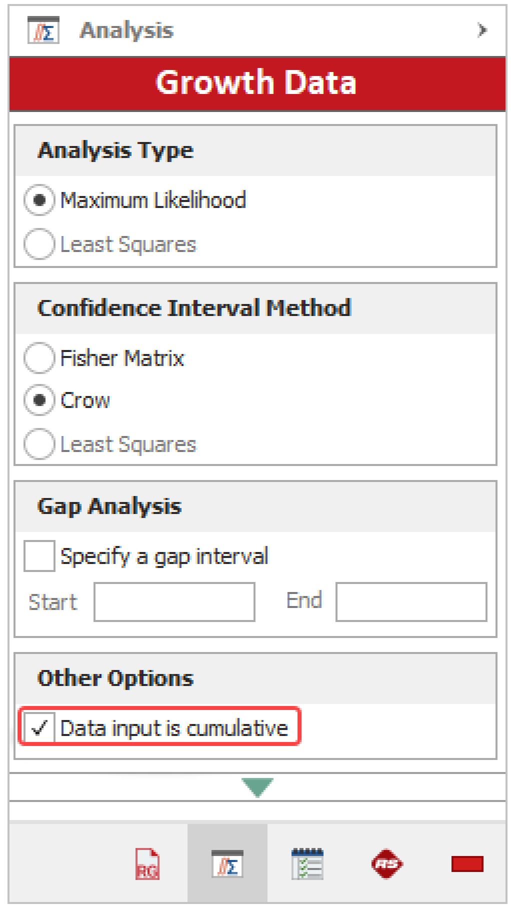
Cumulative

Number of Trials	Number of Failures
10	5
8	3
9	3
9	2
10	2
10	1

Non-Cumulative

You can specify whether the values are cumulative or non-cumulative by clicking the blue text on the **Main** page of the control panel, or by selecting the **Data input is cumulative** check box on the **Analysis** page of the control panel. As an example, the following pictures show these two options for the Failure Times data type.





You can configure Weibull++ to always use a particular format each time you create any of the data sheets mentioned above. You can change this setting by selecting or clearing the **Data input is cumulative** check box on the RGA Growth Data Folios page of the Application Setup.

Normal View and Advanced Systems View

For reliability growth data analysis only.

When you are analyzing data from multiple systems, Weibull++ combines the data into a single representative system that allows you to evaluate all the failures that occurred during the observation period.

- In the following times-to-failure data types, the data sets are combined to form an equivalent system:

- Multiple Systems - Concurrent Operating Times
- Multiple Systems with Dates
- Multiple Systems with Event Codes

- In the Repairable Systems data type, the failure times are placed on a single timeline to form a superposition system.
- In the Fleet data type, the failure times are stacked into a cumulative timeline.

For more information about how the software combines the data for analysis, click one of the links above. This topic focuses on the two different ways that you can view and enter data for all of these data types, regardless of how the analysis is performed.

The **Normal View** displays the data for multiple systems all together in the same data sheet, while the **Advanced Systems View** displays the data one system at a time and provides a navigation panel to select which system you wish to view. Any changes that you make to the data while in one view will be reflected if you switch to the other view; however, only the Advanced System View allows you to select specific systems to be excluded from the analysis (e.g., you may choose to exclude a specific system if that data set is not representative of the entire population).

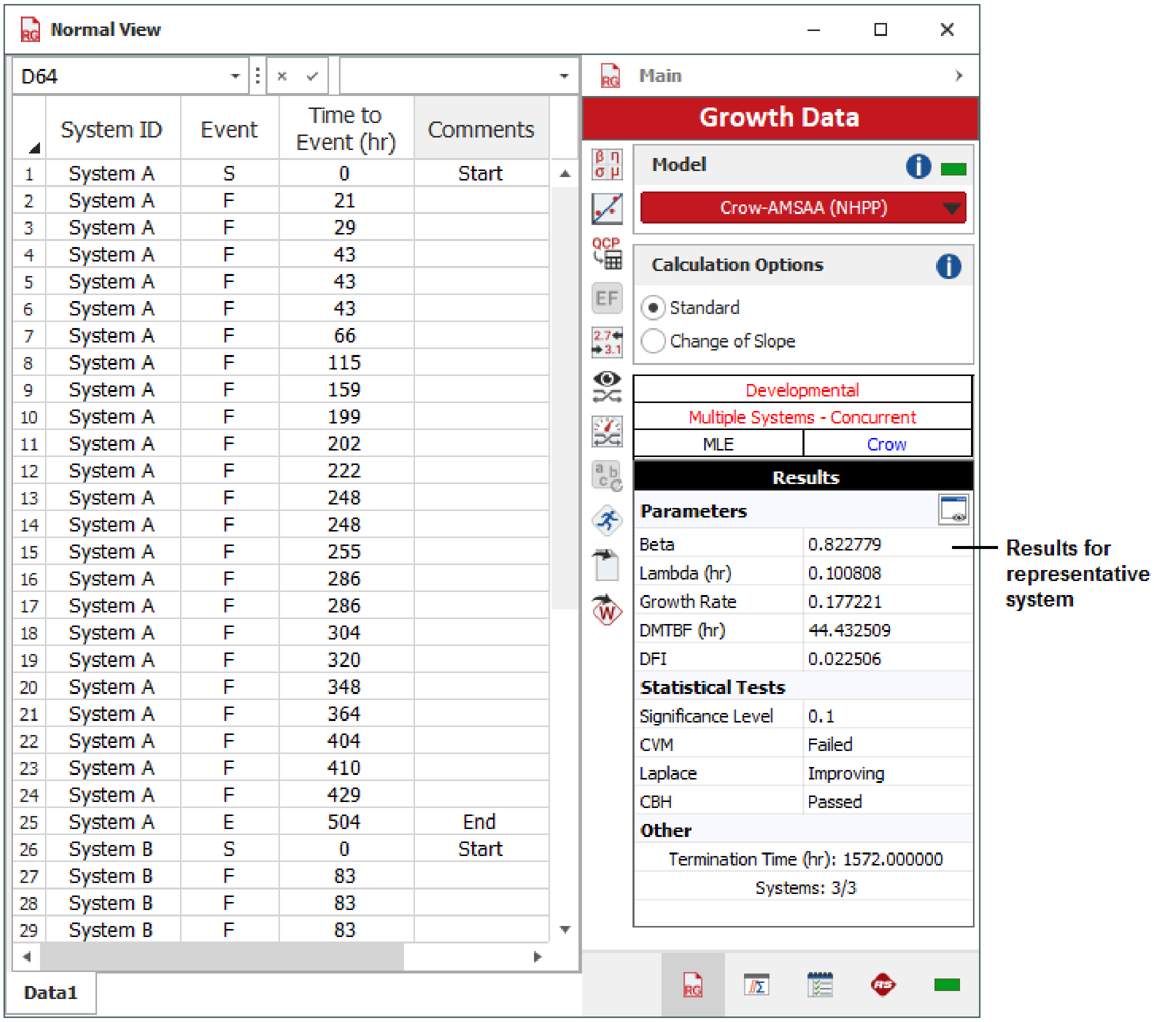
To switch between the two views, choose **Growth Data > Format and View > Switch Systems View** or click the icon on the Main page of the control panel.

Regardless of which view you choose, the System Operation plot shows the failure times of the systems that were included in the analysis, as well as the timeline for the representative system.

NORMAL VIEW

In the Normal View, the **System ID** column is for recording the identifier of the system (e.g., System A, SN#1234, etc.). All events entered for this system must have the same ID. The **Event** column specifies whether the row represents the start time (S), failure time (F) or end time (E) of the system. The **Time to Event** column is for recording the time of the event.

When you calculate the data sheet, the analysis results of the representative system (i.e., the equivalent system, superposition system or cumulative timeline) are displayed in the **Results** area of the control panel, as shown next. If you wish to see the results for an individual system, you must switch to the Advanced Systems View.

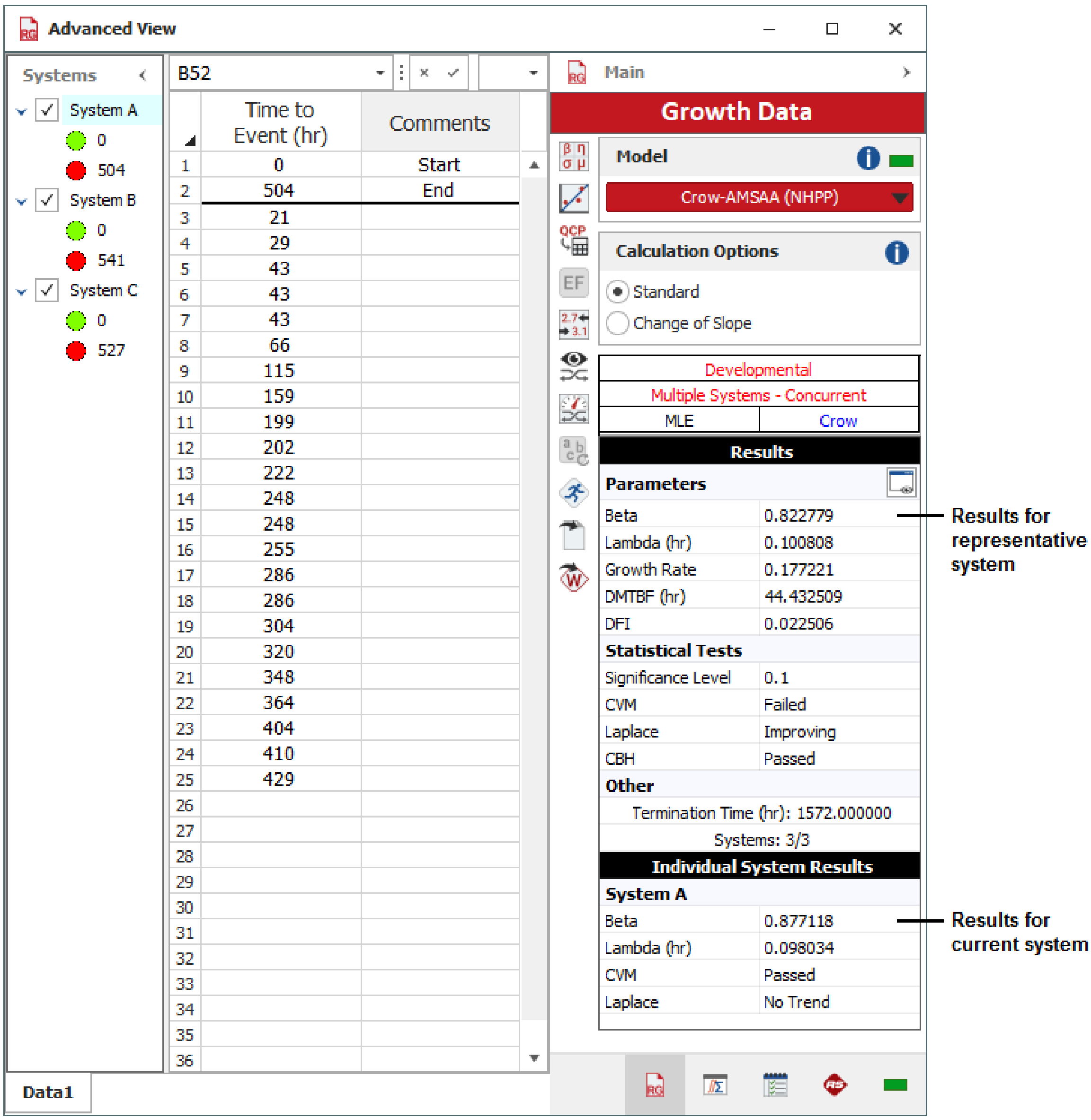


ADVANCED SYSTEMS VIEW

In the Advanced Systems View, the left side of the data sheet shows all the systems that have been defined along with their start times (marked with green bullets) and end times (marked with red bullets). The check box to the left of each system name allows you to select whether to include that system in the representative system for analysis (i.e., the equivalent system, superposition system or cumulative timeline). By default, all systems are selected to be included in the analysis; however, you can clear the check box to remove the system from the combined analysis. You can also edit a system name by right-clicking it and choosing **Rename** on the shortcut menu.

The following example shows the Advanced Systems View with the data for System A visible. In the **Time to Event** column, the start time of the system is entered in row 1, while the end time is entered in row 2 (these rows are reserved for this information). The failure times are entered into the subsequent rows, starting with row 3.

When you calculate the data sheet, the analysis results for the representative system (i.e., the equivalent system, superposition system or cumulative timeline) are shown in the **Results** area of the control panel. In addition, the analysis results for the individual system that is currently selected will be displayed in an **Individual System Results**, as shown next.



Transfer to New Data Type

For reliability growth data analysis only.

In some cases, you can transfer your data from one type of data sheet to another. This option is available only for fielded data and some of the multiple systems data types (Concurrent Operating Times, with Dates and with Event Codes).

To transfer your data, choose **Growth Data > Transfer Data > Transfer to New Data Type** or click the icon on the Main page of the control panel.

You can choose to transfer data between any of the specified data types. A new data sheet with the transferred data will be added to the current folio. In other words, the original data sheet remains unchanged and a new data sheet is created in the same folio.

In addition, the following data sheets give you the option to transfer the data to an equivalent system, which is a single system that represents the combined operating time of all the systems in the data set. Note that this option is available only when the parameters of the data set have been calculated.

- For the Concurrent Operating Times and Multiple Systems with Dates data types, the equivalent system will be transferred into a Failure Times data sheet, which allows you perform gap analysis or link the data set to a mission profile, if desired.
- For the Multiple Systems with Event Codes data type, the equivalent system will be transferred into a Multi-Phase Failure Times data sheet, which allows you to use the Crow Extended - Continuous Evaluation model to analyze the data set.

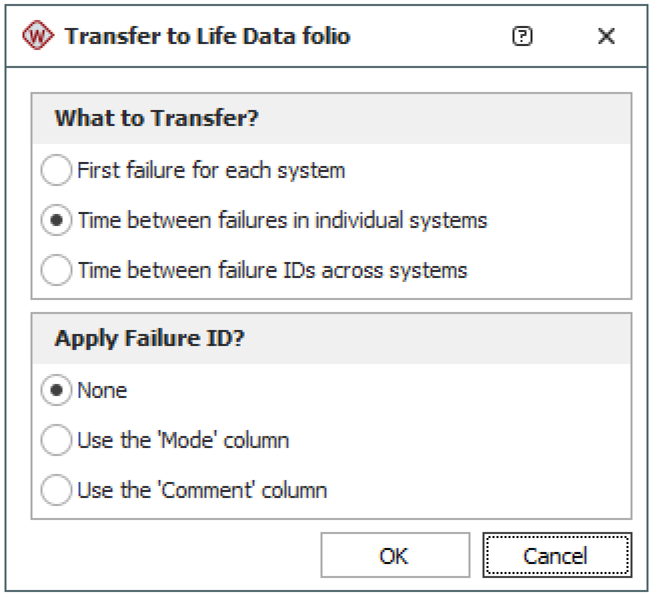
Transfer Data to a Life Data Folio

For reliability growth data analysis only.

You can transfer data from a reliability growth analysis (RGA) folio to a life data folio. This option is available for time-to-failure data (except Multiple Systems – Known Operating Times), multiphase data (except Mixed Data) and fielded data.

To transfer your reliability growth analysis data, first calculate the parameters of the data set, and then choose **Growth Data > Transfer Data > Transfer to Life Data folio** or click the icon on the Main page of the control panel.

The Transfer to Life Data folio window appears, as shown next. The options that are available will depend on the type of reliability growth analysis data sheet and model you are using.



- **What to Transfer?** specifies how your reliability growth analysis data will be handled in the life data folio.
  - **First failure for each system** transfers only the first failure time of each system in the reliability growth analysis data sheet. Choose this option if you wish to use the life data folio to fit a life distribution to the first times to failure and calculate the reliability (i.e., probability of operating without failure for a period of time) of the system.
  - **Time between failures in individual systems** transfers all the failure times of each system in the reliability growth analysis data sheet. Although it is generally not recommended to fit a life distribution to repairable systems data, you might choose this option if you wish to use the life data folio to fit a distribution to the data set for simulation in BlockSim, ReliaSoft's system reliability and maintainability analysis software.

    If the failure times in the data sheet are cumulative, then the process converts the times into their equivalent non-cumulative failure times (because data entry in the life data folio is always non-cumulative). If the data in the reliability growth analysis data sheet are already non-cumulative, then no conversion will occur and the failure times that are transferred will appear as they do in the reliability growth analysis data sheet.
  - **Time between failure IDs across systems** transfers all the failure times in the reliability growth analysis data sheet based on the unique failure modes in each system (you will need to select one of the options in the Apply Failure ID area to serve as the identifier for the failure modes). Choose this option if you wish to use the life data folio to fit a distribution to each failure mode (assuming that the systems are the lowest repairable units).

If a failure mode did not occur in a particular system, then it will be treated as a suspension for that system. Similar to the previous option, if the failure times are cumulative, the process converts the data into their equivalent non-cumulative failure times; if the data are already non-cumulative, then no data conversion will occur.

- **Apply Failure ID?** specifies how each data point in the life data sheet will be grouped into subsets:
  - The **None** option does not group the data points in the life data sheet.
  - The **Use the "Mode" column** option is available only when using the Crow Extended or Crow Extended - Continuous Evaluation models. It transfers the combined information from the Classification and Mode columns in the reliability growth analysis data sheet to the Subset ID column in the life data folio. For example, for a reliability growth analysis data point that has the classification "BD" for mode "100," that data point will be categorized as part of subset "BD100" in the life data folio.
  - The **Use the "Comment" column** option uses any text/values that were entered in the Comments column of the reliability growth analysis data sheet to group the data points in the life data folio.

A NOTE ABOUT SUSPENSION TIMES IN THE TRANSFERRED DATA

When you select to apply a failure ID, all subsets are assumed to end at the same time. For example, if you have two systems where the observation period for System 1 ended at 100 hours and for System 2 at 150 hours, then the transferred data for System 1 will automatically include a suspension time that covers the difference between the end times of the two systems.

Note that if the data were cumulative, then the suspension time would be the difference between the last failure time for System 1 and the last failure time for System 2 (e.g., 150 - 100 = 50 hours), but if the reliability growth analysis data were non-cumulative, then the suspension time would be the difference between the last failure time for system 1 and the total test duration (e.g., if the observation period ended at 250 hours, then the suspension time would be 250 - 100 = 150 hours).

EXAMPLES

The following examples show pictures of how data from a cumulative Failure Times data sheet may be transferred to a life data folio. Here is the original reliability growth analysis data.

	Time to Event (hr)	Classification	Mode	Comments
1	1	BC	1	Mode 1
2	4	BC	2	Mode 2
3	13	BC	3	Mode 3
4	18	BD	4	Mode 3
5	55	BD	5	Mode 3
6	100	BD	6	Mode 3

Reliability Growth Failure Times data sheet (cumulative)

Example 1: Here is the data set transferred using the "Time between failures in individual systems" option with no failure ID applied. The cumulative times have been converted to non-cumulative. All failure times are transferred, but the failure mode is not identified.

	State F or S	Time to F or S (hr)	Subset ID 1
1	F	1	
2	F	3	
3	F	9	
4	F	5	
5	F	37	
6	F	45	

Example 2: Here is the data set transferred using the "Time between failures in individual systems" option and the "Use the Mode column" option. The cumulative times have been converted to non-cumulative. The failure times are grouped into subsets based on the combined information from the Classification and Mode columns in the growth data sheet. The suspension data are added automatically to ensure that all the subsets have the same termination time.

	State F or S	Time to F or S (hr)	Subset ID 1
1	F	1	BC1
2	S	99	BC1
3	F	4	BC2
4	S	96	BC2
5	F	13	BC3
6	S	87	BC3
7	F	18	BD4
8	S	82	BD4
9	F	55	BD5
10	S	45	BD5
11	F	100	BD6

Example 3: Here is the data set transferred using the "Time between failures in individual systems" option and the "Use the Comment column" option. The results are the same as example 2, except that the failure IDs came from the Comments column in the growth data sheet.

	State F or S	Time to F or S (hr)	Subset ID 1
1	F	1	Mode 1
2	S	99	Mode 1
3	F	4	Mode 2
4	S	96	Mode 2
5	F	13	Mode 3
6	F	5	Mode 3
7	F	37	Mode 3
8	F	45	Mode 3

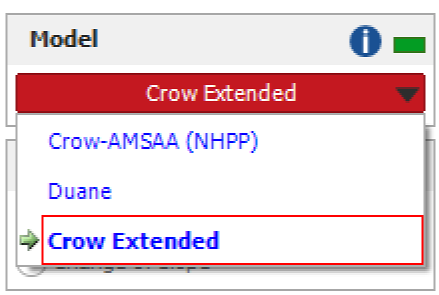
Growth Data Folio Control Panel

For reliability growth data analysis only.

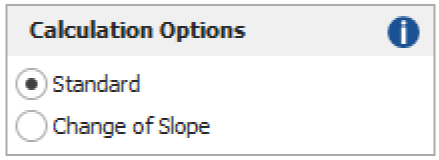
The growth data folio control panel allows you to configure the analysis settings for the data sheet and view/access the results. This topic provides general information that applies to most analyses. For more focused information about performing a particular type of analysis, you can go directly to the topic of interest. (See Traditional Reliability Growth Analysis, Growth Projections, Planning and Management (Crow Extended Model) or Repairable Systems Analysis.)

Control Panel Main Page

- The **Model** area allows you to choose a statistical model to use in analyzing the data set. The list of models available will depend on the data type. See the Reliability Growth Data Types topic for a complete list.



- The **Calculation Options** area allows you to choose the Change of Slope analysis method for situations where there has been a significant change in the failure intensity of the system. (See Change of Slope Analysis.)



- The available analysis settings may vary depending on the data type. The following picture shows an example of the settings that are available for the Failure Times data type. In this example, the settings show the type of data (Developmental – Failure Times), parameter estimation method (MLE), confidence bounds method (Crow), whether a gap interval has been defined in the analysis (No Gap) and whether the failure times are entered as cumulative or non-cumulative (Cumulative).

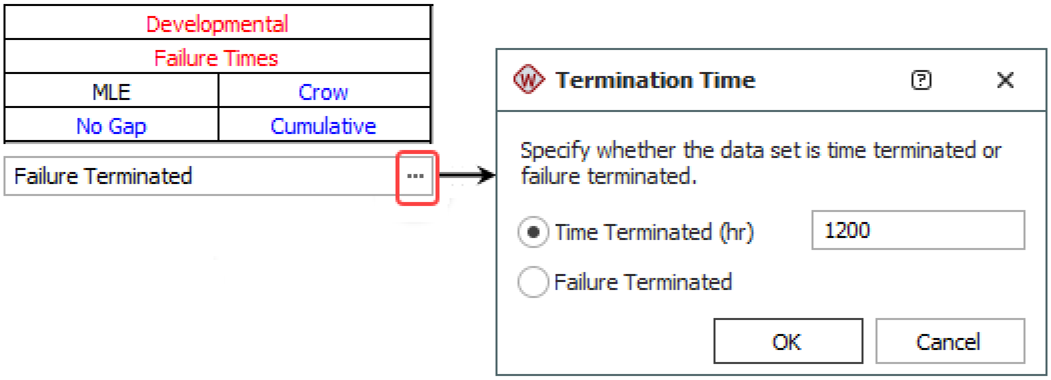
Developmental	
Failure Times	
MLE	Crow
No Gap	Cumulative

  If a setting is displayed in blue text, you can click the label to switch between the available options. These settings are also available on the Analysis page of the control panel. See Control Panel Analysis Page for a description of all available settings.

- The **Termination Time** field is available when you use the Crow-AMSAA (NHPP) or Crow Extended models with any of the following times-to-failure data types:

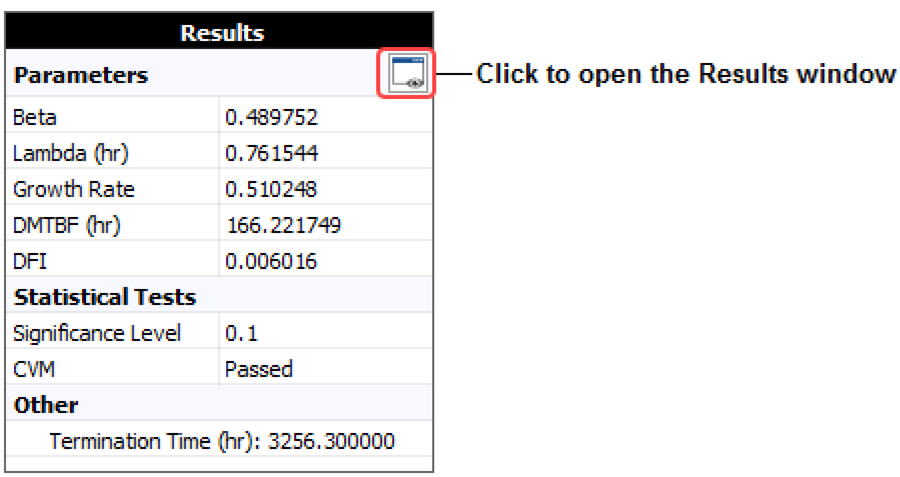
  - Failure Times
  - Grouped Failure Times
  - Multiple Systems - Known Operating Times

  This setting allows you to specify the time when the observation period ended, if applicable. Click the (**...**) button to open the Termination Time window, as shown next. Select the **Time Terminated** option to enter the actual observation time, or select the **Failure Terminated** option to indicate that the observation period ended with the last failure time recorded in the data sheet.



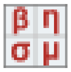
- The **Results** area displays the calculated parameters and other results. The information shown in this area depends on the current data type and model, as well as your selections on the RGA Growth Data Folios page and Calculations page of the Application Setup. If you are analyzing data from multiple systems, this area may show two sets of results (see Normal and Advanced Systems View).

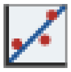
  As an example, the following picture shows the analysis results for one particular data set and model. Click the **Results Report** icon to open the Results window, which shows the results in a worksheet that you can copy or print.



Folio Tools

The folio tools are arranged on the left side of the Main page. Depending on the data type or model used in the analysis, the growth data folio control panel may contain some or all of the following tools:

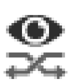

Calculate estimates the parameters based on the selected model and analysis settings. Once the parameters have been calculated, the Results area will show the current results. This command is also available by choosing **Growth Data > Analysis > Calculate**.


Plot creates a new sheet in the folio that provides a choice of applicable plot types. This includes plots such as MTBF


QCP opens the Quick Calculation Pad, which allows you to obtain calculated results based on the analyzed data sheet, such as the cumulative MTBF and the expected number of failures. This command is also available by choosing **Growth Data > Crow Extended > QCP**.


Effectiveness Factors is available only when you use the Crow Extended or Crow Extended - Continuous Evaluation models. It opens the Effectiveness Factors window, which allows you to define the effectiveness factors for each BD mode to estimate the fractional decrease in failure intensity that can be expected after the delayed fix is applied. This command is also available by choosing **Growth Data > Crow Extended > Effectiveness Factors**.


Alter Parameters opens a tool that allows you to alter the values of the calculated parameters, assuming that the likelihood function and Fisher Matrix (evaluated at the original parameter estimates) remain the same. In plots, the position of the data points remain the same, but the solution line and all subsequent analyses 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. This command is also available by choosing **Growth Data > Options > Alter Parameters**.


Switch System View is available only for fielded data and some of the multiple systems data types (Concurrent Operating Times, with Dates and with Event Codes). It allows you to switch between the two complementary views that are available for some multiple systems data types (see Normal and Advanced Systems View). This command is also available by choosing **Growth Data > Format and View > Switch Systems View**.


Change Units is available for all data types except discrete data. It opens the Change Units window, which allows you to change the units of an existing data sheet.

Auto Group Data is available for data types where the exact failure times have been recorded (i.e., Failure Times and Multi-Phase Failure Times) and for the Fleet data type. It opens the Auto Group Data window, which allows you to specify the intervals that will be used to group the data. This command is also available by choosing **Growth Data > Options > Auto Group Data**.

Link Mission Profile is available only for the Failure Times data sheet. It applies to cases where testing involves multiple test profiles. This command allows you to select a mission profile in the project that you want to use to analyze the data set. The data in the current data sheet will be transferred to a new data sheet in the same folio, and then grouped according to the "convergence points" in the specified mission profile. (See Mission Profiles.)

Mode Processing is available for data sets analyzed with the Crow Extended or Crow Extended - Continuous Evaluation models. This utility allows you to extract the first failure time for each unique BC mode and/or unique BD mode in a data set. This allows you to automatically copy the failure times for a particular failure mode classification into a new data sheet for separate analysis.

Event Report is available for all multi-phase data types and for the Multiple Systems with Event Codes data type. It opens a report about the failure modes in the current analysis, such as the classification of the modes, their first times to failure and effectiveness factors. This command is also available by choosing **Growth Data > Analysis > Event Report**. (See Event Reports.)

Batch Auto Run is available only for multi-phase data, fielded data and some of the multiple systems data types (Concurrent Operating Times, with Dates and with Event Codes). It opens the Batch Auto Run window, which allows you to quickly extract data from an existing data sheet based on the system ID, phase or analysis point. This command is also available by choosing **Growth Data > Options > Batch Auto Run**.

Transfer to New Data Type is available only for fielded data and some of the multiple systems data types (Concurrent Operating Times, with Dates and with Event Codes). It opens the Transfer to New Data Type window, which allows you to select the data type into which you would like to transfer the data. This command is also available by choosing **Growth Data > Transfer Life Data > Transfer to New Data Type**.

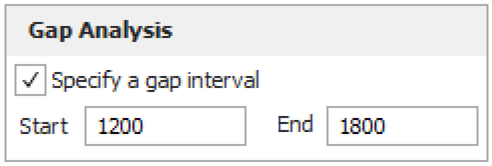
Transfer to Life Data folio is available for time-to-failure data (except Multiple Systems – Known Operating Times), multi-phase data (except Mixed Data) and fielded data. It opens the Transfer to Life Data folio window, which allows you to transfer the current reliability growth data set to a new life data folio data sheet. This command is also available by choosing **Growth Data > Transfer Life Data > Transfer to Life Data folio**.

Growth Data Folio Analysis Settings

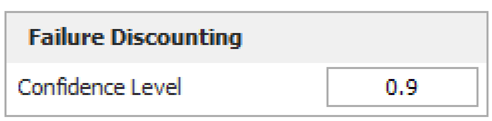
For reliability growth data analysis only.

The Analysis page of the growth data folio control panel may contain some or all of the following options (and some of these settings can also be changed on the Main page of the control panel):

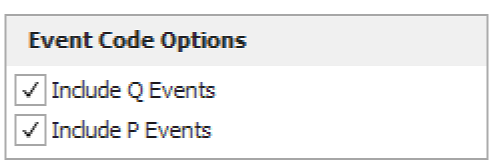
- The **Analysis Type** area allows you to choose the method for estimating the parameters of your chosen model.
  - The **Maximum Likelihood** parameter estimation method is available for the Crow-AMSAA, Crow Extended, Crow Extended - Continuous Evaluation, Power Law and Lloyd-Lipow models.
  - The **Least Squares** method is available for the Lloyd-Lipow, Standard Gompertz, Modified Gompertz, Duane and Logistic models.
- The **Confidence Interval Method** area allows you to choose a method for calculating the confidence bounds. The methods that are available depend on the data type and model.
  - **Fisher Matrix** and **Crow** confidence bounds are available for the Crow-AMSAA (NHPP), Crow Extended, Crow Extended - Continuous Evaluation and Power Law models. There are two exceptions:
    - If you use the Crow-AMSAA model with the Discrete Grouped per Configuration data type, then Least Squares is the only available confidence bounds method.
    - Fisher Matrix is the only available confidence bounds method for the Lloyd-Lipow model.
  - The **Least Squares** method is available for the Standard Gompertz, Modified Gompertz, Duane and Logistic models.
- The **Gap Interval** setting is available only when you use the Failure Times data type with the Crow-AMSAA (NHPP) model. When a gap is defined in the data set, the application assumes that reliable information for that time period is unknown and ignores any entries that have been made for the specified time interval. (See Gap Analysis.)



- The **Data input is cumulative** check box allows you to specify whether the data will be entered cumulatively (where each row shows the total amount of operating time when the failure occurred) or non-cumulatively (where each row shows the incremental operating time from when the last failure occurred). When applicable, this setting also applies to the data entry for number of trials and number of failures in discrete data sheets. (See Cumulative vs. Non-Cumulative Data.) This check box is available for the following data sheets:
  - Times-to-failure data:
    - Failure Times
    - Multiple Systems - Known Operating Times
  - Discrete data:
    - Grouped per Configuration
  - Multi-phase data:
    - Failure Times
- The **Failure Discounting** setting is available only for the Discrete Sequential with Mode data type. It allows you to specify the confidence level that will be used to define the fractional decrease in failure value. (See Failure Discounting.)



- The **Event Code Options** area is available only for the Multiple Systems with Event Codes data type and all multi-phase data types. It allows you to select whether or not to include Q (quality failure) and/or P (performance failure) events in the reliability growth projections analysis.



- The **Statistical Tests** area is available only for the Crow-AMSAA (NHPP) or Crow Extended models. It allows you to specify the significance level used in the statistical tests. (See Statistical Tests.)

Quick Calculation Pad (QCP)

For reliability growth data analysis only.

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.

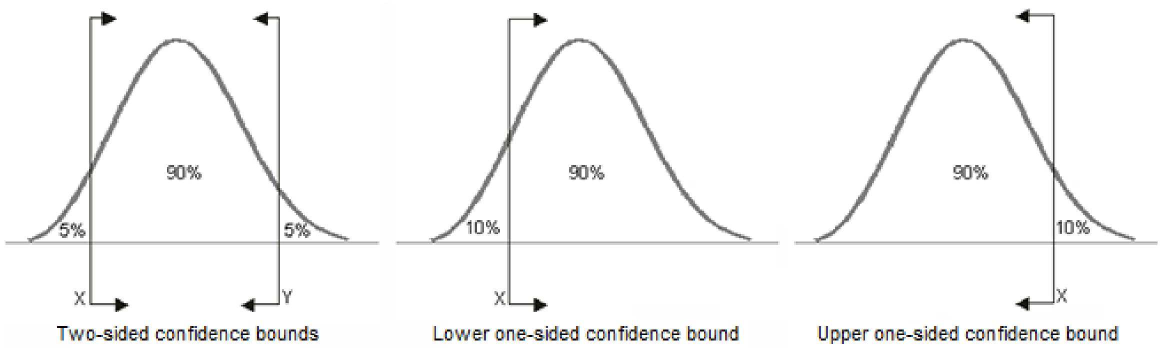
This topic provides a general description of the QCP and how to use it in most analyses. For more focused information about the types of calculations that can be performed for a particular type of analysis/model, you can go directly to the topic of interest:

- Traditional reliability growth analysis (Crow-AMSAA, Duane and discrete models).
- Crow Extended model.
- Crow Extended - Continuous Evaluation model.
- Repairable systems analysis (Crow-AMSAA and Power Law models).

How to Use the QCP

To use the QCP, do the following:

1. Choose a metric in the **Calculate** area. The metrics available depend on the data type and, in the case of the Crow Extended model, which fix strategies were implemented (i.e., BC = fixed during test or BD = delayed fix).
2. If applicable, use the **Units** drop-down list to specify the units for any 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, you could enter the failure times in hours but then calculate the cumulative number of failures for 1 month of operation — the application will convert the times automatically based on the conversion factors specified for the database.
3. If applicable, use the **Bounds** drop-down list to specify what type of confidence bounds to calculate. The following graphics illustrate the types. The ReliaWiki resource portal provides more information on the background theory of confidence bounds at: http://www.reliawiki.org/index.php/Confidence Bounds.



If you select **Both One-Sided**, the QCP will calculate both the lower one-sided bound and the upper one-sided bound.

4. Make any required inputs in the **Input** area.

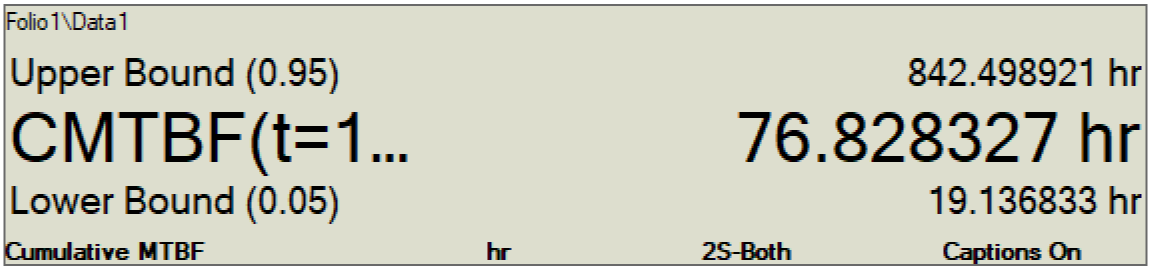
5. Click **Calculate**. The calculated value(s) will always be displayed in the results area at the top of the window.

Other Options

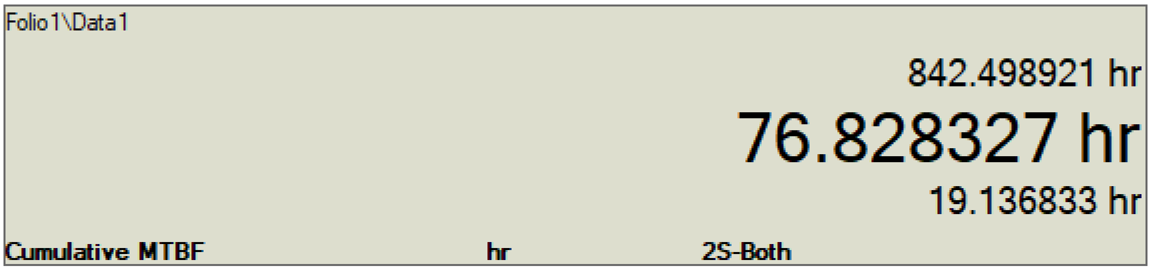
1. Click the **Report** button at the bottom of the window to display a summary of the current calculation input/output in the Results window.

2. 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).
   - 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 cumulative MTBF 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



QCP window with captions off

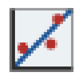
- Select **Show Calculation Log** if you want to display a log on the right side of the window that records some or all of the calculations performed during this QCP session (a green light in the button indicates that this option is selected).
  - If **Auto Print Results** is selected under **Options**, 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 data sheets (a green light in the button indicates that this option is selected). 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. This setting can also be changed by using the **While QCP is open, have access to all folios** option on the Other page of the Application Setup.

Reliability Growth Plots

For reliability growth data analysis only.

Reliability Growth Data folios (RGA) offer a variety of plots to show a visual representation of your analysis results; the available plots depend of what kind of analysis you are performing. To create or view a plot for an analysis, click the **Plot** icon on the Main page of the control panel.



For information about how to use the plot utilities, refer to Plot Utilities.

For focused information on the types of plots that can be generated for a particular analysis, you can go directly to the topic of interest:

- QCP Calculations and Plots for Traditional RGA
- QCP Calculations and Plots for Crow Extended
- QCP Calculations and Plots for Multi-Phase Data
- QCP Results and Plots for Repairable Systems Analysis

Publishing Models from Analysis Results

For reliability growth data analysis only.

Once a data sheet has been analyzed, you can publish the results as a model. To publish a model based on the analysis results, click the **Publish to Model** icon on the Publishing page of the control panel.

The type of model that will be published depends on the data type and growth model that you are working with:

- For discrete data types (Multi-Phase Mixed data sheet and Discrete data sheets), the results are in terms of the demonstrated reliability, so the published model will be a fixed reliability value.

- For all other developmental data, the results are based on the reliability growth process; therefore, the published model will be a 1-parameter exponential distribution with a mean time equal to the demonstrated MTBF.
- For fielded data analyzed with the Power Law model, the published model will be a 2-parameter Weibull distribution, where the beta parameter indicates whether the system is exhibiting wearout, infant mortality or a constant failure rate (beta = 1). If the data are analyzed with the Crow Extended model, the published model will be a 1-parameter exponential distribution with a mean time equal to the instantaneous MTBF at the termination time.

All other tools for working with the published model, such as displaying its properties and tracing its usage, are on the Publishing page of the growth data folio's control panel.

Statistical Tests

For reliability growth data analysis only.

When you use the Crow-AMSAA (NHPP) or Crow Extended models, Weibull++ automatically performs statistical tests on the calculated data set. The results of the tests help you to evaluate how well the model fits the data. Only the test that applies to the data type and model you are using will appear in the results. The results are displayed on the growth data folio control panel, and can also be displayed in a report by choosing **Growth Data > Analysis > Statistical Tests Report**.

The following tests evaluate the hypothesis that the failure times follow a non-homogeneous Poisson process (NHPP). The ReliaWiki resource portal provides more information about these tests at http://www.reliawiki.org/index.php/Crow-AMSAA_(NHPP).

- The **Chi-squared** goodness-of-fit test is applied to grouped failure times.
- The **Cramér-von Mises (CVM)** goodness-of-fit test is applied to non-grouped failure times where there are no gaps in the data.

The following tests apply to multiple systems analysis only. The ReliaWiki resource portal has more information about these tests at http://www.reliawiki.org/index.php/Hypothesis_Tests.

- The **Common Beta Hypothesis (CBH)** test indicates whether all the systems in the data set have similar beta values so you can evaluate whether the systems should be combined into a single representative system (i.e., the equivalent system, superposition system or cumulative timeline).

- The **Laplace Trend** test evaluates the hypothesis that a trend does not exist in the data. It can determine whether the system reliability is improving, deteriorating or staying the same. When the Crow Extended model is used without BC failures modes, it is assumed that there is no trend (i.e., the system is neither improving nor deteriorating).

Tip: You can set the default significance level for the statistical tests by entering a value in the RGA Growth Data Folios page of the Application Setup, or by changing the value on-the-fly from the Analysis page of the growth data folio control panel.

The following tools are also available in Weibull++, but are not automatically performed when you calculate the data set:

- The Interval Goodness-of-Fit Test tool helps you to determine which intervals should be used to group the data so that the goodness-of-fit test passes. This option is available only for fielded data and some of the multiple systems data types (Concurrent Operating Times, with Dates and with Event Codes).
- The Test for Fix Effectiveness tool helps you to assess whether or not applied fixes have been effective across test phases. It is available only for the multi-phase data types.

Interval Goodness-of-Fit Test

For reliability growth data analysis only.

The Interval Goodness-of-Fit Test window helps you to group failure times into intervals so that the chi-squared goodness-of-fit test will pass when you analyze the grouped data. You specify the intervals that you are considering and the tool automatically determines the actual number of failures that were observed at the specified intervals and the number of expected failures based on the Crow AMSAA (NHPP) model. It then compares the observed vs. expected number of failures in order to evaluate how well the Crow-AMSAA (NHPP) model will fit the data when grouped into the proposed intervals.

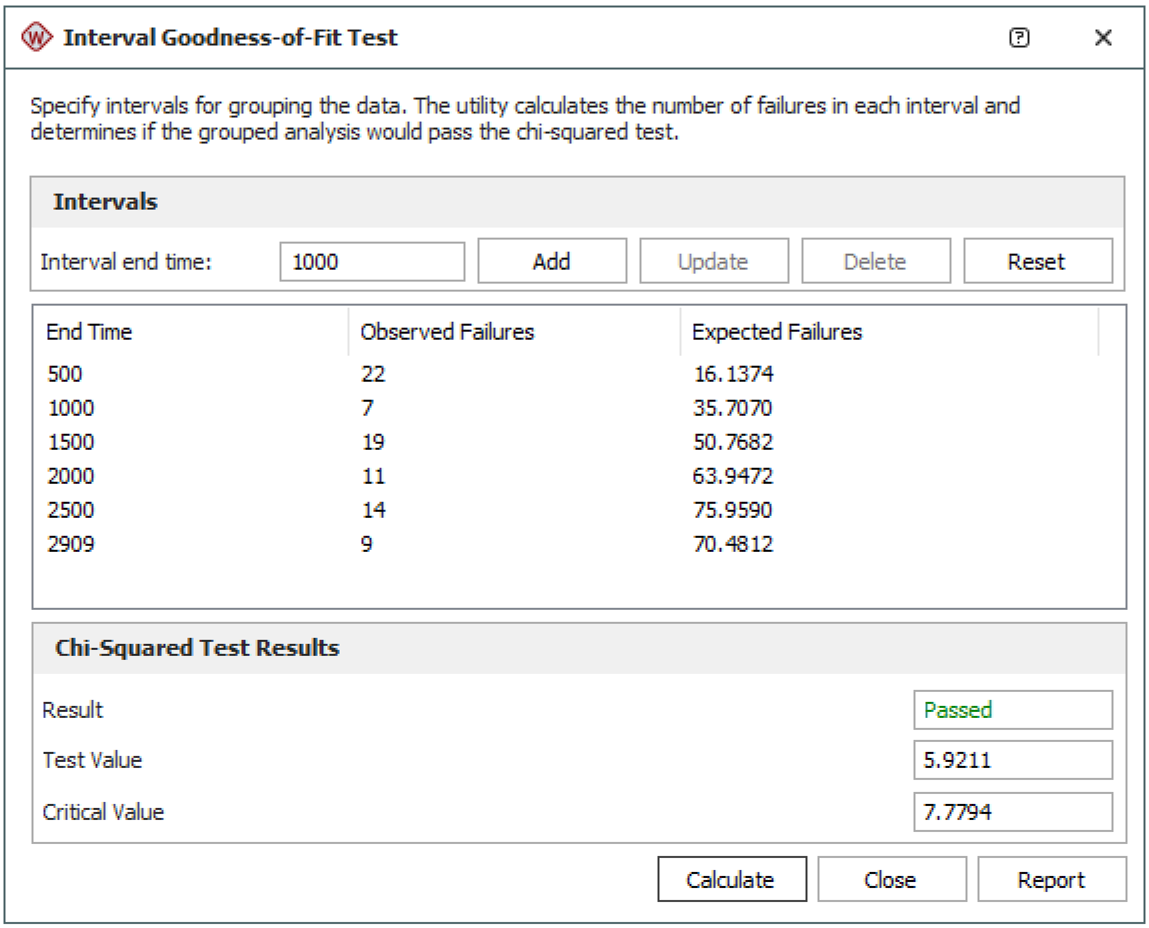
If the model fits the data, the result will indicate that the data set passed the test; but if there is a large difference between the observed and expected number of failures, the test may fail. In this case, the model may not be representative of the data and you will need to adjust the specified intervals such that the test passes.

The ReliaWiki resource portal provides more information about the chi-squared analysis at http://www.reliawiki.org/index.php/Crow-AMSAA_(NHPP).

This tool is available only for fielded data and some of the multiple systems data types (Concurrent Operating Times, with Dates and with Event Codes).

To use the tool, first calculate the parameters of the data set, and then choose **Growth Data > Options > Interval GOF**.

The following picture shows an example of how to use the utility. The **End Time** column displays the interval end times that you want to consider. The **Observed Failures** column displays the observed failures at the specified interval end time. The **Expected Failures** column shows the number of failures that would be expected for that interval based on the Crow-AMSAA (NHPP) model. Note that the termination time of the test is automatically entered for you (in this example, the termination time is 2,909 hours).



- To add an interval to be considered, enter a value in the **Interval end time** field and click **Add**.

- To change an interval, select the row, enter a new value in the **Interval end time** field and click **Update**.
- To remove an interval, select the row and click **Delete**. Alternatively, you can remove all the intervals that were entered by clicking **Reset**.

Once you have specified all of the intervals that you want to consider for grouping the data, click **Calculate** to perform the test. The **Result** field displays whether the test passed or failed.

You can click **Report** to open a summary of the analysis in the Results window. If you want to proceed with the grouped data analysis, you can copy the "End Time" and "Observed Failures" values into a new Grouped Failure Times data sheet.

Select Systems to Plot

For reliability growth data analysis only.

When you are analyzing data from multiple systems, Weibull++ combines the data into a single representative system that allows you to evaluate all the failures that occurred during the observation period. You can view the resulting timeline via the System Operation plot, which also shows the failure times of each available system in the data sheet.

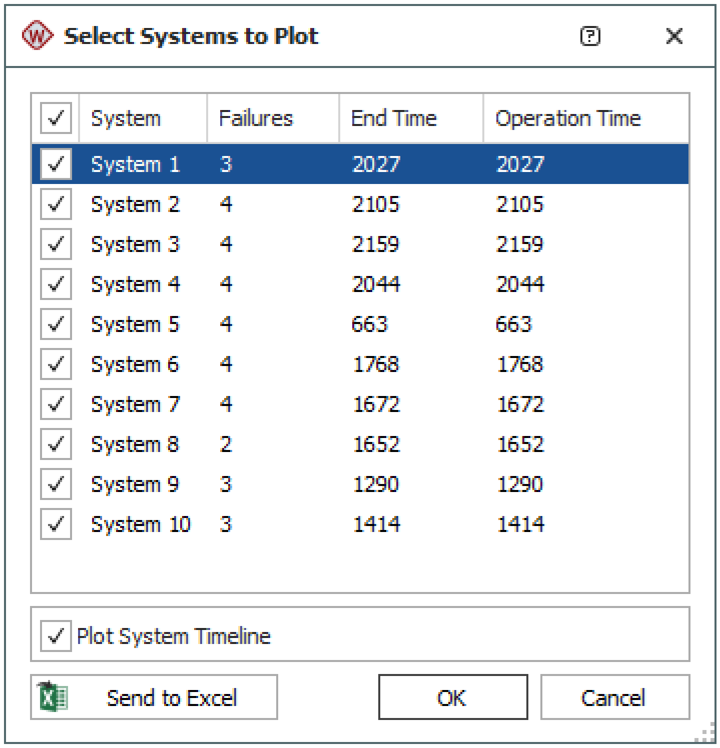
In the Systems Operation plot, you can select to display or hide a system on the plot by choosing **Plot > Options > Plot Systems** or by right-clicking the plot and choosing the option on the shortcut menu.

The Select System to Plot window shows all the systems that were analyzed, including the end time, total operating time and number of recorded failures in each system. To add or remove a system on the plot, select or clear the corresponding check box. Click the column heading above the check boxes to select or clear all check boxes.

To change the order in which the systems appear in the plot, click any of the column headings to sort the values. For example, clicking the **Failures** column heading will sort the systems from the lowest to the highest number of failures. Clicking the heading again reverses the sort order. Clicking one more time returns the order to its original state (i.e., sorted by system).

To show or hide the timeline of the representative system, select or clear the **Plot System Timeline** check box.

To export all the data shown in the window to an Excel file, click the **Send to Excel** button.



Updating RGA Spreadsheets and Reports from Prior Versions

For reliability growth data analysis only.

In versions prior to Version 2020, RGA was a separate desktop application that included both specialized functions for reliability growth analyses as well as modified versions of functions that are also used in Weibull++. While most of the functions in Version 2020 and later work the same, there are some changes that you will need to make before using a general spreadsheet or ReliaSoft Workbook that you created in an earlier version.

New Functions

The following RGA-specific functions have new names in Version 2020 as the names used in older versions are now specific for life data and life-stress data folios.

Old Name	New Name
Spreadsheet Functions	
FCOUNT	FCOUNT_G
PROBFAIL	PROFAIL_G

RELIABILITY	RELIABILITY_G
TIMEATR	TIMEATR_G
Word Processing Functions	
Number of Failures	Number of Failures_G
Probability of Failure	Probability of Failure_G
Reliability	Reliability_G
Time at Reliability	Time at Reliability_G

Data Sheet References

In prior versions, data sheet references looked like this:

```
=MODEL("Folio1!Data1")
```

In Version 2020, you must update the data sheet references to include the folder name, like this:

```
=MODEL("RGA!Folio1!Data1")
```

Data sources that are associated with ReliaSoft Workbooks do not have to be manually updated.

Plots

All reliability growth plots created in the Word Processing Module must be recreated.

Traditional Reliability Growth Analysis

For reliability growth data analysis only.

Depending on the data type and the model selected, the reliability growth data folio can be used to perform several different types of analysis. The Growth Data Folios topic provides general information that is applicable for any type of analysis (e.g., choosing a data type, specifying the time units, using the control panel, etc.). This topic provides more focused information about using the reliability growth data folio specifically for traditional reliability growth analysis.

This type of analysis is performed with data from in-house reliability growth testing that was conducted during the developmental stages for a product. The analysis assumes that all *fixes* (i.e., permanent design improvements) are applied immediately after failure and before testing resumes, and that a reliability growth model can be fitted to the data in order to track how the reliability changes over time. The metrics of interest may include the reliability, MTBF, failure intensity, expected number of failures for a given time or the amount of testing that will be required to demonstrate a specified reliability.

Data Types and Models for Traditional RGA

For reliability growth data analysis only.

The following information provides a summary of the data types and models that are applicable for traditional reliability growth analysis (i.e., the analysis of developmental testing data with the assumption that fixes are applied immediately after a failure and before testing resumes).

Data Types

You can perform traditional reliability growth analysis with any of the following data types:

- All times-to-failure data types (except for Multiple Systems with Event Codes, which is intended only for growth projections analysis)
- All discrete data types
- Reliability data

Reliability Growth Models

Weibull++ includes six reliability growth models that can be used to track how the reliability changes over time during developmental testing. The models available will depend on the data type. The following list provides links to the ReliaWiki resource portal that discusses in detail the assumptions behind each model.

- **Crow-AMSAA (NHPP)** - http://www.reliawiki.org/index.php/Crow-AMSAA_-_NHPP
- **Duane** - http://www.reliawiki.org/index.php/Duane_Model
- **Standard** and **Modified Gompertz** - http://www.reliawiki.org/index.php/Gompertz_Models
- **Lloyd-Lipow** - http://www.reliawiki.org/index.php/Lloyd-Lipow
- **Logistic** - http://www.reliawiki.org/index.php/Logistic

Analysis Results for Traditional RGA

For reliability growth data analysis only.

When you calculate a data set, the analysis results will be displayed in the **Results** area on the control panel. The following pictures show some examples of analysis results.

Results	
Parameters	
Alpha	0.538377
A	0.967358
DMTBF (hr)	174.299275
DFI	0.005737
Other	
Termination Time (hr): 3256.300000	

Duane model
(Failure Times data type)

Results	
Parameters	
Beta	0.461753
Lambda	1.241481
Growth Rate	0.538247
DFP	0.114308
DRel	0.885692

Crow-AMSAA model
(Discreet Sequential data type)

Results	
Parameters	
a	0.987115
b	0.352293
c	0.776624

Standard Gompertz model
(Reliability data type)

Results	
Parameters	
Beta	0.893875
Lambda (hr)	0.065714
Growth Rate	0.106125
DMTBF (hr)	39.687448
DFI	0.025197
Statistical Tests	
Significance Level	0.1
CVM	Passed
Laplace	No Trend
CBH	Passed
Other	
Termination Time (hr): 2909.000000	
Systems: 6/6	
Individual System Results	
1	
Beta	0.877118
Lambda (hr)	0.098034
CVM	Passed
Laplace	No Trend

Crow-AMSAA model
(multiple systems data type in Advanced Systems View)

The available results will vary depending on the data type and model you are working with. The results may include the following:

- The estimated parameters of the model are always displayed. For example, this will be a, b and c for Standard Gompertz; A and Alpha for Duane; Beta and Lambda for Crow-AMSAA; etc.
- The estimated **Growth Rate**. A larger growth rate means faster MTBF growth.

- The instantaneous MTBF and failure intensity at the end of the observation period. An option on the Calculations page of the Application Setup window determines whether you prefer to call these values "demonstrated" (**DMTBF** and **DFI**) or "achieved" (**AMTBF** and **AFI**) for all new folios. For existing folios, you can change the option in the Settings tab of the Item Properties window (**Project > Current Item > Item Properties**).
- The demonstrated reliability (**DRel**) and demonstrated failure probability (**DFP**), which is 1 - DRel.
- Whether the test was **Failure Terminated** or has a specified **Termination Time** for the calculations. (See Termination Time window.)
- Whether the data set passed or failed the applicable goodness-of-fit tests that are performed automatically when you calculate the parameters (applies to the Crow-AMSAA model only). Only the test(s) that apply to the data type you are using will appear in the results. (See Statistical Tests.)
- For data sheets that support the Advanced Systems View, the following information will also be displayed.
  - The number of systems that were included in the analysis (e.g., 4/6 means four out of six systems were analyzed). You can omit a system from the analysis by clearing its check box on the Systems panel while the data sheet is in Advanced Systems View.
  - The control panel will show two sets of results: the larger table at the top shows the results for the *equivalent system* (which represents the combined operating hours of the systems), and the smaller table at the bottom show the results for the individual system that is currently selected in the data sheet. The specific system name is shown in the heading of this table.

Note: If the result of the analysis is associated with a published model, then the model's name will appear as a link at the bottom of the **Results** area. Click the link to view the model's properties. For details on how to publish the results as a model, see Publishing Models.

QCP Calculations and Plots for Traditional RGA

For reliability growth data analysis only.

Weibull++ includes a Quick Calculation Pad (QCP) for computing useful metrics, as well as multiple plots that allow you to visualize the results of your analyses. This topic describes the calculations and plots you can obtain from traditional reliability growth analysis (i.e., the analysis of developmental testing data with the assumption that fixes are applied immediately after a failure and before testing resumes).

Note: When you analyze data from multiple systems, Weibull++ combines the data to create a single *equivalent system*. Any plots generated for the combined data set and analyses via the Quick Calculation Pad will be based on the equivalent system. See Times-to-Failure Data from Multiple Systems for more information about how the software combines the data for analysis.

QCP Calculations

You can open the Quick Calculation Pad (QCP) by choosing **Growth Data > Analysis > Quick Calculation Pad** or by clicking the icon on the control panel.

To perform a calculation, select the appropriate option and enter any required inputs in the **Inputs** area, then click **Calculate**. For more detailed information about all the options available in the QCP, see Quick Calculation Pad (QCP).

The types of calculations available depend on the selected data type.

TIMES-TO-FAILURE DATA TYPES

Two values can be calculated for either the **MTBF** or **Failure Intensity**:

- The **Instantaneous** value is the MTBF/FI over a small interval *dt* that begins at a specified time. For example, an instantaneous MTBF of 5 hours after 100 hours of operation means that, over the next small interval *dt* that begins at 100 hours, the average time between failures will be 5 hours.
- The **Cumulative** value is the MTBF/FI from time = 0 up to a specified end time. For example, a cumulative MTBF of 5 hours from 0 to 100 hours means that the average time between failures was 5 hours over the 100-hour period.

The **Time Given** option allows you to calculate the mission duration given any of the following metrics:

- Cumulative MTBF
- Instantaneous MTBF
- Cumulative failure intensity (FI)
- Instantaneous failure intensity (FI)

Number of Failures is the cumulative number of failures that are expected to occur by a specified time, based on the fitted model.

DISCRETE DATA TYPES

For the Discrete-Sequential data sheet with the Crow-AMSAA (NHPP) or Duane models, the following calculations are available:

- Two values can be calculated for either the **Reliability** or **Probability of Failure**:
    - The **Cumulative** value is the reliability/prob. of failure from time = 0 up to a specified end time. For example, a reliability of 0.85 from 0 to 100 hours means that, on average, the reliability was 0.85 over the 100-hour period.
    - The **Instantaneous** value is the reliability/prob. of failure over a small interval *dt* that begins at a specified time. For example, an instantaneous reliability of 0.85 at 100 hours duration means that, over the next small interval *dt* that begins at 100 hours, the average reliability will be 0.85.
- The **Stage Given** option allows you to calculate at which stage a specific metric will be achieved. You can specify the following metrics:
    - Cumulative reliability
    - Cumulative probability of failure
    - Instantaneous reliability
    - Instantaneous probability of failure
    - **Number of Failures** is the cumulative number of failures that are expected to occur by a specified time, based on the fitted model.

For the Discrete-Mixed data sheet with the Crow-AMSAA or Crow Extended models, the following calculations are available:

- Two values can be calculated for either the **Reliability** or **Probability of Failure**:
    - The **Average** value is the reliability/prob. of failure from time = 0 up to a specified end time.
    - The **Instantaneous** value is the reliability/prob. of failure over a small interval *dt* that begins at a specified time.

- **Number of Failures** is the cumulative number of failures that are expected to occur by a specified time, based on the fitted model.

For all other discrete data sheets and models, the following calculations are available:

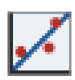
- **Reliability** is the probability of the system operating without failure for a period of time.
- **Probability of Failure** is also known as *unreliability*, and it is the inverse of the reliability.
- The **Stage Given** option allows you to calculate at which stage the reliability will be achieved.

RELIABILITY DATA TYPE

- **Reliability** is the probability of the system operating without failure for a period of time.
- **Probability of Failure** is also known as *unreliability*, and it is the inverse of the reliability.
- **Time Given Reliability** is the mission duration for a specified reliability value.

Plots

You can create plots by choosing **Growth Data > Analysis > Plot** or by clicking the icon on the control panel.



This section describes the types of plots you can create for traditional reliability growth analysis. The scaling, setup, exporting and confidence bounds settings are similar to the options available for all other reliability growth plot sheets. For more information on these common options, see Plots.

The types of plots you can create depend on the selected data type.

TIMES-TO-FAILURE DATA TYPES

- **Cumulative Number of Failures** shows how the number of failures is increasing over time. It plots the failure times on the x-axis and the cumulative number of failures on the y-axis. The points represent the actual failure times in the data set and the solution line represents the expected number of failures. The vertical line represents the test termination time.

- **MTBF vs. Time** shows how the time between consecutive failures increases, decreases or remains constant over time. It plots the cumulative MTBF curve and the corresponding instantaneous MTBF curve on the same plot. The points represent the actual failure times in the data set, while the vertical line represents the test termination time. The horizontal lines represent the instantaneous MTBF over the marked interval, which is obtained by dividing the length of the interval with the number of failures in that interval. You can specify the length of the intervals by right-clicking the plot and choosing **Show/Hide Items** on the shortcut menu. In the window, enter the desired interval length in the **Time Interval** field.

- **Failure Intensity vs. Time** shows how the rate of occurrence of failures increases, decreases or remains constant over time. It plots both the cumulative and instantaneous failure intensity curves on the same plot. The points represent the actual failure times in the data set, while the vertical line represents the failure time. The horizontal lines represent the instantaneous failure intensity over the marked interval, which is obtained by dividing the number of failures in that interval with the length of the interval. You can specify the length of the intervals by right-clicking the plot and choosing **Show/Hide Items** on the shortcut menu. In the window, enter the desired interval length in the **Time Interval** field.
- **System Operation** is available only for multiple systems analysis. It shows the failure times of each system in the data set, along with the timeline for their *equivalent system* that is used for calculating analysis results. See Times-to-Failure Data from Multiple Systems for more information on how the software combines the data to build the equivalent system.

DISCRETE DATA TYPES

The following plots are available when you use the Crow-AMSAA or Duane model with the Discrete-Sequential data sheet, or the Crow-AMSAA or Crow Extended model with the Discrete-Mixed data sheet:

- **Cumulative Number of Failures** shows how the number of failures is increasing over time. It plots the failure times on the x-axis and the cumulative number of failures on the y-axis. The points represent the actual failure times in the data set and the solution line represents the expected number of failures. The vertical line represents the test termination time.
- **Reliability vs. Time** shows how the reliability increases, decreases or remains constant over time. It plots both the cumulative and instantaneous reliability curves on the same plot.
- **Unreliability vs. Time** shows how the probability of failure increases, decreases or remains constant over time. It plots both the cumulative and instantaneous probability of failure curves on the same plot.

For all other models and/or discrete data sheets, the following plots are available:

- **Reliability vs. Time** shows the reliability values over time, capturing trends in the system's failure behavior.
- **Unreliability vs. Time** shows the probability of failure of the system over time.

RELIABILITY DATA TYPE

- **Reliability vs. Time** shows the reliability values over time, capturing trends in the system's failure behavior.
- **Unreliability vs. Time** shows the probability of failure of the system over time.

Tip: Weibull++ includes two additional plot utilities you can use across all types of data: the overlay plot, which allows you to compare different data sets or models; 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.

Example: Failure Times Data

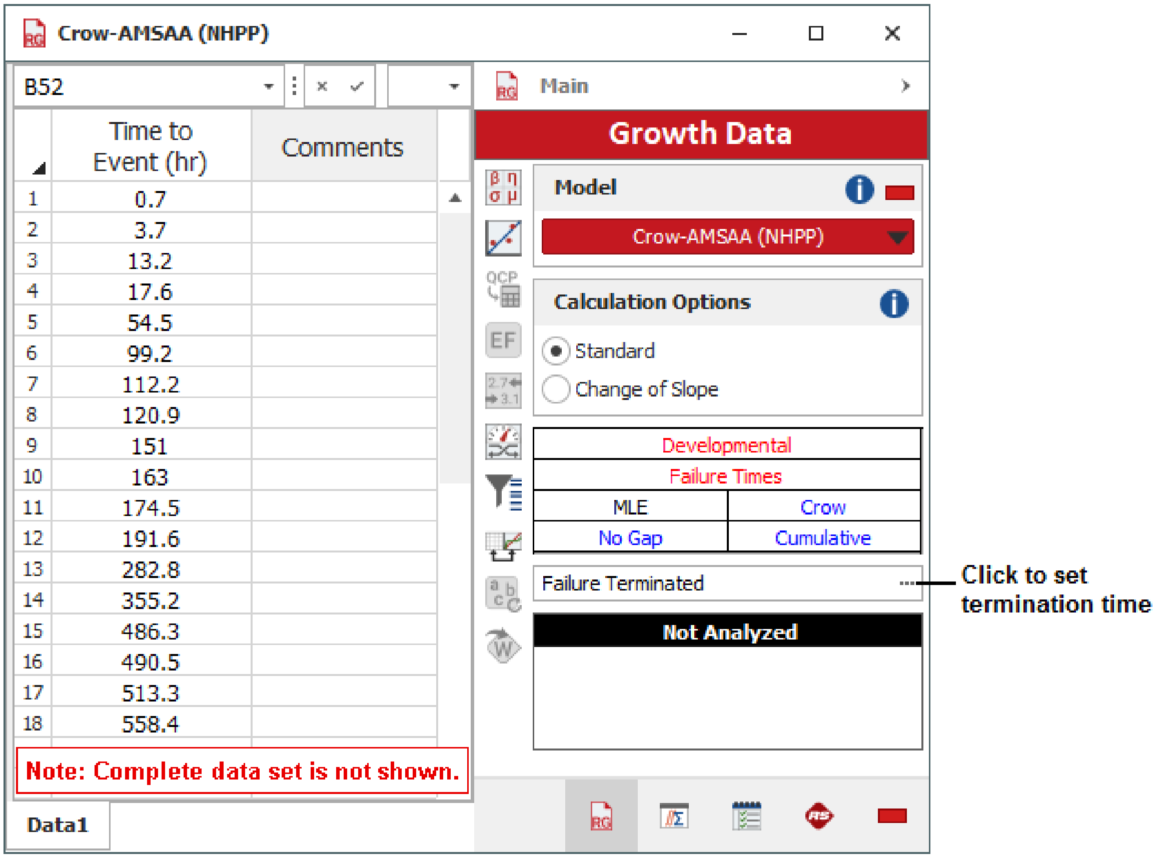
For reliability growth data analysis only.

*The data set used in this example is available in the example database installed with the software (called "Weibull21_ReliabilityGrowth_RGA_Examples.rsgz21"). To access this database file, choose **File > Help**, click **Open Examples Folder**, then browse for the file in the Weibull subfolder.*

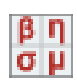
The name of the project is "Failure Times - Duane and Crow-AMSAA," and the folio that contains the data is called "Crow-AMSAA (NHPP)".

A prototype of a system is tested at the end of one of its design stages. When a failure occurs in the system, the time of failure is recorded and then a fix is applied before testing resumes. A total of 40 failures were observed during this period. The prototype has a design specification of an MTBF = 135 hours with a 90% confidence level. The goal is to determine whether the prototype meets the specified MTBF goal.

The data from the test are recorded in a Failure Times data sheet configured for cumulative failure times, as shown next. To specify the time when the test ended, click the (**...**) button on the control panel to open the Termination Time window. Select **Failure Terminated** and click **OK**.



On the control panel, choose the **Crow-AMSAA (NHPP)** model, and then analyze the data by choosing **Growth Data > Analysis > Calculate** or by clicking the icon on the Main page of the control panel.



The results show a demonstrated MTBF (DMTBF) = 166.2217 hours at the end of the test, as shown next.

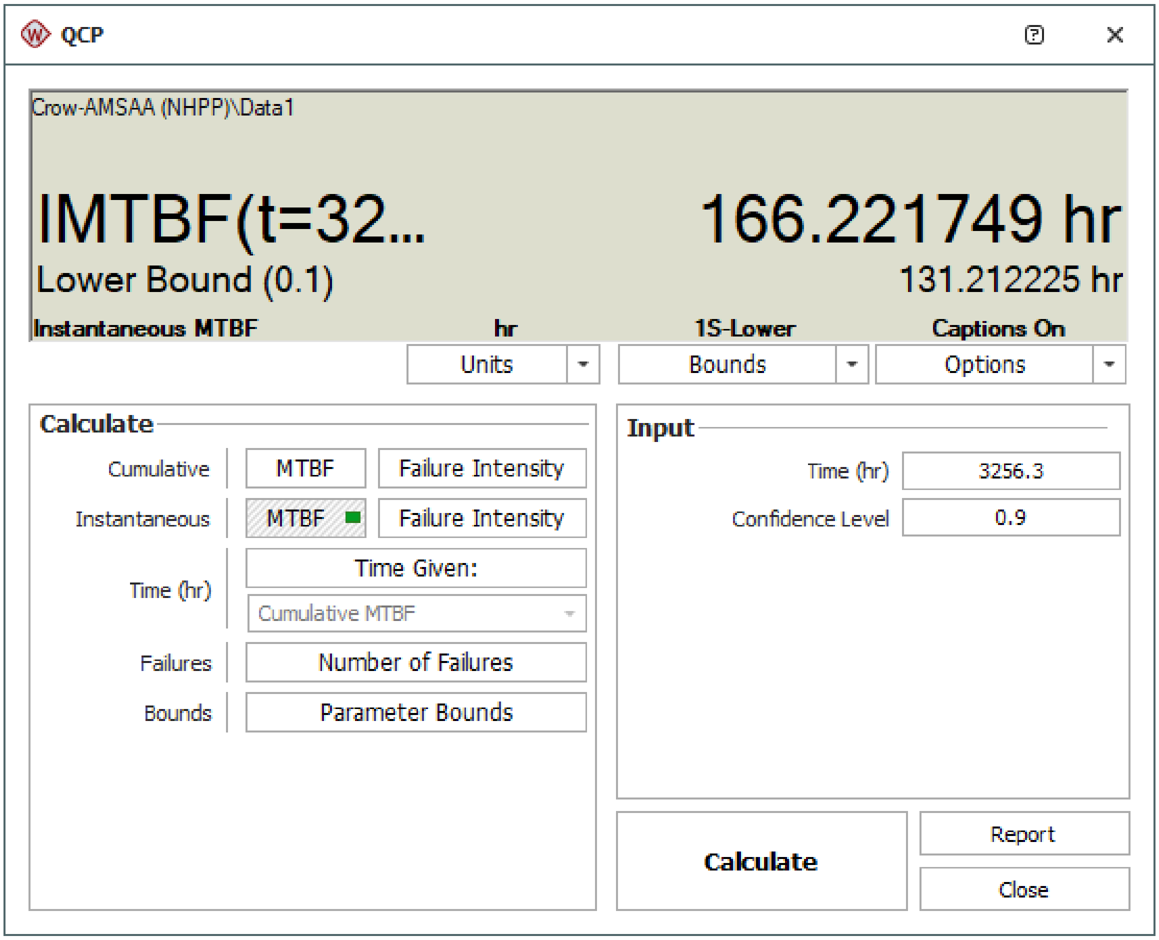
Results	
Parameters	
Beta	0.489752
Lambda (hr)	0.761544
Growth Rate	0.510248
DMTBF (hr)	166.221749
DFI	0.006016
Statistical Tests	
Significance Level	0.1
CVM	Passed
Other	
Termination Time (hr): 3256.300000	

You can use the Quick Calculation Pad (QCP) to obtain the 90% confidence bounds on the demonstrated MTBF. To access the QCP, choose **Growth Data > Analysis > Quick Calculation Pad** or click the icon on the Main page of the control panel.

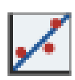
In the QCP, select to calculate the instantaneous **MTBF** with **lower one-sided** confidence bounds. Select **Hour** for the time units and then make the following inputs:

- Time = **3256.3**
- Confidence Level = **0.9**

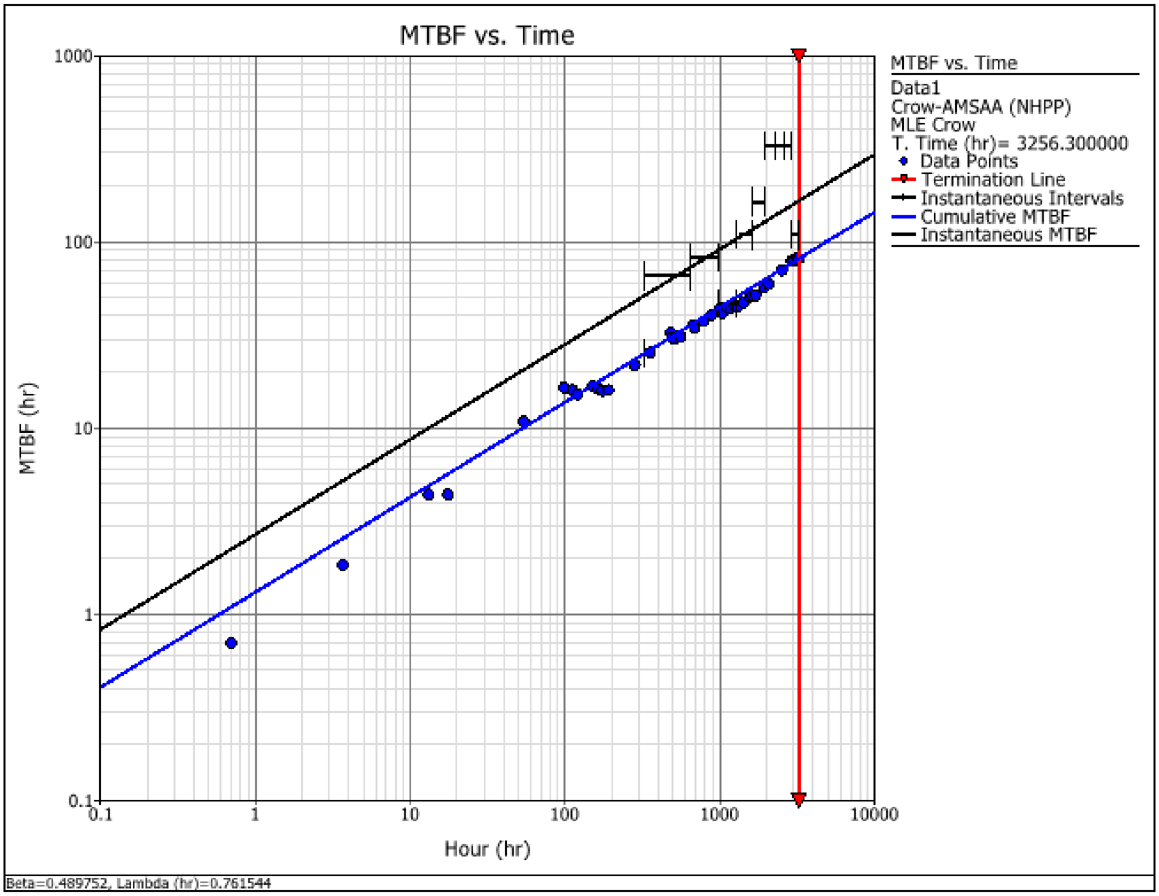
Click **Calculate** to obtain the results. It shows that the lower limit on the MTBF at the specified time with a 90% confidence level is 131.2122 hours. Therefore, the prototype has met the specified goal.



To create a plot of the result, close the QCP, and then choose **Growth Data > Analysis > Plot** or click the icon on the Main page of the control panel.



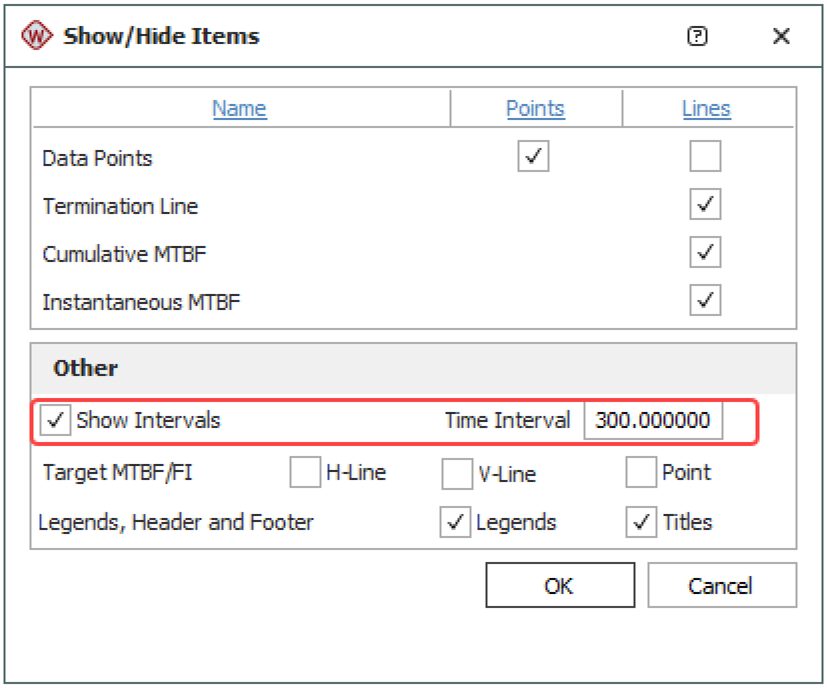
On the plot's control panel, click the **Plot Type** drop-down list and choose **MTBF vs. Time.** The following plot shows the cumulative MTBF and the corresponding instantaneous MTBF on a logarithmic plot. (Note that you can switch to a linear scale by clearing the **Use Logarithmic Axes** check box on the plot's control panel).



The points represent the actual failure times in the data set, while the vertical line represents the test termination time. The horizontal lines on the plot are the instantaneous intervals, which show the MTBF over a specified interval length. The MTBF over an interval is obtained by dividing the length of the interval with the number of failures in that interval. The intervals can be used to observe a general MTBF trend, whether the MTBF increases, decreases or stays the same over time.

You can specify the length of the intervals by choosing **Plot > Actions > Show/Hide Items** or by right-clicking the plot and choosing **Show/Hide Items** on the shortcut menu.

In the window, make sure that the **Show Intervals** check box is selected, and then specify the length of the interval in the **Time Interval** field, as shown next. Click **OK** and the plot will refresh to show the new results.



Example: Discrete (Success/Failure) Data

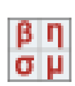
For reliability growth data analysis only.

*The data set used in this example is available in the example database installed with the software (called "Weibull21_ReliabilityGrowth_RGA_Examples.rsgz21"). To access this database file, choose **File > Help**, click **Open Examples Folder**, then browse for the file in the Weibull sub-folder.*

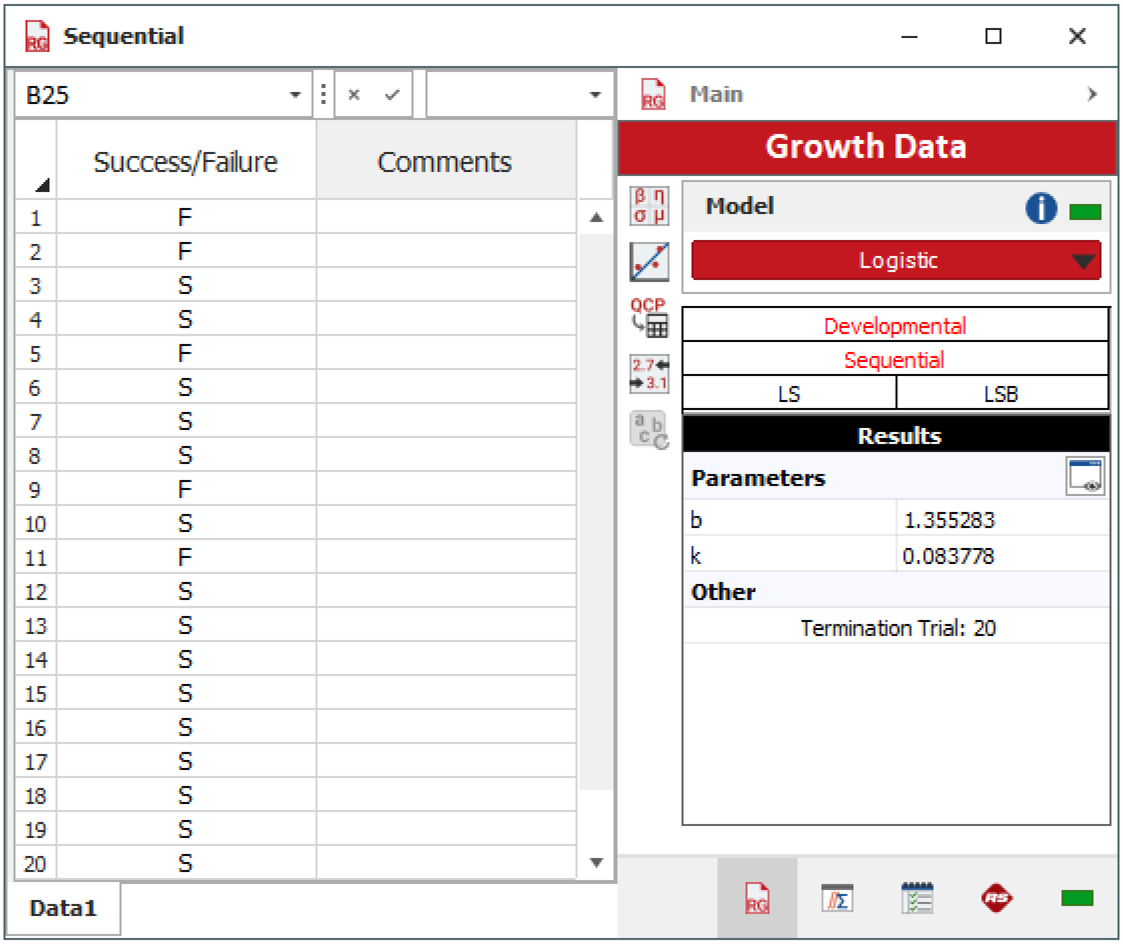
The name of the project is "Discrete Data- Sequential."

A cartridge device is required to have a reliability of 95% at the end of a reliability growth testing program. The device is designed for emergency use and its casing is destroyed when it releases its charge; therefore, the design can be tested and improved only through sequential pass/fail tests. The result of each trial (whether the device passed or failed) is recorded in a Discrete Sequential data sheet.

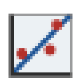
Choose the **Logistic** model, and then analyze the data set by choosing **Growth Data > Analysis > Calculate** or by clicking the icon on the Main page of the control panel.



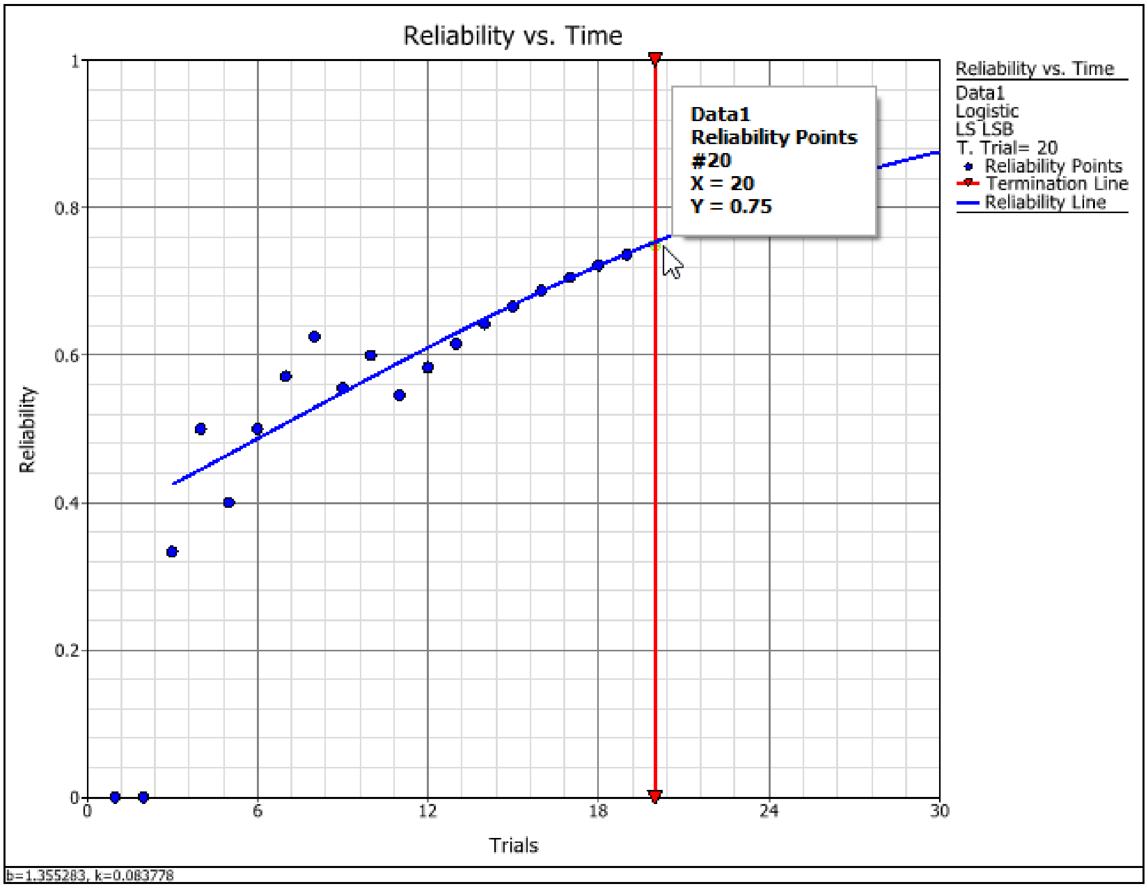
The following picture shows the data set and the calculated parameters of the logistic model.



To create a plot of the results, choose **Growth Data > Analysis > Plot** or click the icon on the Main page of the control panel.

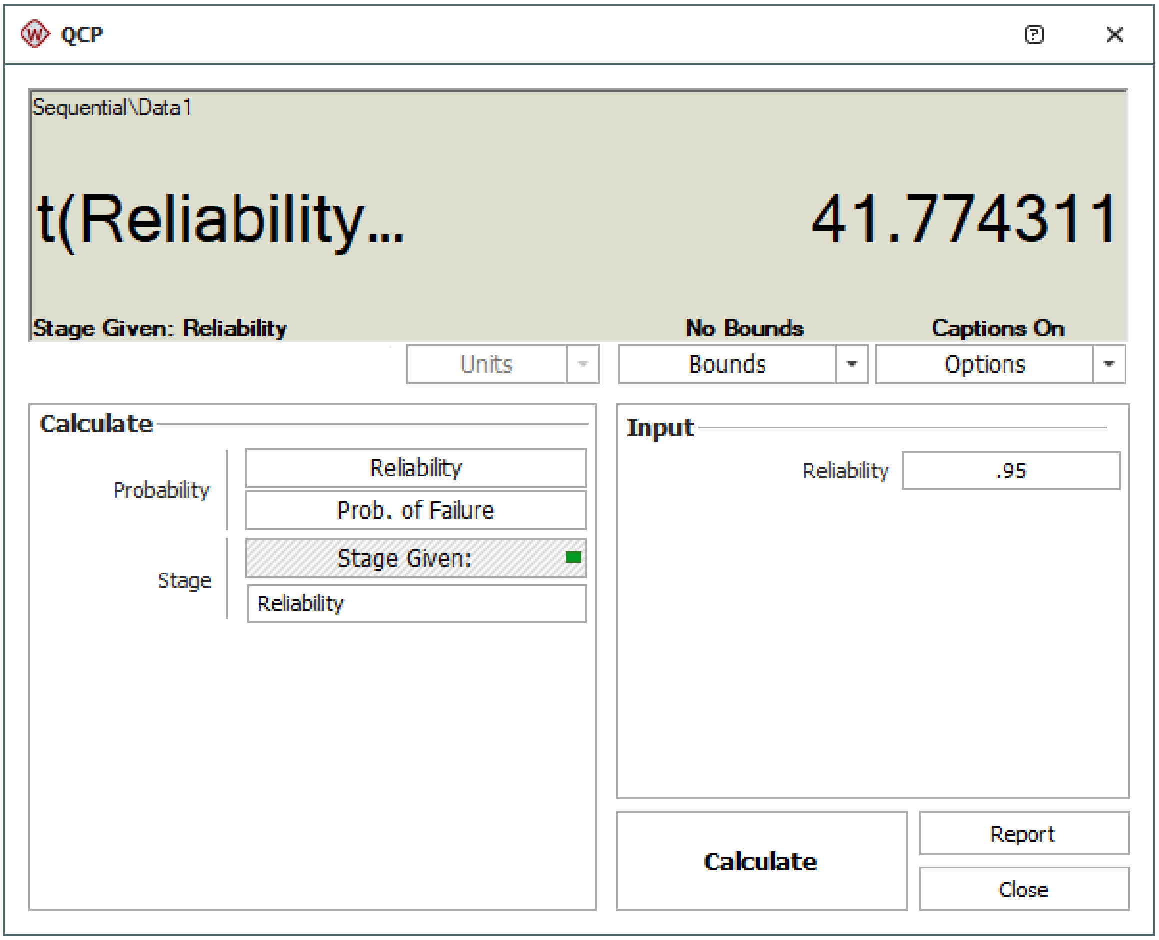


The following reliability vs. time plot shows how the reliability of the device changes with each new improvement in the design. The vertical line represents the stage when the test was terminated. This plot shows that the reliability after 20 trials is 75%.



You can use the Quick Calculation Pad (QCP) to predict when the reliability will reach 95%. To access the QCP, choose **Growth Data > Analysis > Quick Calculation Pad** or click the icon on the Main page of the control panel.

In the QCP, select to calculate the **Stage Given Reliability**, and then enter **0.95**. Select **Hour** for the time units and click **Calculate** to obtain the result. The result indicates that if the reliability keeps its trend, it is expected to reach 95% at the 42nd stage of testing, as shown next.



Change of Slope Analysis

For reliability growth data analysis only.

Change of Slope analysis can be applied to situations where a major change in the system design or operational environment causes a significant change in the failure intensity of the system. In this case, a single model may not provide a good fit for the data set.

To analyze this type of data set, the Change of Slope analysis splits the data set into two segments, based on the time that the change took place, and then applies a separate Crow-AMSAA (NHPP) model to each segment. The time that the change took place can be estimated from the data or based on your own knowledge about the specific change that was applied to the system. Note that although two separate models are applied to the segments, the information collected in the first segment are considered when creating the model for the second segment.

This analysis is available when you use the Crow-AMSAA (NHPP) model with any times-to-failure data type (except Multiple Systems with Event Codes) or with Discrete – Mixed data.

Note: The Change of Slope analysis requires at least three failures up to the break point and two failures after the break point (in the case of grouped data, this would be the number of intervals before and after the break point).

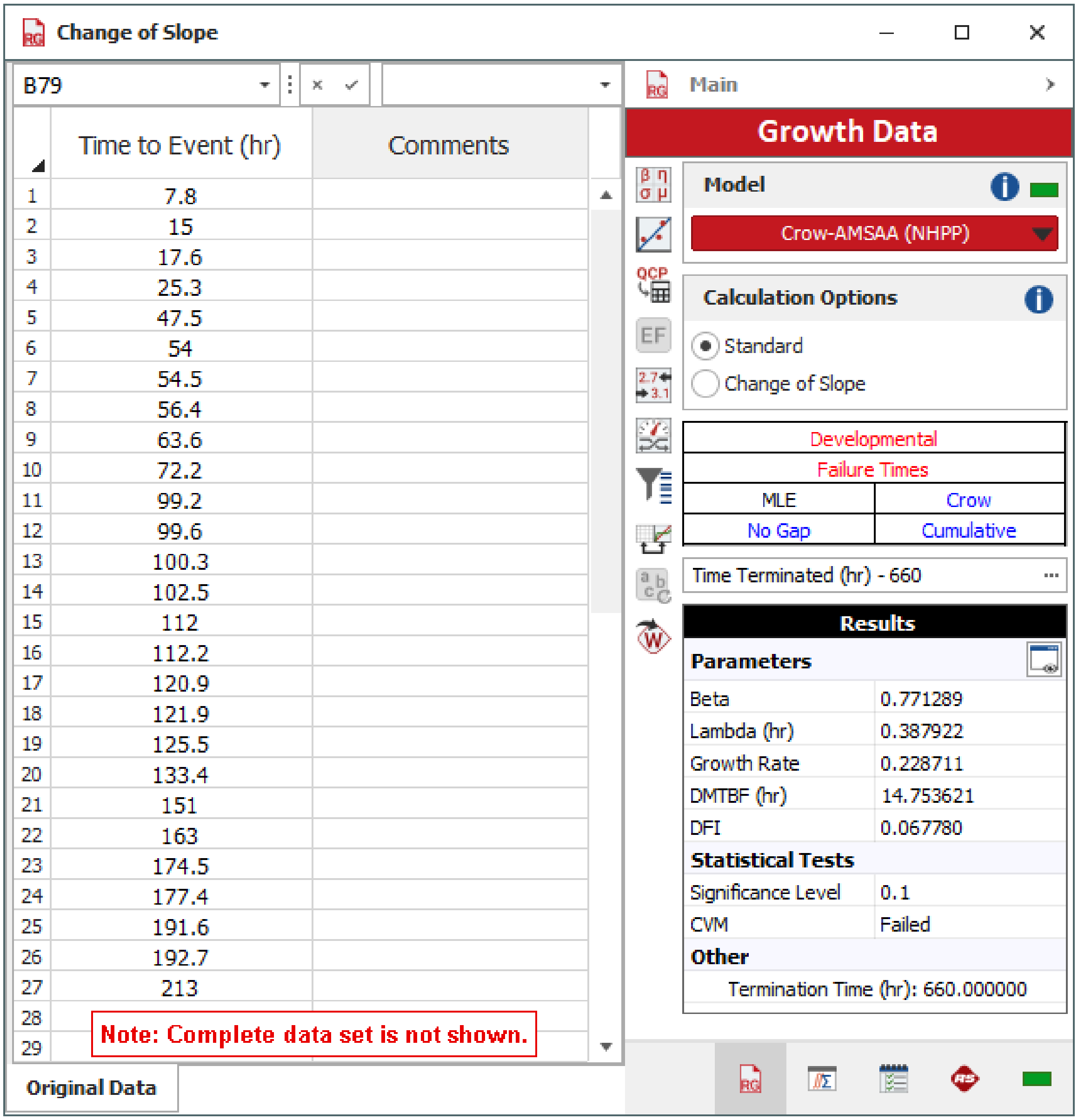
The ReliaWiki resource portal provides more information about the analysis at http://www.reliawiki.org/index.php/Change_of_Slope_Analysis.

Example

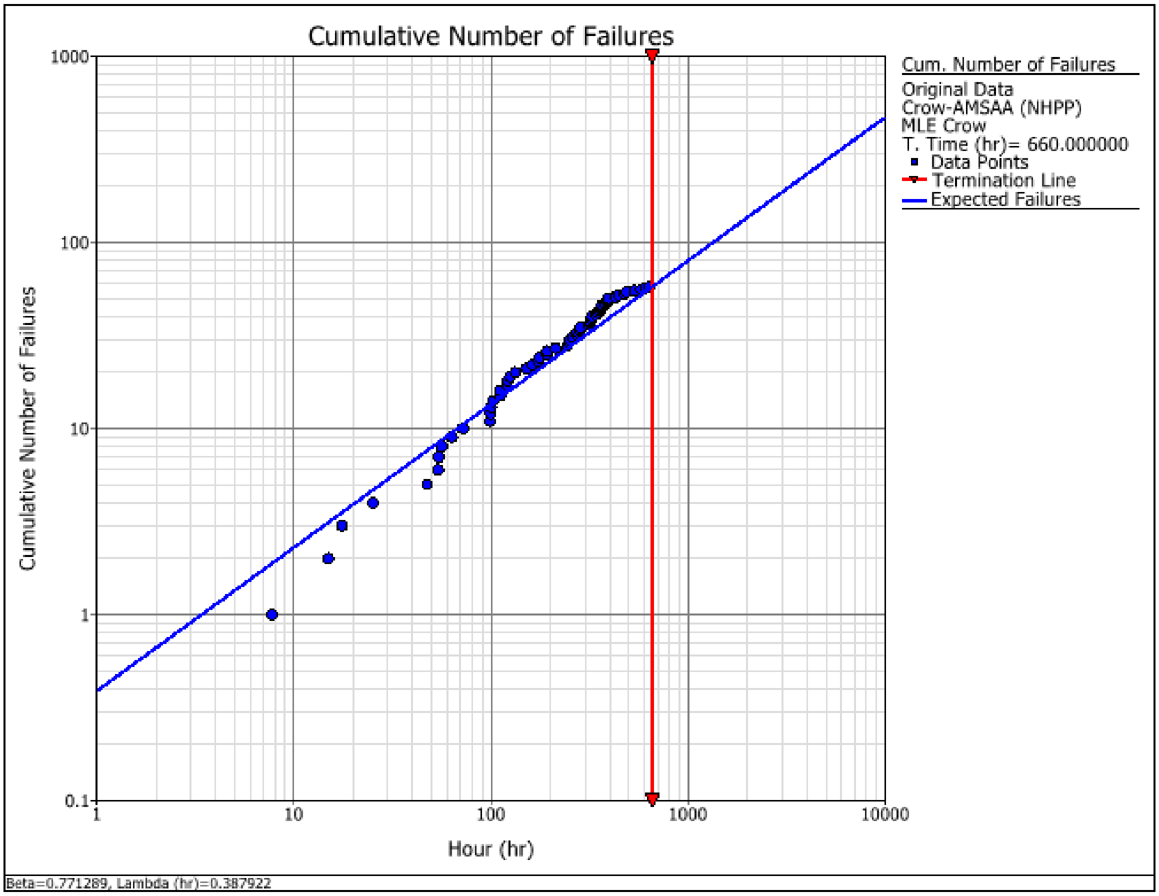
*The data set used in this example is available in the example database installed with the software (called "Weibull21_ReliabilityGrowth_RGA_Examples.rsgz21"). To access this database file, choose **File > Help**, click **Open Examples Folder**, then browse for the file in the Weibull subfolder.*

The name of the project is "Failure Times - Change of Slope."

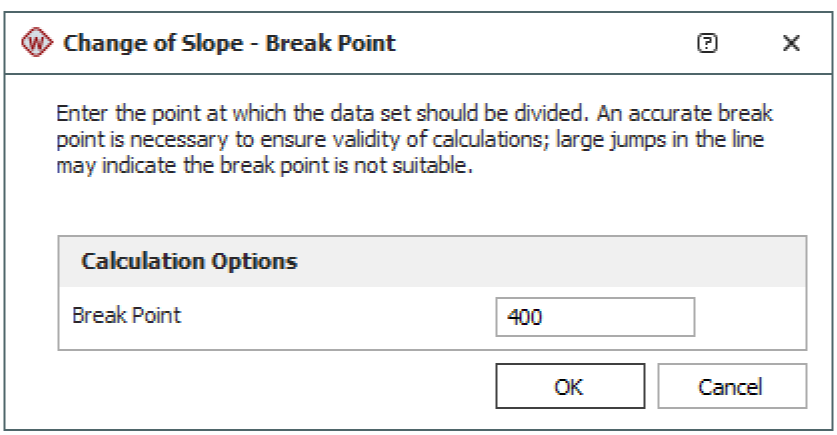
A system is put through a reliability growth testing program. A total of 58 failures are observed during the 660 hours of testing. The data are recorded in a Failure Times data sheet and analyzed with the Crow-AMSAA (NHPP) model, as shown next.



The results show that the data set failed the Cramér-von-Mises (CVM) goodness-of-fit test. This indicates that the model is not a good fit for the data. In addition, the following plot shows that there appears to be a significant change in the slope of the points at around 400 hours. As it turns out, a major design change was implemented at this point in time. Given this scenario, we can repeat the analysis using the Change of Slope methodology with the break point set to 400.



Return to the data sheet. In the **Calculations Options** area of the control panel, select the **Change of Slope** option, and then click the **Calculate** icon. This opens the Change of Slope - Break Point window, which allows you to specify the point at which the data set should be divided. Enter a value of **400**, as shown next.

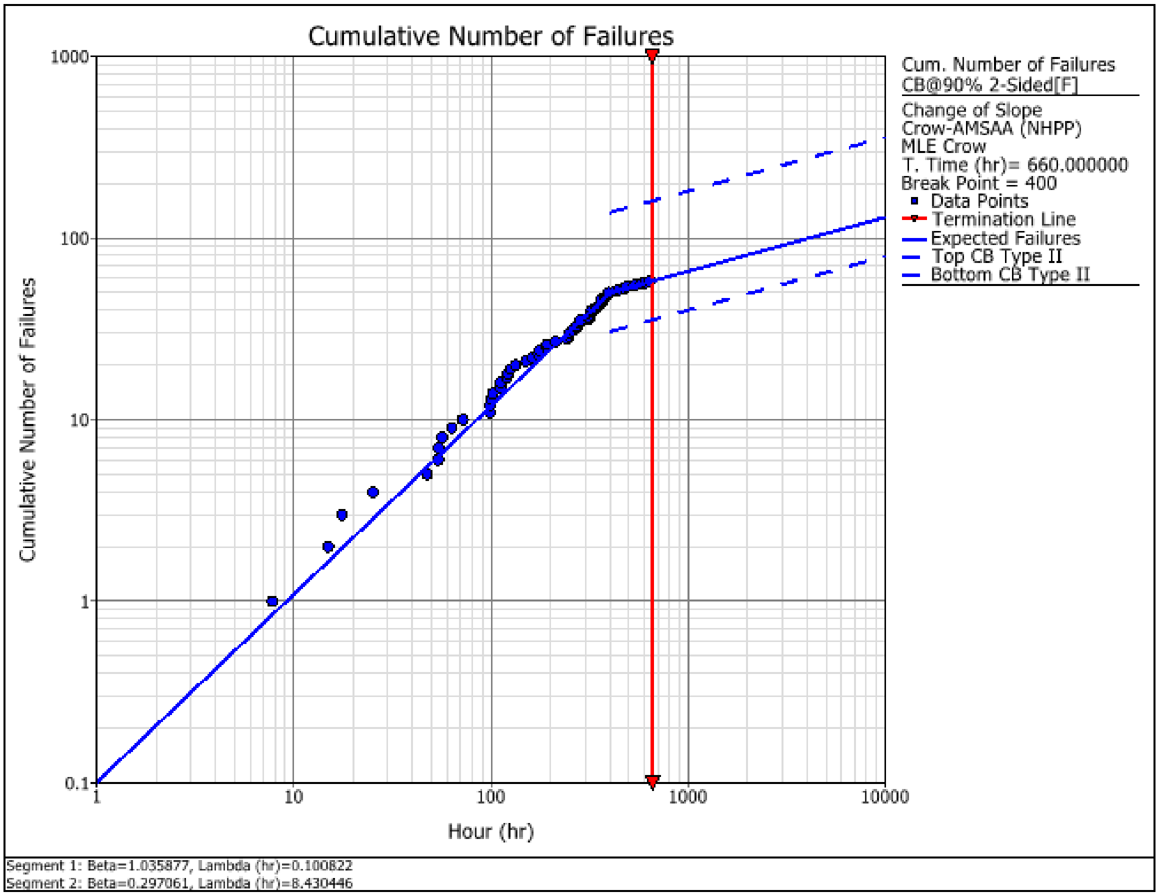


Weibull++ automatically splits the data into two segments and fits a separate Crow-AMSAA (NHPP) model to each segment. For the first segment, the data points up to 400 hours are used to calculate its parameters. For the second segment, the data points after 400 hours plus information from the first segment are used to calculate its parameters.

The following results show the estimated parameters for Segment 2. Based on this model, the demonstrated MTBF (**DMTBF**) of the system at the end of the test is 38.3063 hours. (The Segment 1 model can be used to calculate the MTBF for times up to 400 hours only.) The results also show that the data set has passed the chi-squared goodness of fit test.

Results	
Parameters	
Segment 1	
Beta	1.035877
Lambda (hr)	0.100822
Growth Rate	-0.035877
Segment 2	
Beta	0.297061
Lambda (hr)	8.430446
Growth Rate	0.702939
DMTBF (hr)	38.306267
DFI	0.026105
Significance Level	0.1
Chi-Sq	Passed
Break Point: 400	
Termination Time (hr): 660.000000	

The following plot shows that the model provides a much better fit to the data. If you choose to display the confidence bounds on the plot, only the bounds on Segment 2 (which is the model of interest) will be plotted, as shown next.



Gap Analysis

For reliability growth data analysis only.

Gap analysis is used in situations where issues such as oversight, biases, human error, technical difficulties, etc. cause some portion of the data to be erroneous or completely missing. This causes the analysis to return distorted estimates of the growth rate and actual system reliability. In this type of situation, you can use the Gap Analysis feature in Weibull++ to analyze the data set.

Gap analysis assumes that the information within the problematic time interval is unavailable or unreliable and does not use any entries that have been made for that time period. Instead, the number of failures for that interval is assumed to be unknown, but the rest of the data follow the Crow-AMSAA model. The analysis retains the contribution of the interval to the total test time, but no assumptions are made regarding the actual number of failures over the interval.

This analysis is available when you use the Failure Times data type with the Crow-AMSAA (NHPP) model using standard calculations (gap intervals are not compatible with Change of Slope calculations). You can define one gap interval per data set. Each gap is a time range that exists within the data, and it has a defined start time and end time. There must be at least one failure before and after the specified gap interval.

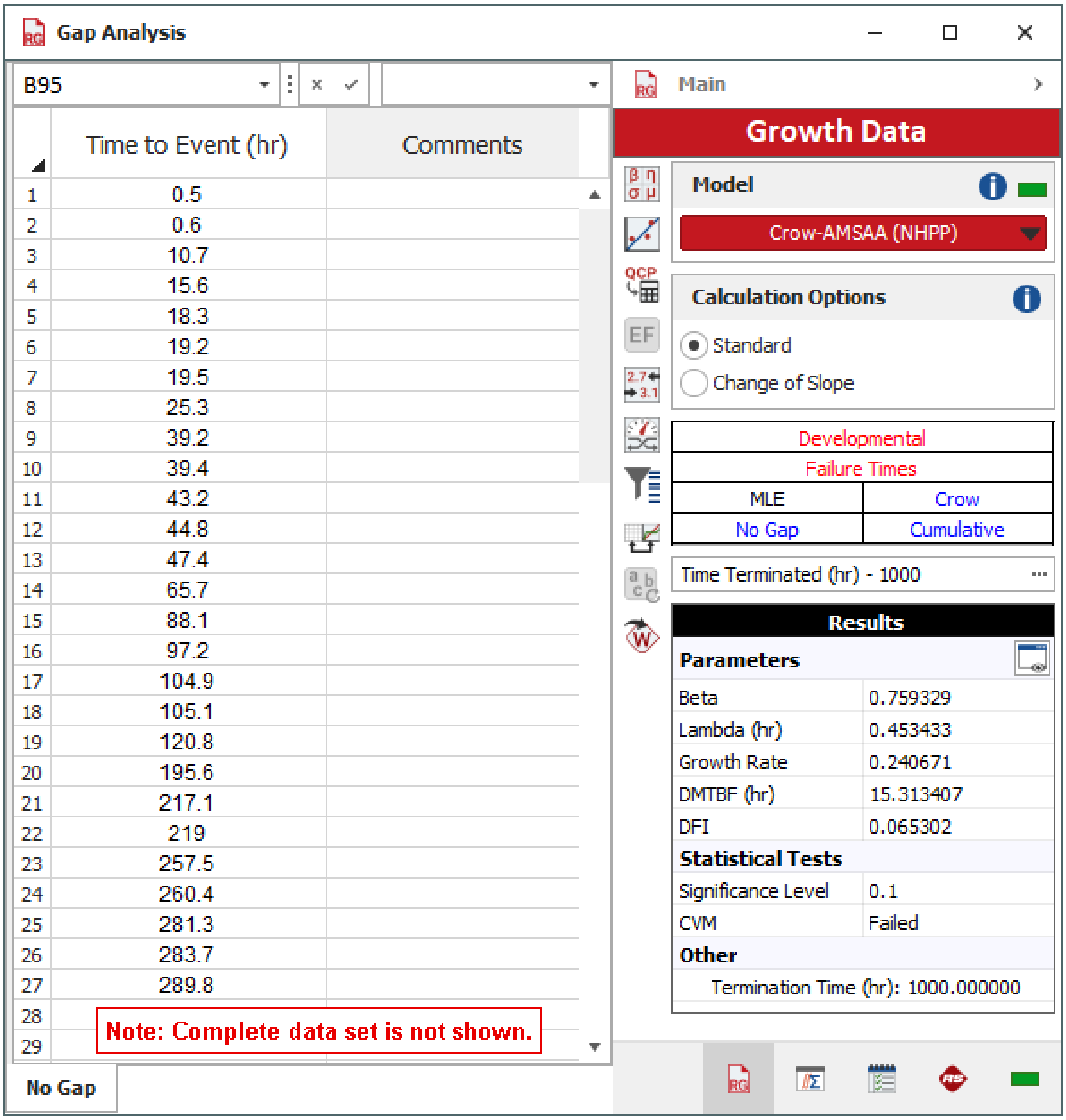
The ReliaWiki resource portal provides more information about the analysis at http://www.reliawiki.org/index.php/Gap_Analysis.

Example

The data set used in this example is available in the example database installed with the software (called "Weibull21_ReliabilityGrowth_RGA_Examples.rsgz21"). To access this database file, choose ***File > Help****, click* ***Open Examples Folder****, then browse for the file in the Weibull subfolder.*

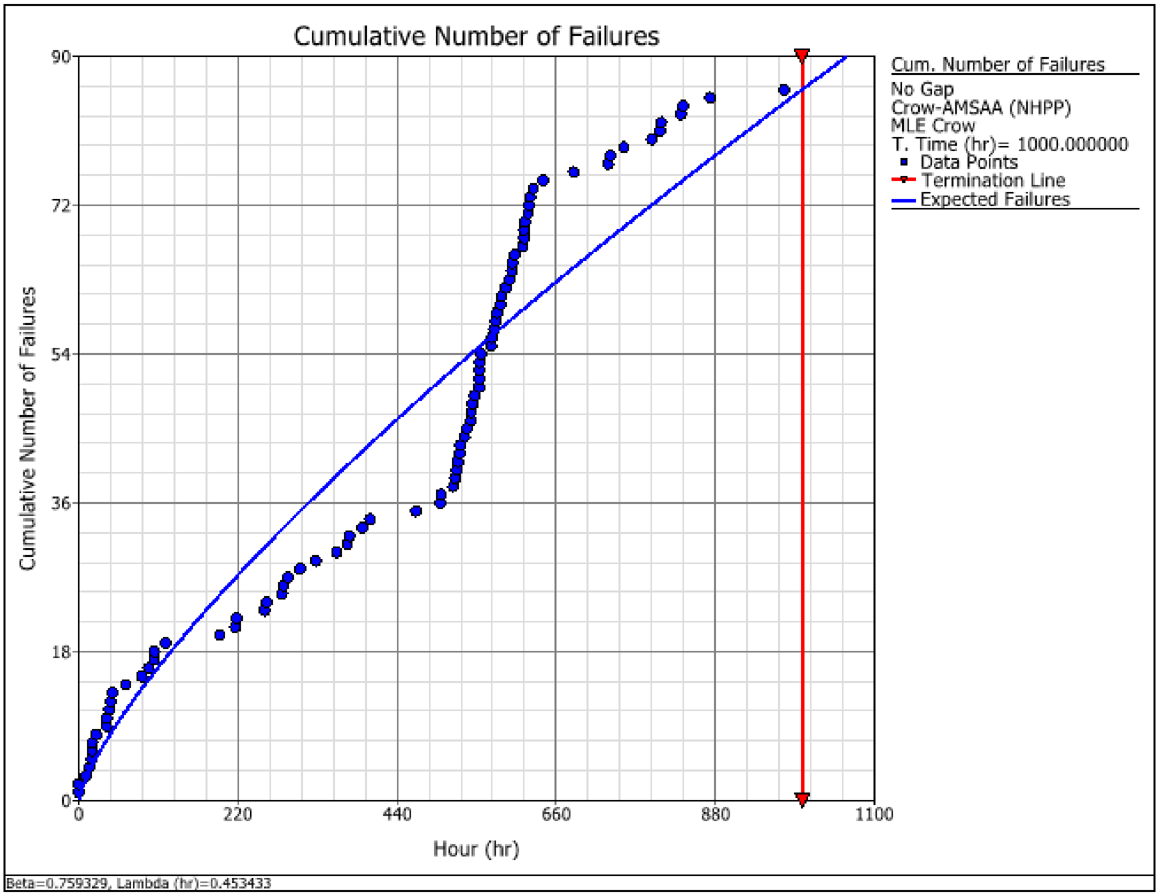
The name of the project is "Failure Times - Gap Analysis."

A system is put through a reliability growth testing program. The test is terminated after 1,000 hours of testing, and a total of 86 failures were observed during that period. The data are recorded in a Failure Times data sheet and analyzed with the Crow-AMSAA (NHPP) model, as shown next.

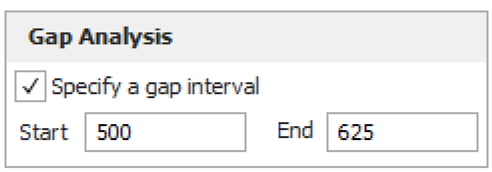


The results show that the data set failed the Cramér-von-Mises (CVM) goodness-of-fit test. This indicates that the model is not a good fit for the data.

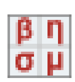
The following plot shows that the number of failures from 500 to 625 hours is abnormally high. A quick investigation found that new data collectors were assigned to the project at around that time and that extensive design changes involving the removal of several parts were also made within that period. It is possible that the parts that were removed were incorrectly reported as failed parts. Based on their knowledge of the system and the test program, the analysts agree that such a large number of failures were extremely unlikely. They decide to repeat the analysis using gap analysis.



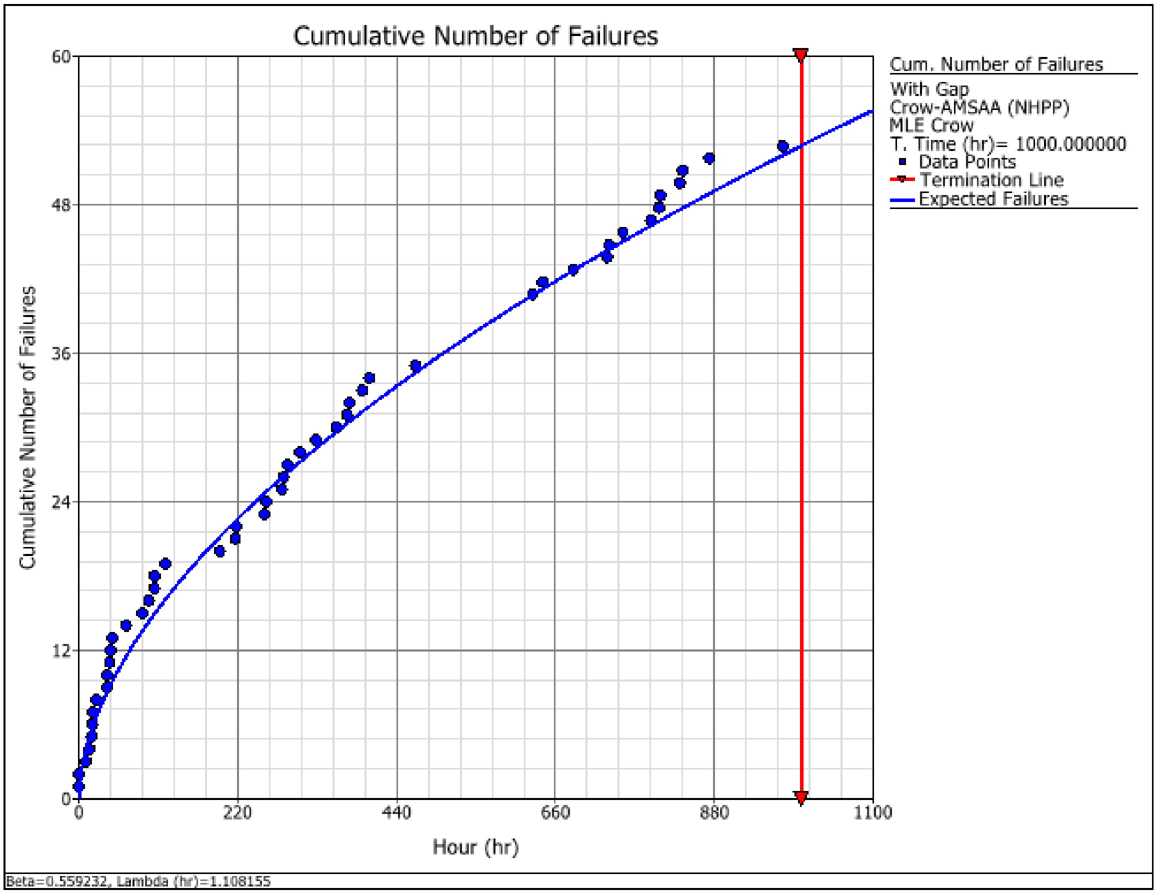
Return to the data sheet. On the Analysis page of the control panel, select the **Specify a gap interval** check box and then enter **500** to **625** for the gap interval, as shown next.



Return to the Main page of the control panel and analyze the data set by choosing **Growth Data > Analysis > Calculate** or by clicking the icon on the Main page of the control panel.



This time, the analysis will ignore the failure times observed between 500 and 625 hours. The resulting plot now shows that the model provides a much better fit to the data.



Failure Discounting

For reliability growth data analysis only.

Failure discounting applies to the analysis of discrete data (also known as *success/failure* or *attribute* data), in which the data set records the success or failure of each one-shot trial (e.g., of a missile or other type of device that either succeeds or fails). In Weibull++, you can perform failure discounting by using the Discrete Sequential with Mode data type.

This analysis is used to track the possible recurrence of a failure mode after a fix has been applied. If the mode does not appear again in the later trials, then the probability of failure due to that mode is reduced. The amount of reduction is related to the consecutive number of successful tests after the fix was applied. The following equation describes this relationship:

$$f = 1 - (1 - CL)^{\frac{1}{S_n}}$$

where *f* is the *failure value* due to the mode, *CL* is the confidence level and *Sn* is the number of additional successful tests after the first success following the fix.

For example:

- In trial#1, failure mode A occurred. The failure value due to this mode is 1 (i.e., counted as one failure). A permanent fix is then applied for mode A.
- In trial#2, mode A did not recur. This marks the first success after the fix was applied. The failure value due to this mode is still counted as 1.
- In trial#3, mode A did not recur; therefore, $Sn = 1$, and the failure value due to mode A is reduced to 0.9, based on a 90% confidence level.
- In trial#4, mode A did not recur; therefore, $Sn = 2$, and the failure value due to mode A is further reduced to 0.684, based on a 90% confidence level.
- In trial#5, mode A occurs again, and because a second fix now needs to be applied, the failure value due to this mode resets back to 1.

Failure discounting has the effect of reducing the total number of failures in the analysis, which results in a higher reliability estimate. The following example illustrates this effect.

Example

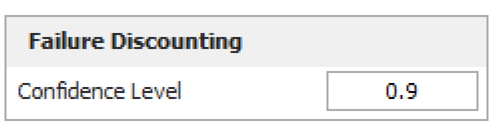
*The data set used in this example is available in the example database installed with the software (called "Weibull21_ReliabilityGrowth_RGA_Examples.rsgz21"). To access this database file, choose **File > Help**, click **Open Examples Folder**, then browse for the file in the Weibull subfolder.*

The name of the project is "Discrete Data - Sequential with Mode."

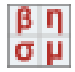
The ReliaWiki resource portal provides the mathematical calculations behind this example at http://www.reliawiki.org/index.php/Failure_Discounting.

A device under development undergoes a reliability growth program. A total of 22 trials are performed and the result of each trial is recorded in the Discrete Sequential with Mode data sheet. Whenever a failure occurs, a fix is applied to the device and the failure mode that caused the failure is recorded.

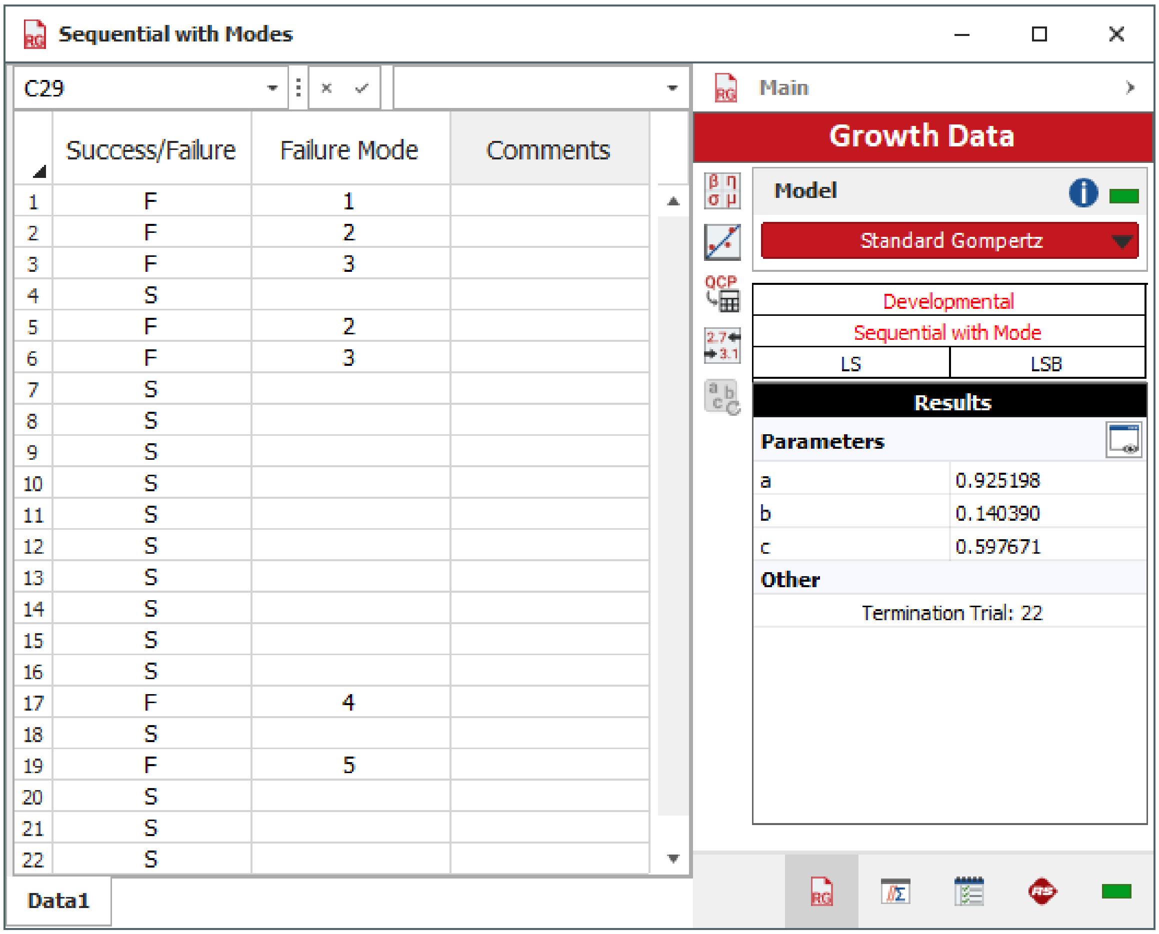
On the Analysis page of the control panel, enter **0.9** for the confidence level in the **Failure Discounting** area, as shown next.



Select the **Standard Gompertz** model and then analyze the data set by choosing **Growth Data > Analysis > Calculate** or by clicking the icon on the Main page of the control panel.

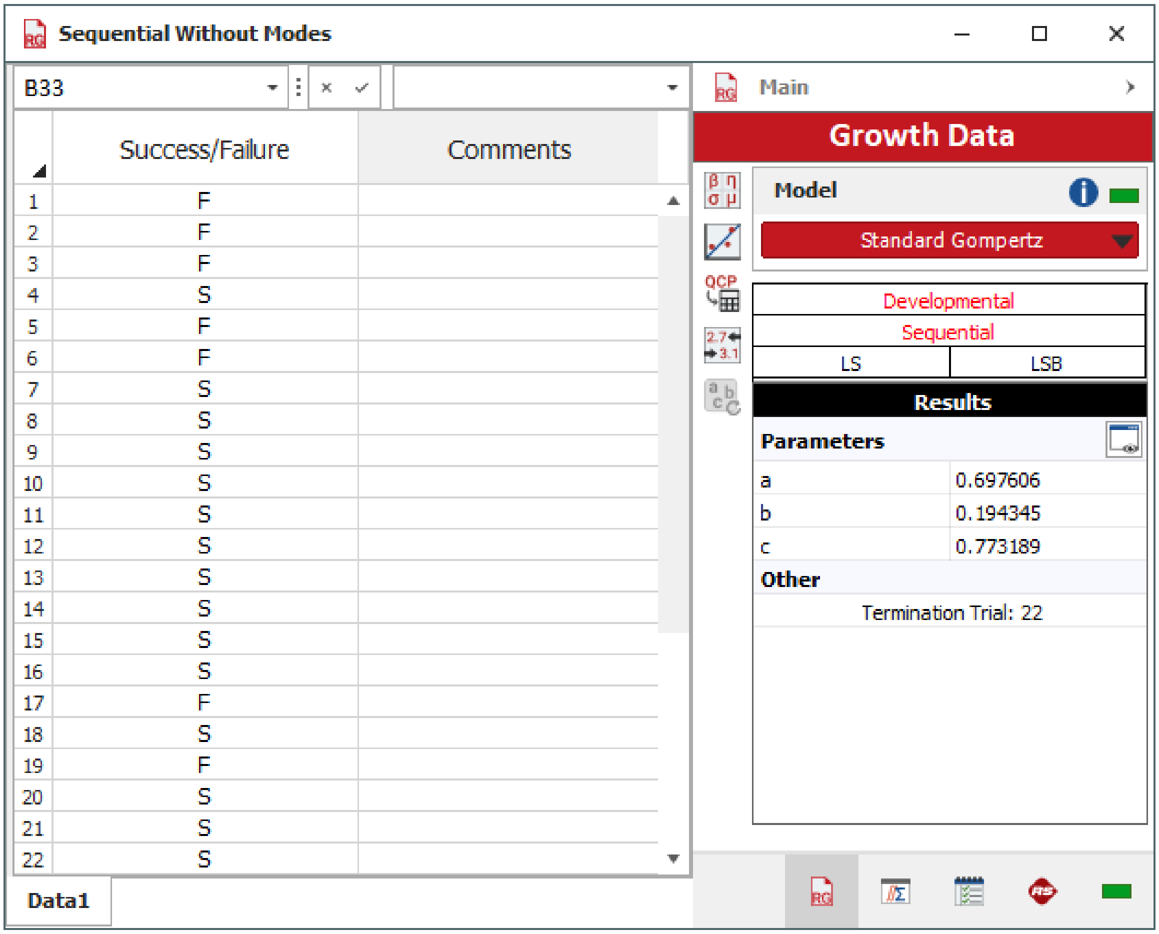


The following picture shows the data set and the parameters of the model.

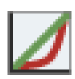


To compare how failure discounting affects the reliability estimates, copy the Success/Failure data and paste them into a Discrete Sequential data sheet. Analyze that data sheet using the **Standard Gompertz** model.

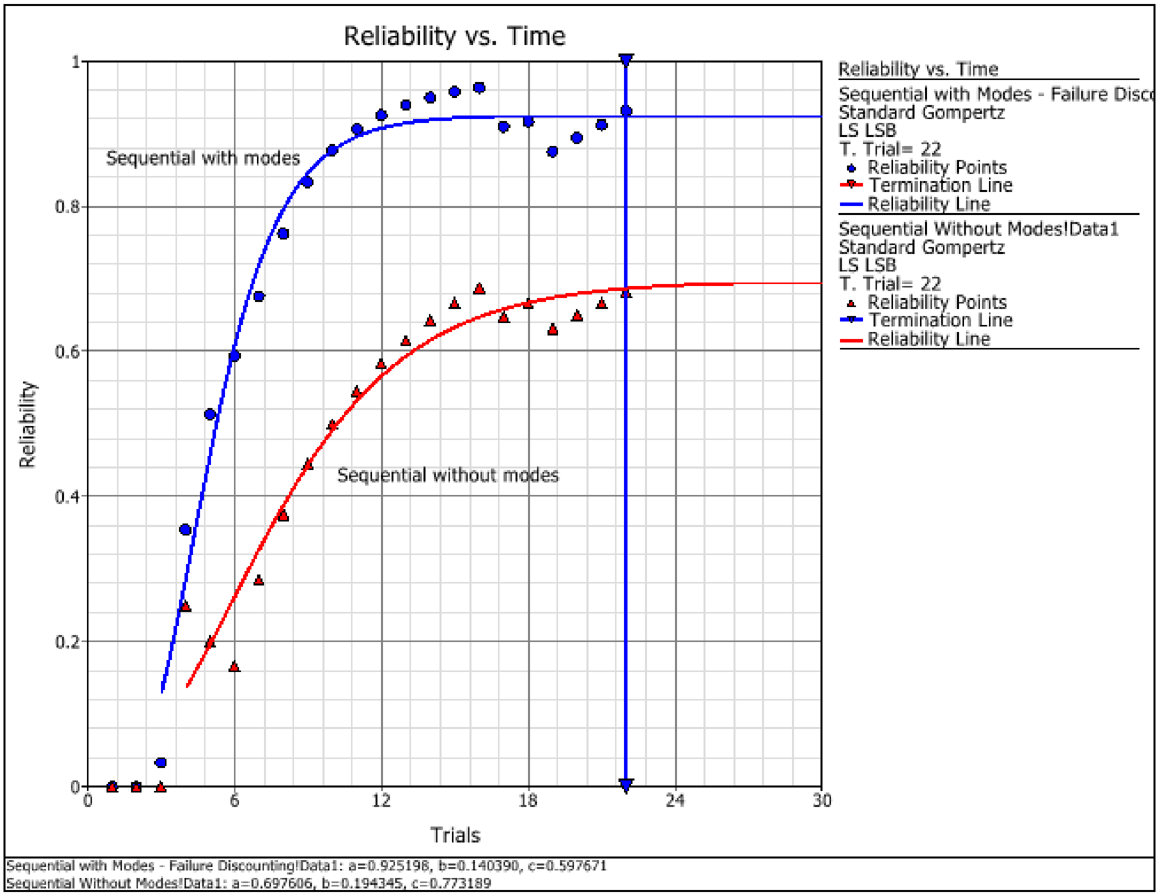
The following picture shows the copied data set and the parameters of the Standard Gompertz model, this time, without failure discounting.



To create an overlay plot that compares the results of the two analyses, choose **Home > Insert > Overlay Plot**.



When prompted to select which data sheets to plot, select the two data sheets you have been working with. The following picture shows the resulting overlay plot (with annotations added via RS Draw to make the plot easier to interpret). It shows that the reliability estimates are higher when failure discounting is used because of its effect of reducing the total number of failures in the analysis.



Analysis with the Crow Extended Model

For reliability growth data analysis only.

Traditional reliability growth analysis is used to analyze data from tests where design fixes are incorporated during the observation period (the *test-fix-test* strategy). However, in actual practice, fixes may be delayed until after the end of the observation period (*test-find-test*), or you may implement some fixes during the observation period while delaying others (*test-fix-find-test*).

With the Crow Extended model, you can perform reliability growth projections, planning and analysis—which allows you to analyze test data from any or all of these strategies by providing additional information about the failure modes and the reliability growth management strategy (i.e., which modes are fixed and how effectively the design improvements reduce failure intensity). You can also use this model for data from fielded repairable systems in order to evaluate the improvement (i.e., the jump in MTBF) that could be achieved by rolling out a set of fixes for all systems operating in the field.

Data Types for Crow Extended Model

For reliability growth data analysis only.

You can apply the Crow Extended model to any of the following data types.

- All times-to-failure data types
- Discrete Mixed data
- All fielded data types

When the Crow Extended model is selected in any of these data sheets, additional Classification and Mode columns will appear in the data sheet. These columns are used to classify and identify the failure mode responsible for each observed failure. To learn how to use these two columns in an analysis, see Failure Mode Classifications. For the background theory behind the Crow-Extended model, the ReliaWiki resource portal provides more information at: http://www.reliawiki.org/index.php/Crow_Extended.

Failure Mode Classifications for Crow Extended

For reliability growth data analysis only.

To use the Crow Extended model, you must classify and identify the failure mode responsible for each observed failure. This information is entered in the Classification and Mode columns in the data sheet. These columns are inserted automatically when you choose the **Crow Extended** model from the **Model** drop-down list on the growth data folio control panel.

Tip: If the Crow Extended model is not selected but still available for the current data type, you can manually add the Classifications and Mode columns to the data sheet by choosing **Growth Data > Format and View > Insert Columns > Projections**. This will allow you to record projections data in case you decide to change the model to Crow Extended later. When Crow Extended is not selected, the projections data will not affect the calculations.

The **Classification** column records the code that describes how the failure mode is treated. The possible codes are described next.

- **A** indicates that no fix will be applied (i.e., management chooses not to address the failure mode because of technical, financial or other reasons).

- **BC** indicates that a fix was applied during the observation period. The analysis assumes that the effect of the design improvement was experienced during the observation period and will be reflected in the data.
- **BD** indicates that the fix will be delayed until after the observation period. You will be required to define the effectiveness factor for each BD mode to estimate the fractional decrease in failure intensity that is expected to occur after the fix has been implemented.

Tip: BC modes are not available for data sheets configured for fielded data or Discrete Mixed data because these data types assume that all design improvements are delayed (i.e., no reliability growth during the observation period). If you are using Discrete Mixed data (i.e., success/failure data from a series of trials on one-shot devices) and you want to account for fixes that were implemented during the test, you can use a data sheet configured for Multi-Phase Mixed data instead. In this sheet, you can enter a BD failure event and its implemented fix event (i.e., "I" event). Such a failure will, in effect, be treated as a BC mode.

The **Mode** column records the specific failure mode identification, which is optional for A modes but required for all BC and BD modes. This can be entered as either a name or a numerical code (e.g. "seal leak," "1," "BD1," etc.).

The next picture shows the first ten rows of a Failure Times data sheet with failure mode identification and mode classification information.

	Time to Event (hr)	Classification	Mode
1	15	BD	seal leak
2	25.3	BD	valve
3	56.4	BD	hose
4	63.6	A	
5	72.2	BD	hose
6	99.6	BD	operator error
7	100.3	BD	bearing
8	102.5	A	
9	112	BD	seal leak
10	112.2	A	

Tip: If you discover that a certain failure mode was classified incorrectly (e.g., if every "operator error" mode in the above data sheet should have been classified as an A classification instead of BD), or if you want to change a particular mode identifier (e.g., you want to change every instance of a particular number in the Mode column to a name), you can use the Convert Modes window to apply the update automatically.

Convert Modes Window

For reliability growth data analysis only.

The Convert Modes window allows you to convert any failure mode classification and mode in the current data set to another classification and mode. For example, suppose a particular failure mode occurs several times during the observation period and is currently classified as BD in the data sheet. If it is later decided to ignore this mode rather than addressing it after the observation period, you could convert every instance of it in the data sheet to an A classification.

To access this window, choose **Growth Data > Crow Extended > Convert Modes**.

Two drop-down lists appear next to **Failure Mode**. Choose the failure mode classification in the first list and the failure mode identifier in the second. Then, in the **Convert To** drop-down list, choose the classification that you want the selected failure mode to be changed to, and if desired, enter a new identifier in the input field.

Click **Convert** to apply the specified changes. The Classification and Mode columns in the data sheet will be updated accordingly, and a message will appear specifying the number of changes that were made. When you are finished converting failure modes, click the **Close** button.

Mode Processing Window

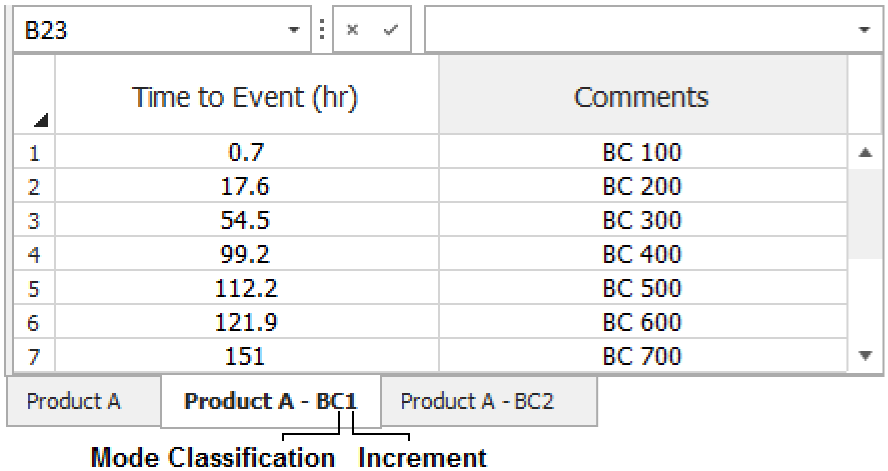
For reliability growth data analysis only.

The Mode Processing utility allows you to extract the first failure time for each unique BC mode and/or unique BD mode in a data set. This allows you to automatically copy the failure times for a particular failure mode classification into a new data sheet for separate analysis. This tool applies only to data sets analyzed with the Crow Extended or Crow Extended - Continuous Evaluation models.

To use the tool, you must first analyze the data sheet (by clicking the **Calculate** icon on the control panel), and then choose **Growth Data > Crow Extended > Mode Processing**.

In the window that appears, select the check box next to each mode classification you wish to extract. If the data set does not contain a particular mode classification, the corresponding check box will be unavailable.

For example, if you select to extract BC modes, the failure times for all BC modes will appear in a new data sheet. (If a particular failure mode (e.g., BC 100) has more than one failure time, only the first time will be extracted.) The new data sheet will have the same name as the source data sheet plus the name of the mode classification and an increment to reflect the number of data sheets that have been created for the same classification.



	Time to Event (hr)	Comments
1	0.7	BC 100
2	17.6	BC 200
3	54.5	BC 300
4	99.2	BC 400
5	112.2	BC 500
6	121.9	BC 600
7	151	BC 700

You can then analyze the extracted data set using traditional reliability growth analysis techniques.

Tip: If you wish to extract subsets of data based on either the system ID recorded in multiple systems data sheets or the phases/analysis points recorded in multi-phase data sheets, use the Batch Auto Run utility.

Effectiveness Factors for Crow Extended

For reliability growth data analysis only.

If you are using the Crow Extended model and have classified some failure modes as BD modes (i.e., the modes will not be addressed until after the current observation period), you will need to specify the expected effectiveness of the delayed fixes. This information is used to estimate the fractional decrease in failure intensity that will occur after the design fixes are implemented. For example, you could indicate that 60% of a BD mode's failure intensity will be removed from the system after the delayed fix has been implemented (i.e., 40% of the failure intensity will remain because the fix is not perfectly effective). This information is used to estimate the fractional decrease in failure intensity that will occur after corrective action is taken.

To specify the effectiveness factors for the BD modes, choose **Growth Data > Crow Extended > Effectiveness Factors** or click the icon on the control panel.

In the spreadsheet area of the window that appears, all BD modes are pre-populated in the BD Mode column. There are two ways to specify the effectiveness of delayed fixes for these modes.

1. You can use the Effectiveness Factor column to set a different effectiveness factor (entered as a decimal) for each unique mode.

   or

2. You can set one general factor that applies to all modes by clicking the **Use Fixed Effectiveness Factor** button on the toolbar and entering the fixed value in the input box, as shown next. Any values entered in the Effectiveness Factor column will then be ignored.

The average of all effectiveness factors will appear in the status bar in the bottom left corner of the window. This value can be used as an input to the growth planning folio, and it can also be useful as a point of reference for estimating the effectiveness factor of a BD mode in a future analysis.

You can enter any additional information about the effectiveness factors in the Comments column. The information in this column does not affect calculations.

If you want to save a record of how the effectiveness factors are currently defined, you can click the **Transfer to Folio** icon . This will add to the folio a general spreadsheet that contains the effectiveness factors you've selected to use, as well as the comments you've entered.

Event Codes for Crow Extended

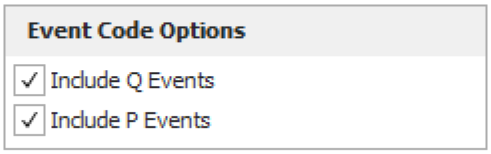
For reliability growth data analysis only.

The Event column that appears in the Multiple Systems with Event Codes data type allows you to specify the type of event that each row in the data sheet represents, so it can be handled appropriately in the calculations. The drop-down list in the column contains the following options:

- **F - Failure** is the default event type. Any event left unspecified will be treated as a normal failure.
- **I - Implemented Fix** indicates the time when the fix for a specified BC failure mode was implemented. This type of event must be preceded by an observed instance of the BC mode in question.

- **P – Performance** and **Q - Quality** indicate failures due to performance or quality issues. You can choose whether or not to consider these events in the analysis.
- Events marked with **X - Exclude** will always be excluded from the analysis. This event code can be used to add comments or a timestamp within the data set. Note that an "X" can be placed in front of any existing event code or entered by itself. For example, "XF" indicates a failure in the data set that should be excluded from the analysis.

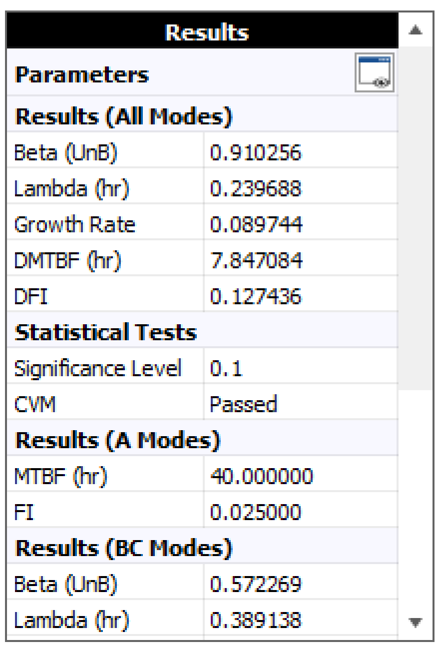
The **Event Code Options** area on the Analysis page of the control panel allows you to choose whether performance (P) and quality (Q) failures will be included in the analysis. By default, both check boxes will be selected.



Crow Extended Analysis Results

For reliability growth data analysis only.

When you calculate a data sheet using the Crow Extended model, the analysis results will be displayed in the **Results** area on the control panel. The results that apply to all failure modes are displayed first. Use the scroll bar to see results that only apply to failure modes with a particular classification (i.e., A Modes = not fixed, BC Modes = fixed during observation period or BD Modes = fixed after observation period), as shown next.



When you use the Crow Extended model, the results may include the following:

- **Beta** and **Lambda** are the two parameters of the Crow Extended model.
  - If beta < 1, the reliability improved during the observation period. If beta > 1, the reliability deteriorated. In addition:
    - If the data set does not include BC modes, **Beta (hyp)** will indicate that the model assumes beta = 1 (i.e., the reliability neither improved nor deteriorated). If this value is shown in red, then the 90% two-sided confidence bounds on beta do not include 1, which means this assumption may not be valid.
    - When working with times-to-failure data, **Beta (Unb)** indicates that the unbiased MLE estimate of the beta parameter was calculated.

      Note that the preference to display Beta (hyp) and/or calculate Beta (Unb) is set on the Calculations page of the Application Setup window for all new folios or in the Settings tab of the Items Properties window for existing folios.
- If the data set includes BC failure modes, the **Growth Rate** will also be displayed, which is equal to 1 - beta. A positive value means the MTBF is improving, and larger values mean faster growth.
- The metrics used to describe the reliability at the end of the observation period will vary depending on the data type you are working with.
  - If you're working with times-to-failure or fielded data, the results will show the mean time between failures and the failure intensity at the end of the observation period. An option on the Calculations page of the Application Setup window (for all new folios) or in the Settings tab of the Items Properties window (for existing folios) determines whether you prefer to call these values "demonstrated" (**DMTBF** and **DFI**) or "achieved" (**AMTBF** and **AFI**).
  - If you're working with Discrete Mixed data, the results will show the demonstrated reliability (**DRel**) and demonstrated failure probability (**DFP**), which is 1 - DRel.
- The **Statistical Tests** area shows whether the data set passed or failed the applicable goodness-of-fit tests that are performed automatically when you calculate the parameters.
  - For failure times and grouped failure times, this will be the Cramer-von Mises test (CVM).
  - For mixed data, it will be the chi-squared test (Ch-Sq).
  - For data from multiple systems, this area may also show the results from the common beta hypothesis (CBH) and/or the Laplace Trend test.

See Statistical Tests for more information.

- Depending on the data type, the **Other** area may display the **Termination Time**:
  - If you are using the Failure Times, Grouped Failure Times or Multiple Systems - Known Operating Times data types, you can use the Termination Time window to specify whether this is the time of the last observed failure, or a specific time that you enter.
  - For all other data types that can be analyzed with the Crow Extended model, the termination time is the end of the observation period.
- Finally, if you are working in a data sheet that supports the Advanced Systems View, the following information will also be displayed.
  - The **Other** area will show the number of systems that were included in the analysis (e.g., 4/6 means four out of six systems were analyzed). You can omit a system from the analysis by clearing its check box on the Systems panel while the data sheet is in Advanced Systems View.
  - The control panel will show two sets of results: the larger table at the top shows the results for the *equivalent system* (which represents the combined operating hours of the systems), and the smaller table at the bottom show the results for the individual system that is currently selected in the data sheet. The specific system name is shown in the heading of this table.

Note: If the result of the analysis is associated with a published model, then the model's name will appear as a link at the bottom of the **Results** area. Click the link to view the model's properties. For details on how to publish the results as a model, see Publishing Models.

QCP Calculations and Plots for Crow Extended

For reliability growth data analysis only.

Weibull++ includes a Quick Calculation Pad (QCP) for computing useful metrics, as well as multiple plots that allow you to visualize the results of your analyses. This topic describes the calculations and plots you can obtain from data sheets analyzed with the Crow Extended model.

Note: When you analyze data from multiple systems, Weibull++ combines the data to create a single representative system (i.e., equivalent system, superposition system or cumulative timeline, depending on the data type). Any plots generated for the combined data set and subsequent analyses via the Quick Calculation Pad will be based on the resulting representative system. See Times-to-Failure Data from Multiple Systems and Fielded Data for more information about how the software combines the data for analysis.

QCP Calculations

You can open the Quick Calculation Pad (QCP) by choosing **Growth Data > Analysis > Quick Calculation Pad** or by clicking the icon on 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 about all the options available in the QCP, see Quick Calculation Pad (QCP).

The Basic Calculations tab of the QCP includes the typical calculations for traditional reliability growth analysis (e.g., cumulative/instantaneous MTBF and expected number of failures). These calculations are applicable only when your data set includes BC failure modes. (See QCP Calculations and Plots for Traditional RGA.)

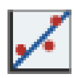
On the Extended Calculations tab, the available calculations will vary depending on whether you're analyzing failure times or mixed (discrete) data. For failure times, the calculated values will be mean time between failures (MTBF) and failure intensity (FI). For mixed data, the values will be reliability and probability of failure. Most of these calculations are applicable only when your data set includes BD failure modes.

- Three values can be calculated for either the mean time between failures (MTBF), failure intensity (FI).
  - The **Demonstrated/Achieved** values reflect the reliability at the end of the observation period, before any delayed fixes have been implemented.
  - The **Projected** values reflect the reliability that will be achieved after the delayed fixes have been implemented.
  - The **Growth Potential** values are the best values that could be achieved by applying the current reliability growth management strategy. In other words, this estimates the maximum reliability growth that you can expect if you continue to find new failure modes at the same rate and make the same types of decisions about which failure modes to fix.
- **Discovery Rate** is the rate at which new BD failure modes are being discovered (i.e., the failure intensity of unseen BD modes) at a specified time. For example, if the discovery rate at 400 hours is 0.02, then 0.02 new BD modes are being discovered every hour (equivalently, 2 new BD modes are discovered every 100 hours).

- **MTBF BD Unseen** is the mean time between failures due to unseen failure modes at a specified time (i.e., BD modes that did not appear during the observation period but are estimated from the analysis).
- **Number of Failures** is the cumulative number of failures that are expected to occur by a specified time, based on the fitted model.
    - If the data set includes BC failure modes, then this value is calculated on the assumption that the reliability changed during the observation period. The option will thus be on the Basic Calculation tab with the other typical calculations for traditional reliability growth analysis (e.g., cumulative/instantaneous MTBF).
    - Otherwise, the calculation assumes that the reliability neither deteriorated nor improved during the observation period (i.e., beta = 1).
- **Expected Fleet Failures** is the number of failures that are expected to occur for all systems by a specified time. This option is available only for fielded repairable data and for fielded fleet data.

Plots

You can create plots by choosing **Growth Data > Analysis > Plot** or by clicking the icon on the control panel.



This section describes the types of plots you can create for the Crow Extended model. The scaling, setup, exporting and confidence bounds settings are similar to the options available for all other reliability growth analysis plot sheets. (See Plots.)

Once again, the available calculations will vary depending on whether you're analyzing failure times or mixed (discrete) data. For failure times, the calculated values will be mean time between failures (MTBF) and failure intensity (FI). For mixed data, the values will be reliability and probability of failure.

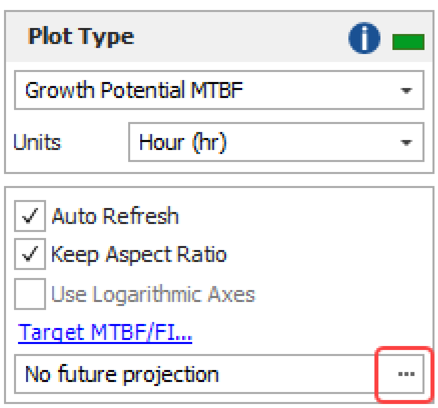
The plots described next apply to data sets that include at least some BD modes. Some of these plots apply specifically to data sets that consist entirely of A/BD modes (e.g., when you use the Crow Extended model to analyze fielded data), where the analysis assumes that there is no reliability growth during the observation.

- **Cumulative Number of Failures** shows the total number of failures versus time. Data points on the plot represent the cumulative number of failures that have been reported by a given time (e.g., the second point marks the time at which the second failure was observed). The lines that can be included will vary depending on whether there are BC modes in the data set.
  - If both BC and BD modes are included, the plot can include the Expected Failures line, which serves as an empirical goodness-of-fit test for the Crow Extended model. It is fitted using the beta that was calculated from the data points.
  - If only A/BD modes are included, the plot can include the Assumed Parameters line based on the assumption that beta = 1 (i.e., no reliability growth was experienced during the observation period) and the Estimated Parameters line based on the beta calculated from the data points. You can then compare the two lines to evaluate whether the beta = 1 assumption is valid.
- The **[Value] vs. Time** plots show how the value increases, decreases or remains constant over time. The points represent the actual failure times in the data set and the plot includes one line for the instantaneous value and one line for the cumulative value. These plots are available when your data set includes BC failure modes.
- **Beta Bounds** is available when only A/BD modes are included in the data set. This plot is used to assess the validity of the assumption that beta = 1. For example, if the two-sided 95% confidence bounds on beta do not include 1, then you can (with 95% confidence) reject the hypothesis that beta = 1. The following lines can be shown in the plot:
  - The Beta Hypothesis line marks beta = 1.
  - The Beta Bounds lines show the confidence bounds for beta at five different confidence levels. You can configure these lines using the Beta Bounds window.
- **System Operation** is available only for multiple systems analysis. It shows the failure times of each system in the data set, along with the timeline for their representative system (i.e., equivalent system, superposition system or cumulative timeline), which is used to evaluate all the failures and fixes that occurred during the observation period. Each type of event is color coded on the plot: failure (red circle), implemented fix (yellow triangle), performance failure (blue circle) and quality failure (green diamond).

  To learn more about how the failure times are combined, see the following topics:

  - For multiple systems times-to-failure data types, see Times-to-Failure Data from Multiple Systems.
  - For fielded data types, see Fielded Data.

- The **Growth Potential** plots are applicable only when the data set includes BD failure modes. The plots can include these items:
    - The Demonstrated/Achieved point represents the value at the end of the test, before any delayed fixes have been implemented.
    - The Projected point represents the expected value after the delayed fixes have been implemented.
    - The Growth Potential line represents the best value that could be achieved by applying the current reliability growth management strategy (i.e., the portion of the system's failure intensity that will be addressed by design fixes).
    - If desired, you can use the Show/Hide Plot Items window to show the Instantaneous line, which shows how the value changes over time during the test. The instantaneous value is calculated over a small interval *dt* that begins at a given time. For example, an instantaneous MTBF of 5 hours at 100 hours duration means that, over the next small interval *dt* that begins at 100 hours, the average MTBF will be 5 hours.
    - If desired, you can estimate the projected MTBF or failure intensity with a specified amount of further testing. This option is applicable only for developmental testing when BD failure modes are included in the data. To enable this option and specify a time greater than the termination time, click the **[...]** button.

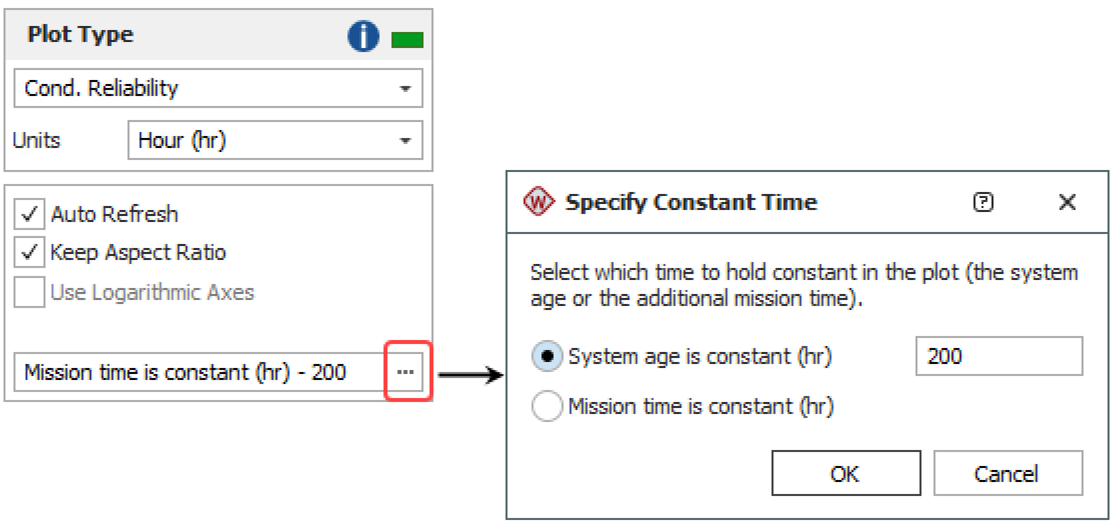


- The **Final** bar charts provide the same information as the Growth Potential plots, but using bars instead of lines.
- **Cumulative Number of BD Modes** shows the total number of unique observed BD modes versus time. This plot can include these items:

  - Data points on the plot represent the cumulative number of BD modes that have been discovered by a given time. For example, the second point marks the time at which the second unique BD mode (e.g., BD2) was observed.
  - The Cumulative Number of BD Modes line is fitted to the data points and shows how the cumulative number of discovered BD modes changes with time.
- **MTBF BD Unseen** and **Discovery Rate** show the rate at which new unique BD failure modes are being discovered at any given time. For example, if the MTBF for unseen BD modes is 50 hours, then 2 more unique BD modes are expected to be observed over the next 100 hours. The discovery rate is the failure intensity of the unseen BD modes (i.e., the inverse of the MTBF). In a successful reliability growth test, the MTBF for unseen BD modes will increase over time and the discovery rate will decrease.
- The **Individual Mode** bar charts show two bars for each failure mode. Use the Plot Modes window to choose which failure modes to include in the chart.
  - The Before bars represents the value at the end of the test, before any delayed fixes have been implemented.
  - The After bars are available only for BD modes and represents the value after the delayed fixes have been implemented.
- The **Failure Mode Strategy** pie chart breaks down the overall failure intensity (FI) into six possible categories:
  - The Type A slice represents the FI that is due to failure modes that are ignored (i.e., no fixes will be implemented).
  - The Type BC - Seen slice represents the FI that is due to BC modes that were observed and for which fixes were implemented.
  - The Type BC - Unseen and Type BD - Unseen slices represents the FI that is due to BC and BD modes that were not observed but are estimated from the analysis. If they were observed (e.g., through future testing), fixes would be implemented.
  - The Type BD - Remained and Type BD - Removed slices represent the FI that is due to BD modes that were observed (i.e., they represent the FI due to seen BD modes). The Remained slice represents the portion of FI that is expected to stay in the system because the fixes are not 100% effective, while the Removed slice represents the FI that is expected to be eliminated.

- **Conditional Reliability/Unreliability** is available only when you are using the Repairable data sheet. These plots show the reliability/unreliability versus system age or mission time.
  - If you choose to hold the system age constant, the plots will show the reliability/unreliability for different mission times. For example, assuming that the system has already operated for 100 hours, the plot can show the reliability for the next 10 hours, 20 hours, 30 hours, etc.
  - If you choose to hold mission time constant, the plots will show the reliability/unreliability for different system ages. For example, assuming that the mission will be 100 hours, the plot can show the reliability for a system that has already operated for 10 hours, 20 hours, 30 hours, etc.

  To specify which value will be held constant, click the **[...]** button on the control panel. In the window that appears, select the type of metric that will be held constant, as shown next. Then enter the constant value in the input field.

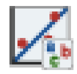


Tip: Weibull++ includes two additional plot utilities you can use across all types of data: the overlay plot, which allows you to compare different data sets or models; and the side-by-side plot, which allows you to display different plots of a single data set all in a single window for easy comparison.

Plot Modes Window

For reliability growth data analysis only.

When you're viewing an Individual Mode MTBF/FI/Reliability plot, you can select which of the failure modes in the current data sheet will appear by choosing **Plot > Options > Plot Modes** or by right-clicking the plot and choosing the option from the shortcut menu.



To show/hide all of the modes, click the column headings above the check boxes.

To select all modes of a given classification (i.e., all A modes, all BC modes or all BD modes), choose the desired classification in the **Select by Mode** field at the bottom of the window.

To export all the data shown in the window to an Excel file, click the **Send to Excel** button.

CHANGING THE ORDER OF THE FAILURE MODES

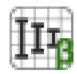
The order of failure modes in this window reflects the order of modes in the plot. By default, the modes are shown in the same order that they occurred during the observational period. To change the order, you can sort the modes by clicking the column heading that you wish to sort by. Click the heading again to reverse the order of the sort, and click it one more time to return the order to its original state.

- **Classification** displays the mode classifications.
- **Mode** displays the failure mode identifiers.
- **Effectiveness Factor** displays the corresponding effectiveness factor for each BD mode, if any factor was specified.
- **Time to First Failure** displays the first time at which the failure mode was observed.
- **Before MTBF** displays the mode's MTBF at the end of the observation period and before any delayed fixes are implemented.
- **After MTBF** displays the mode's MTBF after any delayed fixes are implemented. For BD modes, the "before" and "after" values will be different. For A and BC modes, they will be the same.
- **Change in MTBF** displays the difference between the "before" and "after" values for BD modes.

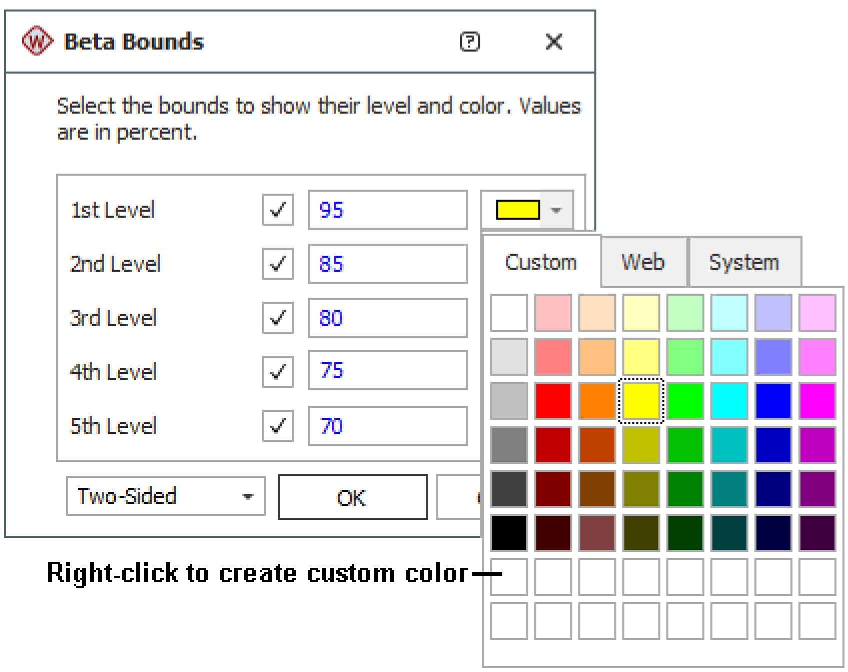
Beta Bounds Window

For reliability growth data analysis only.

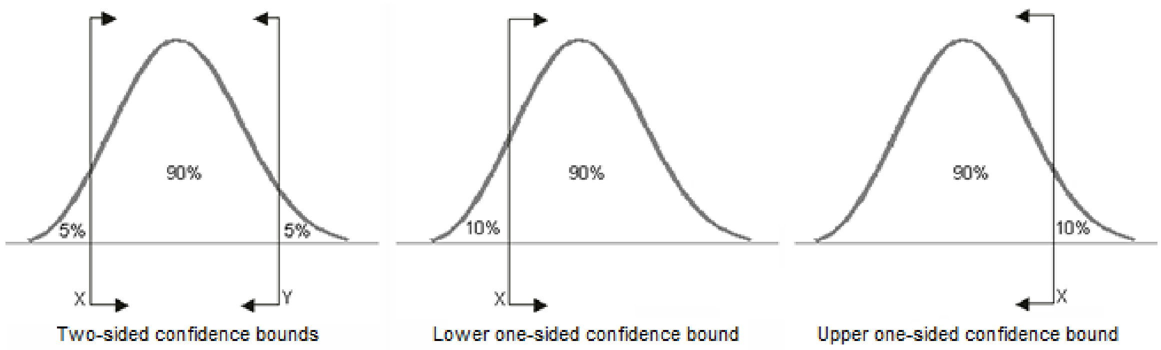
When you're viewing the Beta Bounds plot, you can specify which confidence levels to show by choosing **Plot > Options > Plot Beta Bounds** or by right-clicking plot and choosing the option from the shortcut menu. Beta Bounds is available when only A/BD modes are included in the data set analyzed with the Crow Extended model.



- In the window that appears, you can select to show up to five confidence levels.
- For each beta bound line you have selected to show, enter the confidence level as a percentage in the input field.
- You can click the color field to define the line color for the beta bound. To add a new color to the Custom tab, right-click one of the color boxes in the bottom two rows.



- The drop-down list in the lower left corner of the window allows you to select which type of confidence bounds (one-sided or two-sided) to display in the plot. For example, if you use 90% one-sided bounds, then the upper and lower one-sided 90% bounds will be used (which is the same as using two-sided 80% bounds). The following graphics illustrate the difference between one- and two-sided bounds.



Crow Extended Examples

For reliability growth data analysis only.

The Crow Extended model can be used with data obtained from developmental testing or from systems operating in the field. The following topics provide examples of using the model with these different types of data:

- The developmental testing example uses data obtained from a single system during a developmental test. In this scenario, design fixes are applied during the test. Thus, reliability growth is expected to be observed during the test, and additional growth can be projected for fixes that will be applied after the test.
- The operational testing example uses data obtained from two fielded repairable systems that were tested under customer use conditions. In this scenario, any repairs performed during the test are assumed to be minimal repairs (i.e., they do not improve the system reliability). All permanent design fixes are delayed until after the test. Thus, reliability growth is not expected to be observed during the test, but the model will project the growth due to fixes that will be applied after the test.

Example Using the Crow Extended Model for Developmental Test Data

For reliability growth data analysis only.

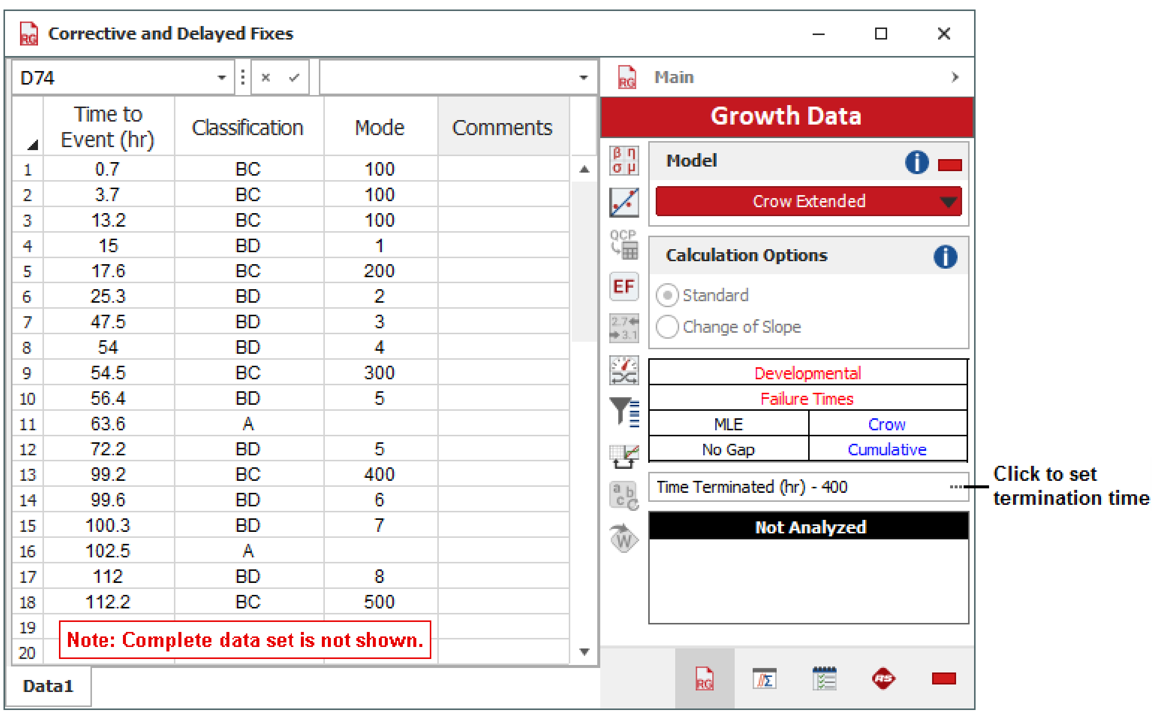
*The data set used in this example is available in the example database installed with the software (called "Weibull21_ReliabilityGrowth_RGA_Examples.rsgz21"). To access this database file, choose **File > Help**, click **Open Examples Folder**, then browse for the file in the Weibull subfolder.*

The name of the project is "Crow Extended - Failure Times," and the folio that contains the data is called "Corrective and Delayed Fixes."

Failure times for a single system undergoing developmental testing were recorded. Some of the observed failure modes were not addressed, some were addressed during the test and some were delayed until after the test. The test ended at 400 hours, and the goal is to estimate the following values:

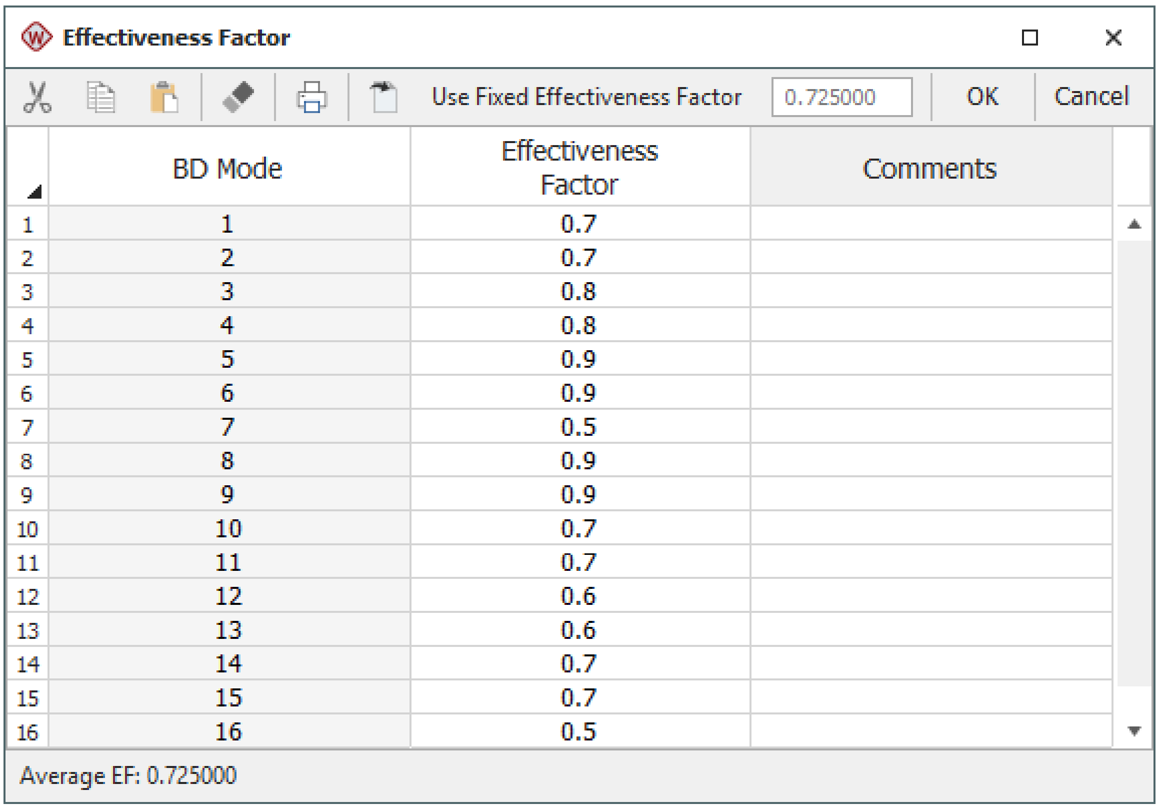
- The MTBF demonstrated at the end of the test.
- The MTBF that can be expected after the BD failure modes are addressed.
- The maximum MTBF that can be achieved if all BD modes that exist in the system were discovered and fixed according to the current maintenance strategy (i.e., given the portion of the system's failure intensity that will be addressed by corrective actions).

The data from the test are recorded in a Failure Times data sheet configured for cumulative failure times and with the Crow Extended model selected, as shown next. Modes classified as "A" are not addressed. "BC" modes are addressed during the test, and "BD" modes are addressed after the test. Numerical codes were used to identify the modes in the order in which they appeared (e.g., 100 and 200 are the first and second BC modes, 1 and 2 are the first and second BD modes, etc.). To specify that the test is time terminated at 400 hours, click the **(...)** button on the control panel to open the Termination Time window.



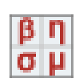
Next, enter an effectiveness factor for each BD mode to specify the effectiveness of each delayed fix (e.g., a factor of 0.7 means the fix will reduce the mode's failure intensity by 70%). To do this, choose **Growth Data > Crow Extended > Effectiveness Factors** or click the icon on the control panel.

Enter the factors shown next. You can see that the average effectiveness factor for all the delayed fixes—displayed in the status bar—is 0.7250. Click **OK** to save the changes and close the window.



	BD Mode	Effectiveness Factor	Comments
1	1	0.7	
2	2	0.7	
3	3	0.8	
4	4	0.8	
5	5	0.9	
6	6	0.9	
7	7	0.5	
8	8	0.9	
9	9	0.9	
10	10	0.7	
11	11	0.7	
12	12	0.6	
13	13	0.6	
14	14	0.7	
15	15	0.7	
16	16	0.5	

Click the **Calculate** icon on the folio's control panel to analyze the data.



Note: This model uses the maximum likelihood estimation (MLE) method to calculate the parameters, which is known to produce a biased value for beta. This bias is more evident when working with small sample sizes. If the folio is not configured to remove this bias, you will be prompted to change the option in the Settings tab of the Item Properties window (**Project > Current Item > Item Properties**). To ensure the unbiased beta is always calculated for all new folios, use the RGA Growth Data Folios page of the Application Setup window to select the option. This example assumes that the unbiased beta is calculated.

The results for all failure modes will appear as shown next.

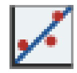
Results	
Parameters	
Results (All Modes)	
Beta (UnB)	0.910256
Lambda (hr)	0.239688
Growth Rate	0.089744
DMTBF (hr)	7.847084
DFI	0.127436
Statistical Tests	
Significance Level	0.1
CVM	Passed
Results (A Modes)	
MTBF (hr)	40.000000
FI	0.025000
Results (BC Modes)	
Beta (UnB)	0.572269
Lambda (hr)	0.389138
MTBF (hr)	58.247704
FI	0.017168
CVM	Passed
Results (BD Modes)	
Beta (UnB)	0.747151
Lambda (hr)	0.181966
MTBF	33.460453
FI	0.029886
CVM	Passed
Other	
Termination Time (hr): 400.000000	

The beta, lambda and growth rate (1 - beta) parameters are shown in the table. In addition to these values, the table provides this information:

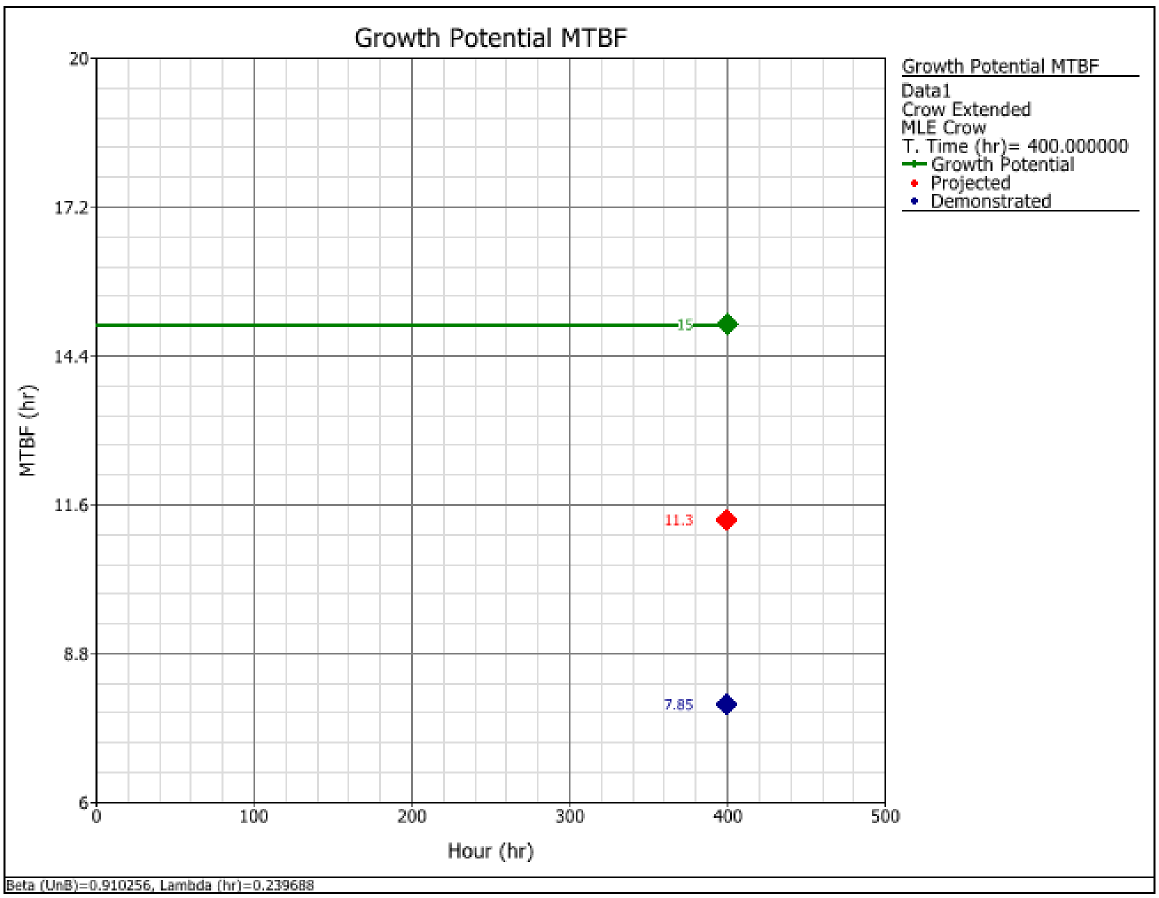
- The demonstrated/achieved mean time between failures (**DMTBF/AMTBF**) at the end of the test is 7.8471 hours.
- The demonstrated/achieved failure intensity (**DFI/AFI**) at the end of the test is 0.1274.

- The data is shown to have passed the Cramér-von Mises statistical test (**CVM**), which means we can accept the hypothesis that the failure times follow a non-homogeneous Poisson process (NHPP). This hypothesis is assumed by the Crow Extended model.

Next, to view all the desired values in a plot, click the **Plot** icon on the control panel.



Then choose **Growth Potential MTBF** from the **Plot Type** drop-down list. The plot appears as shown next.



The following information is shown in the plot:

- The blue Demonstrated point shows the MTBF at the end of the test (before the BD failure modes are addressed) is 7.8471 hours.
- The red Projected point shows that, after the BD failure modes are addressed, the MTBF is expected to be 11.3182 hours.

- The green Growth Potential line shows that the maximum achievable MTBF given the current management strategy is 14.9957 hours.

Example Using the Crow Extended Model for Operational Test Data

For reliability growth data analysis only.

*The data set used in this example is available in the example database installed with the software (called "Weibull21_ReliabilityGrowth_RGA_Examples.rsgz21"). To access this database file, choose **File > Help**, click **Open Examples Folder**, then browse for the file in the Weibull sub-folder.*

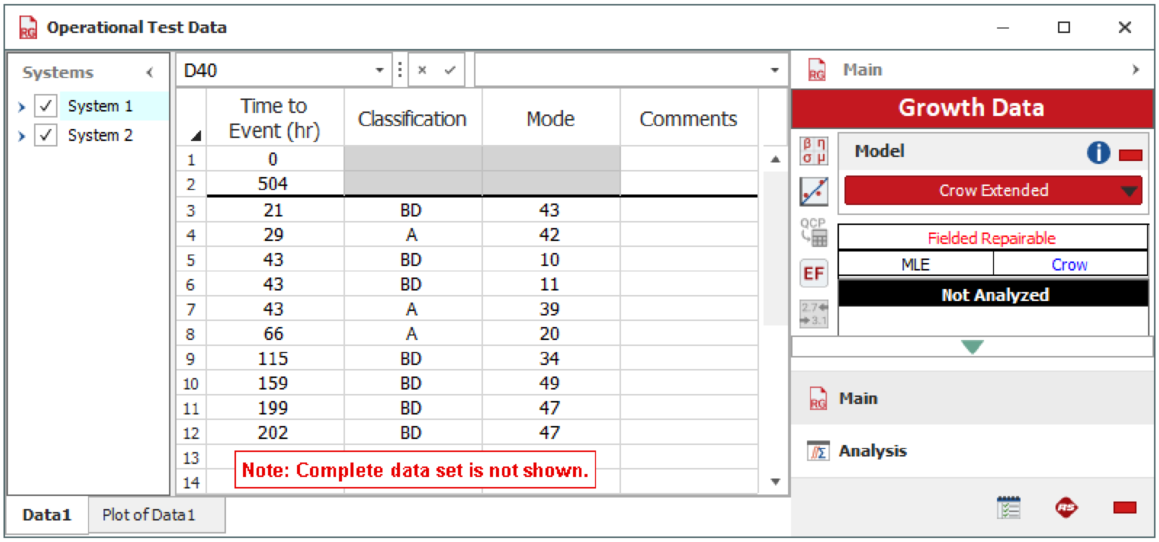
The name of the project is "Crow Extended - Operational Test Data for Two Systems," and the folio that contains the data is called "Operational Test Data."

Operational testing for two systems is performed towards the end of a new product development program. The systems are pilot builds and are subjected to representative customer use conditions during testing. When a system fails, a minimal repair is performed to bring the system back to operating condition (i.e., repairs are only enough to bring the system back to operation). Therefore, the configuration of each system during the test is assumed to not change and the reliability is assumed to neither deteriorate nor improve during the test (i.e., beta = 1). In addition, for each failure time, the associated failure mode is identified and classified.

The analysts have the following goals:

- Determine the MTBF that would be attained at the end of the basic reliability tasks if all the problem failure modes were uncovered in early design and corrected in accordance with the management strategy.
- Create a plot to determine whether new BD modes are being discovered at a decreasing rate throughout the test. If so, this would indicate that the number of remaining unseen BD modes is decreasing with time.

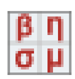
The data from the test are recorded in a Repairable data sheet with the Crow Extended model selected. Modes classified as "A" are not addressed, and "BD" modes are addressed after the test. Numerical codes were used to identify each specific failure mode. Some of the data for System 1 (selected in the Systems panel on the left) are shown next.



Before you can analyze the data, you must enter an effectiveness factor for each BD mode to specify the effectiveness of each corrective action that will be performed after the test (e.g., a factor of 0.7 means the corrective action will reduce the mode's failure intensity by 70%). To do this, choose **Growth Data > Crow Extended > Effectiveness Factors** or click the icon on the control panel.

In the window that appears, select the **Use Fixed Effectiveness Factors** option and then enter **0.6** in the input field. Click **OK** to save the factors and return to the folio.

Click the **Calculate** icon on the folio's control panel to analyze the data.



The results for all failure modes will appear as shown next.

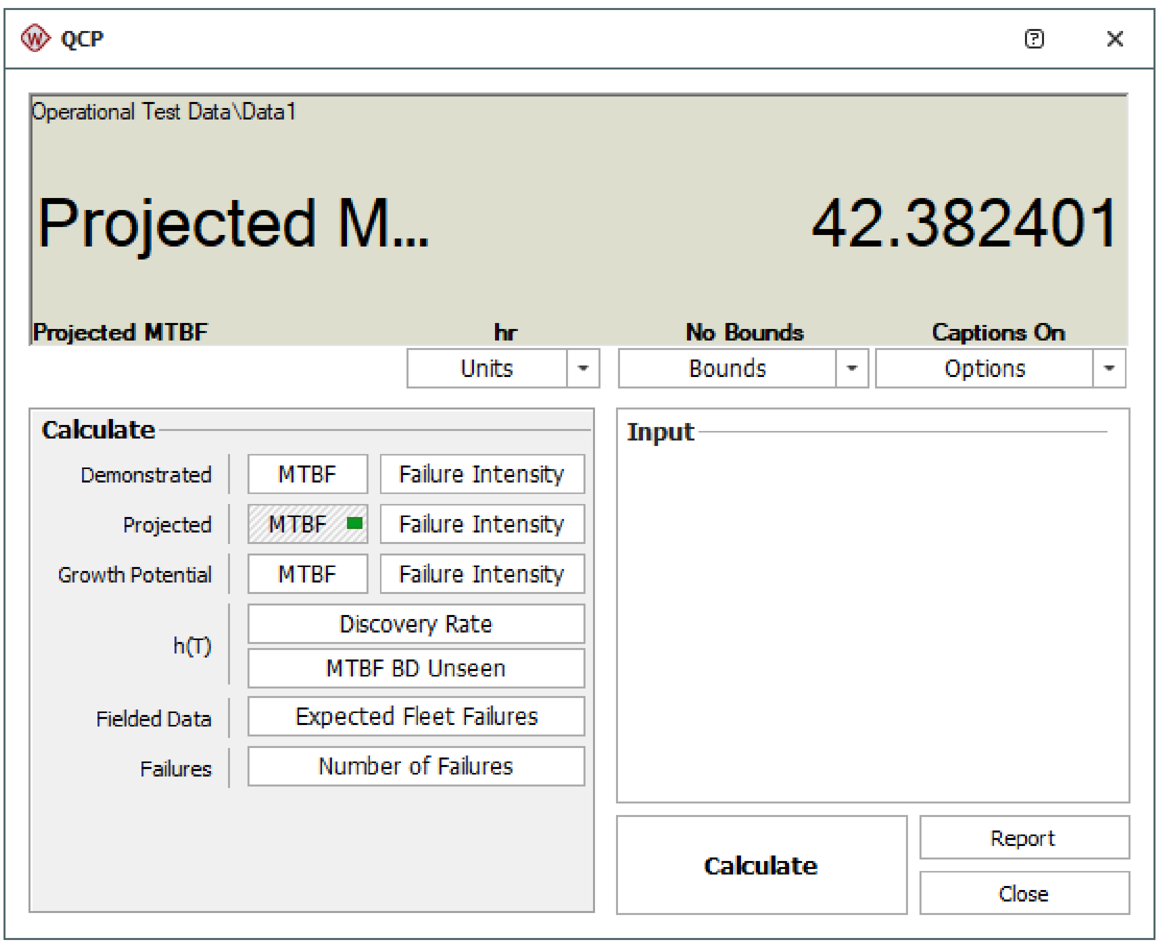
Results	
Parameters	
Results (All Modes)	
Beta (hyp)	1.000000
Beta	0.868505
Lambda (hr)	0.067560
DMTBF (hr)	33.709677
DFI	0.029665
Statistical Tests	
Significance Level	0.1
CVM	Passed
CBH	Passed
Results (A Modes)	
MTBF (hr)	149.285714
FI	0.006699
Results (BD Modes)	
Beta (UnB)	0.789849
Lambda (hr)	0.070114
MTBF	77.825741
FI	0.012849
Other	
Termination Time (hr): 541.000000	
Systems: 2/2	
Individual System Results	
System 1	
Beta (UnB)	0.838982
Lambda (hr)	0.124290
CVM	Passed
Laplace	No Trend

- The **Parameters** values include estimated parameters for the fitted model and other calculations based on those parameters.
  - The beta hypothesis, **Beta (hyp)**, is the assumed beta value for the analysis. If this value is shown in red, then the beta = 1 assumption may not be valid.
  - The calculated **Beta** and **Lambda** parameters are 0.8685 and 0.0676. The calculated beta value is used to determine whether the beta = 1 assumption is valid. (If 0.8685 were significantly different from 1, then the assumption would be invalid.)
  - The demonstrated/achieved mean time between failures (**DMTBF**/**AMTBF**) at the end of the test is 33.7097 hours.
  - The demonstrated/achieved failure intensity (**DFI**/**AFI**) at the end of the test is 0.0297.

- The **Statistical Tests** values indicate whether the data meet various assumptions underlying the analysis.
  - The data is shown to have passed the Cramér-von Mises test (**CVM**), which means we can accept the hypothesis that the failure times follow a non-homogeneous Poisson process (NHPP).
  - The data also passed the Common Beta Hypothesis (**CBH**) test, which means the data sets from both systems can be assumed to have the same beta value.

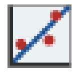
To calculate the maximum attainable MTBF based on the current management strategy, click the **Quick Calculation Pad** icon on the control panel.

In the window that appears, select the **MTBF** option in the **Projected** area. After you click **Calculate**, the QCP will appear as shown next.

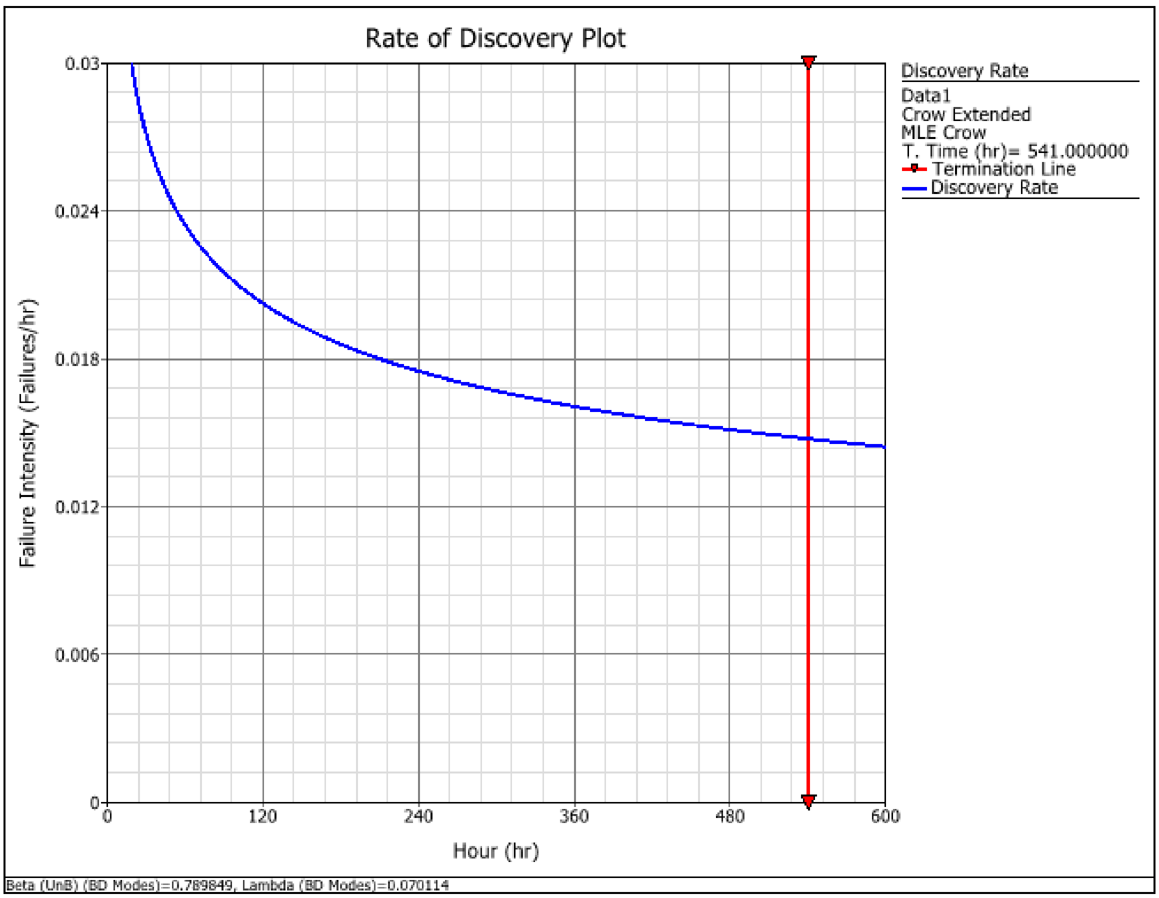


This result means the MTBF demonstrated at the end of the test (the DMTBF of 33.7097 shown on the control panel) is expected to increase to 42.3824 hours after all the BD modes have been fixed.

The next goal is to create a plot to examine how the failure intensity due to unseen BD modes changes over time. To do this, click the **Plot** icon on the control panel.



Then choose **Discovery Rate** from the **Plot Type** drop-down list and clear the **Use Logarithmic Axes** option on the control. The plot will appear as shown next.



As unique BD modes are being discovered during the test, the unseen failure intensity decreases. This indicates that unique BD modes are being discovered at a decreasing rate, which is the desired outcome for a reliability growth test.

Analysis with the Crow Extended - Continuous Evaluation Model

For reliability growth data analysis only.

While the Crow Extended model facilitates reliability growth projections, planning and analysis for a single test phase, the Crow Extended – Continuous Evaluation model is designed for analyzing data across up to 10 test phases.

This model provides increased flexibility to analyze data from practical testing situations in which the failure modes may be addressed a) at the time of the failure, b) later in the same test phase, c) between test phases d) during a subsequent test phase or e) after the completion of all test phases.

The continuous evaluation model is available in three Multi-Phase data types: Multi-Phase Failure Times, Multi-Phase Grouped Failure Times and Multi-Phase Mixed data.

Two related features will be discussed in the next chapter. You can use the growth planning folio to develop the plan for a multi-phase test. Then after you've begun to enter the test data into a multi-phase data sheet, you can use a multi-phase plot to link the plan with your test data. The plot provides a visual way to track the progress toward meeting your MTBF goal and determine whether you will need to make adjustments in the remaining test.

For an example that demonstrates how these three tools can be used together, see Example: Multi-Phase Test Planning and Analysis.

Event Codes for Multi-Phase Data

For reliability growth data analysis only.

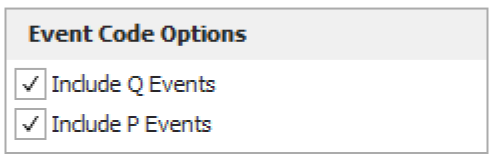
The Event column that appears with all three Multi-Phase data types allows you to specify the type of event that each row in the data sheet represents, so it can be handled appropriately in the calculations.

The drop-down list in the Event contains the following options:

- **F - Failure** is the default event type. Any event left unspecified will be treated as a normal failure.
- **I - Implemented Fix** indicates the time when the fix for a specified BD (delayed fix) failure mode was implemented. This type of event must be preceded by an observed instance of the BD mode in question.

- **P - Performance** and **Q - Quality** indicate failures due to performance or quality issues. You can choose whether or not to consider these events in the analysis.
- Events marked with **X - Exclude** will always be excluded from the analysis. This event code can be used to add comments or a timestamp within the data set. Note that an "X" can be placed in front of any existing event code or entered by itself. For example, "XF" indicates a failure in the data set that should be excluded from the analysis.
- **AP - Analysis Point** indicates a point at which the demonstrated, projected and growth potential MTBF/FI values can be displayed on a multi-phase plot in order to track the progress up to that point.
- **PH - End of Phase** indicates the end of a test phase.

The **Event Code Options** area on the Analysis page of the control panel allows you to choose whether performance (P) and quality (Q) failures will be included in the analysis. By default, both check boxes will be selected.



Failure Mode Classifications for Multi-Phase Data

For reliability growth data analysis only.

Like the failure mode classifications for the Crow Extended model, analysis with the Crow Extended – Continuous Evaluation model will also require you to identify and classify the specific mode responsible for each failure.

For both models, the **Mode** column records the specific failure mode identification, which is optional for A modes but required for all BC and BD modes. This can be entered as either a name or a numerical code (e.g., "seal leak," "1," "BD1," etc.).

However, the classifications are applied a bit differently for the continuous evaluation model in order to support a wider array of possible management strategies. Specifically, for the Classification column in a multi-phase data sheet:

- **A** indicates that no fix will be applied. Management chooses not to address the failure mode because of technical, financial or other reasons. (This is the same as in Crow Extended.)
- **BC** indicates that the fix will be applied at the time of failure and before the testing continues. (Whereas, in Crow Extended, BC indicates that the fix is applied at any time during the test.)

- **BD** indicates that the fix will be delayed until a later time, either in the same test phase, between test phases, in a subsequent test phase or after all test phases are completed. (Whereas, in Crow Extended, BD indicates that the fix is delayed until after the completion of the test.)

The increased flexibility for the treatment of BD failure modes means that the data set can also contain the exact times when the delayed fixes are implemented. Use the **I – Implemented Fix** event code to indicate the time of a delayed fix. The I event must have the exact same classification and mode as the prior failure event (F, P or Q). For example, the following picture shows a highly simplified data set where Mode 1 occurs at 100 hours but the fix is delayed. The first test phase ends at 200 hours and then the fix for Mode 1 is implemented during the second phase, at 250 hours of total test time.

	Event	Time to Event (hr)	Classification	Mode	Comments
1	F	100	BD	Mode 1	Delayed Fix
2	F	120	A	Mode 2	No Fix
3	PH	200			End of Phase 1
4	F	210	BD	Mode 3	Delayed Fix
5	I	250	BD	Mode 1	Implemented Fix
6	F	260	BC	Mode 4	Immediate Fix

If the fix is implemented between test phases or after all test phases are completed, you will not be able to enter a specific I event time in the data sheet. In such cases, you will use the Effectiveness Factors window to indicate when the fix was (or will be) implemented, and supply the estimated effectiveness factor to be used during data analysis. (See Effectiveness Factors for Multi-Phase Data.)

Effectiveness Factors for Multi-Phase Data

For reliability growth data analysis only.

Like the effectiveness factors for the Crow Extended model, analysis with the Crow Extended – Continuous Evaluation model may also require you to specify the decrease in failure intensity for some delayed fixes. However, in the continuous evaluation model, a specified effectiveness factor is required only if the delayed fix time (I event) is not recorded in the data set. (If the fix time is provided, the model can use the data to evaluate the effectiveness of those fixes.)

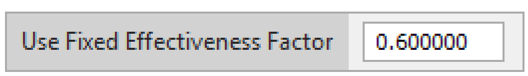
To specify the required effectiveness factors, choose **Growth Data > Crow Extended > Effectiveness Factors** or click the icon on the control panel.

In a multi-phase data sheet, the Effectiveness Factors window will display one or both of the following sheets.

BD Modes Sheet

The BD Modes sheet is always displayed and it shows a list of any BD failure modes in the current data set that do not have a specific fix time (I event) recorded in the data set. For each failure mode in the list:

- Use the **Implemented at End of Phase #** column to specify when the delayed fix was (or will) be applied.
    - If the fix was applied after a specific test phase, select that phase from the drop-down list. For example, if the fix was applied between Phase 1 and Phase 2, select **1 – Phase 1**.
    - If the fix is not yet implemented, select **Not Implemented**.
- Specify the effectiveness factor, entered as a decimal. For example, 0.6 indicates that 60% of the failure intensity will be removed after the fix is applied but 40% will remain. There are two options:
    - If you want to assign different effectiveness factors for different failure modes, use the **Effectiveness Factor** column to specify the factor for each mode.
    - If you prefer to assume the same effectiveness factor for all failure modes shown in the list, click the **Use Fixed Effectiveness Factor** button on the toolbar. This will disable the column in the spreadsheet and you can enter a single value in the input box next to the button, as shown next.



Implemented Fixes Sheet

If you have selected the **Allow EF values for BD modes with implemented fixes** option on the Calculations page of the Application Setup, the Implemented Fixes sheet will show a list of the BD failure modes in the current data set that do have a specific fix time (I event) recorded. This sheet is for your information only. Any inputs you provide here will not affect the calculations.

Analysis Results for Multi-Phase Data

For reliability growth data analysis only.

When you calculate a multi-phase data sheet with the Crow Extended – Continuous Evaluation model, the following results will be displayed on the Main page of the control panel. By default, the Results area displays the results from all failure mode classifications first. Use the scroll bar to see results that only apply to failure modes with a particular classification (A, BC and BD), as shown next.

Results	
Parameters	
Results (All Modes)	
Beta	0.733016
Lambda (hr)	0.116627
p	0.272727
Growth Rate	0.266984
DMTBF (hr)	143.602744
DFI	0.006964
Statistical Tests	
Significance Level	0.1
Chi-Sq	Passed
Results (A Modes)	
MTBF (hr)	631.578947
FI	0.001583
Results (BC Modes)	
Beta	0.811315

- **Beta**, **Lambda** and **p** are the three parameters of the Crow Extended - Continuous Evaluation model.
  - If beta < 1, the reliability improved during the observation period. If beta > 1, the reliability deteriorated. In addition:
    - If the data set does not include any failure modes that were fixed before the end of the last test phase (such as BC modes), **Beta (hyp)** will indicate that the model assumes beta = 1 (i.e., the reliability neither improved nor deteriorated). If this value is shown in red, then the 90% two-sided confidence bounds on beta do not include 1, which means this assumption may not be valid.
    - When working with times-to-failure data, **Beta (UnB)** indicates that the unbiased MLE estimate of the beta parameter was calculated.

      Note that the preference to display Beta (hyp) and/or calculate Beta (UnB) is set on the RGA Growth Data Folios page of the Application Setup window for all new folios or in the Settings tab of the Items Properties window for existing folios.
  - The p value is the probability that the corrective action for an observed BD failure mode will occur after the end of the last test phase. In other words, for every new BD mode that is

discovered during the test, p is the probability that its corrective action will be delayed until after the test.

- The **Growth Rate** is equal to 1 - beta. A positive value means the MTBF is improving, and larger values mean faster growth.
- The metrics used to describe the reliability at the end of the observation period will vary depending on the data type.
    - For failure times and grouped failure times, the results will show the mean time between failures and the failure intensity at the end of the observation period. An option on the Calculations page of the Application Setup window (for all new folios) or in the Settings tab of the Items Properties window (for existing folios) determines whether you prefer to call these values "demonstrated" (**DMTBF** and **DFI**) or "achieved" (**AMTBF** and **AFI**).
    - For mixed data, the results will show the demonstrated reliability (**DRel**) and demonstrated failure probability (**DFP**), which is 1 - DRel.
- The **Statistical Tests** area shows whether the data set passed or failed the applicable goodness-of-fit test that is performed automatically when you calculate the parameters. For failure times and grouped failure times, this will be the Cramer-von Mises test (**CVM**). For mixed data, it will be the chi-squared test (**Ch-Sq**). (See Statistical Tests.)
- For failure times and grouped failure times data sheets, the **Termination Time** is the time of the last data point that is not excluded from the analysis. If this is the end of a phase (PH) or an analysis point (AP), the folio performs a time-terminated calculation. If this is a failure time (F, P or Q), it will be failure-terminated. This is not applicable for mixed data sheets.

Note: If the result of the analysis is associated with a published model, then the model's name will appear as a link at the bottom of the **Results** area. Click the link to view the model's properties. For details on how to publish the results as a model, see Publishing Models.

QCP Calculations and Plots for Multi-Phase Data

For reliability growth data analysis only.

Weibull++ includes a Quick Calculation Pad (QCP) for computing useful metrics, as well as multiple plots that allow you to visualize the results of your analyses. This topic describes the calculations and plots you can obtain from multi-phase data sheets analyzed with the Crow Extended – Continuous Evaluation model.

QCP Calculations

You can open the Quick Calculation Pad (QCP) by choosing **Growth Data > Analysis > Quick Calculation Pad** or by clicking the icon on 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 about all the options available in the QCP, see Quick Calculation Pad (QCP).

The Basic Calculations tab of the QCP includes the typical calculations for traditional reliability growth analysis (e.g., cumulative/instantaneous MTBF and expected number of failures). These calculations are applicable only when your data set includes failure modes that were fixed before the end of the last test phase. (See QCP Calculations and Plots for Traditional RGA.)

On the Multi-Phase Calculations tab, the available calculations will vary depending on whether you're analyzing failure times or mixed (discrete) data. For failure times, the calculated values will be mean time between failures (MTBF) and failure intensity (FI). For mixed data, the values will be reliability and probability of failure. Most of these calculations are applicable only when your data set includes BD failure modes. The projected and growth potential calculations apply only when the data set includes BD modes that will be fixed after the end of the last test phase (as specified in the Effectiveness Factors window).

- The **Demonstrated** values reflect the reliability at the end of the last test phase, before any delayed fixes are implemented.
- The **Nominal** values represent the best case scenario in which a fix is applied to every BD failure mode, even if the fix has not yet been implemented or planned. These calculations consider the implemented fix times (I events) and all of the effectiveness factors (even if the fix is marked "Not Implemented").
    - **Nominal Projected** are the expected values after the delayed fixes for all seen BD failure modes.
    - **Nominal Growth Potential** are the best possible values that could be achieved if all delayed fixes are implemented for all seen and unseen BD failure modes.
- The **Actual** values take into account which delayed fixes have actually been implemented in the current management strategy. These calculations consider the implemented fix times (I events) and the effectiveness factors for the fixes that were marked to be implemented after a

specific test phase. However, any BD failure mode that is marked as "Not Implemented" in the Effectiveness Factors window is given an EF=0.

- **Actual Projected** are the expected values after the delayed fixes that are actually implemented for seen BD failure modes.

- **Actual Growth Potential** are the best possible values that could be attainable by applying the current management strategy for both seen and unseen BD failure modes.

- **Discovery Rate** is the rate at which new BD failure modes are being discovered (i.e., the failure intensity of unseen BD modes) at a specified time. For example, if the discovery rate at 400 hours is 0.02, then 0.02 new BD modes are being discovered every hour (equivalently, 2 new BD modes are discovered every 100 hours).

- **MTBF BD Unseen** is the mean time between failures due to unseen failure modes at a specified time (i.e., BD modes that did not appear during the observation period but are estimated from the analysis).

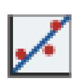
- **Number of Failures** is the cumulative number of failures that are expected to occur by a specified time, based on the fitted model.

  - If the data set includes some failure modes that were fixed before the end of the last test phase, then this value is calculated on the assumption that the reliability changed during the test. The option will thus be on the Basic Calculations tab with the other typical calculations for traditional reliability growth analysis (e.g., cumulative/instantaneous MTBF).

  - Otherwise, the calculation assumes that the reliability neither deteriorated nor improved during the observation period (i.e., beta = 1).

Plots

You can create plots by choosing **Growth Data > Analysis > Plot** or by clicking the icon on the control panel.



This section describes the types of plots you can create for the Crow Extended – Continuous Evaluation model. The scaling, setup, exporting and confidence bounds settings are similar to the options available for all other reliability growth analysis plot sheets. (See Plots.)

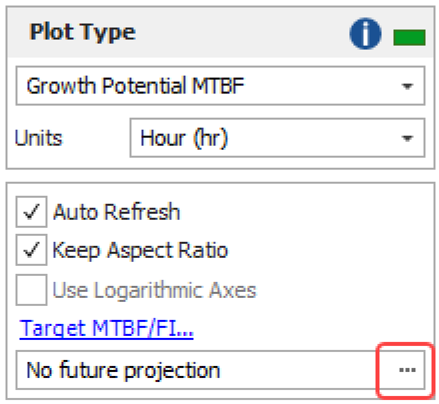
Once again, the available calculations will vary depending on whether you're analyzing failure times or mixed (discrete) data. For failure times, the calculated values will be mean time between failures (MTBF) and failure intensity (FI). For mixed data, the values will be reliability and probability of failure.

Also note that all projected and growth potential values displayed in the plots are based on the "actual" (not "nominal") calculations.

- **Cumulative Number of Failures** shows the total number of failures versus time. Data points on the plot represent the cumulative number of failures that have been reported by a given time (e.g., the second point marks the time at which the second failure was observed). The lines that can be included will vary depending on whether there the data set includes failures modes that will be fixed before the end of the last test phase.
  - If some failure modes will be fixed before the end of the last test phase, the plot can include the Expected Failures line, which serves as an empirical goodness-of-fit test for the Crow Extended model. It is fitted using the beta that was calculated from the data points.
  - If no failure modes will be fixed before the end of the last test phase, the plot can include the Assumed Parameters line based on the assumption that beta = 1 (i.e., no reliability growth was experienced during the observation period) and the Estimated Parameters line based on the beta calculated from the data points. You can then compare the two lines to evaluate whether the beta = 1 assumption is valid.
- The **[Value] vs. Time** plots show how the value increases, decreases or remains constant over time. The points represent the actual failure times in the data set and the plot includes one line for the instantaneous value and one line for the cumulative value. These plots are available when your data set includes failure modes that were fixed before the end of the last test phase.
- The **Growth Potential** plots are applicable only when the data set includes failure modes that will be fixed after the end of the last test phase (as specified in the Effectiveness Factors window). The plots can include these items:
  - The Demonstrated/Achieved point represents the value at the end of the test, before any delayed fixes have been implemented.
  - The Projected point represents the expected value after the delayed fixes have been implemented.
  - The Growth Potential line represents the best value that could be achieved by applying the current reliability growth management strategy (i.e., the portion of the system's failure intensity that will be addressed by design fixes).

- If desired, you can use the Show/Hide Plot Items window to show the Instantaneous line, which shows how the value changes over time during the test. The instantaneous value is calculated over a small interval *dt* that begins at a given time. For example, an instantaneous MTBF of 5 hours at 100 hours duration means that, over the next small interval *dt* that begins at 100 hours, the average MTBF will be 5 hours.

- If desired, you can estimate the projected MTBF or failure intensity with a specified amount of further testing. This option is applicable only for developmental testing when BD failure modes are included in the data. To enable this option and specify a time greater than the termination time, click the **[...]** button.



- The **Final** bar charts provide the same information as the Growth Potential plots, but using bars instead of lines.

- **Cumulative Number of BD Modes** shows the total number of unique observed BD modes versus time. This plot can include these items:

  - Data points on the plot represent the cumulative number of BD modes that have been discovered by a given time. For example, the second point marks the time at which the second unique BD mode (e.g., BD2) was observed.

  - The Cumulative Number of BD Modes line is fitted to the data points and shows how the cumulative number of discovered BD modes changes with time.

- **MTBF BD Unseen** and **Discovery Rate** show the rate at which new unique BD failure modes are being discovered at any given time. For example, if the MTBF for unseen BD modes is 50 hours, then 2 more unique BD modes are expected to be observed over the next 100 hours. The discovery rate is the failure intensity of the unseen BD modes (i.e., the inverse of the MTBF). In a successful reliability growth test, the MTBF for unseen BD modes will increase over time and the discovery rate will decrease.

- The **Individual Mode** bar charts show two bars for each failure mode. Use the Plot Modes window to choose which failure modes to include in the chart.
  - The Before bars represents the value at the end of the test, before any delayed fixes have been implemented.
  - The After bars are available only for BD modes and represents the value after the delayed fixes have been implemented.
- The **Failure Mode Strategy** pie chart breaks down the overall failure intensity (FI) into six possible categories:
  - The Type A slice represents the FI that is due to failure modes that are ignored (i.e., no fixes will be implemented).
  - The Type BC - Seen slice represents the FI that is due to BC modes that were observed and for which fixes were implemented.
  - The Type BC - Unseen and Type BD - Unseen slices represents the FI that is due to BC and BD modes that were not observed but are estimated from the analysis. If they were observed (e.g., through future testing), fixes would be implemented.
  - The Type BD - Remained and Type BD - Removed slices represent the FI that is due to BD modes that were observed (i.e., they represent the FI due to seen BD modes). The Remained slice represents the portion of FI that is expected to stay in the system because the fixes are not 100% effective, while the Removed slice represents the FI that is expected to be eliminated.

Tip: Weibull++ includes two additional plot utilities you can use across all types of data: the overlay plot, which allows you to compare different data sets or models; 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.

Event Reports

For reliability growth data analysis only.

Once you have calculated the parameters for the Multiple Systems with Event Codes data type or any multi-phase data type, you can choose **Growth Data > Analysis > Event Report** to view a special report in the Results window. The Event Report can consist of up to five tabs, depending on the types of events defined in the data set. The tabs are described next.

Mode Summary Tab

The Mode Summary tab is always displayed and it provides the following information for each unique failure mode in the data sheet:

- **Mode and Classification**: The unique identifier and classification (A, BC or BD) for the failure mode.
- **First Failure**: The time of the first failure due to that mode.
- **Total Failures**: The total number of failures due to that mode.

For the Multiple Systems with Event Codes data type, there is one additional column.

- **Effectiveness Factor**: The effectiveness factor assigned for each BD failure mode.

For multi-phase data types, there are three additional columns. These columns are applicable only for BD failure modes that do not have an implemented fix time (I event) in the data set. For these failure modes, the Effectiveness Factors window records the phase after which the fix will be applied, if any, along with the estimated effectiveness factor. Therefore, the report displays:

- **Nominal EF**: The effectiveness factor that will be used for "nominal" calculations (e.g., nominal projected MTBF, etc.). These calculations represent the best case scenario in which a fix is applied to every BD failure mode, even if the fix is marked "Not Implemented."
- **Phase Fixed**: A number indicates the phase after which the fix was applied (e.g., the number 2 indicates that the fix was applied after the end of Phase 2 but before the start of Phase 3). "Not Implemented" indicates that the fix has not been applied.
- **Actual EF**: The effectiveness factor that will be used for "actual" calculations (e.g., actual projected MTBF, etc.). These calculations take into account which delayed fixes have actually been implemented in the current management strategy. Therefore, if the fix is "Not Implemented," the actual EF will be zero.

Implemented Fixes Tab

The Implemented Fixes tab is present only if the Multiple Systems with Event Codes data set contains BC failure modes, or if the multi-phase data set contains BD failure modes that have an implemented fix time (I event) in the data set. For each applicable failure mode, this tab provides the following information:

- **Mode and Classification**: The unique identifier and classification (A, BC or BD) for the failure mode.

- **First Failure**: The time of the first failure due to that mode.
- **Total Failures**: The total number of failures due to that mode.
- **Before Fix Failures** and **After Fix Failures**: The number of failures due to the mode that occurred before and after the fix was implemented.
- **Before Fix Runtime** and **After Fix Runtime**: The amount of operating time before and after the fix was implemented. In a Multiple Systems with Event Codes data sheet, this is the sum of the operating times for all affected systems.

For the Multiple Systems with Event Codes data type, there is one additional column:

- **Systems With Fixes**: The number of systems for which the fix was implemented.

For multi-phase data types, if the **Allow EF values for BD modes with implemented fixes** option is selected on the on the Calculations page of the Application Setup, there are two additional columns:

- **Effectiveness Factor** and **Comments** display any additional information that you have entered on the Implemented Fixes tab in the Effectiveness Factors window. This information is for your information only and is not considered in the data analysis.

Quality Events and Performance Events Tabs

The Quality Events tab and the Performance Events tabs are present only if the data set contains events of these types. These tabs provide summaries for the events that have been marked with the Q and P event types.

Mode Fix List Tab

The Mode Fix List tab is present only if fixes have been implemented to Multiple Systems with Event Codes data. The tab provides a summary of all systems that have a fix recorded for each failure mode.

Test for Fix Effectiveness

For reliability growth data analysis only.

The Test for Fix Effectiveness window allows you to apply Dr. Crow's "Statistical Test for Effectiveness of Corrective Actions" test for an applicable data set. This test helps you to determine whether the fixes applied during or at the end of a phase have effectively reduced a system's failure intensity.

The utility is available for:

- Any of the multi-phase data types (as long as the parameters have been calculated and the data set contains at least two phases).
- Any data sheet that has been analyzed with the Change of Slope calculation option.

To access the window, choose **Growth Data > Crow Extended > Test for Fix Effectiveness**.

If you are working with a multi-phase data sheet, use the **Phase 1** and **Phase 2** drop-down lists to choose the two phases you want to compare. If you are working with a Change of Slope analysis, Segment 1 is the data before the specified Break Point, while Segment 2 is the data after.

Specify the **Significance Level** (a decimal between 0 and 1) and click **Test**. The utility gives two results:

- The first result compares the average failure intensities of the two phases (or segments). If the average failure intensity during the second is less than during the first (at the specified significance level), the utility displays "Passed." If not, it displays "Failed."
- The second result compares the average failure intensity during the second phase (or segment) with the instantaneous failure intensity at the end of the first. If the average failure intensity during the second is less than the instantaneous failure intensity at the end of the first (at the specified significance level), the utility displays "Passed." If not, it displays "Failed."

Click the **Report** button to see a more detailed summary of the analysis in the Results window. This includes the failure intensities that are calculated for both phases.

Growth Planning Folio and Multi-Phase Plots

For reliability growth data analysis only.

This chapter describes two features that can be used in conjunction with data analyses performed with the Crow Extended and Crow Extended – Continuous Evaluation models. Starting in Version 2019, you can also use the MIL-HDBK 189 model with a continuous growth planning folio.

You can:

- Use the growth planning folio to develop an effective reliability growth test management plan for a single- or multi-phase test.

- Use the multi-phase plot to view the actual test results (analyzed with either the Crow Extended model or the Crow Extended – Continuous Evaluation model) across multiple test phases. If desired, you can also use the plot to compare the actual results against the original test plan.

There is also a case study example that demonstrates how the three tools (test plan, test data analysis and multi-phase plot) can be used together for multi-phase test planning and management.

Growth Planning Folio

For reliability growth data analysis only.

Weibull++'s growth planning folio helps you to plan a single- or multi-phase reliability growth test program that is designed to achieve a specific MTBF or reliability goal. For the most advanced functionality, this typically uses the Crow Extended growth planning model. Starting in Version 2019, the MIL-HDBK 189 planning model is also available in the Continuous Growth Planning folio, allowing you to compare the results between the two models.

Creating a Growth Testing Plan

To create a growth planning folio, right-click the **Test and Planning** folder in the current project explorer and choose either **Add Continuous Growth Planning** (if working with times-to-failure data) or **Add Discrete Growth Planning** (if working with one-shot devices).

Important: Starting in Version 2019, when you add a Continuous Growth Planning folio, you need to select the model that you will be using on the control panel. Your choice will determine the required inputs.

In the data sheet of the folio:

1. **Do you know how the test time will be accumulated across each test phase?** Choose **Yes** if you want to specify the end time (and other details) for each phase of testing. Choose **No** if you want to create a plan for a single test phase with no specific end time.
2. **Planned test phases** is available at the bottom of the data sheet only if you chose Yes to the question in step 1 above. Specify the number of phases and then define each phase:
   - **Phase Name** can be any alphanumeric identifier, up to 50 characters.
   - **Cumulative Time** is the amount of cumulative test time at the end of each phase. Because this represents cumulative test time across all test phases, the number must be higher for each subsequent phase.

- **Average Fix Delay** is the amount of test time from when a failure mode is discovered until a fix is likely to be implemented. (Not available when using the MIL-HDBK 189 model.)

As an example, the following picture shows the inputs for a multi-phase test with 5 test phases that last 1,000 hours each. In the first phase, the planners estimate that delayed fixes (for BD failure modes) will, on average, take 500 hours each. The fix time is expected to increase in later phases.

Planned test phases

Number of Phases		5	
	Phase name	Cumulative Time (hr)	Average Fix Delay (hr)
Phase 1	Phase 1	1000	500
Phase 2	Phase 2	2000	550
Phase 3	Phase 3	3000	600
Phase 4	Phase 4	4000	700
Phase 5	Phase 5	5000	850

If there are a lot of test phases and/or if you have this information available in an external data file, you can click the **Edit Phase Data** icon to open a more flexible spreadsheet for entering/editing these details. The **Import** icon at the bottom of this window allows you to import data from an Excel spreadsheet (*.xls or *.xlsx). Note that if you import from Excel, it will overwrite anything that you may have already entered (i.e., the data from Excel replaces entries rather than being appended).

3. Each growth planning model considers a set of variables; you will select to solve for one of them (i.e., **Which value would you like to calculate?**) based on your inputs for the rest of them (i.e., **Assumed inputs for the reliability growth test plan**.)

   For the Crow-Extended model, the following variables are considered. (Note that the calculations in the continuous growth planning folio are in terms of MTBF, while the calculations in the discrete growth planning folio are in terms of reliability.)

   - **Initial MTBF/Reliability** is the value before the test program begins. It can be determined by some initial testing or through historical information, engineering expertise and/or reliability predictions.

   - **Goal MTBF/Reliability** is the requirement that you need to achieve by the end of the test program.

   - **Growth Potential Design Margin** estimates the amount by which the growth potential value (i.e., the value that would be achieved if you continued testing until all failure modes are observed and corrected according to the current maintenance strategy) exceeds the goal. This value provides a "safety factor" to ensure that the requirement is met. A higher GP design margin means there's a smaller risk that the reliability observed in the field will not meet the requirement, but it also means a more rigorous reliability growth program will be required. Typically, the GP design margin is between 1.2 and 1.5.

- **Average EF** (where EF = effectiveness factor) estimates the fractional decrease in failure intensity you expect to achieve by implementing design fixes. Typically, about 30% of the failure intensity for the failure modes that are addressed will remain in the system after implementing all of the design fixes; therefore, in many reliability growth programs, the average effectiveness factor is 0.7. The value must be greater than 0, and less than 1.
- **Management Strategy Ratio** determines the percentage of the unique failure modes discovered during the test that will be fixed. This is an important variable in reliability growth planning because the management strategy can be changed to address a larger percentage of the discovered failure modes if the goal cannot be reached with the current strategy. Generally, the management strategy is recommended to be above 90%.
- **Discovery Beta** is the rate at which new, unique B failure modes are being discovered during testing. These are the failure modes that will be fixed. In growth planning, it is assumed that most failures will be identified early on, and that new failure modes will be discovered at a decreasing rate as the test proceeds. Therefore, this value must be greater than 0 and less than 1. This input is always required; you cannot solve for the discovery beta.
- **Average Fix Delay** is used only for single-phase plans with no specific end time (i.e., you chose No to the question in step 1 above). This value is the amount of test time from when a failure mode is discovered until a fix is likely to be implemented. If this value is considered, it is required as an input; you cannot solve for the average fix delay.

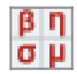
For the MIL-HDBK 189 model, the following variables are considered:

- **Average MTBF over the First Test Phase** is the value before the test program begins. It can be determined by some initial testing or through historical information, engineering expertise and/or reliability predictions.
- **Growth Parameter** is how fast the rate is expected to grow.
- **Final MTBF** is the requirement that you need to achieve by the end of the test program.
- **Test Time for First Phase** is used only for single phase plans with no specific end time (i.e., you chose No to the question in step 1 above). This value is the time it will take for the first phase to be completed.

- **Total Test Time** is used only for single phase plans with no specific end time (i.e., you chose No to the question in step 1 above). This value is the total time the test will take.

Analysis Results

Once you have entered all of the required inputs, calculate the results by choosing **Test and Planning > Analysis > Calculate** or clicking the icon in the control panel.



The result shows the estimated value for the metric you have selected to solve and other results that may be used to evaluate the test plan. You can then use the Plot sheet to visualize the results expected if you implement the test plan, and the QCP to calculate a variety of metrics based on the test plan.

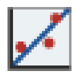
- Additional results in continuous growth planning folios when using the Crow Extended model:
  - **Time at which growth begins** is the estimated time when the first B failure mode (i.e., the first failure mode that will be fixed) is expected to occur. Reliability growth is expected to begin after this time.
  - **Final MTBF** is the expected MTBF at the end of the test, taking into account the average fix delay.
  - **Time to reach goal** is the test time at which the goal MTBF is expected to be reached. The **Actual** value takes into account the average fix delay, and the **Nominal** value does not. If the current test plan will not meet the goal, this field displays "Goal not met."
- Additional results in discrete growth planning folios:
  - **Trial at which growth begins** is the estimated number of trials when the first B failure mode (i.e., the first failure mode that will be fixed) is expected to occur. Reliability growth is expected to begin after this time.
  - **Initial MTrBF** (where MTrBF = mean trials before failure) is the initial estimated system MTrBF.
  - **Growth Potential MTrBF** is the expected MTrBF that could be reached if you continued testing until all B failure modes are observed and fixed according to the management strategy.
  - **Final MTrBF** is the expected MTrBF at the end of the test.

- **Num. trials to reach goal** is the number of trials at which the goal reliability is expected to be reached. The **Actual** value takes into account the average fix delay, and the **Nominal** value does not. If the current test plan will not meet the goal, this field displays "Goal not met."

Growth Planning Folio Plots

For reliability growth data analysis only.

The Plot sheet in the growth planning folio shows a variety of plots that display the results that are expected if you implement the plan. To view the growth planning folio plots, choose **Test and Planning** > **Analysis** > **Plot** or click the icon on the control panel.



The following is a description of the different types of plots you can create for growth planning. For general information on working with plots, see Plot Utilities.

PLOT TYPES

The following plots show the expected value of the specified metric across all planned test phases:

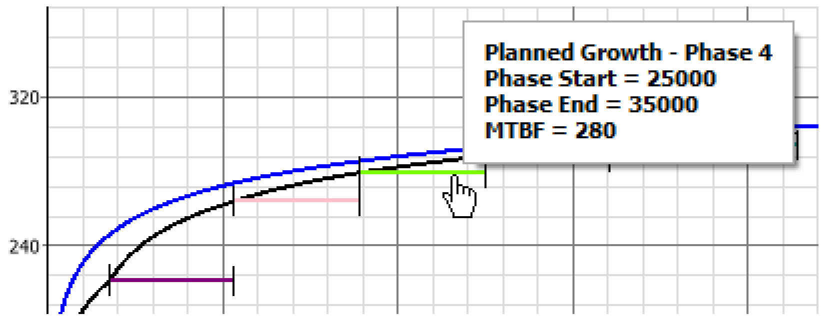
- **MTBF vs. Time** (available only for the continuous growth planning folio)
- **Reliability vs. Time** (available only for the discrete growth planning folio)
- **MTrBF vs. Time** (available only for the discrete growth planning folio)
- **Failure Intensity vs. Time**

For the plots above, you can use the check boxes in the control panel to select which of the following elements will be displayed in the plot:

For the Crow Extended model, the options are:

- **Nominal Idealized** and **Actual Idealized** lines (also called *idealized growth curves*) show the overall characteristic pattern for the reliability growth across all phases of testing. The *nominal* curve is the best case scenario, which assumes that the average fix delay is zero. The *actual* curve takes into account the average fix delay. If fixes are not incorporated instantaneously, the actual line will show slower growth compared to the nominal line.
- **Planned Growth** lines display information for each phase of testing (and thus are available only when you have provided information for each phase). These horizontal lines mark the

time when each phase will begin and end, while the vertical position of each line shows the expected value of the metric at the beginning of the phase. You can point to the line to see the specific values. (Note that in the continuous growth planning folio, you can show confidence bounds on the planned growth lines, but only when you have at least 3 test phases specified on the folio's data sheet.)



- The **Termination Line** shows the end time of the final test phase.
- The **Goal** line shows the target value that you hope to achieve by the last phase of testing.

For the MIL-HDBK 189 model, the options are:

- The **Idealized Growth Curve** line shows the overall characteristic pattern for the reliability growth across all phases of testing.
- If the test phase settings are specified, the **Average per Test Phase** line shows either the average MTBF for each test phase on the MTBF vs. Time plot or the average failure rate for each test phase on the Failure Intensity vs, Time plot.

For the Crow Extended model, the following plots are also available:

- **Cumulative Number of B Modes** shows the cumulative number of observed unique B failure modes versus time. These are failure modes that have been discovered and will be fixed.
- **MTBF B Unseen** shows the mean time between failures due to B failure modes that have not yet appeared in testing. These are failure modes that are anticipated based on the analysis and will be fixed when/if they are discovered.
- **Discovery Rate** shows the rate at which unique B failure modes are being discovered. In other words, it is the failure intensity of the unseen B failure modes.

Growth Planning Folio QCP

For reliability growth data analysis only.

After you have used the data sheet in the growth planning folio to create a reliability growth program test plan, you can use the Quick Calculation Pad (QCP) to calculate a variety of results that are expected if you implement the plan.

To open the QCP for growth planning folio calculations, choose **Test and Planning > Analysis > Quick Calculation Pad** or click the icon on the control panel.

To perform a calculation, click one of the buttons on the left side of the window and enter any required inputs in the **Input** area, then click **Calculate**. For more detailed information about all the options available in the QCP, see Quick Calculation Pad (QCP).

CONTINUOUS GROWTH PLANNING FOLIO

For the Crow Extended model, the options are:

- **Nominal MTBF** and **Nominal Failure Intensity** calculations reflect the best case scenario. They assume that the average fix delay is zero.
- **Actual MTBF** and **Actual Failure Intensity** take into account the average fix delay. If fixes are not incorporated instantaneously, the actual value will show slower growth compared to the nominal value.
- **Planned MTBF** and **Planned Failure Intensity** return the expected MTBF or failure intensity at the beginning of each test phase. To perform this calculation, you must select the phase from a drop-down list. Note that, unlike the other calculations, you can calculate confidence bounds on the planned growth values, but only when you have at least 3 test phases specified on the folio's data sheet.
- **Discovery Rate** is the rate at which new BD failure modes are being discovered at the specified time.
- **MTBF B Unseen** is the mean time between failures due to B failure modes that have not yet appeared in testing but are estimated from the analysis. These are failure modes that would be fixed if they were discovered.
- **Cum. Num. B Modes** is the cumulative number of B failure modes that are expected to have been seen during testing by the specified time (i.e., failure modes that will be fixed).

For the MIL-HDBK 189 model, the options are:

- **Idealized Growth MTBF** and **Idealized Growth Failure Intensity** calculations reflect the best case scenario.
- **Average MTBF** and **Average Failure Intensity** return the expected MTBF or failure intensity at the beginning of each test phase. To perform this calculation, you must select the phase from a drop-down list. These options are available only if the accumulated test time is known.

DISCRETE GROWTH PLANNING FOLIO

- **Nominal Reliability**, **Nominal MTrBF** and **Nominal Failure Intensity** calculations reflect the best case scenario. They assume that the average fix delay is zero.
- **Actual Reliability**, **Actual MTrBF** and **Actual Failure Intensity** take into account the average fix delay.
- **Planned Reliability**, **Planned MTrBF** and **Planned Failure Intensity** return the expected value of the metric at the beginning of each test phase. To perform this calculation, you must select the phase from a drop-down list.
- **Discovery Rate** is the rate at which new B failure modes are being discovered at the specified time.
- **MTBF B Unseen** is the mean time between failures due to B failure modes that have not yet appeared in testing but are estimated from the analysis. These are failure modes that would be fixed if they were discovered.
- **Cum. Num. B Modes** is the cumulative number of B failure modes that are expected to have been seen during testing by the specified time (i.e., failure modes that will be fixed).

Multi-Phase Plot

For reliability growth data analysis only.

The multi-phase plot displays key analysis results from data analyzed with the Crow Extended model or Crow Extended – Continuous Evaluation model so you can see how the demonstrated, projected and growth potential MTBF or failure intensity changes across multiple analysis points and/or test phases.

If desired, you can also link the plot to a growth planning folio so you can see how the actual test results compare to the plan, and determine if it’s necessary to make adjustments in subsequent test phases in order to meet your reliability growth goals.

To create a multi-phase plot, choose **Home > Insert > Multi-Phase Plot**.

Multi-Phase Plot Wizard

For reliability growth data analysis only.

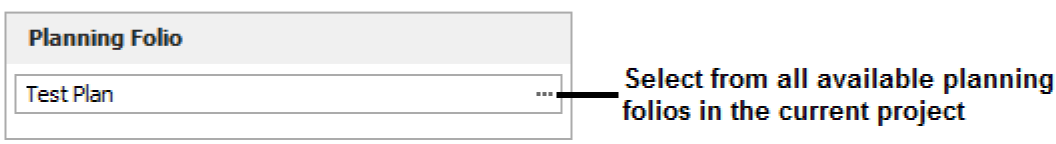
When you create a new multi-phase plot or click the **Multi-Phase Plot Setup** button in an existing plot, the Multi-Phase Plot Wizard allows you to specify the data source(s) for the plot. The test data displayed in the plot can come from:

- One **Multi-Phase** data sheet that includes all of the analysis points and phase data defined together in a single data source. [See details below.]

  or

- **Multiple Data Sheets** that each contain the data from one particular analysis point or phase. This requires you to assign a specific data sheet to each analysis point and phase you want to display in the plot. [See details below.]

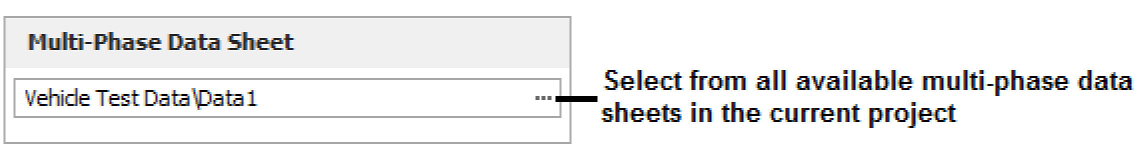
After you have defined the test data source(s), the wizard also allows you to select an applicable growth planning folio, if desired. Use this option if you want to use the plot to compare the test results against what was expected based on the reliability growth program plan.



Click **Finish** when you're ready to save the changes and create or return to the Multi-Phase Plot window.

OPTION 1: GETTING THE TEST DATA FROM ONE MULTI-PHASE DATA SHEET

If you select **Multi-Phase** and proceed to the next page in the wizard, you will be prompted to select a single analyzed multi-phase data sheet that contains all of the test phases and analysis points already defined in a single data sheet.

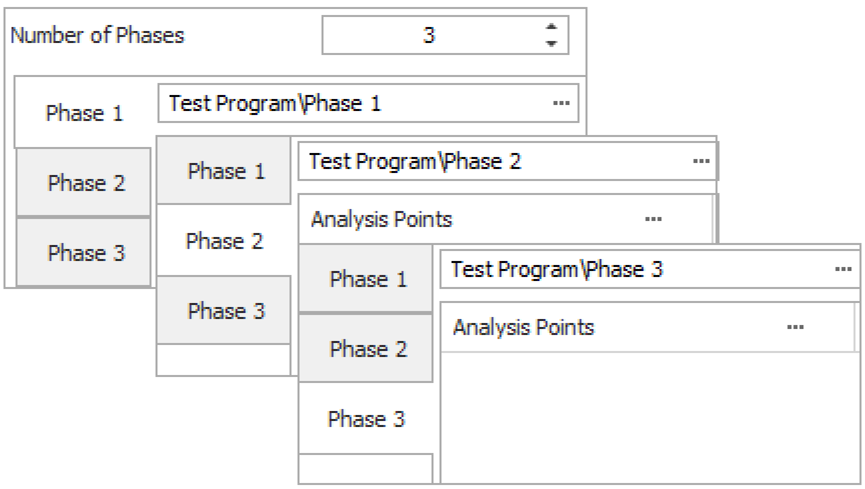


OPTION 2: GETTING THE TEST DATA FROM MULTIPLE DATA SHEETS

If you select **Multiple Data Sheets** and proceed to the next page in the wizard, you will be prompted to select the separate data sheets that contain the data for each analysis point and phase you wish to display in the plot. The data sheets must be calculated with either the Crow Extended or Crow Extended – Continuous Evaluation model, and each sheet can be used only once in the same plot.

One Sheet per Phase

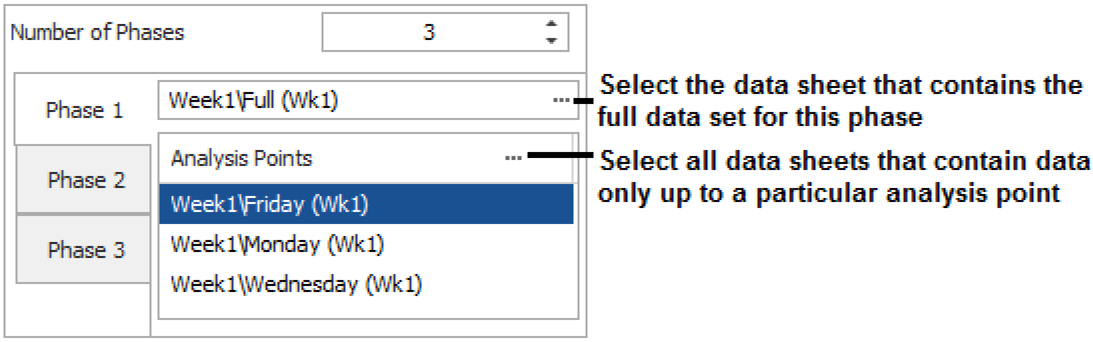
If you don't wish to include analysis points in the plot, you can simply specify the number of phases at the top of the window and then select one data sheet on the tab for each phase.



Separate Sheets for Analysis Points

If you want to include analysis points, there must be a separate data sheet that contains the data for each analysis point, in addition to the data sheets that contains all of the data for each entire test phase.

For example, in the following picture, the data from the first week of testing were recorded and analyzed in four separate data sheets (in which each sheet contains all of the data up to a specific point in time).



In other words:

- "Monday (Wk1)" contains the data from Sunday and Monday.
- "Wednesday (Wk1)" contains all of the data from Monday's data sheet, plus the additional data collected on Tuesday and Wednesday.
- "Friday (Wk1)" contains all of the data from Wednesday's data sheet, plus the additional data collected on Thursday and Friday.
- "Full (Wk1)" contains all of the data from Friday's data sheet, plus the additional data collected on Saturday. This data sheet contains all of the data from phase 1.

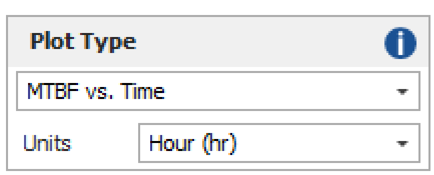
To select the data sheets for the analysis points, click the **(...)** button in the Analysis Points bar. In the Select Data Sheets window that opens, select the check box for all the data sheets that contains analysis point data for the current phase.

To remove a data sheet from the Analysis points list, you can return to the Select Data Sheets window and clear the check box, or simply double-click the data sheet name in the list.

Multi-Phase Plot Control Panel

For reliability growth data analysis only.

In the Multi-Phase Plot window’s control panel, the **Plot Type** drop-down list allows you to select whether the plot will display the mean time between failures (MTBF) or failure intensity, while the **Units** drop-down list allows you to select the units, which are defined at the database level.



The **Multi-Phase Plot Setup** button opens the Multi-Phase Plot Wizard, which allows you to view or change the data source(s) that are displayed in the plot.

The **Show** area at the bottom of the control panel allows you to select which results and other elements will be shown in the plot.

- The options under the **Phases** and **Analysis Points** headings are based on the test data from each phase.
  - The **Demonstrated** values reflect the reliability before any delayed fixes are implemented.
  - The **Projected** values reflect the expected jump in reliability after delayed fixes are implemented.
  - The **Growth Potential** values are the maximum system reliability that can be attained with the current system design and reliability growth management strategy (i.e., all B failure modes have been found and fixed).
  - The **Estimated Parameters Line** is applicable only for the test phases (not analysis points). It shows MTBF or failure intensity vs. time within each individual test phase.
  - The **Phase Termination Line** is applicable only for the test phases (not analysis points). It provides a long vertical line to more clearly mark the end of each test phase.
- The options under the **Planning** heading are based on the growth planning folio that's been linked to the plot, if any, and on the planning model used with the planning folio.

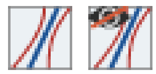
  These options are available if the planning folio uses the Crow Extended model.

  - The **Nominal Idealized** and **Actual Idealized** lines (also called *idealized growth curves*) show the overall characteristic pattern for the reliability growth across all phases of testing. The *nominal* MTBF/FI is the best case scenario, which assumes that the average fix delay is zero. The *actual* MTBF/FI takes into account the average fix delay.
  - The **Planned Growth** lines mark the time when each phase will begin and end, while the vertical position of each line shows the expected MTBF or failure intensity at the beginning of the phase.
  - The **Goal Value** line shows the target MTBF or failure intensity that you hope to achieve by the last phase of testing.

  These options are available if the planning folio uses the MIL-HDBK 189 model.

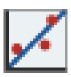
- The **Idealized Growth Curve** line shows the overall characteristic pattern for the reliability growth across all phases.
- If the test phase settings are specified, the **Average per Test Phase** line shows the calculated MTBF at each phase.

If you want to hide/display confidence bounds, choose **Plot > Confidence Bounds > [Show/Hide] Confidence Bounds**.



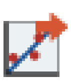
The remaining options in the control panel are similar to other plot sheets throughout the application. Specifically:

The folio tools are arranged on the left side of the control panel.

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

 **Plot Setup** opens the Plot Setup window, which allows you to customize most aspects of the plot including the titles, colors, sizes, etc.

 **RS Draw** launches ReliaSoft Draw, which allows you to view the plot in greater detail, add annotations and modify selected plot elements.

 **Export Plot Graphic** 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.

The **Options** area provides:

- **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** maintains the ratio of the horizontal size to the vertical size of the plot graphic when you resize the plot sheet.

In the **Scaling** area:

- The **X** and **Y Scaling** boxes show 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.

Example: Multi-Phase Test Planning and Management

For reliability growth data analysis only.

*The data set used in this example is available in the example database installed with the software (called "Weibull21_ReliabilityGrowth_RGA_Examples.rsgz21"). 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 "Multi-Phase Test Planning and Management."

A manufacturer is developing a small electric vehicle for indoor use, such as airports. The goal MTBF for the vehicle is 300 operating hours. 20 developmental prototypes are going to be tested for 3,000 hours each, for a total of 60,000 hours. The first phase of testing is planned to end at 5,000 hours of cumulative test time; the second phase will end at 15,000 hours; the third phase will end at 25,000 hours; the fourth will end at 35,000 hours; the fifth will end at 45,000 hours and the sixth will end at 60,000 hours.

The analysts wish to:

- Create a test plan that will achieve the desired MTBF goal.
- Use the Crow Extended - Continuous Evaluation model to analyze the data at the end of each phase of testing.
- Use the multi-phase plot to track the actual progress against the original test plan.

Continuous Growth Planning Folio

The first step of creating an overall reliability growth program plan is to set an idealized growth curve and planned MTBF goal at each stage of the program. This will help determine the management strategy that may be needed to achieve the final MTBF goal. (The reliability growth program plan for this example is located in the growth planning folio called "Test Plan.") The inputs on the data sheet are shown next.

Design a reliability growth test plan			
Do you know how the test time will be accumulated across each test phase?			
Answer	Yes		
Which value would you like to calculate?			
Answer	Initial MTBF		
Assumed inputs for the reliability growth test plan			
Goal MTBF	300		
Growth Potential Design Margin	1.3		
Average EF	0.7		
Management Strategy Ratio	0.95		
Discovery Beta	0.71		
Planned test phases			
Number of Phases	6		
	Phase name	Cumulative Time (hr)	Average Fix Delay (hr)
Phase 1	Phase 1	5000	3000
Phase 2	Phase 2	15000	5000
Phase 3	Phase 3	25000	6000
Phase 4	Phase 4	35000	7000
Phase 5	Phase 5	45000	8000
Phase 6	Phase 6	60000	9000

You can see that the cumulative test time at the end of each test phase has been defined under the **Planned test phases** heading, along with the average fix delay for failure modes that are discovered in each phase. In this case, the team expects that it will be easier to implement fixes earlier in the process; therefore, the estimated fix delay increases in each phase. For example, the fix for a failure mode discovered during Phase 1 is expected to be implemented approximately 3,000 test hours later; whereas a failure mode discovered during Phase 5 might not be able to be fixed until approximately 8,000 test hours later because the design is more mature at that point.

The growth planning folio also shows the inputs for the Crow Extended growth planning model. In this case, the analysts have selected to calculate for the **Initial MTBF** that the product must have before reliability growth testing begins in order to achieve the **Goal MTBF** of 300 hours by the end of the test program, given the following growth management strategy:

- A **GP Design Margin** of 1.3, which indicates that the growth potential MTBF (i.e., the maximum achievable MTBF if you continue until all modes are observed and corrected according to the current maintenance strategy) must equal 130% of the goal MTBF. This value provides a "safety factor" to ensure that the reliability requirement is met.
- An **Average EF** of 0.7, which indicates that the team expects that the fixes applied to failure modes discovered during testing will remove about 70% of the failure intensity due to those modes.

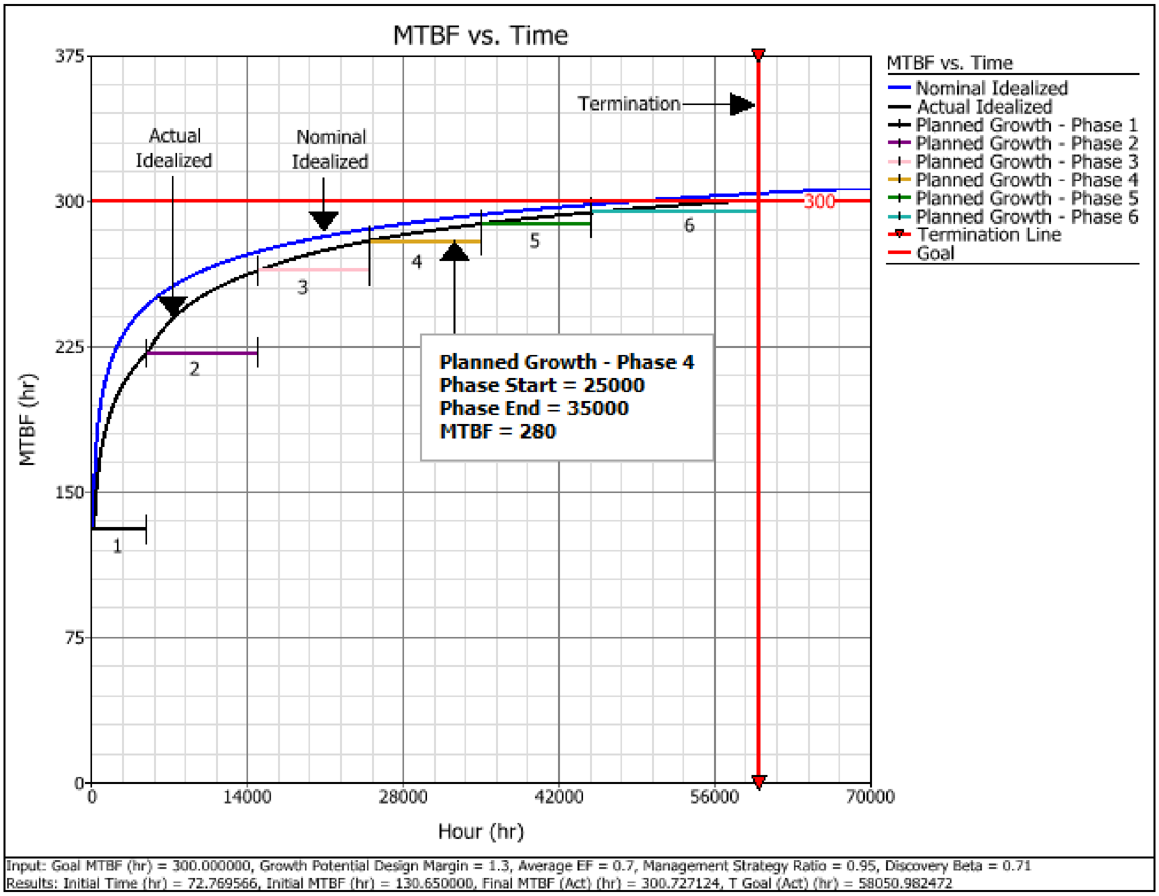
- A **Management Strategy** of 0.95, which indicates that the team plans to fix about 95% of the failure modes that are discovered during testing (while about 5% may be classified as A modes, which are not fixed due to technical, financial or other reasons).
- A **Discovery Beta** of 0.71, which indicates that the inter-arrival times between unique failure modes discovered during testing will become larger as the test progresses.

As shown next, the results indicate that the initial MTBF must be about 131 hours when the first test phase begins. Furthermore, the plan estimates that the MTBF goal will be achieved after about 58,000 hours of test time.

Results	
Time at which growth begins (hr)	72.769566
Initial MTBF (hr)	130.650000
Final MTBF (Act) (hr)	300.727124
Time to reach goal (Actual) (hr)	58050.982472
Time to reach goal (Nominal) (hr)	49180.916974

Click the **Plot** icon to see a visual representation of the results that are expected in each phase of testing. The following picture contains annotations to help you identify the following:

- The **Actual Idealized** and **Nominal Idealized** lines show the overall characteristic pattern for the reliability growth across all phases of testing.
  - The *actual* MTBF takes into account the average fix delay. If fixes are not incorporated instantaneously, the actual line will show slower growth compared to the nominal line.
  - The *nominal* MTBF is the best case scenario, which assumes that the average fix delay is zero.
- The horizontal **Planned Growth** lines mark the duration of each test phase, as well as the MTBF that is expected to be achieved by the beginning of the phase. For example, the planned growth line for Phase 4 begins at 25,000 hours, ends at 35,000 hours and shows that the MTBF is planned to be about 280 hours by the beginning of that phase.
- The **Goal** line marks the MTBF the analysts plan to achieve.
- The **Termination** line marks the end of the last test phase at 60,000 hours of cumulative test time.



Analyze the Test Data with the Continuous Evaluation Model

When the testing began, the team used the Multi-Phase Failure Times data type in Weibull++ to record the data. For demonstration purposes, the sample project contains separate folios with the data up to the end of each individual test phase. As an example, you can open the "Test Data - Through Phase 2" folio, which contains all of the data through the end of the second phase. The last twenty data points from this folio are shown next.

	Event	Time to Event (hr)	Classification	Mode
79	F	12147	BC	104
80	I	12500	BD	5005
81	F	12664	BD	5006
82	F	12720	BD	5003
83	F	12932	BD	5001
84	I	12935	BD	5001
85	AP	13000		
86	F	13064	A	1
87	F	13289	BC	102
88	F	13676	BD	5017
89	F	13983	BD	5009
90	AP	14000		
91	F	14133	BD	5004
92	F	14333	BD	5007
93	F	14395	BD	5003
94	F	14633	BD	5016
95	F	14733	BD	5013
96	F	14736	BD	5006
97	F	14933	A	1
98	F	14998	BD	5010
99	PH	15000		

As you can see, in addition to recording the time and mode classification for each failure event (**F**)—where A = no fix, BC = immediate fix before testing resumes and BD = delayed fix—this data type also allows the team to identify:

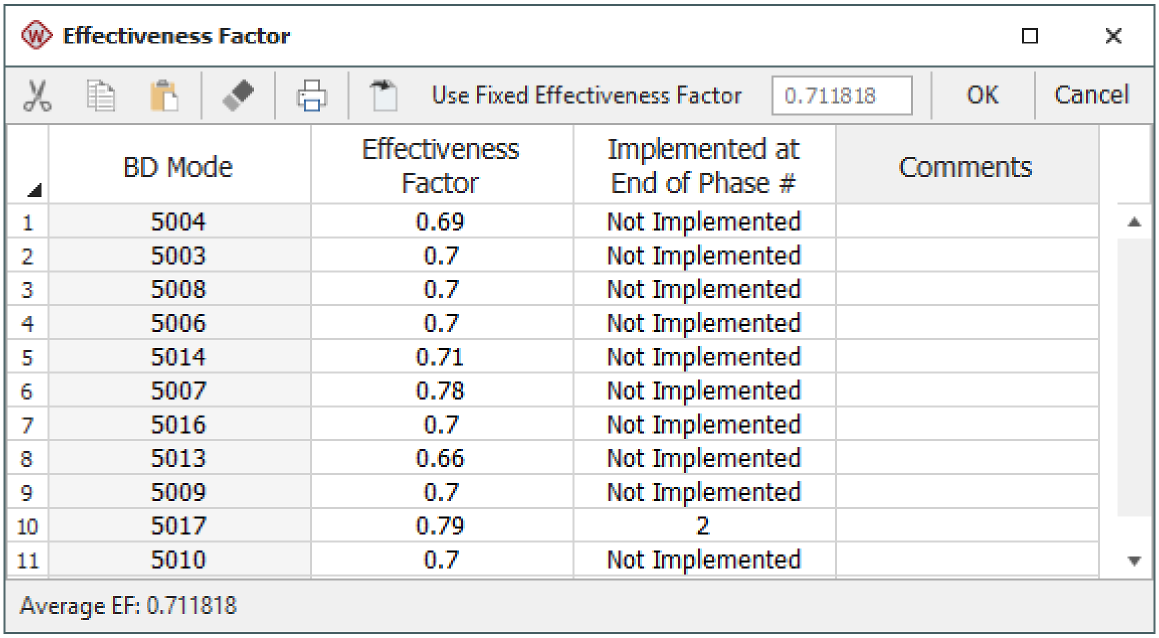
- **PH**: The end time of each phase, which allows for the data from multiple test phases to be entered and analyzed together in the same data sheet.
- **I**: The specific time when delayed fixes are implemented during testing. For example, on row 84 in this picture, you can see that a fix was applied at 12,935 hours for mode 5001.
- **AP**: Specific analysis points at which the team wants to calculate results in order to track the progress of the test plan.

(Although they are not shown in this picture, this data type also allows you to mark events as performance (**P**) or quality (**Q**) issues that can be excluded from the analysis if desired, You can also mark any event with an **X** to exclude it from the analysis.)

For the BD (delayed fix) failure modes that do not have a specific fix time (I event) recorded in the data set, the Effectiveness Factors window allows the team to specify when the fix was (or will be) implemented. To open the window, choose **Growth Data > Crow Extended > Effectiveness Factors**.

EF

As you can see, the fix for mode 5017 was implemented at the end of the second phase, and it's expected to have an effectiveness factor of 0.79 (i.e., only 79% of the mode's failure intensity is expected to be removed because the fix is not perfectly effective). The remaining modes have not yet been fixed before the start of Phase 3; they may be fixed during or between later test phases.



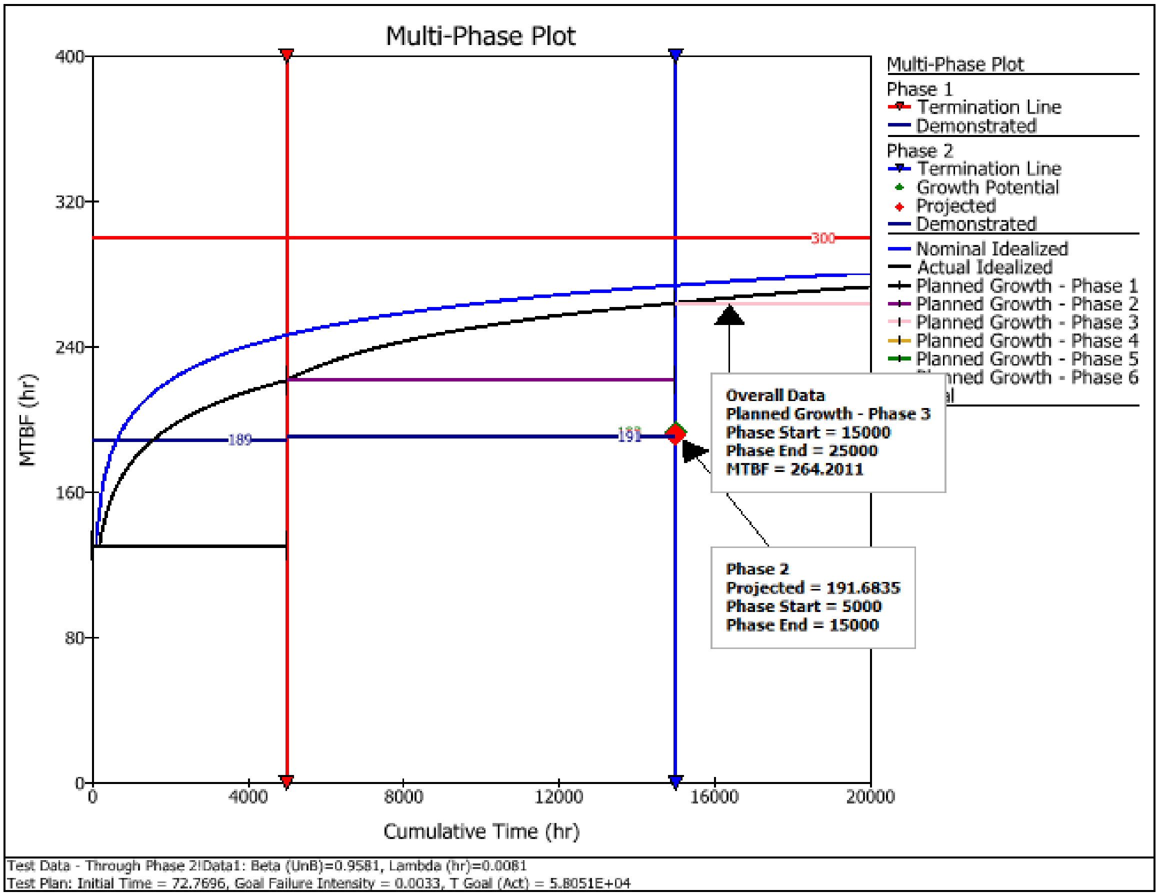
	BD Mode	Effectiveness Factor	Implemented at End of Phase #	Comments
1	5004	0.69	Not Implemented	
2	5003	0.7	Not Implemented	
3	5008	0.7	Not Implemented	
4	5006	0.7	Not Implemented	
5	5014	0.71	Not Implemented	
6	5007	0.78	Not Implemented	
7	5016	0.7	Not Implemented	
8	5013	0.66	Not Implemented	
9	5009	0.7	Not Implemented	
10	5017	0.79	2	
11	5010	0.7	Not Implemented	

Average EF: 0.711818

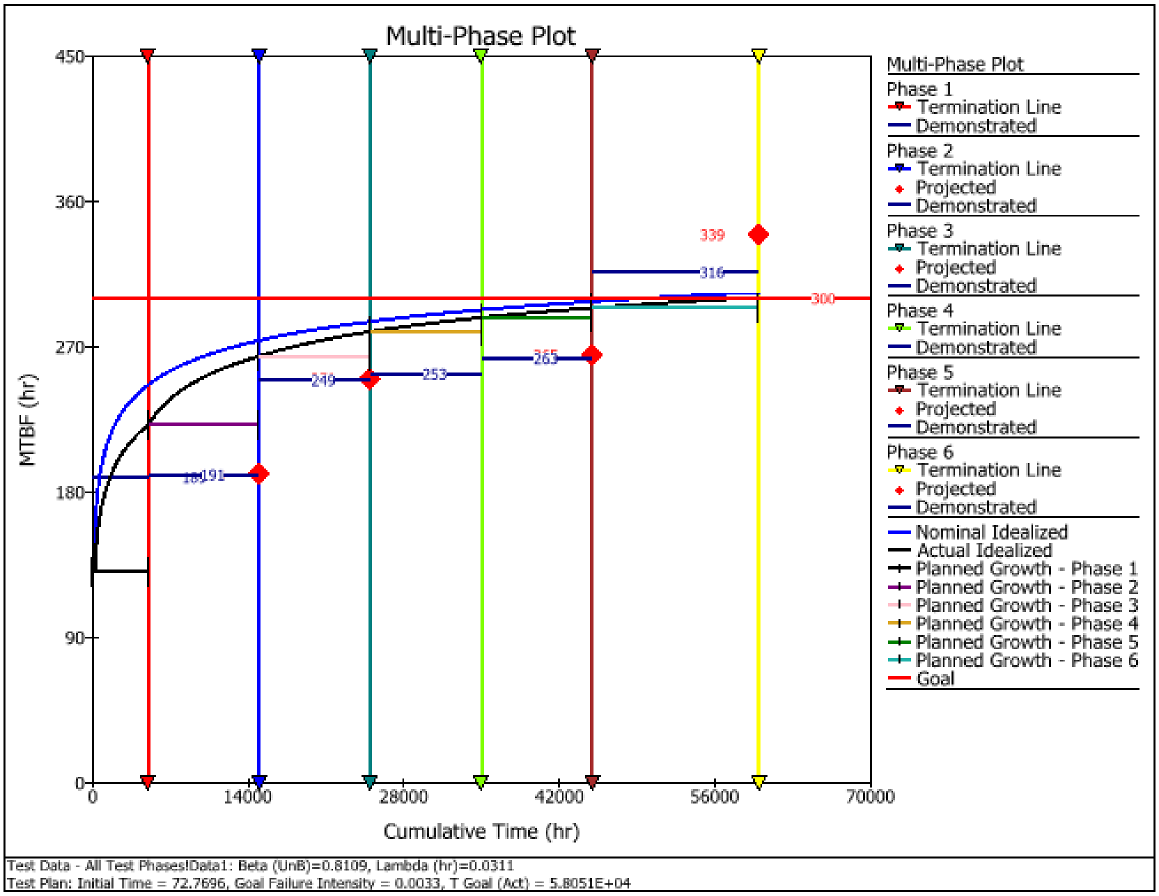
Multi-Phase Plot

As the testing progressed, the team used Weibull++'s multi-phase plot to track the progress against the original test plan. As an example, you can open the "Through Phase 2" multi-phase plot. This plot shows that the MTBF expected after the delayed fixes are implemented after Phase 2 (194.0895 hours) is less than the MTBF planned for the beginning of Phase 3 (264.2011 hours). The team can use this information to determine whether the growth management strategy needs to be adjusted in subsequent phases.

Note: The multi-phase plots in this example are based on analyses that use the maximum likelihood estimation (MLE) method to calculate the parameters, which is known to produce a biased value for beta. In this example, all these plots are based on unbiased estimates of beta. If the folio is not configured to remove this bias, you will be prompted to change the option in the Settings tab of the Item Properties window (**Project > Current Item > Item Properties**). To ensure the unbiased beta is always calculated for all new folios, use the RGA Growth Data Folios page of the Application Setup window to select the option.



Finally, if you open the "All Test Phases" multi-phase plot, you can see that the demonstrated MTBF eventually exceeded the goal, indicating a successful reliability growth plan.



Repairable Systems Analysis

For reliability growth data analysis only.

Some of the models in Weibull++ can be used to analyze data from repairable systems operating in the field under typical customer usage conditions. Such data might be obtained from a warranty system, repair depot, operational testing, etc.

Specifically, you can use the power law or Crow-AMSAA (NHPP) models for repairable system analysis based on the assumption of *minimal repair* (i.e., the system is "as bad as old" after each repair) to calculate metrics such as the expected number of failures, rate of wearout or the optimum time to replace or overhaul a system to minimize life cycle costs.

The ReliaWiki resource portal has more information on repairable systems analysis at http://www.reliawiki.org/index.php/RGA_Models_for_Repairable_Systems_Analysis.

Tip: For even more advanced repairable systems analysis capabilities, you may wish to use ReliaSoft's BlockSim software. With BlockSim, you can use discrete event simulation to perform a wide variety of reliability, availability, maintainability and supportability (RAMS) analyses for repairable systems.

Data Types and Models for Repairable Systems Analysis

For reliability growth data analysis only.

There is a choice of two data sheets for analyzing data from repairable systems. In general:

- Use the Repairable data type for analyzing the individual failure times for multiple repairable systems operating in the field. The analysis models the number of individual system failures vs. system time.
- Use the Fleet data type for analyzing the failure times for multiple repairable systems from a fleet (rather than individual system) perspective. The analysis groups the data and models the number of fleet failures vs. fleet time.

If you wish to perform repairable systems analysis based on the assumption of minimal repair (i.e., the system is "as bad as old" after each repair), use the **Power Law** model in a Repairable data sheet. If the power law model does not provide a good fit, you can transfer the data to a Fleet data sheet and use the **Crow-AMSAA (NHPP)** instead.

For information about using the Crow Extended model for fielded systems, see Crow Extended.

QCP Results and Plots for Repairable Systems Analysis

For reliability growth data analysis only.

Weibull++ includes a Quick Calculation Pad (QCP) for computing useful metrics, and multiple plots that allow you to visualize the results of your analyses. The following calculations and plots are available when you are performing repairable systems analysis with the power law or Crow-AMSAA (NHPP) model.

Note: When you analyze data from fielded systems, Weibull++ combines the data to create a single superposition system or cumulative timeline. Any plots generated for the combined data set and subsequent analyses via the Quick Calculation Pad will be based on the resulting system/timeline. See Fielded Data for more information about how the software combines the data for analysis.

QCP Calculations

To open the Quick Calculation Pad (QCP), choose **Growth Data > Analysis > Quick Calculation Pad** or click the icon on the control panel.

This section describes the types of calculations you can perform in the QCP for repairable systems analysis. For general information on how to use the QCP, see Quick Calculation Pad (QCP). For information about the metrics and plots available for other types of analyses that you can perform in Weibull++, see QCP Calculations and Plots for Traditional RGA and QCP Calculations and Plots for Crow Extended.

When you are performing repairable systems analysis with a Repairable or Fleet data sheet, the following calculations will be available:

- Two values can be calculated for either the **MTBF** or **Failure Intensity**:
  - The **Instantaneous** value is the MTBF/FI over a small interval *dt* that begins at a specified time. For example, an instantaneous MTBF of 5 hours after 100 hours of operation means that, over the next small interval *dt* that begins at 100 hours, the average time between failures will be 5 hours.
  - The **Cumulative** value is the MTBF/FI from time = 0 up to a specified end time. For example, a cumulative MTBF of 5 hours from 0 to 100 hours means that the average time between failures was 5 hours over the 100-hour period.
- The **Time Given** option allows you to calculate the mission duration given any of the following metrics:
  - Cumulative MTBF
  - Instantaneous MTBF
  - Cumulative failure intensity (FI)
  - Instantaneous failure intensity (FI)
- **Number of Failures** is the cumulative number of failures that are expected to occur by a specified time, based on the fitted model.

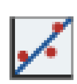
- **Expected Fleet Failures** is the number of failures that are expected to occur for all systems by a specified time. This option is available only for fielded repairable data and for fielded fleet data.

The following calculations are available only for the Repairable data sheet (which models the number of individual system failures vs. system time, rather than the number of fleet failures vs. fleet time).

- **Reliability** is the probability that a system that has already operated for a certain amount of time will successfully complete a mission of a certain duration. Enter the amount of time that the system has already operated for in the **System Age** field, and enter the duration of the mission in the **Mission Time** field. For example, if the system age = 2 years and mission time = 1 year, the QCP will calculate the reliability for one additional year of operation after the system has already operated for two years (i.e., conditional reliability). To calculate the standard (i.e., non-conditional) reliability, enter 0 for the system age.
- **Prob. of Failure** is similar to reliability, except that it is the probability that the system will *not* complete the mission. Thus, it is equal to 1 - Reliability.
- **Mission Time** is the duration of a mission that a system can complete while still maintaining a certain reliability. Enter the required reliability in the **Reliability** field, and if you will assume that the system has already operated for a certain time prior to the mission, enter the prior operating time in the **System Age** field. For example, if system age = 1 year and reliability = 0.9, then the QCP will calculate the longest mission that the system can complete while maintaining a reliability of at least 90%, under the assumption that the system that has already operated for one year.
- **Optimum Overhaul** is the best time to use between regularly scheduled renewals of the system. By overhauling the system at the right times, you can minimize the total life cycle cost. The optimum overhaul time is calculated using the average **Repair Cost** and the **Overhaul Cost**. For example, if it costs $1,000 to repair the system and $3,000 to overhaul it, the QCP can calculate the optimum time to overhaul the system (e.g., every 5 years).

Plots

You can create plots by choosing **Growth Data > Analysis > Plot** or by clicking the icon on the control panel.

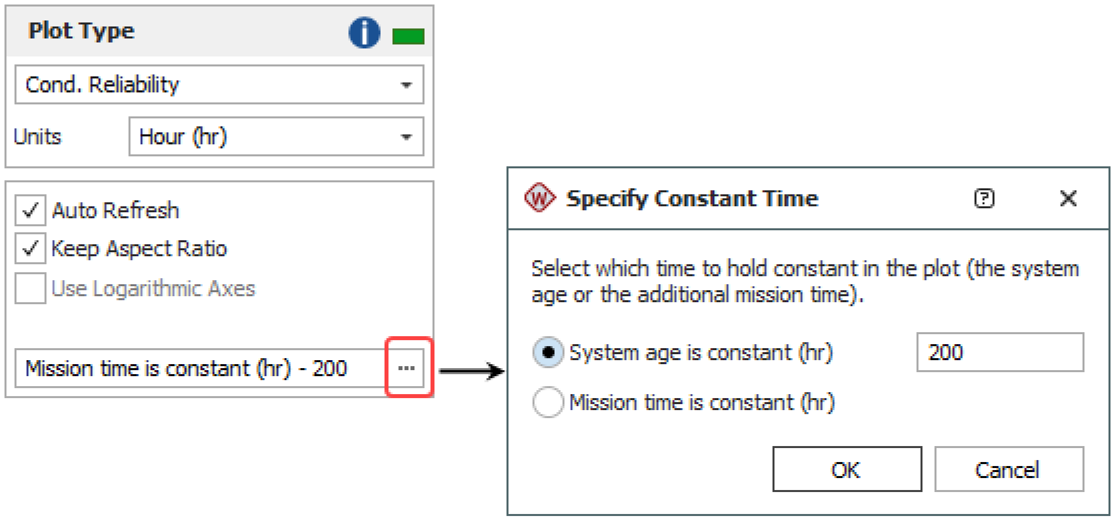


This section describes the types of plots you can create for repairable systems analysis. The scaling, setup, exporting and confidence bounds settings are similar to the options available for all other reliability growth analysis plot sheets. For more information on these common options, see Plot Utilities.

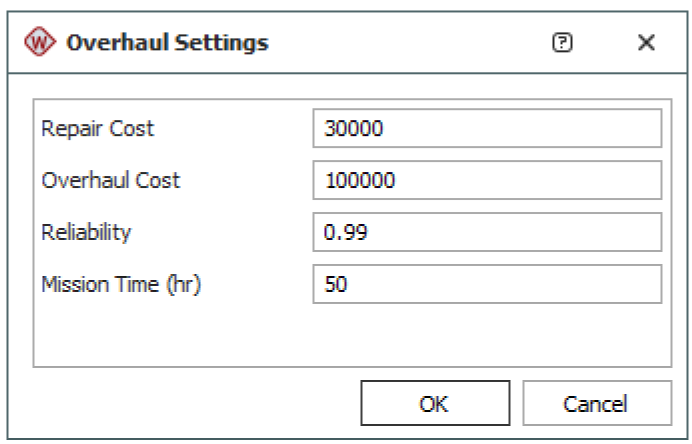
When you are performing repairable systems analysis with a Repairable or Fleet data sheet, the following plots may be available.

- **Cumulative Number of Failures** shows the total number of failures versus time. Data points on the plot represent the cumulative number of failures that have been reported by a given time (e.g., the second point marks the time at which the second failure was observed). The Expected Failures line is fitted to the data points and serves as an empirical goodness-of-fit test for the model.
- The **[Value] vs. Time** plots show how the value increases, decreases or remains constant over time. The points represent the actual failure times in the data set and the plot includes one line for the instantaneous value and one line for the cumulative value.
- **System Operation** shows the failure times of each system plotted on separate lines. The last line is the timeline for the representative system (i.e., superposition system or cumulative timeline), which is used to evaluate all the failures that occurred during the observation period. To learn more about how the failure times are combined, see Fielded Data.
- **Conditional Reliability/Unreliability** is available only when you are using the Repairable data sheet. These plots show the reliability/unreliability versus system age or mission time.
  - If you choose to hold the system age constant, the plots will show the reliability/unreliability for different mission times. For example, assuming that the system has already operated for 100 hours, the plot can show the reliability for the next 10 hours, 20 hours, 30 hours, etc.
  - If you choose to hold mission time constant, the plots will show the reliability/unreliability for different system ages. For example, assuming that the mission will be 100 hours, the plot can show the reliability for a system that has already operated for 10 hours, 20 hours, 30 hours, etc.

  To specify which value will be held constant, click the **[...]** button on the control panel. In the window that appears, select the type of metric that will be held constant, as shown next. Then enter the constant value in the input field.



- **Optimum Overhaul** plots are only available when Beta > 1 and only when you are using the Repairable data sheet. These plots show two values; economic life and useful life. Economic life is the overhaul time that minimizes cost (same as the QCP calculation). Useful life takes into account the reliability requirement for the given mission duration. Click **Overhaul Settings** on the plot control panel to change the parameters.



Tip: Weibull++ includes two additional plot utilities you can use across all types of data: the overlay plot, which allows you to compare different data sets or models; 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.

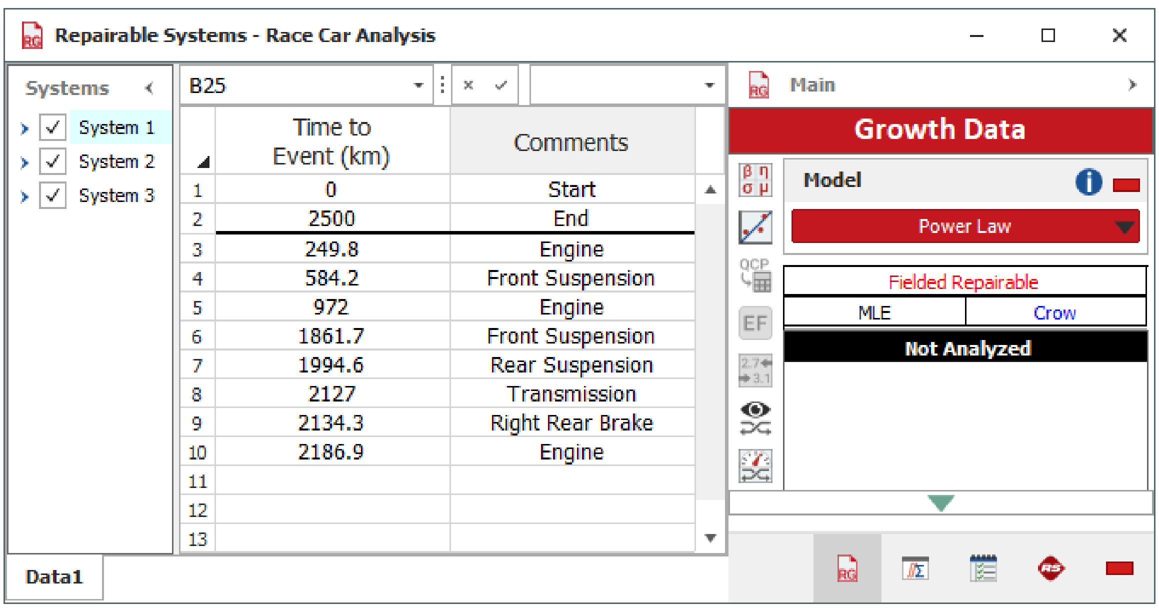
Example Using Power Law Model for Repairable Systems Analysis

For reliability growth data analysis only.

*The data set used in this example is available in the example database installed with the software (called "Weibull21_ReliabilityGrowth_RGA_Examples.rsgz21"). 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 "Repairable Systems - Race Car Analysis."

Failure times were recorded for three Formula 1 race cars operating in the field. Each car failed multiple times during the observation period. When a failure occurred, the failure time was recorded, along with the name of the failed component that was replaced. For example, the data set for the first car is shown next using the Advanced Systems View.



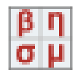
According to this data set, the first car was observed during its first 2,500 kilometers of operation. The engine was replaced when the car failed after 249.8 kilometers, the front suspension was replaced after 584.2 kilometers, and so on. Similar data sets are available for two others cars.

The goals of the analysis are to analyze the combined data from all three repairable systems in order to:

- Estimate the total number of failures that you could expect for a car that competes in ten 200-kilometer races.

- Determine the probability that a car that has finished two races will complete a third without any failures.
- Determine if it would be feasible to overhaul the entire car at regular intervals in order to minimize its total life cycle cost. If it is feasible, calculate the overhaul time, assuming an average repair cost of $192,000 and an overhaul cost of $500,000.

On the control panel, choose the **Power Law** model, and then analyze the data by choosing **Growth Data > Analysis > Calculate** or by clicking the icon on the Main page.



A summary of the calculation results will then be shown on the control panel.

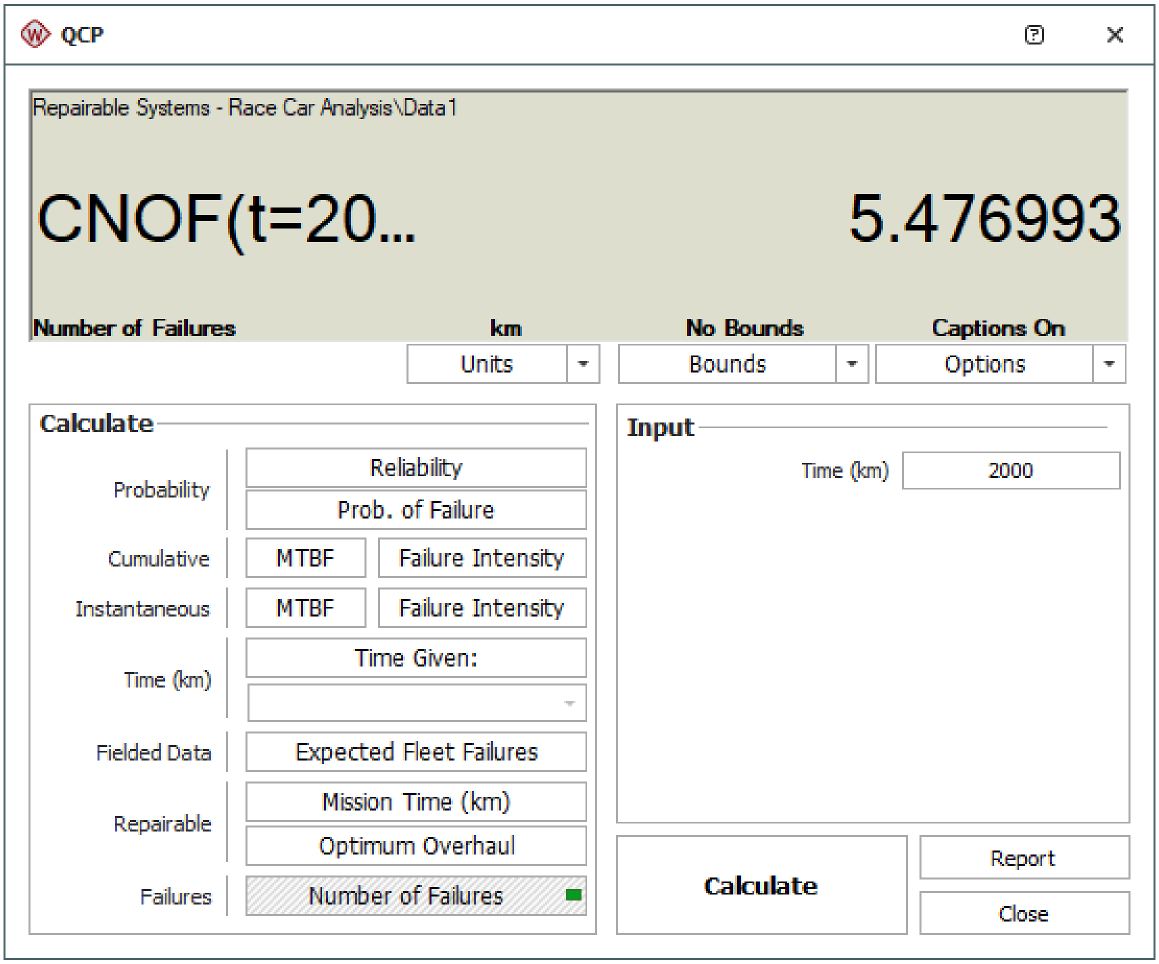
Results	
Parameters	
Beta	1.678224
Lambda (km)	0.000016
Statistical Tests	
Significance Level	0.1
CVM	Passed
CBH	Passed
Other	
Termination Time (km): 2500.000000	
Systems: 3/3	

You can see in the results that the data passed the Cramér-von Mises (**CVM**) test, which indicates that the power law model provides a good fit for the data. (If the power law model did not provide a good fit, you could transfer the data to a Fleet data sheet and use the Crow-AMSAA model instead. The Fleet data sheet allows you to group the data and combine them into a cumulative timeline for analysis.)

To estimate the total number of failures expected for one car over the period of ten 200-kilometer races, open the QCP by choosing **Growth Data > Analysis > Quick Calculation Pad** or by clicking the icon on the Main page of the control panel.

In the QCP, choose to calculate the **Number of Failures** with **None** for the confidence bounds. Select **km** for the time units and then enter **2000** (i.e., the equivalent of ten 200-kilometer races) in the **Time** field.

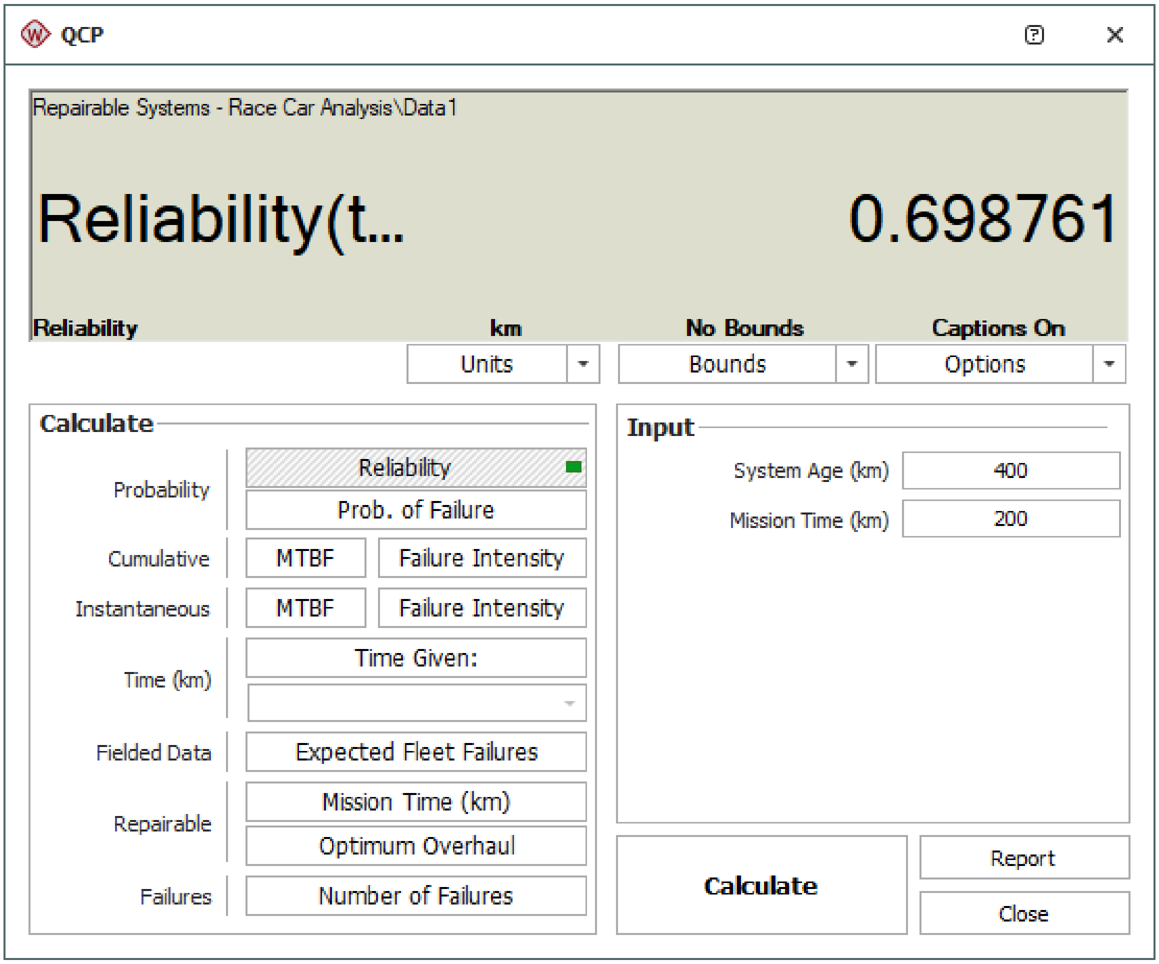
Click **Calculate** to obtain the result, as shown next. This analysis estimates about 5 failures for a car that competes in ten races.



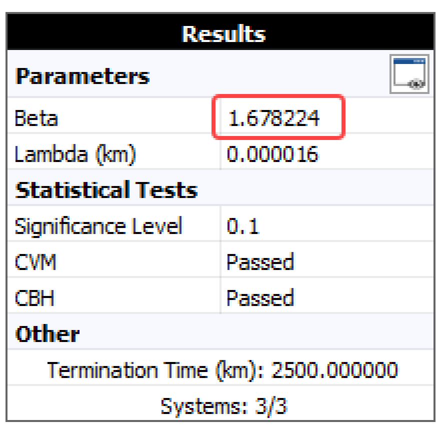
You can also use the QCP to determine the probability that a car that has finished two 200-kilometer races will complete a third race. Choose to calculate **Reliability** and make the following inputs:

- System Age = **400** (i.e., two races, each 200 kilometers)
- Mission Time = **200** (i.e., a third 200-kilometer race)

According to this result, the reliability of the car for the third race is estimated to be 69.88%.



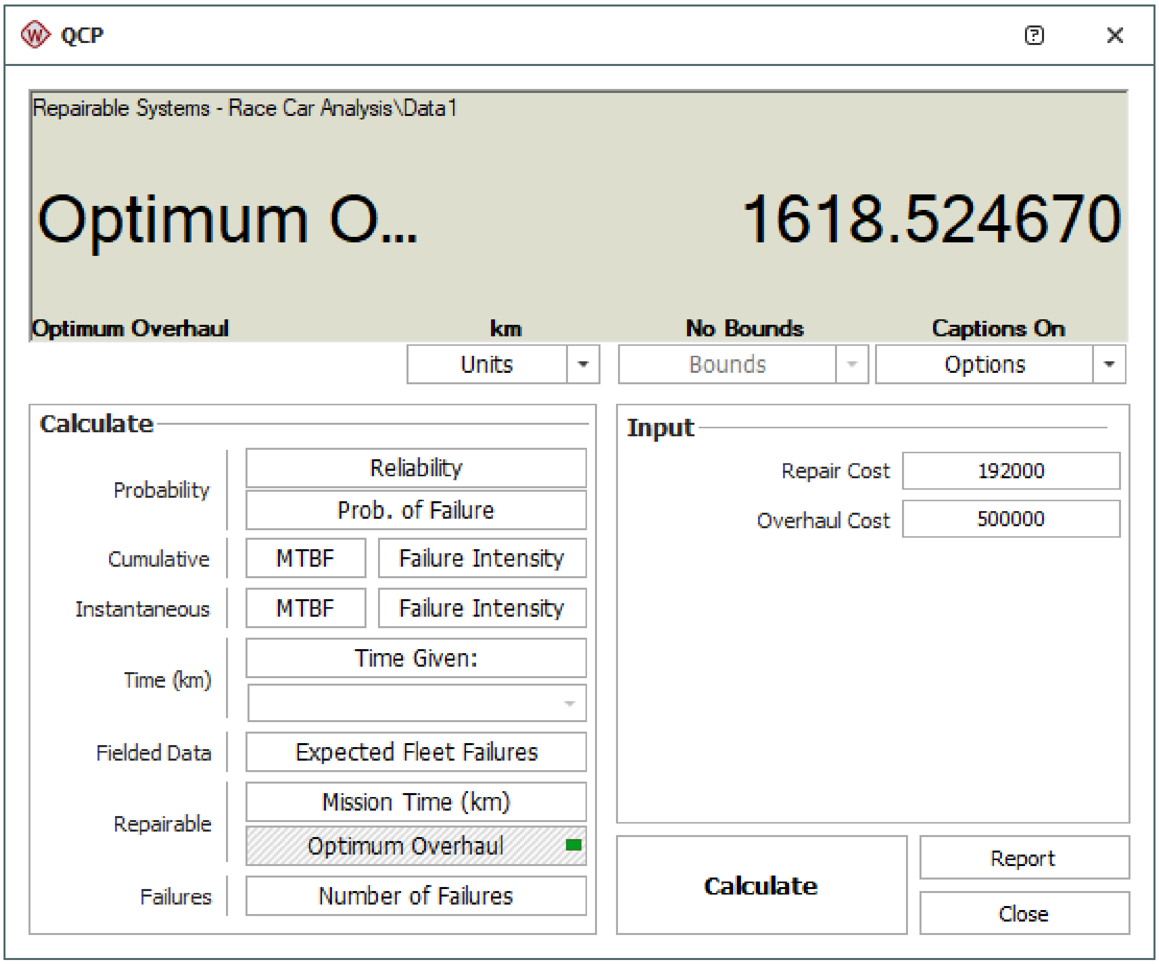
Finally, to calculate an optimum overhaul time, first make sure the beta value of the fitted power law model is greater than 1 (indicating wearout). If the system has a constant failure rate (beta = 1) or a decreasing rate (beta < 1), then it will not be cost-effective to implement an overhaul policy.



Since beta is greater than 1, you can use the QCP to solve for the optimum overhaul time. In the QCP, choose to calculate **Optimum Overhaul**, and then make the following inputs:

- Repair Cost = **192000**
- Overhaul Cost = **500000**

According to this result, the total life cycle cost of the car can be minimized by overhauling it every 1,619 kilometers.

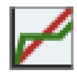


Mission Profiles

For reliability growth data analysis only.

Mission Profiles can help you to ensure that your testing is representative of the expected conditions of actual use by checking, at defined "convergence" points, whether expected usage and actual usage are acceptably close. The ReliaWiki resource portal provides more information about operational mission profile testing at: http://www.reliawiki.org/index.php/Operational_Mission_Profile_Testing.

To add a mission profile folio to a project, choose **Home > Insert > Mission Profile** or right-click the **Mission Profile** folder in the current project explorer and choose **Add Mission Profile**.



To use a mission profile folio in Weibull++, do the following:

- Set the convergence points
- Define the profile for each separate characteristic that will be tested
- Validate the mission profiles
- Create a plot that allows you to visually compare the actual vs. expected usage in any or all of the defined profiles

If desired, you can also link a profile with a data sheet in order to group the data.

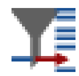
Convergence Points Sheet

On the Convergence Points sheet, enter the times at which you want to check that the expected usage and the actual usage either meet their expected averages or fall within an *accepted range* (i.e., the amount that the usage can vary from the expected value, while still being acceptable. For example, if the expected number of miles traveled is 1,000 and the allowed variance is 100, then usages of 1,051 and 913 are acceptable, while usages of 1,103 and 879 are not). You must include a minimum of three convergence points in the analysis. The test times between the convergence points do not have to be the same.

The following example shows a convergence points sheet with four times defined.

	Convergence Points	Comments
1	100	
2	250	
3	320	
4	400	
5		
6		
7		

All of the defined convergence points must be included in all of the profile sheets that you add to the folio. To automatically add the defined convergence points to each profile sheet, choose **Mission Profile > Mission Profile > Transfer Convergence Points to Profile Sheets** or click the icon on the control panel.



Profile Sheet

Each profile sheet represents a different part of testing (e.g., a different task that the system performs, etc.). By default, each new mission profile folio starts with one profile sheet. You can add additional profile sheets to the folio by choosing **Mission Profile > Mission Profile > Insert Profile Sheet**.

At a minimum, each profile sheet must contain the times that you entered on the convergence points sheet. It may also contain any additional times at which usage is observed and recorded.

The profile sheet contains the following information:

- The **Cumulative Time** column indicates the time at which the usage is measured. The input is cumulative, which means that the times must be increasing and each row represents the total test time up to that point.
- The **Expected Usage** column is the usage value that you expect at that time (e.g., the distance that the system should have traveled, the number of hours it should have operated or the number of times it should have performed a certain action). These inputs are also cumulative, which means that the values must be increasing and each row represents the total distance, hours operated, times used, etc. up to that point in time.
- The **Actual Usage** column is the usage value recording during the testing. These inputs are also cumulative.
- In the **Plus/Minus Range** field, specify the accepted variance from the expected usage values.

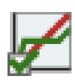
The following example shows part of the data in a profile sheet.

	Cumulative Time	Expected Usage	Actual Usage
1	5	50	0
2	10	100	0
3	15	150	0
4	20	200	0
5	25	250	100
6	30	300	150
7	35	350	400
8	40	400	600
9	45	450	600
10	50	500	600
11	55	550	800
12	60	600	800
13	65	650	800
14	70	700	800
15	75	750	800
16	80	800	900
17	85	850	950
18	90	900	1000
19	95	950	1000
20	100	1000	1000
21	105		
22	110		

Note: Complete data set is not shown.

Validating the Mission Profile

After you have entered all of the data, you can validate the profile to make sure that all necessary data points have been entered. To do so, choose **Mission Profile > Mission Profile > Validate Mission Profiles** or click the icon on the control panel.



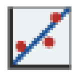
This feature checks that all convergence points are in all the profile sheets and sorts the data by the Cumulative Time column.

Note: Validation also occurs when you generate a plot of the mission profile. If all of the data points have been entered, all of the profile sheets are marked as being verified.

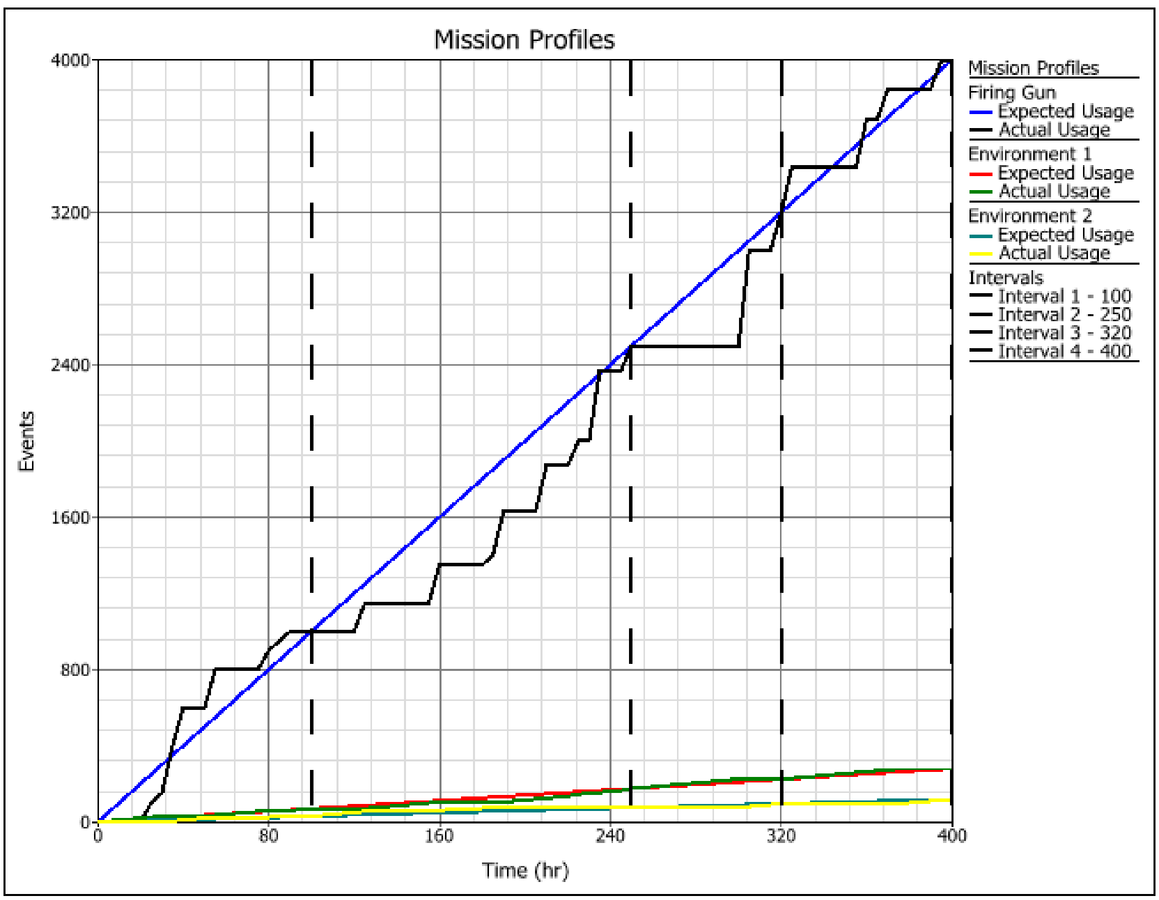
Mission Profile Plots

For reliability growth data analysis only.

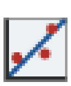
You can visually compare the expected usage and the actual usage in any or all of the mission profiles defined in the folio by choosing **Mission Profile > Mission Profile > Plot Mission Profiles** or clicking the icon on the control panel. A plot sheet will be added to the folio.



An example of a mission profile plot is shown next:



The plot sheet control panel includes the following options:

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

Plot Setup opens the Plot Setup window, which allows you to customize most aspects of the plot including the titles, colors, sizes, etc.

RS Draw launches ReliaSoft Draw, which allows you to view the plot in greater detail, add annotations and modify selected plot elements.

Export Graphic allows you to save the current plot graphic in one of the following formats: *.jpg, *.gif, *.png or *.wmf.

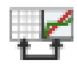
- **Auto Refresh** automatically redraws the plot when something changes. If not selected, you must click the **Redraw Plot** icon to refresh the display.
- **Keep Aspect Ratio** maintains the ratio of the horizontal size to the vertical size of the plot graphic when you resize the plot sheet.
- **Units** allows you to choose which units you want the plot to show from the drop-down menu.
- In the **Scaling** area, the **X** and **Y Scaling** boxes show 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.
- **Show**
    - Select **Show Plus/Minus Range** if you want to display lines that show the allowed *accepted variance* from the expected average usage values on the plot. (Accepted variance is the amount that the usage can vary from the expected value, while still being acceptable. For example, if the expected number of miles traveled is 1,000 and the allowed variance is 100, then usages of 1,051 and 913 are acceptable, while usages of 1,103 and 879 are not.)
    - Select **Intervals** if you want to display the convergence points on the plot.
- **Available Profiles** allows you to select which of the profile sheets in the folio will be included in the plot.

Linking a Mission Profile to a Data Sheet

For reliability growth data analysis only.

If your actual testing follows the balanced test plan that you established in a Weibull++ mission profile folio, then it is likely that you will get valid results from analyzing the test data directly with one of the reliability growth models in a Failure Times data sheet. However, if there is a possibility that the test data might not be appropriate for direct analysis, you can use the mission profile folio to group the data based on the specified convergence points so it is suitable for reliability growth analysis.

To use this feature, first enter the test data in a Failure Times data sheet, then click the **Link Mission Profile** icon on the growth data folio control panel.



After you select one of the mission profiles in the current project, the application will automatically transfer the data to a new Grouped Failure Times data sheet in the folio. The data will we grouped according to the times in the specified mission profile.

Example: Using the Mission Profile Folio

For reliability growth data analysis only.

*The data set used in this example is available in the example database installed with the software (called "Weibull21_ReliabilityGrowth_RGA_Examples.rsgz21"). To access this database file, choose **File > Help**, click **Open Examples Folder**, then browse for the file in the Weibull subfolder.*

The name of the example is "Mission Profiles," and the folio that contains the data is called "Tank Testing Profile."

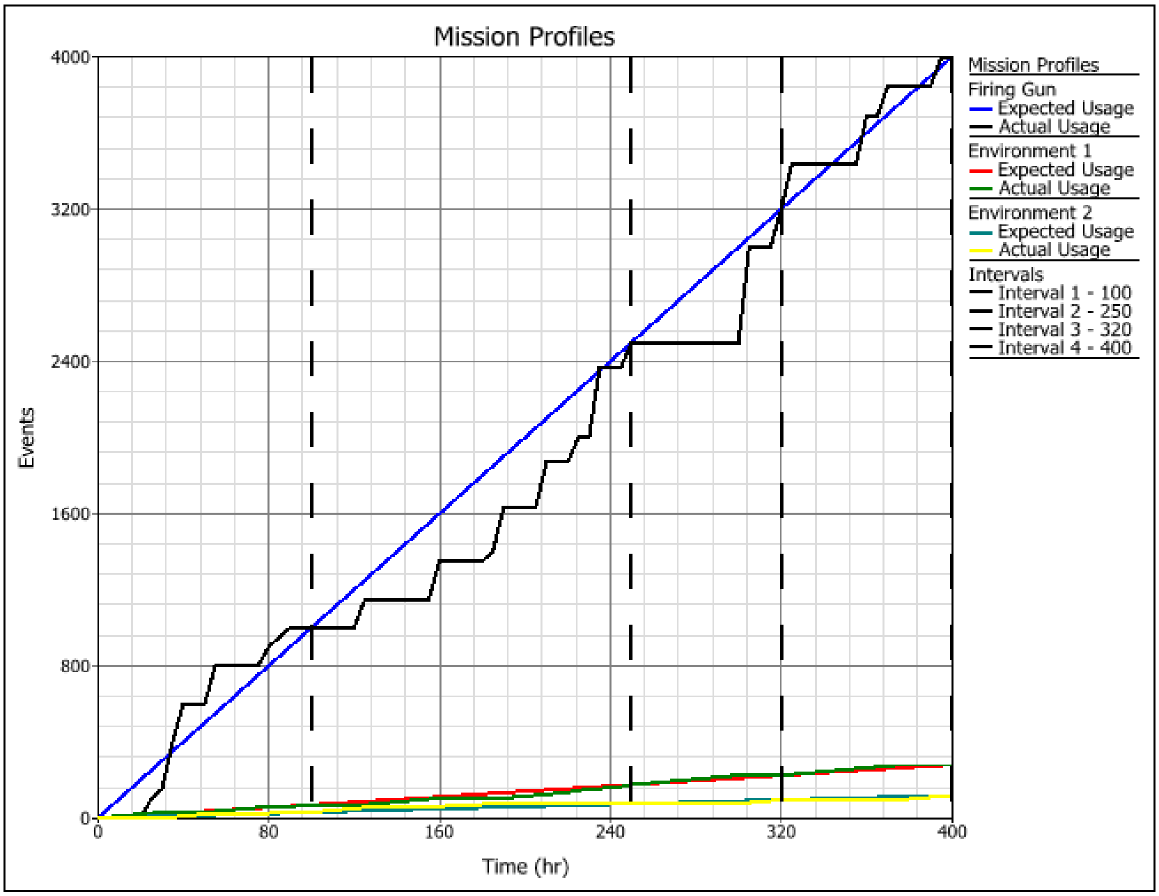
A team is working on a military system that must perform three tasks:

- firing a gun
- moving under environment 1
- moving under environment 2

They use a mission profile to create a test plan that defines the expected accumulated usage values for each task, and includes the same four convergence points (100, 250, 320 and 400 hours) in each task's profile. The purpose of these convergence points is to make sure that the usage levels for all three tasks match up at those specific times (even if there is some variation from the test plan in between convergence points).

When the team performs the test, they record the actual usage values in the mission profile folio, and they record the failure times data in a reliability growth data folio.

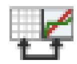
The mission profile folio called "Tank Testing Profile" shows the actual and expected usage for all three tasks. As shown next, the plot shows all three planned mission profiles together with the actual usage during the test. For example, the straight blue line shows the planned usage for the gun firing task and the jagged black line shows the actual usage during the test. You can see that the actual usage tends to fluctuate around the plan (due to normal variations under actual test conditions), but always meet the target at each of the four convergence points. The same applies for the actual usage on the environment 1 and environment 2 tasks.



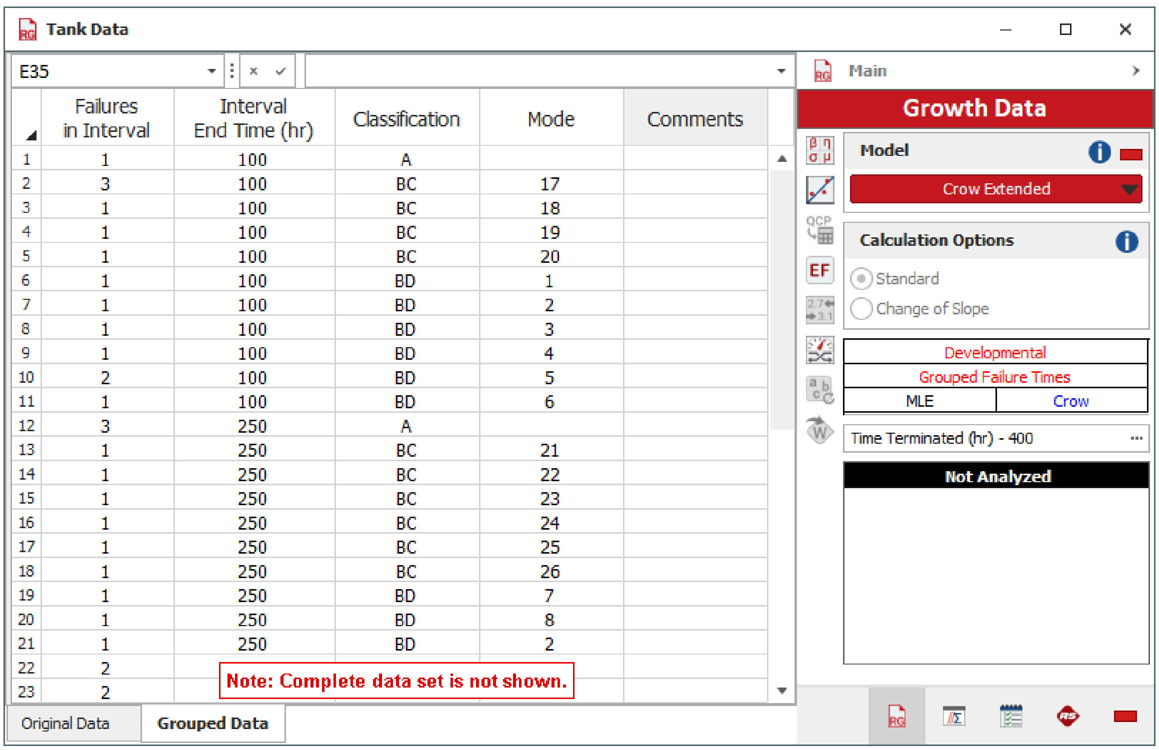
The plot can be used to track whether the test is proceeding according to plan. In addition, the mission profile folio can also assist with data analysis by automatically grouping the data based on the specified convergence points.

If there has been significant variation from the test plan in between the convergence points, this means that the usage values for the three tasks may not have been properly balanced at certain points in time, and an analysis of the exact failure times may give misleading results. For this example, the team decides to group the data at the convergence points so they can be sure that the usage of all three tasks has been synchronized according to the plan.

The "Tank Data" growth data folio contains the failure times observed during the test. The first data sheet (called "Original Data") contains the exact failure times. To create the second sheet (called "Grouped Data"), the analyst simply opened the original data sheet and clicked the **Link Mission Profile** icon on the control panel.

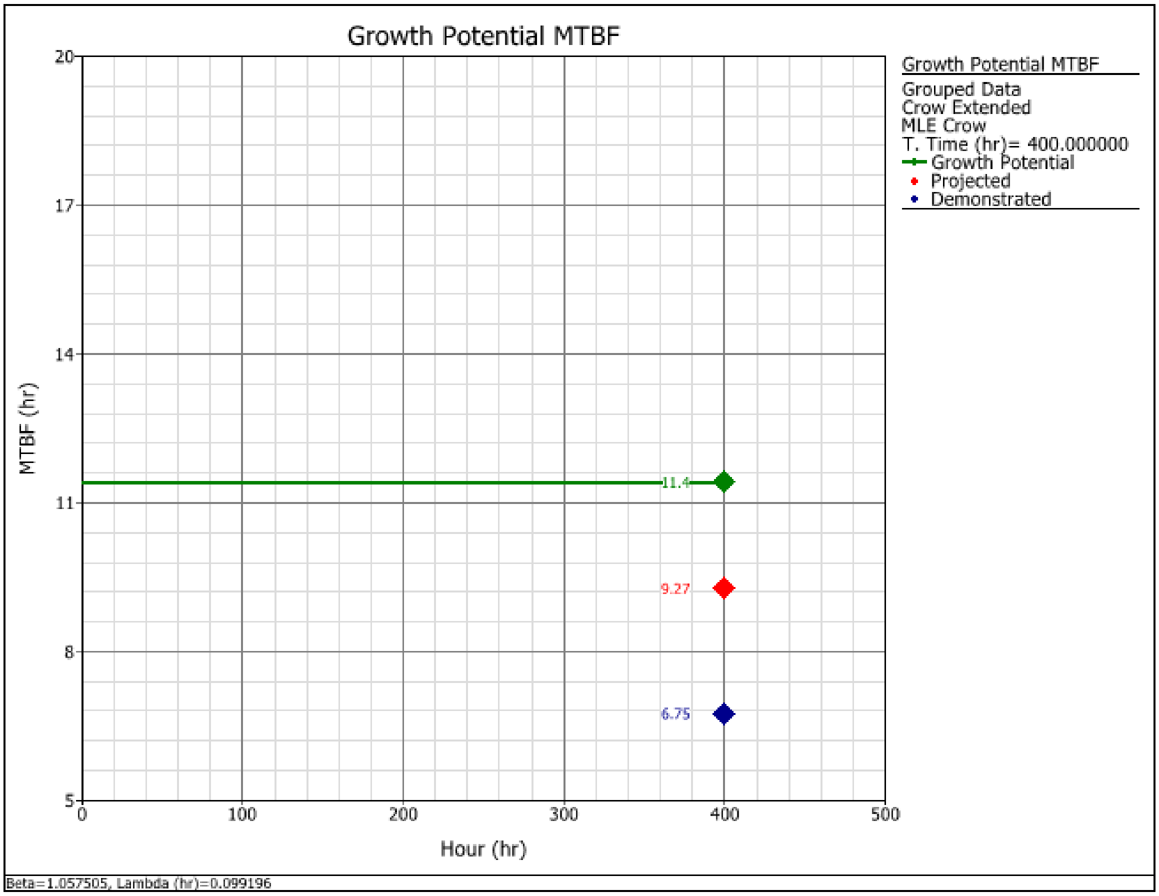


When prompted, he selected the applicable mission profile and the application automatically grouped the data. The following picture shows the grouped data analysis, which is based on balanced usage values for the three different tasks.



	Failures in Interval	Interval End Time (hr)	Classification	Mode	Comments
1	1	100	A		
2	3	100	BC	17	
3	1	100	BC	18	
4	1	100	BC	19	
5	1	100	BC	20	
6	1	100	BD	1	
7	1	100	BD	2	
8	1	100	BD	3	
9	1	100	BD	4	
10	2	100	BD	5	
11	1	100	BD	6	
12	3	250	A		
13	1	250	BC	21	
14	1	250	BC	22	
15	1	250	BC	23	
16	1	250	BC	24	
17	1	250	BC	25	
18	1	250	BC	26	
19	1	250	BD	7	
20	1	250	BD	8	
21	1	250	BD	2	
22	2				
23	2				

The following plot shows some useful planning metrics obtained from the grouped data analysis with the Crow Extended model: the demonstrated, projected and growth potential MTBFs.


Growth Potential MTBF
Growth Potential MTBF
Grouped Data
Crow Extended
MLE Crow
T. Time (hr)= 400.000000
Growth Potential
Projected
Demonstrated
MTBF (hr)
Hour (hr)
20
17
14
11
8
5
0
100
200
300
400
500
11.4
9.27
6.75
Beta=1.057505, Lambda (hr)=0.099196

DOE Design Folios

IN THIS CHAPTER

DOE Background

In order to develop the best possible products and processes in the most cost-effective way, it is necessary to plan the testing process carefully so that all relevant factors and potential interactions of factors are considered without wasting time, effort or money. Design of Experiments (DOE) is widely used across the natural and social sciences to develop experimentation strategies that maximize learning using a minimum of resources.

Designed experiments are usually carried out in five stages: planning, screening, optimization, robustness testing and verification. These stages are described next.

Planning Stage

Careful and thorough planning prior to embarking upon the process of testing and data collection can save time and resources by eliminating unnecessary work and by helping the analyst to avoid making costly mistakes. Well-planned experiments are easy to execute and analyze, whereas experiments that are poorly planned may result in data sets that are difficult or impossible to analyze with even the most sophisticated statistical tools or, if they can be analyzed, give inconclusive results. Essential points to consider during the planning stage include:

- **Objective:** What do you hope to learn through the experiment?
- **Team:** Individuals from different disciplines related to the product or process should be chosen, in order to incorporate the broadest possible range of knowledge.
- **Factors and Responses:** The team should identify a pool of factors that will be investigated and the response(s) that will be measured. For each response, the team should identify a goal. This might be to minimize or maximize the response, to bring it as close as possible to a target value, to minimize variability or some combination of these.

Screening Stage

The goal of the screening stage is to determine which factors out of the pool of potential factors identified during the planning stage are important enough to examine in greater detail (i.e., >to extract the "vital few" from the "trivial many"). To this end, screening experiments are carried out to identify factors with a significant effect on the measured response(s). Typically, these experiments are efficient designs that require only a few executions and focus on the main effects of factors rather than on interactions between the factors. These experiments, in conjunction with prior knowledge of the process, help in eliminating unimportant factors and focusing attention on the factors that require more detailed analysis.

In DOE, factorial designs are well suited for use in the screening stage.

Optimization Stage

Once the important factors have been identified, the objective is to determine the settings of these factors that, taken together, will yield the desired outcome. The analyst will need to consider the goal set for each response and, in cases where multiple responses are measured, may need to consider the relative importance of each response when determining optimal solutions.

Response surface method design types are often used in the optimization stage, when the focus is on the nature of the relationship between the response and the factors rather than on the identification of important factors. The purpose of response surface methods is to examine this relationship, or "surface."

In DOE, after analyzing data from an experiment with at least two factors (one of which must be quantitative), the optimization folio can be used to optimize the factor settings.

Robustness Testing Stage

While the experimental environment can be carefully controlled, it is likely that there will be factors that affect the product or process in the application environment and are beyond the control of the analyst. Such factors, referred to as noise or uncontrollable factors, may include humidity, ambient temperature, variation in material, etc. The goal of robustness testing is to identify these factors and ensure that the product or process is made as insensitive, or robust, as possible to them.

In DOE, you can use robust parameter designs for robustness testing. In addition, response variability analysis is available for two level factorial and Plackett-Burman designs.

Verification Stage

In the verification stage, the analyst confirms the results drawn from the previous phases by performing a few follow-up experimental runs to see if the observed response values are close to the predicted value(s). The goal is to validate the best settings that have been determined, making sure that the product or process functions as desired and that all objectives are met.

The ReliaWiki resource portal provides more information about Design of Experiments at http://www.reliawiki.org/index.php/DOE_Overview.

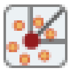
Design Folio

Weibull++ includes three kinds of folios for design of experiments:

- **Standard design folios** are used for identifying important factors (i.e., "screening") and determining the best settings for these factors (i.e., "optimization"). They can also be used for simple one factor designs that allow you to compare the response at different factor levels.
- **Robust design folios** are used to minimize the system's sensitivity to noise factors (i.e., "robustness testing"). Robust design folios include all the functionality of the standard design folio, with the addition of some features specific to the robust design methodology.
- **Mixture design folios** are used when the factors in an experiment are *components* in a mixture, and when you wish to determine the best proportions to use for each component.

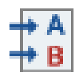
To create a DOE folio, right-click the **DOE** folder in the current project explorer and choose the folio type.

 Standard Design

 Robust Design

 Mixture Design

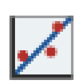
When the folio first appears, only the Design tab will be displayed, which allows you to configure an experiment design. After you have specified all of the settings on the Design tab, choose **Design > General > Build Design** or click the icon on the control panel.



This will create a second tab called Data that a) shows a "plan" for how the experiment needs to be performed and b) allows you to enter and analyze response data from an experiment that follows that plan. (If you want to change the experiment design settings after the Data tab has already been created, see Changing Design Settings.) After you analyze the data, a summary of results (including the factors that were found to be significant) will appear on the control panel.

Tip: This documentation assumes that you will first design an experiment, then perform it, and then enter and analyze the resulting data. However, if you already have data available from a prior experiment, you can also use the Design tab to build a design that reflects the prior experiment and then build the design so you can enter and analyze the results on the Data tab.

After you have entered the response information in the Data tab, you can choose **Data > Analysis > Plot** or click the icon on the control panel.



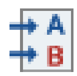
This will create a third tab called Analysis Plot that allows you to view all of the graphical plots that are relevant to the current analysis of the response data.

The information presented in the topics that follow describes interfaces and procedures that are common to all design types. To learn more about a specific design type, see one of the following topics:

- Factorial Designs
- Response Surface Method Designs
- One Factor Designs
- Robust Parameter Designs
- Mixture Designs
- Reliability DOE

Design Tab

When you first create a design folio, the Design tab will be shown. After you specify the settings on this tab, choose **Design > General > Build Design** or click the icon on the control panel.



This will create the Data tab, which shows the new "plan" for the experiment and allows you to record and analyze response data.

Design Types

The following design types are supported:

Factorial Designs are used for screening vital factors, and they are available in standard design folios and robust design folios. The links below explain these designs in the context of standard design folios.

- Two Level Factorial: You can have 2 to 15 factors; each factor must have exactly two levels. This design type can be used to investigate all main effects and their interactions (full factorial) or just a subset of them (fractional factorial).
- Plackett-Burman Factorial: This is a special category of two level fractional factorial design, where only a few specifically chosen runs are performed to investigate the main effects (i.e., no interactions). You can have 2 to 47 factors; each factor must have exactly two levels.
- General Full Factorial: Different factors can have different numbers of levels. All effects and their interactions are investigated. You can have 2 to 7 factors; each factor can have 2 to 20 levels.
- Taguchi Orthogonal Array (OA) Factorial: This is a highly fractional design type. It can investigate main effects with just a few runs, and different factors can have different numbers of levels.

Response Surface Method Designs are used for system optimization and are available only in standard design folios.

- Central Composite Design Response Surface Method: This is the most commonly used response surface methodology design. It is typically used to study the quadratic effects of factors.
- Box-Behnken Response Surface Method: This design type allows you to consider from three to nine quantitative factors at three levels each (one level being a center point). If setting all factors at the high level at the same time carries a risk of equipment damage or violates other constraints, these designs provide a matrix that avoids setting all factors at extreme values simultaneously.

One Factor Designs are used for comparison, and they are available only in standard design folios.

- Only one factor is investigated, and the response is compared at different factor levels. The factor levels must be qualitative. (To perform one factor analysis with quantitative levels, use the Multiple Linear Regression folio.)

Mixture Designs are used when the factors in an experiment are components in a mixture, and when you wish to determine the best proportions to use for each component.

- Simplex Designs include:
    - **Simplex Lattice**: With this design, the blends in the experiment are determined by the specified number of levels of each component (i.e., the degree of design + 1). Since it includes all the reasonable combinations of components, this design is useful when the number of components is not large, and a higher polynomial equation is needed for optimization.
    - **Simplex Centroid**: By default, this design includes single-component blends (vertices) and all centroids up to the dimension *g*, where *g* is the number of components. Users can specify the degree of design (i.e., the dimension of centroids). Since it usually has fewer test runs than a simplex lattice design, this design is useful when the number of components is larger, but a lower polynomial equation will suffice for optimization.
    - **Simplex Axial**: This design includes vertices, blends with an absent component (edge points), center points and interior points between the center point and vertices (axial points). It is mainly used for screening components.
- Extreme Vertex Designs: This design allows you to impose additional limits on the component values by specifying upper bounds on components and defining linear constraints for blends.

Building a New Design

The Design tab of the design folio is used to define and then build an experiment design. When you build the design, its "plan" will be shown on the Data tab.

- If you are working with a standard design folio, you will use the Design tab to select a design type, define responses/factors and choose additional settings for the whole design.
- If you are working with a robust design folio, you will make these selections for each of the inner and outer arrays of the design.
- If you are working with a mixture design folio, you will make these selections to specify the design for mixture factors and, if applicable, process factors.

After you create the folio, follow the steps outlined next to build a design from scratch.

Tip: If you wish to use saved settings instead, choose **Design > Import/Export > Import Template** or click the icon on the control panel. After you import the settings, you can proceed to the last step to review your settings and build the design. (See Importing/Exporting Design Settings.)

CHOOSE THE DESIGN TYPE(S)

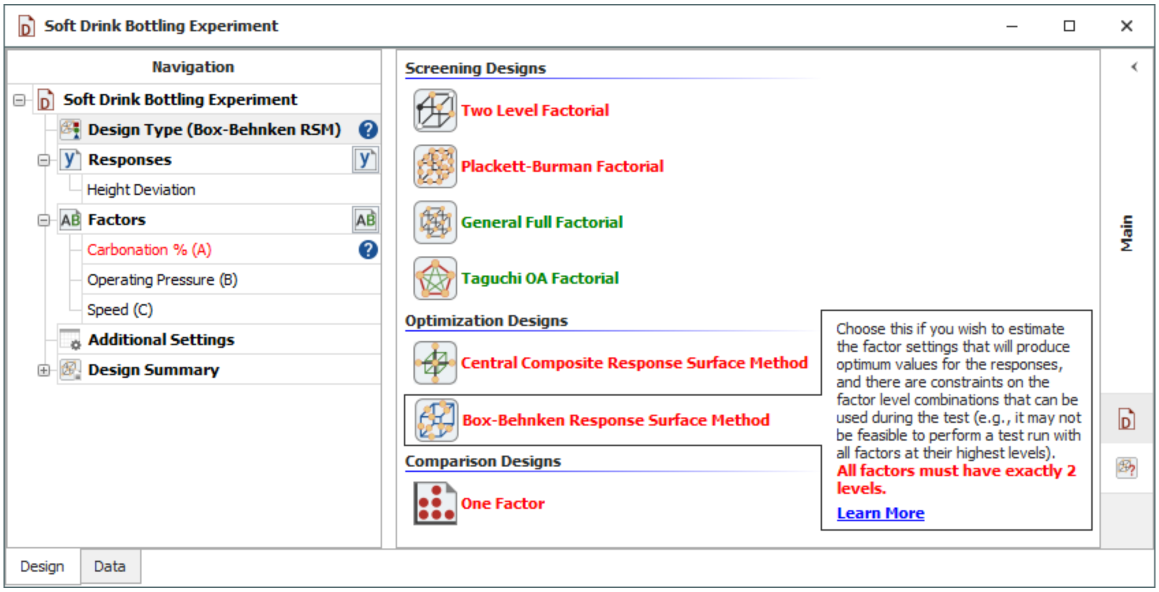
If you are working with a standard design folio, choose one design type for the experiment by clicking the **Design Type** heading in the navigation panel on the left and choosing an option in the input panel on the right.

If you're using a robust design folio, you need to select two design types (one under the **Inner Array** heading and another under the **Outer Array** heading). For considerations in choosing the design types and additional information about settings used in configuring the robust design, see Robust Parameter Designs.

When you first create a design folio, the current factor settings will change as required depending on the design you select. After you directly make changes to the factors, responses or additional settings, the factor settings will no longer change automatically, and the design types will be color-coded in green or red.

If a design is red, then one of the current factor settings is incompatible with it. When you select a red design type, the factor(s) it is incompatible with will become red as well. To find out why the setting is incompatible, read the design type's description. For example, the red text in the design description below says, "All factors must have exactly 2 levels," which indicates that the

Carbonation factor (also shown in red) does not have the required number of levels for a Box-Behnken design.



DEFINE THE RESPONSE(S)

If you are using a standard design folio, you must configure the response(s) that will be measured in the experiment by selecting the response in the navigation panel and editing its properties in the input panel. (See Adding, Removing and Editing Responses.) To perform reliability DOE, select **Life Data** as the response type for one response. The resulting Data tab will be configured for reliability DOE.

For robust design folios, default response properties will be used to build the design.

DEFINE THE FACTOR(S)

You can change the number of factors by clicking the **Factors** heading in the navigation panel and choosing an appropriate number in the input panel. You can then define individual factors by selecting the factor in the navigation panel and editing its properties. (See Adding, Deleting and Editing Factors.)

SPECIFY ADDITIONAL SETTINGS

The options available when you click the **Additional Settings** heading will vary depending on which design type is currently selected. The following options may be available:

- **General Settings**
  - **Replicates** allows you to specify the number of test samples you want to observe at each combination. In other words, the number of replicates is the number of times you want to perform the complete set of runs required by the design. Using replicates helps to separate error caused by a poor model fit from error caused by natural variation (i.e., experimental error).
  - **Repeated Measurements** allows you to specify the number of measurements to use for each test run. The mean and standard deviation of the measurements are calculated for each run, and either can be selected as the analyzed response for the experiment. This option is not available for R-DOE or designs with multiple responses.
  - **Block on Replicates** allows you to choose whether you will use each replicate of the design as a block. By blocking on replicates, you can account for "nuisance factors" (i.e., effects that are not of interest to the investigator) due to conditions that differ from one replicate of the design to another. For example, if the first replicate of the design is performed on Day 1, and the second is performed on Day 2, you could account for variation in the response due to the day. This option is available only if the number of replicates in the design is greater than 1.
- **Block Settings**
  - **Number of Blocks** allows you to specify the total number of blocks that will be used in the design. **Blocks per Replicate** (shown when you choose to block on replicates) allows you to specify the number of blocks that will be used per replicate. The more blocks you use in the design, the more potential nuisance factors you can account for. Note, however, that using blocks requires aliasing them with one or more effects.

    Select a number of blocks from the drop-down list; the options available in this list depend on the type of design and the number of factors. Using 1 block in the entire design is equivalent to not using blocks.

    With Taguchi OA designs, you must block on replicates to use blocks, and you can use at most 1 block per replicate.
  - **Center Points per Block** allows you to specify the number of center points that will be used per block. Center points are used primarily to help you determine if there is curvature in the relationship between the factors and the response. In addition, using several center points provides an estimate of pure error.
  - **Block Name** allows you to enter a name for each block.

  - **Design Generators** will appear when you have multiple blocks. This field is used to specify the factorial interaction effects that will be aliased with that of the nuisance factor for which you are blocking the design. (See Specifying Generators.)

- **Factorial Settings**

  - **Fraction** allows you to select the number of base runs you want to use in the experiment, based on a fraction of the total number of runs that would be required for a full factorial design. For example, if your design includes five factors with two levels each, a full factorial design would require $2^5 = 32$ base runs. For a fractional factorial design in this case, you can select to use a one-half fraction (16 runs) or a one-fourth fraction (8 runs). The total number of runs required for the experiment may be greater than the number of base runs (e.g., if you use replicates and/or center points). (See Two Level Factorial Designs and Central Composite Designs.)

  - **Design Generators** will appear when you are using a fractional design, not a full factorial. It allows you to specify the factorial interaction effects that will be aliased with the main effects of particular factors. (See Specifying Generators.)

- **Plackett-Burman Settings** (Plackett-Burman designs only)

  - **Base # of Runs** allows you to specify the number of runs you want to use in the experiment. The available choices are determined by the number of factors you specify and the design matrix developed by Robin L. Plackett and J. P. Burman.

- **Taguchi OA Settings** (Taguchi OA designs only)

  - The options available in the **Taguchi Design Type** drop-down list will depend on the numbers of levels for each factor. For example, if all your factors have three levels, then you could choose the L9 (3^4) design, which uses nine runs to investigate up to 4 three-level factors. If some factors have two levels and the rest have four, then you could choose the L16 (2^9*4^2) design, which uses 16 runs to investigate up to 9 two-level factors and up to 2 four-level factors.

  - Click the **View Alias Table** icon to see the alias table for the selected Taguchi design type. The table shows what aliases result from assigning factors to particular columns in the orthogonal array. See Taguchi OA Alias Table.

  - **Taguchi Column Indices** allows you assign factors to columns of the selected orthogonal array. No two factors can have the same index.

- The **Specify Interaction Terms** link opens a window that helps you assign factors to columns without aliasing any main effects or any interaction terms of interest. (See Specify Interaction Terms Window.)

- **Central Composite Settings** (Central Composite designs only)

  - **Alpha value** allows you to specify the alpha value that you want to use to determine the axial points in the design. Options include:

    - **Rotatable Design**: This alpha value ensures that the variation in the response will be the same for all points that are the same distance from the design center point. This makes the model provide equally precise predictions throughout the region of interest, which means that regardless of the direction taken to search for the optimum solution, the result is equally accurate.

    - **Spherical Design**: This alpha value places the axial points on the surface of the sphere defined by the factorial design points (i.e. all design points have the same distance to the center point).

    - **Orthogonal Block**: This alpha value ensures that the effect of the blocking is unconfounded with other effects. This option is available only if you have selected to block the design.

    - **Face Centered Design**: In this case, alpha = 1, which means that all design points will be in the range from -1 to 1.

    - **Custom**: Allows you to specify your own alpha value to meet the constraints of your experiment.

  - If you select **Yes** in the **Add axial point block** drop-down list, an extra block will be added to the design and used for the axial points.

  - **Center points / axial block** allows you to select how many center points will be added to the axial block.

- **Mixture Settings** (Mixture designs only)

  - **Mixture Total** allows you to specify the total amount of the mixture used for each test run.

  - **Degree of Design** is available only for simplex lattice designs. With a degree of n, each factor will use the values 0, 1/n, 2/n, …, n/n. Therefore, choosing n means each mixture factor will have n + 1 factor levels.

  - **Degree of Centroid** is available only for simplex centroid and extreme vertex designs. With a degree of n, the centroid of n-dimensional space will be added to the design. (For example, the center point of a line is the centroid of a 2-dimensional space.)

- **Additional Runs** (Mixture designs only)

  - **Center Points** allows you to specify the number of center points to add to the overall design. A center point is a test run that uses a blend with all components set to the same value.

  - **Axial per Component** (simplex designs only) allows you to specify the number of axial points to add for each component in the design. An axial point is a test run that uses a blend between a center point and a vertex (i.e., a blend with one component set to its maximum value).

  - **Vertex per Component** (simplex designs only) allows you to specify the number of vertex runs to add for each component in the design.

  - **Axial per Vertex** (extreme vertex designs only) allows you to specify the number of axial points to add for each vertex in the design (i.e., a blend with one component set to its maximum value). An axial point is a test run that uses a blend between a center point and a vertex.

  - The **Blend Combinations** settings (e.g., **2-Blend Combinations**) are available if you have at least three components in your simplex design. They allow you to specify how many combinations of N-blend points to add to the design. For example a 2-blend point is a test run that includes two components set to equal values, and any remaining components set to zero. 3-, 4- and 5-blend combinations may also be available depending on the number of components in the design.

    Note that the specified number is the number of combinations of points to add, not individual points. Thus, if you have three components and choose **1** for the **Double Blend**, three runs with the following values will be added to the design: (1/2, 1/2, 0), (1/2, 0, 1/2) and (0, 1/2, 1/2).

  - **Linear Constraints** (extreme vertex designs only) allow you to define limits on how the components can be combined. See Defining Linear Constraints.

DESIGN FOR PROCESS FACTORS

For mixture designs, click **Design for Process Factors** in the navigation panel and specify whether process factors (e.g., temperature) will also be included in this design. If you choose **Yes**, more settings become available to specify the design for process factors. The steps for defining this part of the design are similar to those for creating a factorial design.

RENAME THE FOLIO

If you wish to rename the folio, click the heading at the very top of the navigation panel and edit the text in the **Name** field in the input panel. Alternatively, you can right-click the folio in the project explorer and choose **Rename** from the shortcut menu.

REVIEW YOUR SETTINGS AND BUILD THE DESIGN

You can review some properties of your new design in the **Design Information** area of the control panel, as shown next.

Design Information	
Design Type	General Full Factorial
Responses	1
Factors	2
Base Runs	4
Replicates	2
Total Runs	8
Detailed Summary	

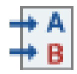
In addition to the **Design Type**, **Responses**, **Factors** and **Replicates** you chose for the design, the following calculated values are shown:

- **Base Runs** is determined by the number of factors, the number of levels for each factor and whether you are using a fractional design. For example, if your design is a full factorial design with 3 factors that each have 2 levels, then the number of base runs is 2^3, or 8. If the factors/levels are the same but you are using a fractional design that uses 1/2 of the runs required for a full design, then the number of base runs is 4 (i.e., 1/2 of the 8 required for the full design).
- **Total Runs** is the total number of runs that are required for the experiment. This value is determined by the number of base runs, replicates, center points per block, axial points per block and other settings. For extreme vertex designs, this information will not be available until you have validated the constraints.

If you click any link in the first column of this table, you will be able to edit the relevant setting in the input panel. If you wish to see more detailed information about your current design settings, click the **Detailed Summary** link at the bottom of the table. The Design Summary will provide further information about the design's properties.

You can also use the Design Evaluation tool to help you determine whether a design with the current settings would, for example, be likely to detect the main effects and factorial interactions you are interested in. (See Design Evaluation.)

When you are ready to build the design, choose **Design > General > Build Design** or click the icon on the control panel.



This will create a new Data tab in the folio that will be used to enter and analyze response data.

Adding, Removing and Editing Responses

Before building your design, make sure you have the correct number of responses (i.e., the types of output that you want to measure) and that each response is defined appropriately. These settings are selected on the Design tab.

ADDING AND REMOVING RESPONSES

There are three ways to add or remove responses from a design.

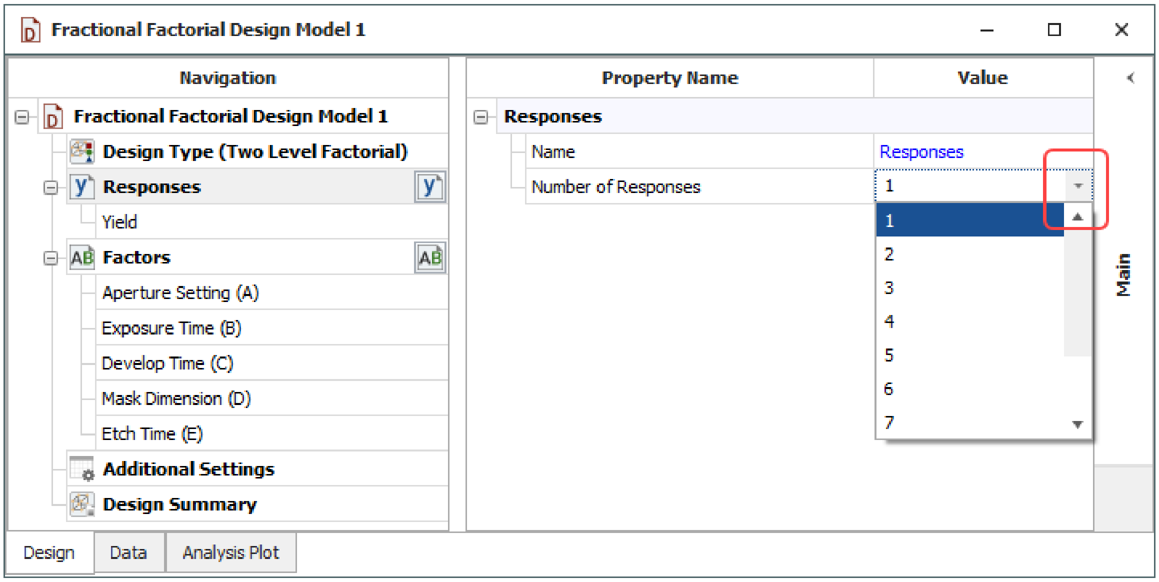
- You can choose **Design > Responses > Add Response**.

  If two or more responses are included in the design, you can remove a response by selecting it in the navigation panel and choosing **Design > Responses > Remove Response**.

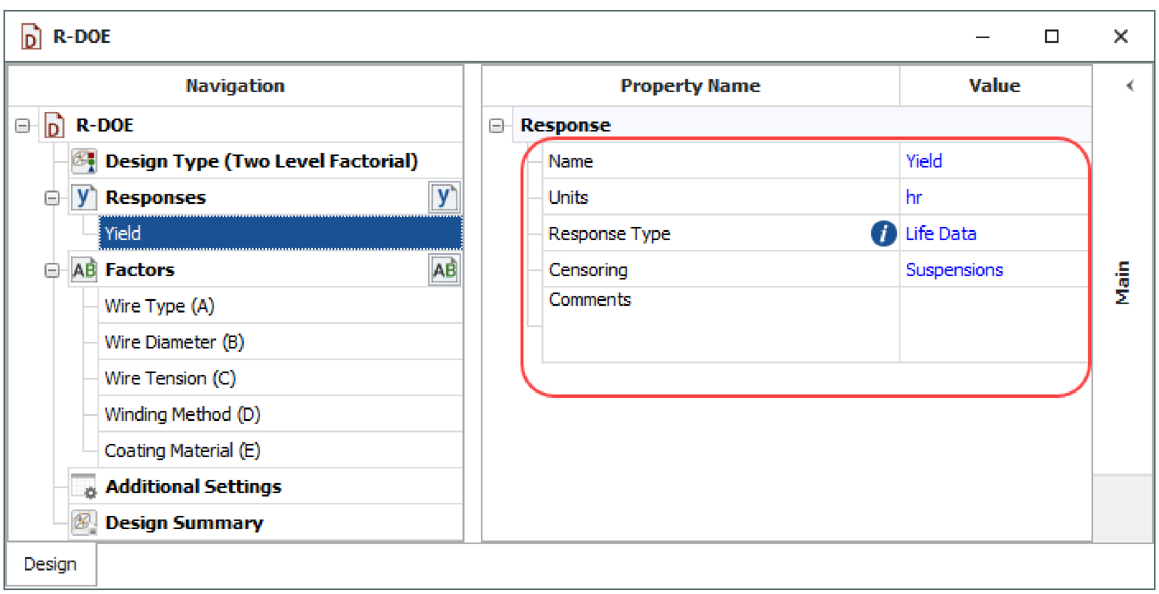
- You can also add/remove responses by right-clicking a response and choosing the appropriate option from the shortcut menu.
- To quickly add or remove several responses at once, click the **Responses** heading and then change the number in the **Number of Responses** drop-down list in the input panel, as shown next.



Note that when you decrease the number of responses using the drop-down list, the responses at the bottom of the list will be deleted first.

EDITING RESPONSES

You can define the properties of an individual response by selecting it in the navigation panel and editing its properties in the input panel, as shown next.



Alternatively, you can edit all the responses simultaneously by clicking the **Modify Responses** icon in the navigation panel ().

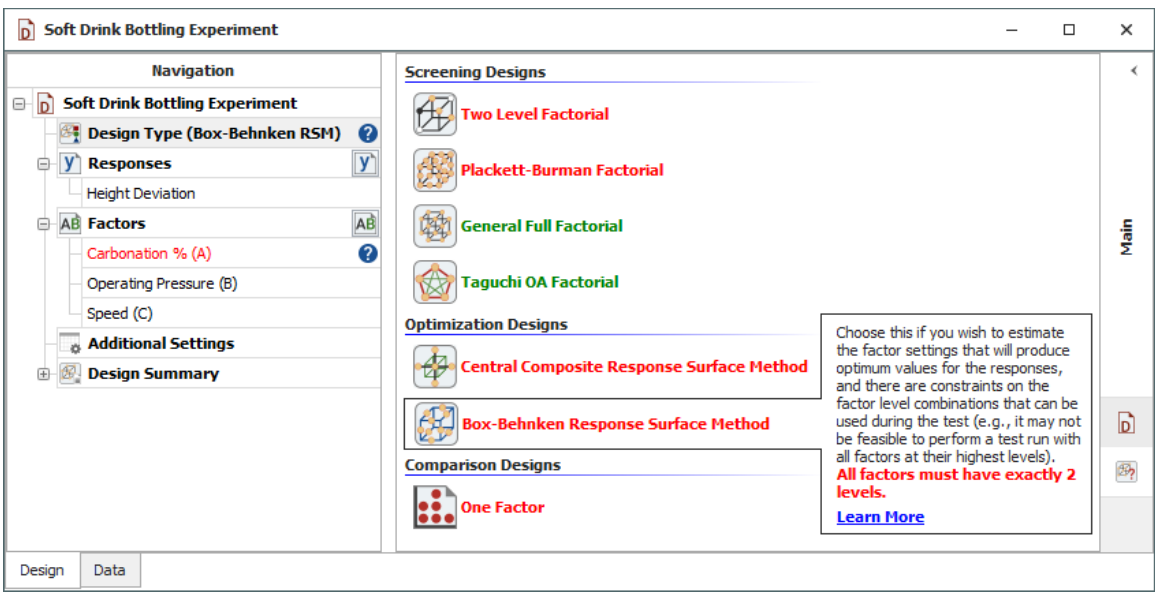
The following properties are available:

- **Name** is the label for the response.
- **Units** are the units of measurement that will be used for the response.
- **Response Type** specifies whether you will be using life data (e.g., failure times) for the response values. If you select **Life Data** from the drop-down list, the data sheet will be configured for reliability DOE. Note that only one response in a given design can contain life data.
- **Censoring** is applicable only if the response contains life data. The option you select from the drop-down list will determine which columns will be added to the Data tab so you can enter the selected type of censored data. For example, if you select **Intervals**, then the Data tab will include a Last Inspected column and a Time Failed column. (See Reliability Design Data Types.)
- The **Comments** field allows you to save notes or other text about the response. This information is not used in any calculations, and it can be accessed only via the input panel.

Adding, Removing and Editing Factors

Before building your design, make sure you have the correct number of factors and that each factor is defined appropriately.

Note that if a factor appears red in the navigation panel (as shown next), then it is incompatible with the selected design type. The reason will be displayed in red text next to the selected design.



Note: For robust design folios, factors are configured separately for the inner array and the outer array. For example, you could use two factors (e.g., pressure and viscosity) for the inner array and one factor (e.g., humidity) for the outer array. Similarly, for mixture design folios, factors are configured separately for mixture factors and process factors, if applicable.

ADDING AND REMOVING FACTORS

There are three ways to add or remove factors from a design.

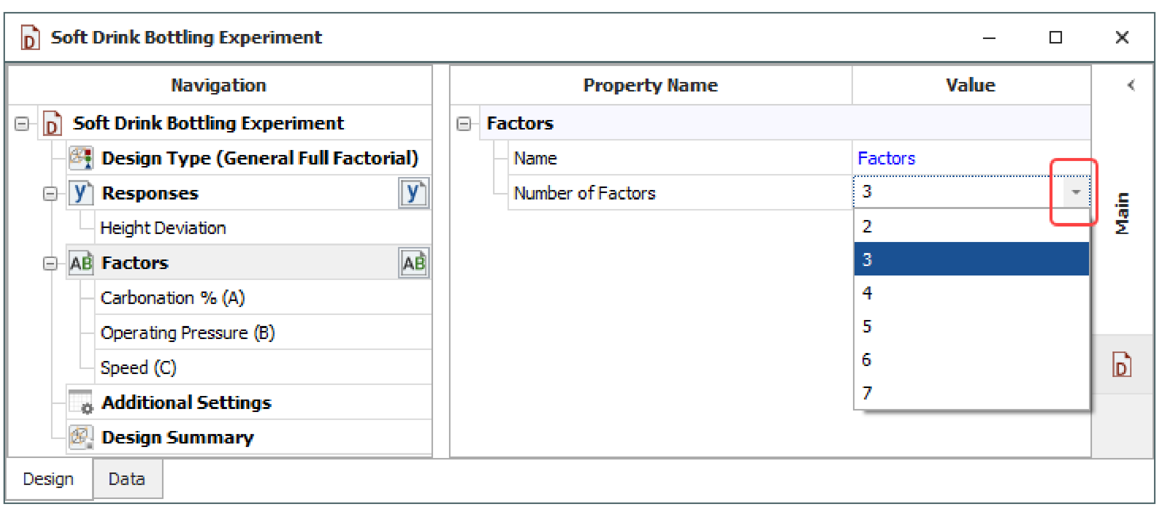
- You can choose **Design> Factors** > **Add Factor**.

  You can remove a factor by selecting it in the navigation panel and choosing **Design** > **Fact-ors** > **Remove Factors**.

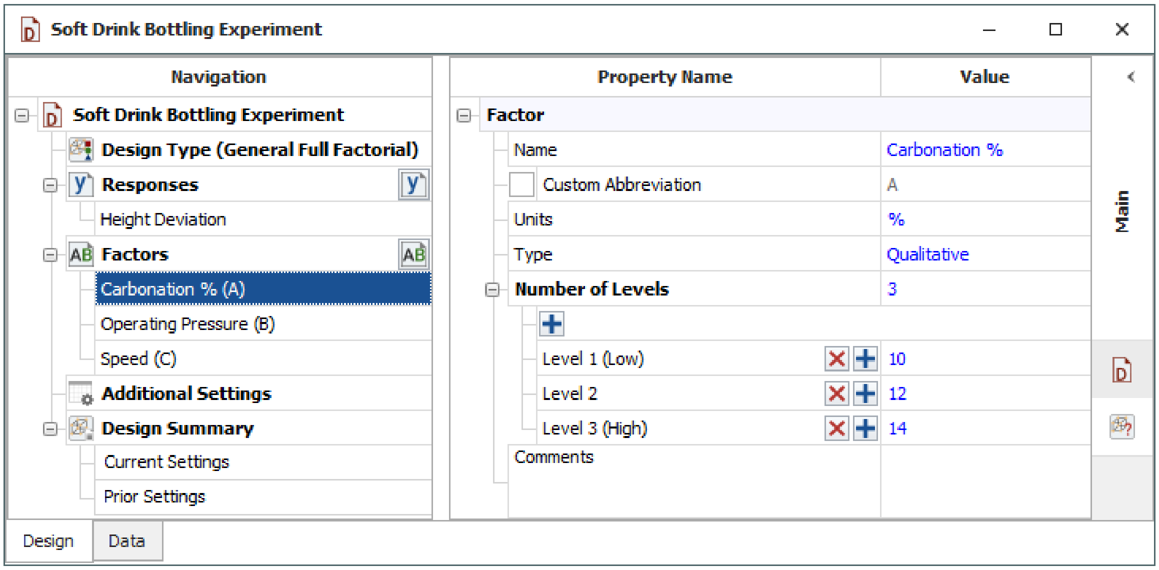
- You can also add/remove factors by right-clicking a factor and choosing the appropriate option from the shortcut menu.
- To quickly add or remove several factors at once, you can click the **Factors** heading and then change the number in the **Number of Factors** drop-down list in the input panel, as shown next.



Note than when you decrease the number of factors using the drop-down list, the factors at the bottom of the list will be deleted first.

EDITING FACTORS

You can define the properties of an individual factor by selecting it in the navigation panel and editing its properties in the input panel, as shown next for a standard design folio.



Alternatively, you can edit all the factors simultaneously by clicking the **Modify Factors** icon in the navigation panel ().

The following properties are available for factors in all design folios:

- **Name** is a label for the factor.
- **Custom Abbreviation** allows you to enter your own abbreviated name for the factor. If you clear this check box, the default abbreviation "A" will be used for the first factor, "B" will be used for the second factor, and so on. The abbreviated name is used when referring to interactions in plots and when specifying generators.
- **Units** are the units of measurement that will be used for the factor (e.g., "K" if the applied temperature is measured in kelvins).
- The **Comments** field allows you to save notes or other text about the factor. This information is not used in any calculations, and it can be accessed only via the input panel.

These properties are available only for factors in standard design and robust design folios.

- **Type** specifies whether the factor will be treated as having quantitative or qualitative values (e.g., "300" vs. "Small"). Note that predictions are possible only with quantitative factors.

- You can change the **Number of Levels** for the factor using the drop-down list, or by clicking the **[-]** and **[+]** icons.
  - For central composite designs, you can view the factor's **Range Based on Alpha Values**, which shows the full range of values that the factor will have in the experiment. Select **Define levels using range** to set the range directly.

These properties are available only for factors in mixture design folios.

- **Lower Bound** and **Upper Bound** set the bounds on the amounts of each mixture factor. You can change the lower bounds in extreme vertex designs only.

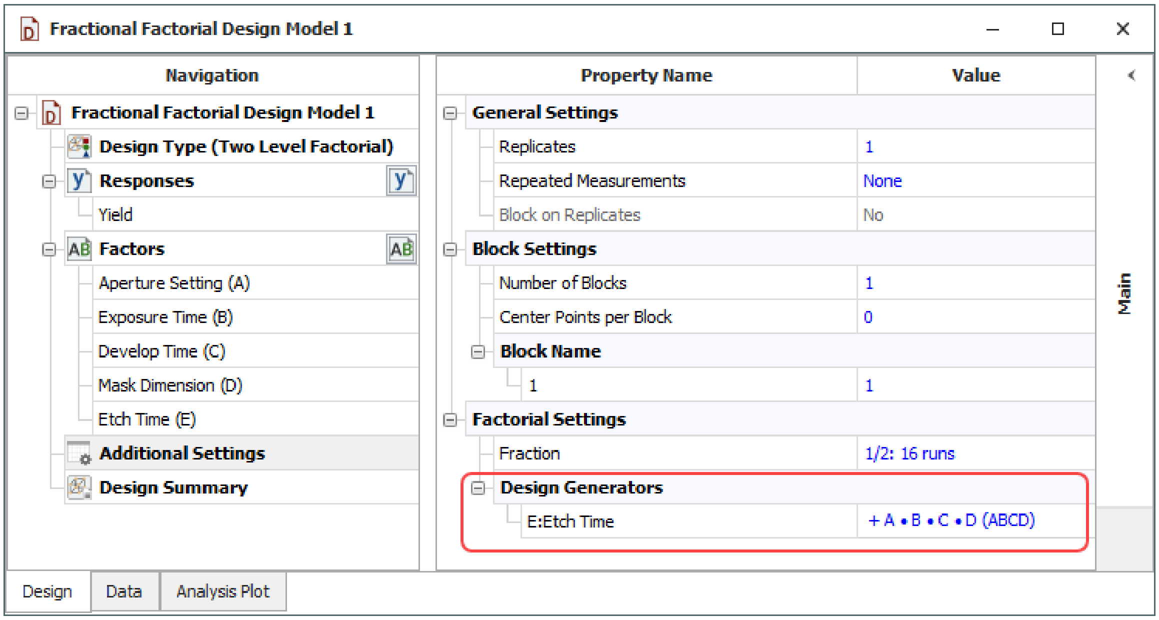
Specifying Generators

If you are working with a fractional factorial and/or blocked design, selecting the **Additional Settings** heading in the Design tab's navigation panel will allow you to specify *generators* for the design.

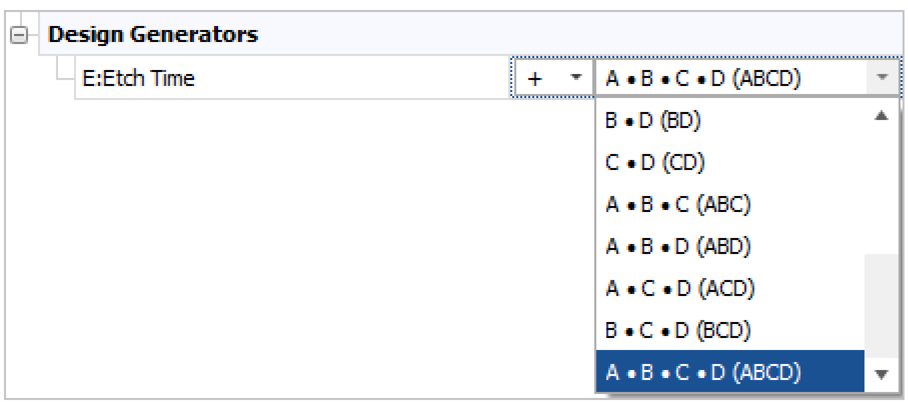
- *Fractional generators* are needed when working with a fractional factorial design. These generators are factorial interactions whose effects will be *aliased* (or *confounded*) with the main effects of particular factors. In other words, by specifying a generator, you determine which interaction effects will be indistinguishable from the main effects of certain factors.
- *Block generators* are needed when working with a blocked design (i.e., a design that divides runs into blocks in order to screen out nuisance factors). These generators are factorial interactions whose effects will be aliased with that of the nuisance factor.

This topic will focus on specifying fractional generators, but the underlying principles are the same for block generators.

In the example shown next, the main effect of "Etch Time" is aliased with the interaction of the first four factors. Note that the abbreviated names of the factors (e.g., "B" for "Exposure Time") are specified when you edit each factor. By default, the first factor is abbreviated as "A," the second as "B," and so forth.



The software automatically selects the fractional generators that maximize the design's resolution given the number of factors included in it. However, if you click inside the field(s) in the **Design Generators** area, you can change the sign of the generator using the first drop-down list. You can also use the second drop-down list to change the interaction that is used for the generator, as shown next.



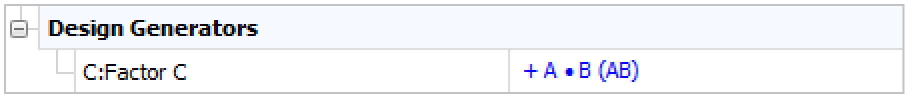
As an example, consider an experiment involving three factors (A, B and C) at two levels each (1 and -1). Running all possible combinations of factor settings without any replication requires eight runs, as shown next.

	Factor A	Factor B	Factor C
Run 1	-1	-1	-1
Run 2	1	-1	-1
Run 3	-1	1	-1
Run 4	1	1	-1
Run 5	-1	-1	1
Run 6	1	-1	1
Run 7	-1	1	1
Run 8	1	1	1

To create a half-fraction design (i.e., one with half as many runs), it will be necessary to alias the effect of one factor with the interaction of the other two. For example, if you are confident that there are no interaction effects between any of the factors, then you can alias C with the interaction of A and B, as shown next. With this alias structure, the test design will allow you to evaluate all three main effects (but no interactions).

	Factor A	Factor B	Factor C = Factor A x Factor B
Run 1	-1	-1	1
Run 2	1	-1	-1
Run 3	-1	1	-1
Run 4	1	1	1

The selected generator for this case is shown next.



Alternatively, you could use the rule C = -AB, aliasing the effect of Factor C with the negative (or opposite) effect of the interaction of the other two factors, as shown next.

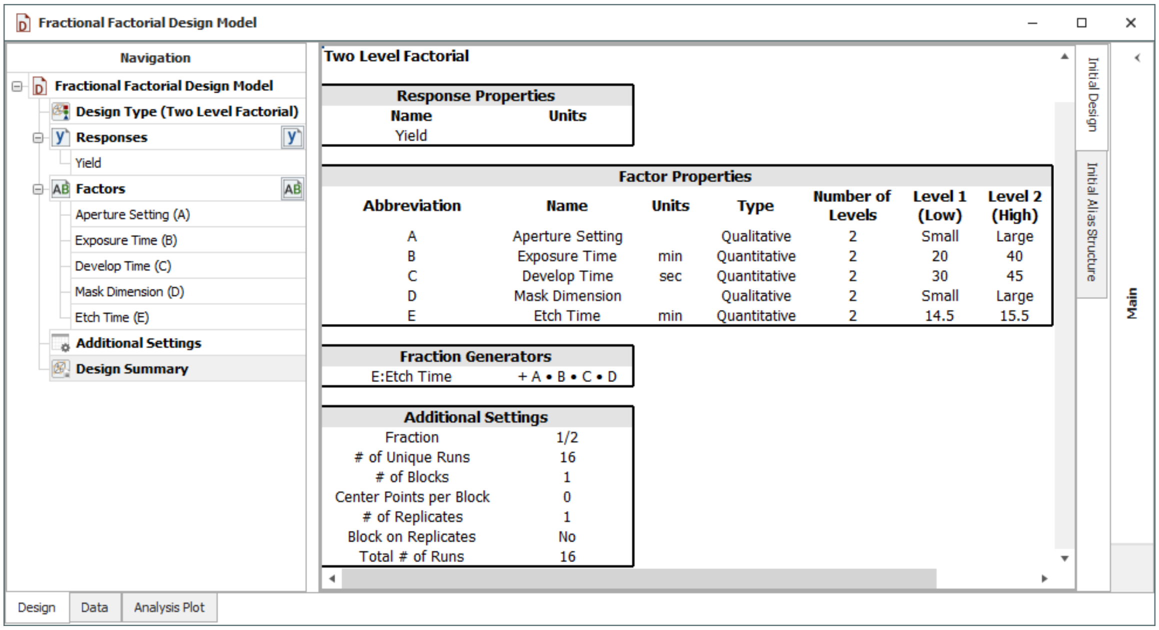
	Factor A	Factor B	Factor C = –(Factor A x Factor B)
Run 1	-1	-1	-1
Run 2	1	-1	1
Run 3	-1	1	1
Run 4	1	1	-1

To use the negative effect of the factors, change the sign of the generator as shown next:

Design Summary

The Design Summary provides details for the settings currently selected on the Design tab, including the design type, response/factor details, blocks, etc. If the folio includes an experiment plan (i.e., Data tab) that is no longer in sync with the Design tab settings, a second "Prior Settings" summary will provide a record of the design settings that were used to build the experiment.

The Design Summary is accessed by clicking the **Design Summary** heading in the navigation panel of the Design tab or by clicking the **Detailed Summary** link on the control panel. An example summary is shown next.

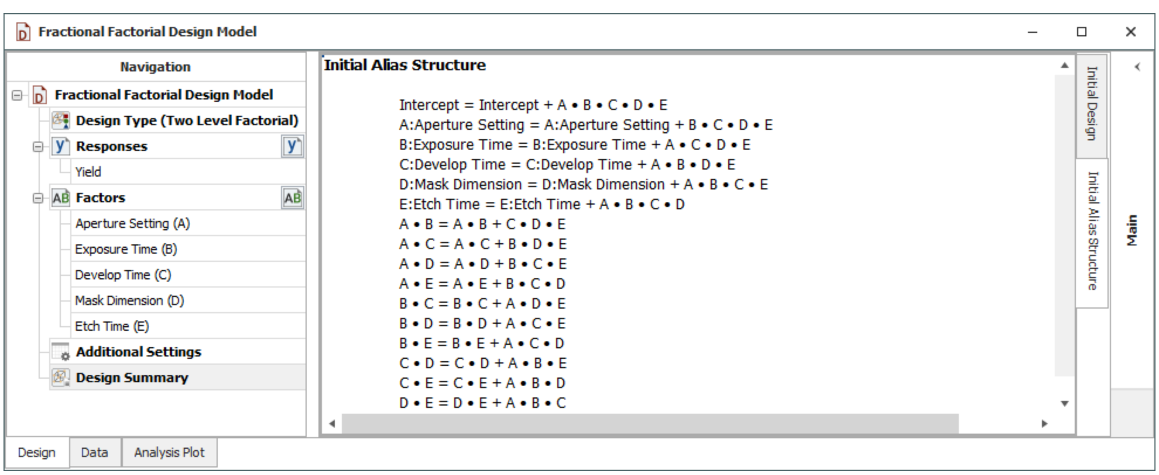


Note: For extreme vertex designs, information on the number of runs will not be available until you have validated the constraints.

ALIAS STRUCTURE FOR TWO LEVEL FACTORIAL DESIGNS

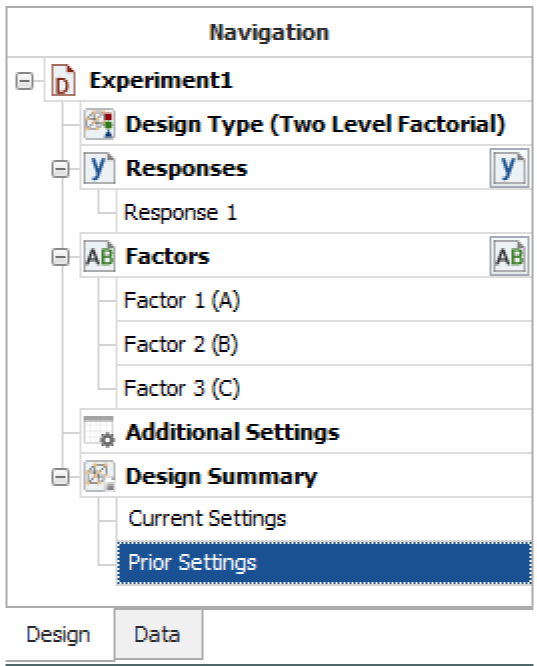
If you are using a two level factorial design with aliased factors (i.e., two level fractional factorial design), the summary will consist of two pages, accessed by clicking two tabs. The first tab will display the design properties summary and the second tab will display the initial alias structure, which describes which effects are indistinguishable from other effects in the current design. The alias structure is set automatically, but it can be changed by manually specifying generators.

The initial alias structure of a design is shown next.



PRIOR SETTINGS SUMMARY

If the Design tab is out of sync with the Data tab (as indicated at the top of the Design tab control panel), two summaries will be available under the **Design Summary** heading. The Current Settings summary provides information about the settings currently selected on the Design tab (as described above). The Prior Settings summary provides a record of the design settings that were originally used to build the experiment plan.



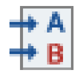
If any changes were made to the experiment plan via the Data tab (e.g., runs were manually deleted), the Prior Settings summary will describe the experiment as "customized." In other words, the Design tab settings that were originally used to create the experiment may no longer accurately represent it. In addition, depending on the change that was made, it may no longer be possible to recreate the current experiment using the Design tab settings (e.g., the experiment may no longer fit into any of the basic design types). For these reasons, the Design tab will be considered permanently out of sync with the current experiment.

If no changes were made via the Data tab, then you can use the Prior Settings summary as a reference while you modify the settings on the Design tab. For example, if you want to undo some of the changes you've made to the Design tab settings since the experiment plan was built, you can compare the two summaries to see what has changed, or you can choose **Design > General > Undo Changes** to quickly bring the Design tab back in sync with the Data tab (i.e., the Design tab settings will once again accurately describe the built experiment).

Changing Design Settings

After building a design (i.e., creating a Data tab in the design folio), you may wish to change some of its settings (e.g., the number of replicates that are used, or the number of factors). There are several ways to do this:

- In most cases, you can return to the Design tab of the folio, change the settings and then rebuild the design by choosing **Design > General > Build Design**.

  In the window that appears, select whether you wish to create a new folio for the modified design, or whether you wish to overwrite the experiment plan on the Data tab. If you select the **Maintain response data** check box, the response data from the prior design will be included in the modified design.

  Note that when you modify the experiment using the Design tab, two design summaries will become available under the **Design Summary** heading. The Current Settings summary shows the settings that are currently selected on the Design tab. The Prior Settings summary shows the settings that were originally used to build the experiment on the Data tab.

  - If, after changing the settings on the Design tab, you want to revert to the settings that were originally used, choose **Design > General > Undo Changes**.

  - You can also choose **Design > General > Clear Settings** to put the Design tab in its original state (i.e., using default settings that change depending on the design type you choose).

- If you wish to make changes that aren't possible with the Design tab settings, you can customize the experiment by:

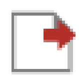
  - Manually changing any of the values (e.g., factor levels) that are shown on the Data tab. To do this, simply click inside a cell, change the value and press **ENTER**.

- Using one of the following features to modify the experiment automatically.
  - Use the augment design feature to add new blank rows to the Data tab, or to add replicates of the entire design.
  - Use the fold design feature to add a replicate to the design with runs that use the opposite levels for all the factors you specify.
  - Use the tool for creating an optimal design. The new design will have fewer runs, and when creating the design the software can consider the number of runs you wish to use, the effects you wish to investigate and which factor level combinations you wish to use in the experiment.

Importing/Exporting Design Settings

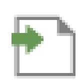
When the Design tab of a design folio is active, you can choose to save the current design settings or load the settings from a design that was previously saved.

- To save all your current design settings for use in future designs, choose **Design > Import/Export > Export Template** or click the icon on the control panel.

  Then specify the name and location of the experiment file (*.dex) that will store all the current design settings.

- To overwrite all the settings of your current design and use the settings from an experiment file instead, choose **Design > Import/Export > Import Template** or click the icon on the control panel.

  Then select the *.dex file that has all the desired settings.

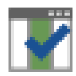
Data Tab

After you build a design with the Design tab, you can use the Data tab to enter response data and change settings in the built design. By analyzing the data, you can investigate the relationship between factors and responses. Depending on the data you provide and the type of design you use, you may then be able to use this information to determine the optimal response and/or predict response levels using the analysis results.

Data Tab Columns

After you build a design, the Data tab will include a test plan with Response columns for entering response data. Each cell in a Response column is used to record the response value obtained at the combination of factor settings (green columns) specified in the current row. Note that robust parameter designs work differently; for more information on using these designs, see Using Robust Parameter Designs.

The columns shown in the test plan will vary depending on the design type you are working with. To hide/show columns, choose **Data > Format and View > Visible Columns** or click the **Visible Columns** icon on the control panel.



The available columns are described below.

- **Standard Order** displays the basic order of runs, as specified in the design type, without randomization. This pattern is provided as a reference (e.g., textbook examples often use the standard order).

  Tip: If you need to use a non-randomized order in an actual experiment, you can sort the design by clicking a cell in the column you wish to sort by and choosing **Sheet > Sheet Actions > Sort > [Order]**. For example, if changing levels in a randomized experiment would be too expensive, you can sort the design table by factor levels.

- **Run Order** displays the randomized order generated by the software. By conducting experiments using a randomized order, you can reduce the effect of factors not being investigated.
- **Point Type** indicates what kind of point the run is. 0 indicates that the run is a center point. 1 is used for all other points.
- **Block Value** displays the block to which the run belongs. If the design is not blocked, all rows will display the same default value.
- **Life Data Columns** are present only for reliability DOE. The columns that are available depend on the type of life data specified when the design was created. These columns take the place of the response column(s) in a standard design, and are used to calculate the underlying response for the design, which is considered to be time.

- **Time Failed Column** allows you to enter the failure time for each unit. This column is available only for times-to-failure data with no censoring (complete data).
- **State F or S Column** allows you to specify whether the unit is failed (F) or suspended (S). This column is available only for data with right censoring.
- **Time to F or S Column** allows you to enter the failure time for the unit or, in the case of suspended units, the time at which the unit was removed from the test while still operating. Note that you can enter a negative number in this column and it will automatically be marked as a suspension in the State F or S column. This column is available only for data with right censoring.
- **Last Inspected Column** allows you to enter the last time the unit was inspected prior to the time it failed. This column is available only for data with interval and/or left censoring.
- **State End Time Column** allows you to enter the time at which the unit's state was discovered (i.e., the first inspection time after the unit's failure). This column is available only for data with interval and/or left censoring and without right censoring.

To change the type of life data used in the experiment, use the Alter Data Type window.

- **Factor Columns** display the settings used for each factor in the run. By default, these columns are titled using the format "X: Factor Name," where X is the abbreviated name. If you have defined the units for the factor, that will also be displayed in the column heading. Factor columns are shown in green by default.

  Factor levels can be displayed as the actual values assigned in the factor properties or as coded values. When you use coded values for two level factorial or Plackett-Burman designs, the low level will be displayed as -1 and the high level will be displayed as 1. When you use coded values for general full factorial or Taguchi OA factorial designs, the lowest level will be displayed as 1, the next level as 2, and so on. To switch between coded and actual values, choose **Data > Display > [Value Type]**.
- **Ignore / Include** allows you to specify whether the run will be included in the analysis. You can also enter an asterisk (*) in a response column to ignore that data point.
- **Comments** allows you to enter notes or other text for each run. This information is not used in any calculations.

If your design includes only one observation per test run, the following columns may be available:

- **Response Columns** allow you to enter the response(s) for each test run. A response column heading will include the response name and (if specified) units you have defined on the Design tab.

Each column contains a check box in the column heading that specifies whether the response will be included in the analysis. If a column is selected, the color of the check box border will indicate the status of that response's analysis. A green border indicates that the response has been analyzed using the current analysis settings; a red border indicates that it has not. Note that if you clear the check box, you will not be able to specify analysis settings for the response, nor will you be able to display it in plots or use it in optimization.

- **Response Standard Deviation Columns** are available only when you select to perform variability analysis. These columns are populated by the software when the data set is analyzed, and they show the standard deviation of the selected response at each factor level combination. (Asterisks indicate that the standard deviation was shown in a previous row.)

If your design includes multiple measurements per test run, the following columns will be available:

- **Repeated Measurements Columns** are available if you chose to include repeated measurements under the Additional Settings on the Design tab. The measurements for each test run are used to generate an average and standard deviation value for that run, either of which can be selected as the analyzed response for the experiment.
- The **YMean** and **Y Standard Deviation Columns** are populated by the software when the data set is analyzed. They show the mean and standard deviation of the measurements taken during each test run.

Data Tab Control Panel

The design folio control panel allows you to control the settings for analyzing response data and evaluating an experiment design. It also displays the results of the analysis/evaluation. The panel is composed of the following pages:

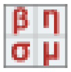
- The **Main Page** contains most of the tools you will need to analyze response data.
- The **Analysis Settings Page** contains the analysis settings that are also displayed on the Main page.
- The **Evaluation Settings Page** contains the settings that are used when you perform a design evaluation.
- The **Comments Page** allows you to enter notes or other text that will be saved with the folio.
- If your design includes a life data response, the **Identifiers** and **Publishing** pages will also be available. These pages allow you to publish the results as a model.

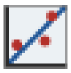
MAIN PAGE

The Main page of the control panel includes the following tools and areas.

Folio Tools

The folio tools are arranged on the left side of the control panel. Use these tools to configure your analysis and experiment with the analysis results. Depending on the type of design you are working with, the control panel may contain some or all of the following tools:

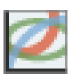

Calculate analyzes the data for each response that is selected to be included in the analysis. To exclude a response from the analysis, clear the check box in its column heading.


Plot creates a plot based on the analysis. If you click this icon before the current data set has been analyzed, an analysis will be performed automatically.

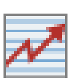

Select Terms opens a window that allows you to specify the terms (i.e., factors, factorial interactions, blocks and quadratic effects) to be considered for each response.


View Analysis Summary opens a window that contains detailed information about current and past analysis results.


Optimization adds an optimal solution plot to the Optimization folio that is linked to the design folio. If no linked Optimization folio exists, the software will create one.


Overlaid Contour Plot adds an overlaid contour plot to the Optimization folio that is linked to the folio. If no linked Optimization folio exists, the software will create one.


Select Transformation opens a window that allows you to select a transformation to apply to each response.


Prediction opens a window that allows you to enter values for each factor and see predicted results for the selected response.


Transfer to Simulation Worksheet transfers the factor settings to a new simulation worksheet.

Visible Columns opens a window that allows you to choose which columns to hide/show in the Data tab.

Transfer Data to Life Data Folio is available only for reliability designs. It transfers the data from the design folio to a new life data folio. The factor settings are noted in a subset ID column in the new folio.

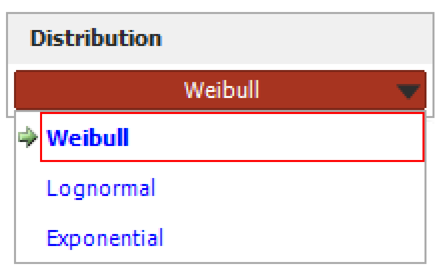
Transfer Data to Life-Stress Folio is available only for reliability designs. It transfers data from the design folio to a new life-stress folio. The factor settings are entered in stress columns; each factor is considered a separate stress.

Settings Area

The Main page of the control panel may also contain some or all of the following settings. Information and examples on how to use these settings for particular designs are available in the documentation for the various design types.

- The **Response** drop-down list is used to select the response that all the displayed settings and analysis results will apply to.

- The **Distribution** area is used to select a life distribution for reliability DOE.



If you already have a reliable estimate for the selected distribution's shape parameter, you can fix the parameter at that value by choosing **Data > Design > Set Shape Parameter**.

Setting this parameter can reduce the uncertainty in your results, since the analysis will solve for fewer variables.

- The **Settings** area is used to view the available analysis settings, which vary depending on the design type. The following picture shows an example of the settings that are available in a two level factorial design.

Settings	
Partial SS	Grouped Terms
Risk Level: 0.1	
Transformation: Y' = Y	
Analyzed	
12 Observations	

In this example:

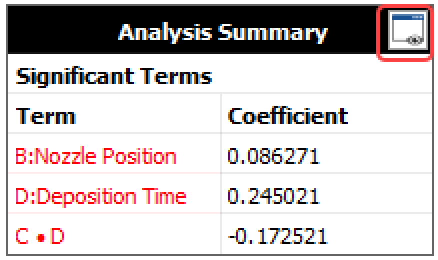
- The partial sum of squares ("Partial SS") and grouped terms are used.
- The specified risk level is 10%.
- No transformation is used (i.e., the "transformed" response, *Y'*, is equal to the non-transformed response, *Y*).
- "Analyzed" is shown in green, which indicates that the current response data has been analyzed using the current settings. Otherwise, this would show "Modified" in red.
- The last row indicates that the analyzed data consists of 12 observations.

To configure the settings used in the analysis, click the blue text. These settings are also accessible on the Analysis Settings page of the control panel. See the Design Folio Control Panel Analysis Settings Page section for a description of all available settings.

Note: If you are using a robust parameter design and the selected response is signal noise ratio, this area will also include a row to indicate which kind of signal noise ratio is being analyzed (i.e., nominal, smaller or larger).

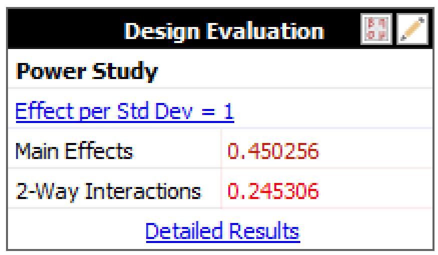
Analysis Summary Area

The **Analysis Summary** area is shown when the current response data has been analyzed using the current settings. This area displays the terms that were found to be significant, as well as their associated regression coefficients. Click the **View Analysis Summary** icon in this area to view the details of the analysis results (including an ANOVA table).



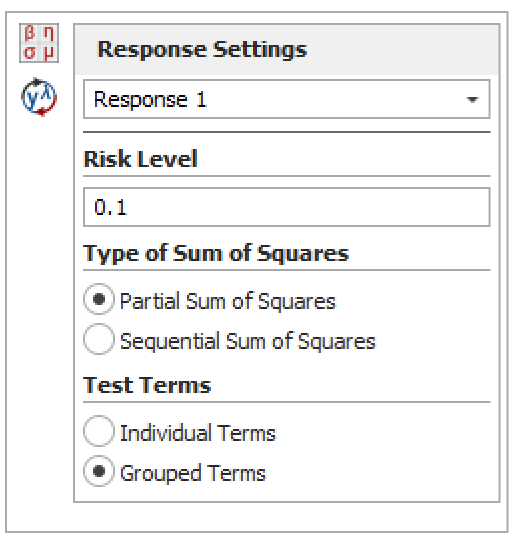
Design Evaluation Area

The **Design Evaluation** area is used to help you determine the efficiency of the current design. In the picture shown next, the main effect that is the least likely to be detected by design has a 45.02% of being detected. Click the **Detailed Results** link in this area to view details of the evaluation results (including the probability of detection, or *power*, for individual effects and interactions).



ANALYSIS SETTINGS PAGE

The Analysis page of the design folio control panel includes additional settings that may be applicable for the current analysis. It may contain some or all of the following options (which can also be changed on the Main page of the control panel).



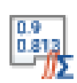
- The **Response Settings** drop-down list is used to choose the response that all the visible settings and analysis results will apply to. When you choose a response here, it will automatically be chosen when you return to the Main page.
- The **Risk Level**, or alpha value, is a measure of the risk that the analysis results are incorrect (i.e., alpha = 1 - confidence level).
- The next area allows you to specify the **Type of Sum of Squares** to use in the analysis. Select the **Partial Sum of Squares** option to test if a term is significant given that all other terms are considered in the model. Select the **Sequential Sum of Squares** option to test if an additional term is significant given that the terms before it are already in the model. This area is not applicable to R-DOE.
- The **Test Terms** area specifies what the information in the ANOVA table in the Design Folio Analysis Results applies to. If you select **Individual Terms**, the software will separately examine the effects of each individual factor and/or factorial interaction. If you select **Grouped Terms**, it will examine groups of effect types, such as main effects, two-way interactions, etc.

You can also analyze the selected response or open the Select Transformation window by clicking the icon to the left of the **Response Settings** area.

Design Folio Analysis Results

When accessed from a design folio, the Analysis Summary window will contain detailed information about analysis results, including information that describes how each factor and factorial interaction affects the variation of the response that is currently selected in the Data tab control panel.

If the current response data has been analyzed, you can open the window by clicking the **View Analysis Summary** icon on the control panel.



If the current response data have not been analyzed, the icon will still be available so you can view the folio's analysis history.

ANALYSIS RESULTS

Depending on the type of design you are working with, the analysis results may contain some or all of the following:

The **Analysis of Variance (ANOVA) table** provides general information about the effects of the factor(s) and factorial interactions on the selected response. For designs with multiple factors, this information may be presented for individual factors and interactions or for groups of factors and interactions, depending on your analysis setting on the control panel.

ANOVA Table Columns

- **Source of Variation** is the source that caused the difference in the observed output values. This can be a factor, factorial interaction, curvature, block, error, etc. If your design includes more than one factor and you have selected to use grouped terms in the analysis (specified on the Analysis Settings page of the control panel), the effects will be grouped by order (i.e., main effects, two-way interactions, etc.). Sources displayed in red are considered to be significant.

- The number of **Degrees of Freedom** for the **Model** is the number of regression coefficients for the effects included in the analysis (e.g., two coefficients might be included in the regression table for a given main effect). The number of degrees of freedom for the **Residual** is the total number of observations minus the number of parameters being estimated.

- **Sum of Squares** is the amount of difference in observed output values caused by this source of variation.

- **Mean Squares** is the average amount of difference caused by this source of variation. This is equal to Sum of Squares/Degrees of Freedom.

- **F Ratio** is the ratio of Mean Squares of this source of variation and Mean Squares of pure error. A large value in this column indicates that the difference in the output caused by this source of variation is greater than the difference caused by noise (i.e., this source affects the output).

- **P Value** (alpha error or type I error) is the probability that an equal amount of variation in the output would be observed in the case that this source does not affect the output. This value is compared to the risk level (alpha) that you specify on the Analysis Settings page of the control panel. If the *p* value is less than alpha, this source of variation is considered to have a significant effect on the output. In this case, the term and its *p* value will be displayed in red.

- The following values are shown underneath the ANOVA table, and they indicate how well the model fits the data:

  - **S** is the standard error of the noise. It represents the magnitude of the response variation that is caused by noise. Lower values indicate better fit.

- **R-sq** is the percentage of total difference that is attributable to the factors under consideration. It is equal to Sum of Squares(factor)/Total Sum of Squares. Higher values usually indicate better fit.
- **R-sq(adj)** is an R-sq value that is adjusted for the number of parameters in the model. Higher values indicate better fit.

The **Data Summary table** is available only for one factor designs. It gives the mean and standard deviation of the output at each level of the factor.

Data Summary Table Columns

- **Factor Level** is the name of the qualitative level.
- **Number in Level** is the number of data points obtained at the factor level.
- **Estimated Mean** is the average of the data points obtained at the level.
- **Standard Deviation** is the standard deviation of the data points obtained at the level.

The **Mean Comparisons table** is available only for one factor designs. It provides information on comparisons between factor levels. The table includes the following columns:

Mean Comparisons Table Columns

- **Contrast** gives the paired comparison of any two levels. Level 1 - Level 2 means the difference between Level 1 and Level 2. Contrasts displayed in red are considered to be significant.
- **Mean Difference** is the mean value of the difference in output between the two levels.
- **Pooled Standard Error** is the standard error of the mean difference in output between the two levels.
- **Low Confidence** is the lower confidence bound of the mean difference.
- **High Confidence** is the upper confidence bound of the mean difference.
- **T Value** is the normalized difference, which is equal to Mean Difference/Pooled Standard Error.
- **P Value** is the probability that an equal amount of variation in the output would be observed in the case that there is no significant difference between the levels. This value is compared to the risk level (alpha) that you specify on the Analysis Settings page of the

control panel. If the *p* value is less than alpha, there is considered to be a significant difference between the levels. In this case, the contrast and its *p* value will be displayed in red.

The **Regression table** provides specific information on the contribution of each factor or factorial interaction to the variation in the response and an analysis of the significance of this contribution.

Note: For each factor with n levels, n-1 effects are estimated (e.g., one effect is estimated for a two level factor, four effects are estimated for a five level factor). For a two level factor, the effect is the difference of the average response at the two levels. For factors with more levels, the average response at one level is considered to be a baseline and the average responses at other levels are compared with that baseline. If more than one effect exists for a factor, the effects are differentiated with a number along with the factor's letter designation (e.g., A[1], A [2]).

Regression Table Columns

- **Term** is the factor, factorial interaction, curvature, block, etc. under consideration. Terms displayed in red are considered to be significant. In cases where there is no error in the model, significant effects are determined according to Lenth's method and the term names are displayed in red and followed by an asterisk (*).
- **Effect** is a measure of how much the response value (Y) changes when the value of the corresponding term in the model changes from the low level to the high level.
- **Coefficient** is the regression coefficient of the term, which represents the contribution of the term to the variation in the response.
- **Standard Error** is the standard deviation of the regression coefficient.
- **Low Confidence** and **High Confidence** are the lower and upper confidence bounds on the regression coefficient.
- **T Value** is the normalized regression coefficient, which is equal to Coefficient/Standard Error.
- **P Value** (alpha error or type I error) is the probability that an equal amount of variation in the output would be observed in the case that this term does not affect the output. This value is compared to the risk level (alpha) that you specify on the Anaysis Settings page of the control panel. If the *p* value is less than alpha, this source of variation is considered to

have a significant effect on the output. In this case, the term and its *p* value will be displayed in red.

The **Likelihood table** is available only for reliability designs. It provides general information about the factor's effects on the times-to-failure.

Likelihood Table Columns

- **Model** displays the model for which the results apply.
  - **Reduced** assumes that the product life is the same at different levels of the factor.
  - **Full** assumes that the product life is different at different levels of the factor.
- **Degrees of Freedom** is the degrees of freedom of this source of variation. This is also the number of parameters in the model for this source.
- **Ln(Likelihood Value)** is the logarithm transformation of the likelihood value for this source of variation.
- **Likelihood Ratio** is the likelihood ratio value for this source of variation.
- **P Value** is the probability that LR is from a chi-squared distribution, which would indicate that the factor has no effect on product life. This value is compared to the risk level (alpha) that you specify on the Analysis Settings page of the control panel. If the *p* value is less than alpha, then the factor is considered to have a significant effect on product life. In this case, the effect and its *p* value will be displayed in red.

The **MLE Information table** is available only for reliability designs. It provides specific information on the contribution of each factor or factorial interaction to the variation in the times to failure and an analysis of the significance of this contribution.

MLE Information Table Columns

- **Term** is the factor, factorial interaction, etc. under consideration. Terms displayed in red are considered to be significant.
- **Effect** is a measure of how much the response value (Y) changes when the value of the corresponding term in the model (using coded values) changes from -1 to 1.
- **Coefficient** is the regression coefficient of the term, which represents the contribution of the term to the variation in the response.
- **Standard Error** is the standard deviation of the regression coefficient.

- **Low Confidence** and **High Confidence** are the confidence bounds on the regression coefficient, using Fisher bounds.
- **Z Value** is the normalized regression coefficient, which is equal to Coefficient/Standard Error.
- **P Value** is the probability that an equal amount of variation in the output would be observed in the case that this source does not affect the output. This value is compared to the risk level (alpha) that you specify on the Analysis Settings page of the control panel. If the *p* value is less than alpha, this source of variation is considered to have a significant effect on the output. In this case, the term and its *p* value will be displayed in red.

The **Life Characteristic Summary table** is available only for one factor reliability designs. It gives the characteristic life and standard deviation for the product at each factor level, along with lower and upper confidence bounds.

Life Characteristic Summary Table Columns

- **Factor Level** is the name of the level.
- **Number in Level** gives the number of failures (F) and suspensions (S) in the level.
- The next column is the characteristic life at that level.
  - **Eta** is shown for the Weibull distribution, and it is equal to the time at which unreliability = 63.2%.
  - **Ln-Mean** is shown for the lognormal distribution, and it is equal to the time at which unreliability = 50%.
  - **MTTF** (i.e., the mean time to failure) is shown for the exponential distribution.
- The **Standard Deviation** is also shown for the characteristic life.
- **Low Confidence** and **High Confidence** give the two-sided confidence bounds on the characteristic life, based on the risk level entered on the Analysis Settings page of the control panel (e.g., if the risk level is 0.1, then 90% two-sided bounds will be shown).

The **Life Comparisons table** is available only for one factor reliability designs. It provides information on comparisons between levels of the factor, allowing you to determine whether one particular level is significantly different from another.

Life Comparisons Table Columns

- **Contrast** gives the paired comparison of any two levels. Level 1 - Level 2 means the difference between Level 1 and Level 2. Contrasts displayed in red are considered to be significant.
- **Mean Difference** is the mean value of the difference in output between the two levels.
- **Pooled Standard Error** is the standard error of the mean difference in output between the two levels.
- **Low Confidence** is the lower confidence bound of the mean difference.
- **High Confidence** is the upper confidence bound of the mean difference.
- **T Value** is the normalized difference, which is equal to Mean Difference/Pooled Standard Error.
- **P Value** is the probability that an equal amount of variation in the output would be observed in the case that there is no significant difference between the levels. This value is compared to the risk level (alpha) that you specify on the Analysis Settings page of the control panel. If the *p* value is less than alpha, there is considered to be a significant difference between the levels. In this case, the contrast and its *p* value will be displayed in red.

The **Regression Equation** information is presented using multiple tables. The available tables will vary depending on the design type you are working with. The results that could be available include:

Regression Equation Tables

- The **Response** table displays the response that the regression equation applies to and the units of measurement that were entered for the response (if any).
- The **Additional Settings** table shows the transformation and risk level you entered for the response.
- The **Significant Terms** table is applicable only when at least one term was found to be significant. It shows the significant terms in the Name column and the associated regression coefficients in the Coefficient column.
- The **Equation** tables show the regression coefficients for the model of the selected response. For example, consider this table:

Equation (Coded values)	
Response 1 =	
9.3903	
-0.8467	A:Factor 1
+1.2813	B:Factor 2
+0.7307	A • B

The corresponding model for this table is $y = 9.3903 - 0.8467x_1 + 1.2813x_2 + 0.7303x_1x_2$.

- The first Equation table shows the coefficients for a model that uses coded values for the factors.
- The second Equation table shows the coefficients for a model that uses the actual factor values.

ADDITIONAL RESULTS

All of the following tables provide information that was generated from the main calculations. The available tables will vary depending on the design type you are working with. The results that could be available include:

Alias Structure

This item is available for all designs with at least two factors. It describes the alias structure for the design, taking into account only the terms you've selected to include in the analysis. Together with your engineering knowledge, you can use this information to help determine whether any important interaction information was lost due to aliasing. When aliased terms exist, the following areas will be shown:

- **Terms selected to be in the model** lists all the terms that are considered for inclusion in the regression model (i.e., the selections in the Select Terms window).
- **Terms included in the model** lists all the selected terms that are included in the model. The alias structure determines which terms are excluded.
- **Alias Structure** lists the aliased effects based on the selected terms. For example, A • B = A • B + C • D means the interaction effect A • B is aliased because it is indistinguishable from effect C • D. Therefore, the model cannot include both interaction terms; it will include only one (e.g., A • B).

Alias Summary

The terms in the first column of this table are aliased with the terms shown in the second column. Only the terms in the first column are included in the model.

Var/Cov Matrix

This shows the variance/covariance matrix, which is available for one factor R-DOE designs and all other designs with two or more factors. The diagonal elements in this matrix are used to calculate the coefficients in the MLE or Regression Information table.

Diagnostic Information

This table is available for one factor R-DOE designs and all other designs with two or more factors. It displays various analysis results for each run and highlights significant values. The following columns are included:

- **Run Order** is the randomized order, generated by the software, in which it is recommended to perform the runs to avoid biased results. Note that any changes made to the Run Order column on the Data tab will be reflected here.

- **Standard Order** is the basic order of runs, as specified in the design type, without randomization. Note that any changes made to the Standard Order column on the Data tab will be reflected here.

- **Actual Value (Y)** is the observed response value for the run, as entered in the response column on the Data tab.

- **Predicted Value (YF)** is the response value predicted by the model given the factor settings used in the run.

- **Residual** (or "regular residual") is the difference between the actual value (Y) and the predicted value (YF) for the run.

- **Standardized Residual** is the regular residual for the run divided by the constant standard deviation across all runs.

- **Studentized Residual** is the regular residual for the run divided by an estimate of its standard deviation.

- **External Studentized Residual** is the regular residual for the run divided by an estimate of its standard deviation, where the run in question is omitted from the estimation.

- **Leverage** is a measure of how much the run influences the predicted values of the model, stated as a value between 0 and 1, where 1 indicates that the actual response value of the run is exactly equal to the predicted value (i.e. the predicted value is completely dependent upon the observed value).

- **Cook's Distance** is a measure of how much the output is predicted to change if the run is deleted from the analysis.

Values that are considered to be significant, or outliers, are displayed in red. For the residual columns, significant or critical values are those that fall outside the residual's upper or lower bounds, calculated based on the specified alpha (risk) value.

The ReliaWiki resource portal has more information on how significant values are determined for the Leverage and Cook's Distance columns at: http://www.reliawiki.org/index.php/Multiple_Linear_Regression_Analysis.

Least Squares Means

This table shows the predicted response values for the given factor levels. It includes the following columns:

- **Effect** is the main effect or interaction used to predict the response. The coefficients for effects not used in the prediction are set to zero.
- **Level** is the combination of factor levels used to predict the response.
- **Mean** is the predicted response value.

Regressor

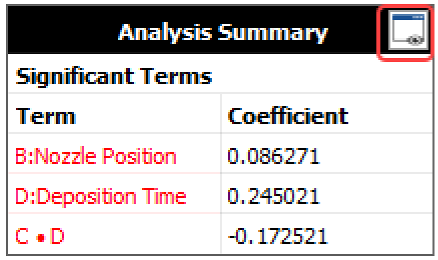
This table is available only for mixture designs. It shows the values of each term used in the regression equation (whereas the Regression Equation table shows the coefficients).

Analysis Summary Window

The Analysis Summary window allows you to display a detailed summary of current and past analysis results. This window is available from any of the following folios:

- The standard, robust or mixture design folios
- The free form folio
- The multiple linear regression folio
- The one-way ANOVA folio
- All of the measurement system analysis folios

To access the window, click the **View Analysis Summary** icon.



In addition to a spreadsheet that displays analysis information, the window includes two panels and a ribbon, as described next.

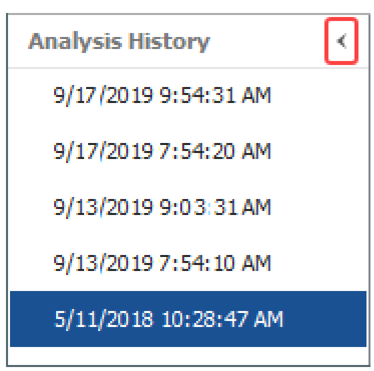
AVAILABLE REPORT ITEMS PANEL

The Available Report Items panel is used to determine which analysis results will display in the spreadsheet. The items you see in this panel will vary depending on the design type and/or folio that you are working with. If desired, you can edit the results anywhere in the spreadsheet by clicking a cell. These changes can then be copied to the Windows Clipboard or exported to an Excel spreadsheet using the ribbon.

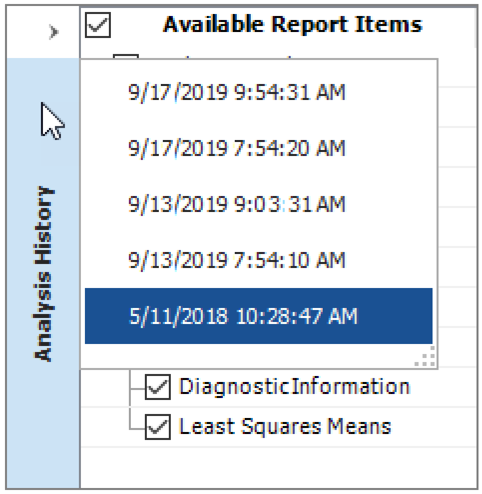
ANALYSIS HISTORY PANEL

The Analysis History panel includes a history of all the results for all analyses that have been performed in the folio. To display the results of a prior analysis, click the date and time of that analysis in the list.

You can switch between the panel's hidden and displayed states by clicking the **Hide/Show** icon in its title bar.



When the panel is in its hidden state, a bar containing the analysis history will be displayed on the left side of the window, and the list of results will be hidden until you click the bar, as shown next. To hide the results again, click outside the list.



THE RIBBON

The ribbon of the Analysis Summary window allows you to cut, copy, paste and print the displayed analysis results, as well as save them in an Excel spreadsheet.

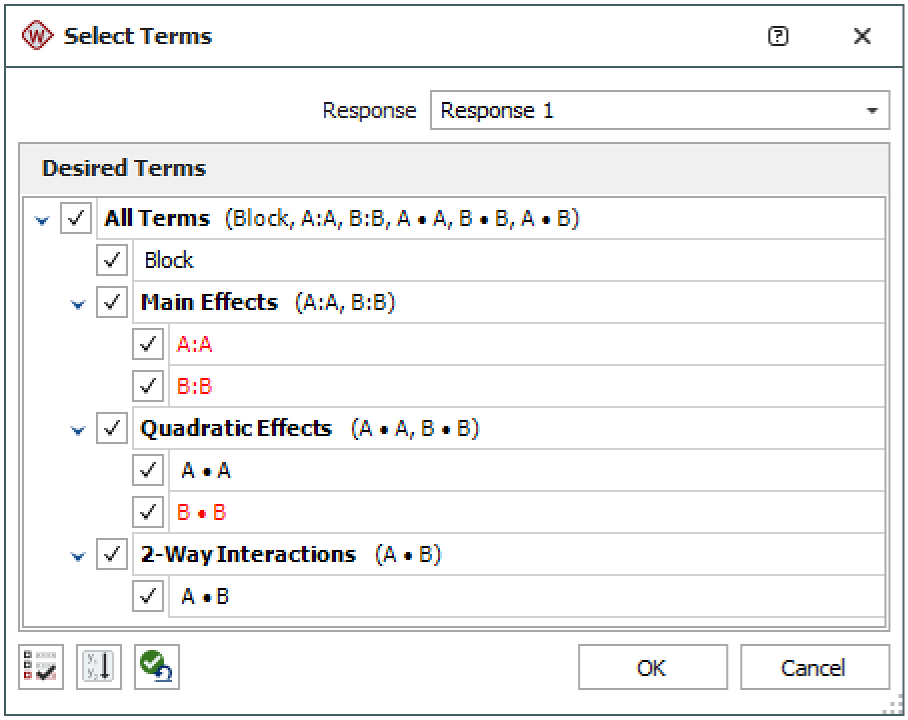
If you choose **Settings > Apply Settings** from the ribbon, the analysis settings that were used to obtain the displayed results will be applied to the current analysis. For example, if you used the Analysis History panel to view results that were obtained using a risk level of 0.5, and if you wanted to use that risk level and all the other settings from that prior analysis, you would choose this command and then recalculate the folio.

Note that you can apply the settings from a prior analysis only if it is based on the same design and response data as the current analysis. If this isn't the case for the displayed analysis results, then the results will include red text that says, "The design has been modified since this history was created. These settings can no longer be applied to this response."

Select Terms Window

The Select Terms window allows you to specify, for each response, which main effects and interaction effects you want to include in the analysis of each response and in the evaluation of the design for each response. It is accessed by choosing **Data > Design > Select Terms** or clicking the icon on the Data tab control panel.

The window will appear as shown next. If the response data has been analyzed already, any terms that were found to be significant will be highlighted in red.



- The **Response** drop-down list allows you to select the response you want to select regression terms for.

- If you are working with a mixture design that includes process factors, the **Include process factors up to** drop-down list will be available. Use this drop-down to automatically select only those process factor terms that are required to fit a model of the order you choose. After the terms are automatically selected, you can add/remove individual terms as desired.

- The check boxes in the **Desired Terms** area allow you to select which terms you want to include in the analysis/evaluation. Note that if you select an interaction term, you must also select the interacting effects under the Main Effects heading (e.g., if you select interaction AC, then you must select the main effects A and C individually).

  - The **Block** check box directly underneath the All Terms heading allows you to specify whether you want to include blocks in the model. This option is available only if the design you are working with is blocked.

  - **Quadratic Effects** check boxes are available only for factors with three or more unique values. They are used to determine whether the factors have curvature effects.

  - The following terms are available for mixture designs only:

    - **Mixture Factors** includes all the terms for fitting different types of models (linear, quadratic, etc.). By default, the terms required to fit a quadratic model are selected.

- **Process Factors** include all the terms for investigating the effects of mixture factors under the influence of the process factors. Instead of selecting process factors individually, you can use the **Include Process Variables up to order** drop-down list at the top of the window to automatically select the process variables needed to investigate effects up to a given order.

- The **Select Significant Terms** icon ( ) automatically selects all terms that were found to be significant when you last analyzed the data. All other terms will be cleared.

- The **Apply to All Responses** icon ( )applies the current selections to all other responses (if any) instead of just to the response you selected.

- The **Reset Defaults** icon ( ) selects all terms for all responses.

After you select the desired terms, click **OK** and recalculate the folio to update the analysis or evaluation.

Select Transformation Window

Analysis of variance is based on two assumptions:

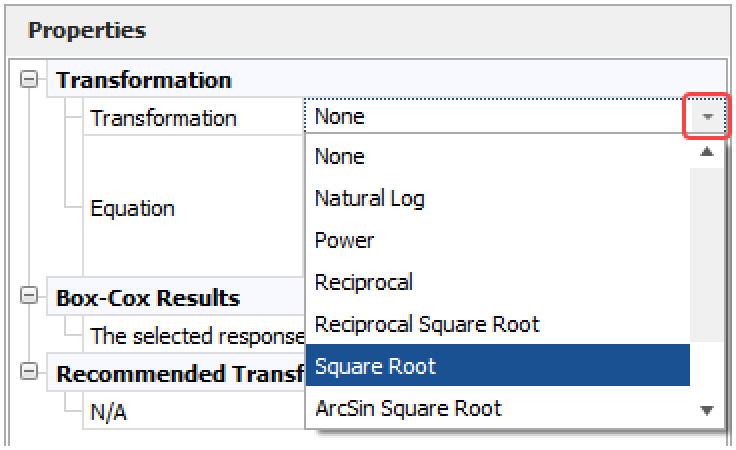
1. The response in question is normally distributed.
2. The error variance of each observation is constant (i.e., it does not change with either the observation time or the observed response value).

If either of these assumptions is violated, applying a transformation to the response may be useful.

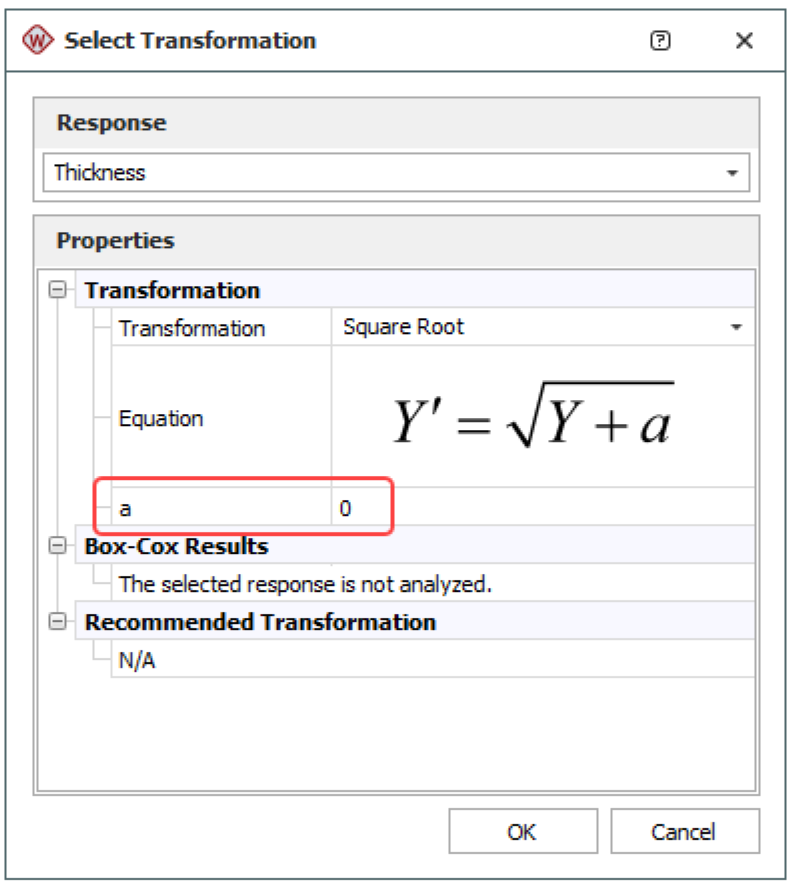
The Select Transformation window allows you to select a transformation to apply to each response. It is accessed by choosing **Data > Design > Select Transformation** or clicking the icon on the Data tab control panel.

To use the window, first choose the response you want to work with from the **Response** drop-down list.

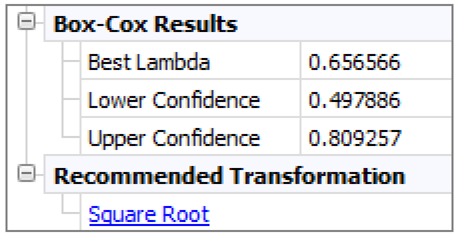
Next, select a transformation from the **Transformation** drop-down list.



The equation for the transformation will be displayed and input areas for any required inputs will be available under the **Transformation** heading. In the example shown next, 0 was entered for the A parameter in the equation $Y' = \sqrt{Y + a}$.



If you are not certain what transformation, if any, to apply to the selected response, you can calculate the data in the folio and then obtain information on the Box-Cox transformation results for the selected response, which helps in selecting an appropriate transformation. The **Box-Cox Results** area will display the best lambda value for the response, including the two-sided confidence bounds at the level determined by the risk level (alpha) you have specified. The recommended transformation will be displayed in the **Recommended Transformation** area, as shown next.



Click the link to automatically choose it in the **Transformation** drop-down list.

Repeat the process for any other responses you want to apply a transformation to, and then click **OK** to return to the folio, where you can recalculate using the selected transformation(s).

Available Transformations and Requirements

The following table displays the transformations that can be applied to responses in DOE folios and specifies any requirements for using the transformations.

Transformation Name	Transformation Equation	Data Requirements
None	$Y' = Y$	None
Square Root	$Y' = \sqrt{Y+a}$	$Y + a \geq 0$
Natural Log	$Y' = \ln(Y+a)$	$Y + a > 0$
Base 10 Log	$Y' = \log_{10}(Y+a)$	$Y + a > 0$
Inverse Square Root	$Y' = \frac{1}{\sqrt{Y+a}}$	$Y + a > 0$
Inverse	$Y' = \frac{1}{Y+a}$	$Y + a \neq 0$

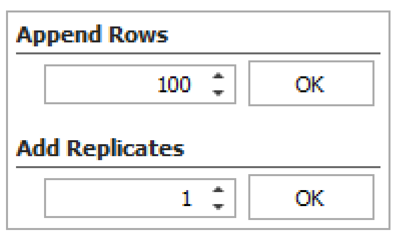
Power	$Y' = (Y+a)^{\lambda}$	$\lambda \neq 0$ $Y + a \neq 0$ [λ negative integer] $Y + a > 0$ [λ negative fraction] $Y + a \geq 0$ [λ positive fraction]
Logit	$Y' = \ln\left(\frac{Y-a}{b-Y}\right)$	$a < Y < b$
ArcSin Square Root	$Y' = \sin^{-1}\sqrt{Y}$	$0 < Y < 1$

Augmenting Designs

After building an experiment design, you may wish to augment the design by adding individual runs or replicates. For example, after performing all the runs for a design and entering the response data, you may decide to perform one more replicate of the entire design to get more precise results. Or you may decide to add rows to the data sheet for additional runs that include center points, etc.

To augment a built design, choose **Data > Design > Augment Design** or right-click a cell in the design and choose **Augment Design** from the shortcut menu.

The following options will appear:



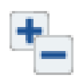
- To add a specific number of blank rows to the design, enter a number in **Append Rows** field and click **OK**. The new rows will be added at the bottom of the table.
- The **Add Replicates** field is used to add replicates of the entire current design. For example, if there are 20 rows in the current design, then adding 1 replicate will add 20 new rows that use the same factor combinations as the original design.

Note: After you augment a design, the selected options on the Design tab will no longer fully reflect the experiment described on the Data tab. In other words, rebuilding the design from the Design tab will remove the appended rows/replicates.

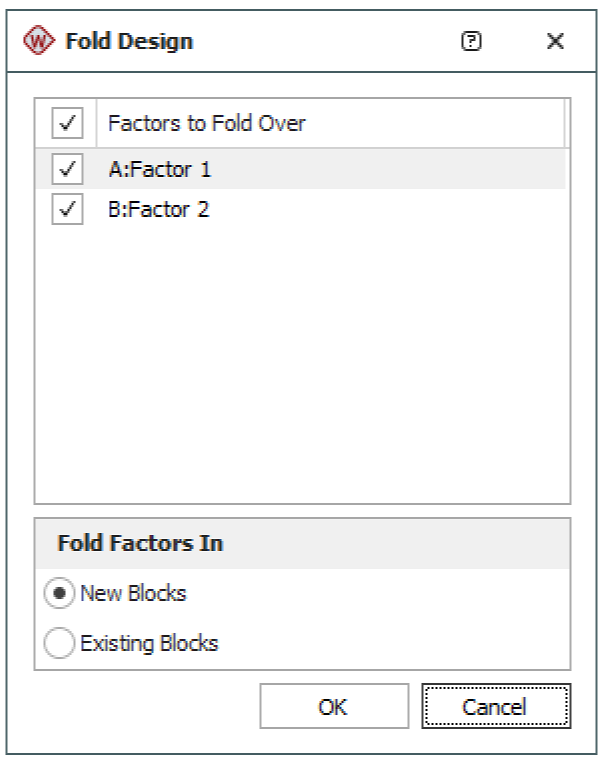
Fold Design Window

When you fold a factor over in a design, you replicate the design with runs that use the opposite levels for that factor. For example, if you fold over Factor A, the design will be replicated, and in each new row the value of Factor A will be opposite of what it was in the original row that was copied (i.e., the -1 values will be replaced with the 1 values and vice versa). The ability to fold the design is available only for designs that require two-level factors (i.e., two level factorial and Plackett-Burman designs).

To fold a design, choose **Data > Design > Fold Design**.



The Fold Design window will appear as shown next.

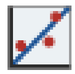


The first area in the window allows you to select the **Factors to Fold Over**.

The **Fold Factors In** area allows you to select whether you would like to add the new rows to a new block or to the existing blocks.

Design Folio Plots

The Analysis Plot tab is added to a design folio the first time you choose **Data > Analysis > Plot** or click the icon on the Data tab control panel.



The options available on the Analysis Plot control panel vary depending on the selected plot. For general information on working with plots, see Plot Utilities.

The available plots will vary depending on the design type you are working with. The following plots may be available.

Level Plots (One Factor Designs Only)

Level plots allow you to visually evaluate the effects of different factor levels on the selected response.

- The **Comparison Chart** shows the standardized difference for each paired comparison of factor levels. Use the **Contrasts** area of the control panel to select which pair of levels to show on the plot.
- The **Response vs. Level** plot shows the observed output, or response, as well as the calculated mean output, at each level of the factor.
- The **Level Mean** plot shows the mean output at each level of the factor. The center point of each level line is the calculated mean and the end points represent the high and low confidence bounds on the mean based on the alpha (risk) value specified on the Analysis Settings page of the control panel.
- The **Box Plot** shows the output at each level of the factor. The top and bottom points at each level represent the highest and lowest responses. The points within the box represent the responses at the 25th, 50th and 75th percentile.
- The **Mean PDFs** plot shows the *pdf* of the mean response at the selected factor levels.
- The **Life Characteristic** plot (reliability DOE only) shows the calculated life characteristic at each factor level. The top and bottom tick marks on the vertical lines mark the two-sided confidence bounds on the life characteristic. The confidence level for the bounds is determined by the risk level specified on the Analysis Settings page of the Data tab control panel (e.g., if the

risk level is 0.1, then 90% two-sided bounds would be shown).The value of the characteristic life depends on the selected distribution.

- **Eta** is used for the Weibull distribution, and it is equal to the time at which unreliability = 63.2%.
- **Ln-Mean** is used for the lognormal distribution, and it is equal to the time at which unreliability = 50%.
- **MTTF** (i.e., the mean time to failure) is used for the exponential distribution.

Effect Plots

Effect plots allow you to visually evaluate the effects of factors and factorial interactions on the selected response.

- The **Pareto Chart - Regression** plot shows the standardized effects of the selected terms (i.e., factor or combination of factors). The vertical blue line is the threshold value. If a bar is beyond the blue line, it will be red, indicating that the effect is significant.
- The **Pareto Chart - ANOVA*** plot shows the inverse *p* value (1 - *p*) of each selected term. The vertical blue line is the threshold value. If a bar is beyond the blue line, it will be red, indicating that the term is significant.
- The **Pareto Chart - LRT** (reliability DOE only) shows the inverse *p* value for the reduced model in the likelihood table. The vertical blue line is the threshold value. If the bar is beyond the blue line, it will be red, indicating that the factor has a significant effect on reliability.
- The **Effect Probability** plot is a linear representation of probability versus the standardized effect (i.e., the probability that any term's standardized effect will be lower than the given value). The points on this plot represent the values for each term in the T Value column of the Regression Table in the detailed analysis results. If there is no error in the design, then the probability versus the effect is shown and the points on this plot represent the values for each term in the Effect column of the Regression Table in the analysis results.
  - Select **Normal** in the **Scale Type** area to display the negative and positive values of the effects (coefficients). The negative effects will appear to the left of the probability line.
  - Select **Half-normal** to display the absolute values of all the effects, which allows you to compare the size of each effect. All the effects will appear to the right of the probability line.

- The **Main Effects** plot shows the mean effect of the selected factor(s). The points are the observed Y values at the low and high level for each factor. The line connects the mean value at each factor level, and you can specify how the means are calculated in the **Calculation Options** area. Note that if you are using actual factor values in the plot, you can plot only one factor at a time. If you are using coded values, you can plot multiple factors simultaneously. For mixture designs, this plot applies only to process factors (i.e., process variables).

- The **Interactions** plot shows the mean effect of a selected factor versus another selected factor at each level. If the green and red mean effect lines are parallel, there is no interaction between the two factors. You can specify how the means are calculated in the **Calculation Options** area. For mixture designs, this plot applies only to process factors (i.e., process variables).

- The **Interaction Matrix** shows multiple Interactions plots. The plots shown depends on the factors you select. For example, if you select factors A and B, then two interactions plots will be shown: one showing A versus B and another showing B versus A.

- The **Term Effect Plot** shows the fitted means for all combinations of all factor levels for each selected term. You can specify how the means are calculated in the **Calculation Options** area.

- The **Cube Plot** shows the mean values of the selected response for the combinations of the low and high levels of three selected factors. You can specify how the means are calculated in the **Calculation Options** area. Note that you have the option of selecting "none" for the third factor, generating a square (2-dimensional) plot. Only two level factors can be included in the Cube plot, and at least two quantitative factors (each run at two levels) must be included in the model for the Cube plot to be available.

- The **Scatter Plot** shows the observed values of the currently selected response plotted against the levels of the selected factor. A 3-dimensional version of this plot is available in the 3D plot folio.

- The **Contour Plot** shows how varying two selected factors affects the predicted response values, which are represented as colors. See Contour Plots. A 3-dimensional version of this plot ("Surface Plot") is available in the 3D plot folio. For mixture designs, this plot applies only to process factors (i.e., process variables).

Note that for mixture designs, main effects cannot be shown in the Pareto Chart - Regression and Effect Probability plots. This is because the *T* value (or standardized effect) is based on a comparison with 0, which is not appropriate for the coefficients of main effects in mixture designs.

The following effect plots are available only for mixture designs and apply only to mixture factors (i.e., components):

- The **Simplex Design Plot** is available only when there are at least three components in the experiment, and it shows the different blends that were included in the experiment.
- The **Response Trace Plot** is used to see the effect of each component. By default, the center of the design is set as the reference point (which you can change by clicking **Set Factor Levels** on the control panel). As one component's proportion moves away from the reference point, the relative ratios of other components are kept constant. The x-axis is the amount of change of each component, and the y-axis displays the corresponding response values when x is changed. Each component has its own curve on the plot.
- The **Mixture Contour Plot** shows how varying the three selected components affects the predicted response values, which are represented as colors. See Contour Plots. This plot uses the same principles as the simplex design plot to represent different mixtures (e.g., the center of the plot represents a blend with all three components in equal proportions). A 3-dimensional version of this plot ("Mixture Surface Plot") is available in the 3D plot folio.

Residual Plots

Residuals are the differences between the observed response values and the response values predicted by the model at each combination of factor values. Residual plots help to determine the validity of the model for the currently selected response. When applicable, a residual plot allows the user to select the type of residual to be used:

- **Regular Residual** is the difference between the observed Y and the predicted Y.
- **Standardized Residual** is the regular residual divided by the constant standard deviation.
- **Studentized Residual** is the regular residual divided by an estimate of its standard deviation.
- **External Studentized Residual** is the regular residual divided by an estimate of its standard deviation, where the observation in question is omitted from the estimation.

The plots are described next.

- The **Residual Probability*** plot is the normal probability plot of the residuals. If all points fall on the line, the model fits the data well (i.e., the residuals follow a normal distribution). Some scatter is to be expected, but noticeable patterns may indicate that a transformation should be used for further analysis. Two additional measures of how well the normal distribution fits the data are provided by default in the lower title of this plot. Smaller values for the Anderson-Darling test indicate a better fit. Smaller *p* values indicate a worse fit.

- The **Residual vs. Fitted*** plot shows the residuals plotted against the fitted, or predicted, values of the selected response. If the points are randomly distributed around the "0" line in the plot, the model fits the data well. If a pattern or trend is apparent, it can mean either that the model does not provide a good fit or that Y is not normally distributed, in which case a transformation should be used for further analysis. Points outside the critical value lines, which are calculated based on the specified alpha (risk) value, may be outliers and should be examined to determine the cause of their variation.

- The **Residual vs. Order*** plot shows the residuals plotted against the order of runs used in the design. If the points are randomly distributed in the plot, it means that the test sequence of the experiment has no effect. If a pattern or trend is apparent, this indicates that a time-related variable may be affecting the experiment and should be addressed by randomization and/or blocking. Points outside the critical value lines, which are calculated based on the specified alpha (risk) value, may be outliers and should be examined to determine the cause of their variation.

- The **Residual vs. Factor*** plot shows the residuals plotted against values of the factor selected in the **Residual Factor** area. It is used to determine whether the residuals are equally distributed around the "0" value line and whether the spread and pattern of the points are the same at different levels. If the size of the residuals changes as a function of the factor's settings (i.e., the plot displays a noticeable curvature), the model does not appropriately account for the contribution of the selected factor. Points outside the critical value lines, which are calculated based on the specified alpha (risk) value, may be outliers and should be examined to determine the cause of their variation.

- The **Residual Histogram*** is used to demonstrate whether the residual is normally distributed by dividing the residuals into equally spaced groups and plotting the frequency of the groups. The **Residual Histogram Settings** area allows you to:

  - Select **Custom Bins** to specify the number of groups, or bins, into which the residuals will be divided. Otherwise, the software will automatically select a default number of bins based on the number of observations.

  - Select **Superimpose pdf** to display the probability density function line on top of the bins.

- The **Residual Autocorrelation*** plot shows a measure of the correlation between the residual values for the series of runs (sorted by run order) and one or more lagged versions of the series of runs. The default number of lags is the number of observations, n, divided by 4. If you select **Custom Lags** in the **Auto-Correlation Options** area, you can specify up to n -1 lags. The correlation is calculated as follows:

$$r_k = \frac{\sum_{i=1}^{N-k}\left(Y_i - \overline{Y}\right)\left(Y_{i+k} - \overline{Y}\right)}{\sum_{i=1}^{N}\left(Y_i - \overline{Y}\right)^2}$$

where:

- k is the lag.
- $\overline{Y}$ is the mean value of the original series of runs.

For example, lag 1 shows the autocorrelation of the residuals when run 1 is compared with run 2, run 2 is compared with run 3 and so on. Lag 3 shows the autocorrelation of the residuals when run 1 is compared with run 4, run 2 is compared with run 5 and so on. Any lag that is displayed in red is considered to be significant; in other words, there is a correlation within the data set at that lag. This could be caused by a factor that is not included in the model or design, and may warrant further investigation.

- The **Fitted vs. Actual** plot shows the fitted, or predicted, values of the currently selected response plotted against the observed values of the response. If the model fits the data well, the points will cluster around the line.

Diagnostic Plots

- The **Leverage vs. Order** plot shows *leverage* plotted against the order of runs used in the design. Leverage is a measure (between 0 and 1) of how much a given run influences the predicted values of the model, where 1 indicates that the actual response value of the run is exactly equal to the predicted value (i.e. the predicted value is completely dependent upon the observed value). Points that differ greatly from the rest of the runs are considered outliers and may distort the analysis.

- The **Cook's Distance*** plot can show Cook's distance (i.e., a measure of how much the output is predicted to change if each run is deleted from the analysis) plotted against either the run order or the standard order for the currently selected response. Points that differ greatly from the rest of the runs are considered outliers and may distort the analysis.

- The **Box-Cox Transformation*** plot can help determine, for the currently selected response and model, what transformation, if any, should be applied. The plot shows the sum of squares of the residuals plotted against lambda. The value of lambda at the minimum point of this curve is considered the "best value" of lambda, and indicates the appropriate transformation, which is also noted by default in the lower title of the plot.

* These plots are available only when there is error in the design, indicated by a positive value for sum of squares for Residual in the ANOVA table of the analysis results.

Contour Plots

To view the contour plot, create the Plot tab in your design folio, then choose **Contour Plot**. You can specify how the means are calculated in the **Calculation Options** area.

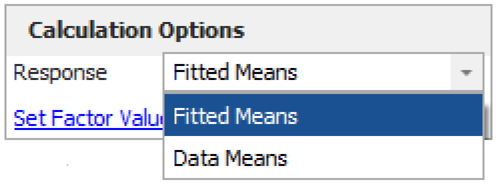
When you are working with this type of plot, the control panel includes an Additional Settings page with the following options:

- The **Contour Options** area lets you specify the color gradient for the contour bands and the resolution of the lines. The colors for low, medium and high response values can be modified by clicking the colored **Low**, **Mid** and **High** icons. You can also change the **Resolution** of the contour lines (higher resolution means smoother curves, but the plot may take longer to generate).
- The **Band Settings** area lets you choose how many contour bands will be used and what response values they will represent. Three options are available:
  - Select **Count** to enter the number of contour bands directly in the **Contour Count** field. More bands means more fine-grained results (i.e., more colors for marking response ranges). The minimum and maximum response values are set by the software automatically.
  - Select **Range** to manually set the minimum and maximum response values to be shown in the plot. Then enter the **Increment** that will be used to create the contours. A lower increment means more fine-grained results.
  - Select **Values** to define each contour manually. For example, if you enter the values 2 and 3, then the plot will have three contours: one for values less than 2, another for values between 2 and 3, and another for values greater than 3. Use the [+] and [-] icons in the **Contour Values** area to add a new value or remove a selected one.

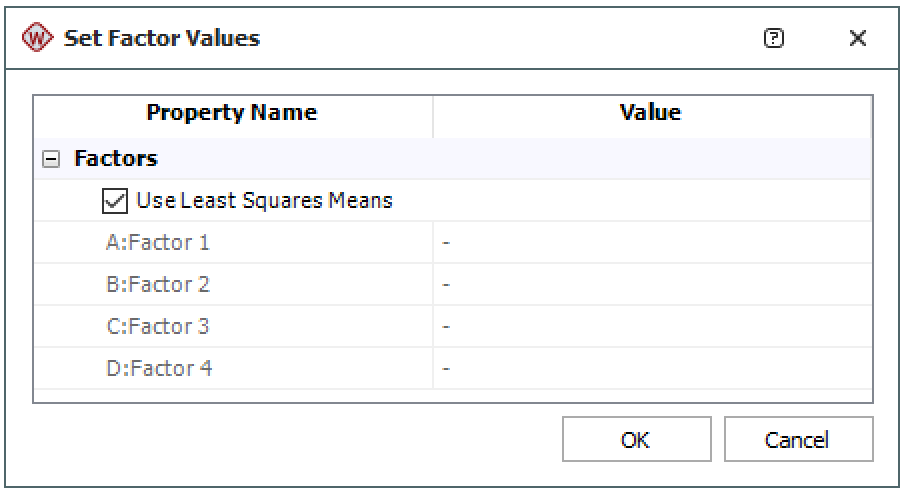
Effect Plot Calculation Options

For effect plots that show mean response values, the **Response** drop-down list in the **Calculation Options** area allows you to specify how the means will be calculated at the different factor levels.

Use the drop-down list in this area to select the type of mean value that is used. Note that, for some effect plots, only fitted means will be available.



- **Fitted Means** uses mean values based on the fitted regression model. When you use this option, you can click **Set Factor Values** to specify how factors will be treated in the calculation.



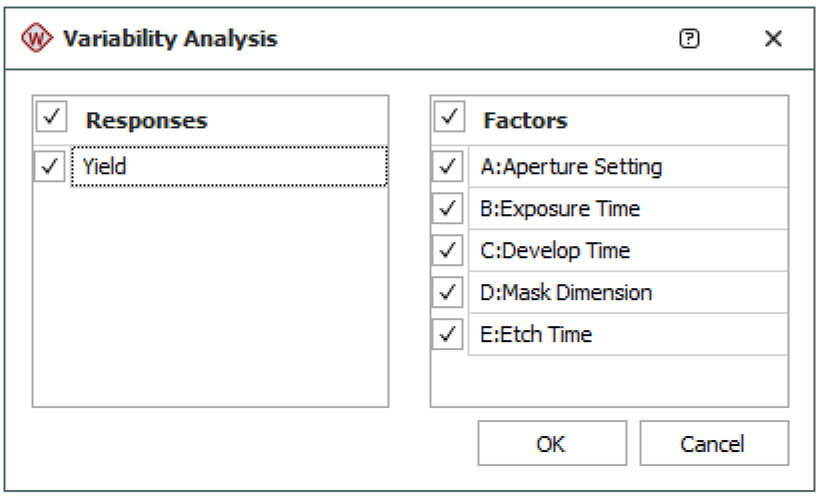
  - Select **Use Least Squares Means** to set all non-varied factors to 0 (i.e., remove them from the response calculation).
  - For some plots created in the mixture design folio, you can select **Use Centroid** to set the plot's reference point to the center of the design for the response trace plot, or to set the values of any non-varied mixture components.
  - Clear the option if you want to manually specify the values of non-varied factors.
- **Data Means** uses the mean values from the observed data entered on the Data tab. (If a transformation has been applied to the response during analysis, data means will be based on the transformed values.)
  - When this option is used for response surface method designs, the mean values will not include curvature because only a fitted regression model can account for quadratic effects.

Variability Analysis Window

Variability analysis is available only for two level factorial and Plackett-Burman designs with standard response data, and it is used to identify the experimental run that results in the least amount of variation in the response. To perform this analysis, the design must have 2 or more runs that use the same factor level combinations for the factors that you choose to consider in the analysis.

The Variability Analysis window allows you to select the response column(s) that you want to see variability analysis for and which factors to consider. To open the window, choose **Data > Design > Variability Analysis** or click the icon on the Data tab control panel.

The Variability Analysis window is shown next.



For each response that you select in this window, a standard deviation column will be inserted beside it on the Data tab, displaying the standard deviation of the observations taken for each factor setting combination across replicates. Such columns can be selected for inclusion in the analysis (i.e., used as responses).

The **Factors** area allows you to specify which factors are considered in the variability analysis. If, for example, only Factor A is selected, there will be only two rows of data in the Response Standard Deviation column: one for the runs that used A's low level and one for the runs that used the high level. The remaining rows would include an asterisk to indicate the standard deviation was displayed in a previous row, as shown next.

	A:Factor 1	B:Factor 2	Response 1	Response 1 Standard Deviation
1	-1	-1	519.474	462.0921742
2	1	-1	502.688	369.1617447
3	-1	1	965.384	*
4	1	1	824.312	*
5	-1	-1	454.064	*
6	1	-1	1354.839	*
7	-1	1	1451.071	*
8	1	1	1120.477	*

If the design includes blocks, the **Factors** area will include a **Block** check box that allows you to consider them in the variability analysis.

Variability Analysis Example

*The data set used in this example is available in the example database installed with the software (called "Weibull21_DOE_Examples.rsgz21"). 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 example project is "Factorial - Two Level Full Factorial Design."

In a separate example, two optimal solutions were found for the factor level combination that optimize the process of growing epitaxial layers on polished silicon wafers. The two solutions are shown next.

Solutions

	Solution Name	B:Nozzle Position	C:Deposition Temperature	D:Deposition Time	Thickness Predicted	Thickness Desirability	Global Desirability
1	Optimal Solution 1	2	1210.73922	High	14.5	1	1
2	Optimal Solution 2	6	1214.82346	High	14.5	0.999999	0.999999

OK Cancel

Although both solutions give the same predicted thickness (14.5 µm), it is important to identify which solution is better in terms of variability. To do this, you need to perform a variability analysis, where the analyzed response is the standard deviation of the thickness at each factor level combination.

The data and results for the variability analysis are given in the "Variability Analysis" folio of the example file. On the Design tab, you can see the various settings that were used to generate the experiment design.

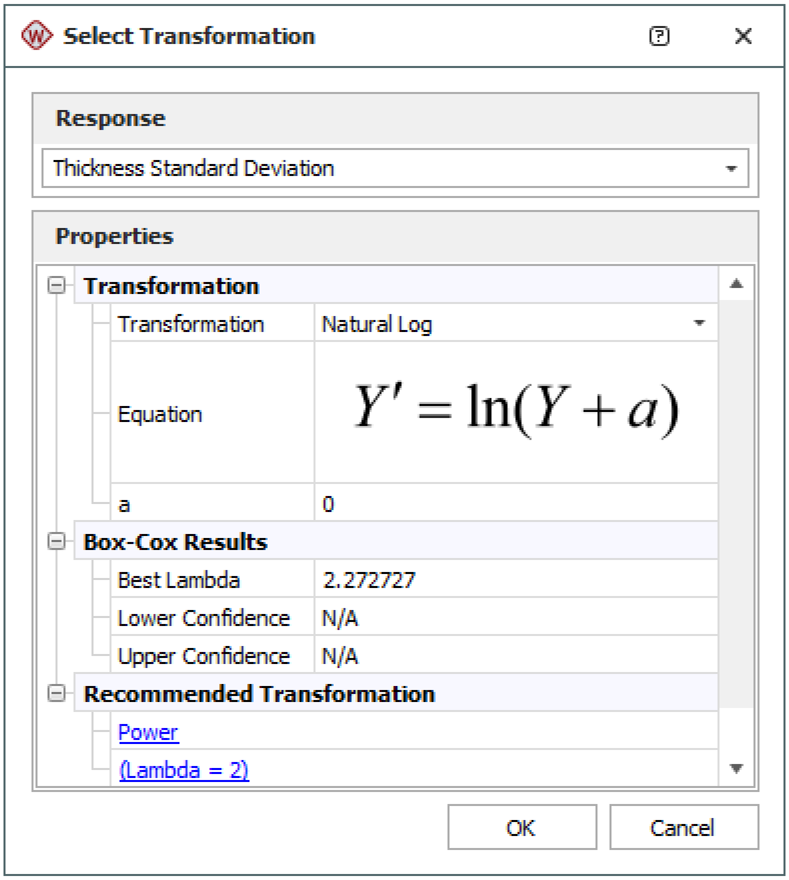
Each row on the Data tab shows the factor level combination and recorded thickness for a single run. In addition, the yellow Thickness Standard Deviation column, which was added using the Variability Analysis window, shows the standard deviation of the thickness at the given factor level combination. (If a cell in this column does not display a value, that's because the variation in response for that factor level combination is already shown in a previous cell.) The Thickness column was removed from the analysis by clearing the check box in the column heading.

	A:Susceptor-Rotation Method	B:Nozzle Position	C:Deposition Temperature	D:Deposition Time	Thickness	Thickness Standard Deviation
1	Continuous	2	1210	Low	12.886	0.539396113
2	Oscillating	2	1210	Low	14.249	0.521644323
3	Continuous	6	1210	Low	14.059	0.452595183
4	Oscillating	6	1210	Low	13.775	0.503089422
5	Continuous	2	1220	Low	13.758	0.443804649
6	Oscillating	2	1220	Low	13.605	0.478569535
7	Continuous	6	1220	Low	13.707	0.463682398
8	Oscillating	6	1220	Low	14.031	0.438270883
9	Continuous	2	1210	High	14.506	0.519414286
10	Oscillating	2	1210	High	15.05	0.518401003
11	Continuous	6	1210	High	14.629	0.470301605
12	Oscillating	6	1210	High	14.274	0.476438873
13	Continuous	2	1220	High	13.926	0.517595981
14	Oscillating	2	1220	High	13.327	0.468886944
15	Continuous	6	1220	High	13.8	0.471131829
16	Oscillating	6	1220	High	13.723	0.500249538
17	Continuous	2	1210	Low	12.963	*
18	Oscillating	2	1210	Low	13.9	*
19	Continuous	6	1210			
20	Oscillating	6	1210			

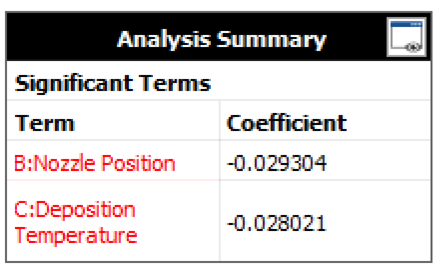
Note: Complete data set is not shown.

To transform the standard deviation values, choose **Data > Design > Select Transformation**, or click the icon on the Data tab's control panel.

Then choose the natural log transformation (which is typically used when studying variability) in the window that appears.



The results for the variability analysis are shown in the **Analysis Summary** area on the control panel.



The values of the regression coefficients show that nozzle position and deposition temperature have a negative effect on the variation in thickness.

CONCLUSION

Since the significant factors have a negative effect on variability, higher values will produce lower standard deviation. Therefore, of the two optimal solutions, the second solution is better in terms of variability.

Nozzle Position	Deposition Temperature	Deposition Time	Predicted Thickness	Predicted Thickness Std
Position 2	1210.7392	High	14.5	0.512
Position 6	1214.8235	High	14.5	0.472

Optimal Design

The many DOE design types in Weibull++ are sufficient for most applications. However, sometimes limitations on resources and factor level combinations will require you to build a custom design that is based on one of the included design types.

You can use the Optimal Design tool to build a custom design that is optimized for a selected regression model (i.e., the design will minimize the uncertainty of the regression coefficients) or optimized to cover as large of a design space as possible (i.e., the distance between design points is maximized). For example, if you only have 11 units available to test using a two level factorial design with five factors, you could build a two level full factorial design (which would require 32 runs), and then use this tool to build a modified version of the design that uses 11 runs and takes into account only the effects you want to investigate, as well as any restrictions on the factor level combinations that can be used in the experiment.

Optimal Design Window

The Optimal Design window is used to select test runs from a design table in order to minimize the estimation uncertainty of the coefficients in a given regression model. It takes into account a) the specific effects that are under investigation, b) limitations on the number runs that can be performed and c) constraints on the factor level combinations that can be used during the experiment. The ReliaWiki resource portal provides more information on optimal custom designs at: http://www.reliawiki.org/index.php/Optimal_Custom_Designs.

To access the Optimal Design window, build a design and then choose **Data > Design > Optimal Design**.

The window includes two tabs discussed next.

DESIGN SETTINGS TAB

This tab is used to specify the method that will be used to create the design. Follow the steps outlined below to enter the settings.

- If you wish to optimize the design for a regression model, select the **Regression Model-Based Optimal Design** option at the top of the window.
  - In the **Desired Model Terms** area, select which terms will be included in the regression model.
  - In the **Number of Runs in the Optimal Design** area, specify the number of runs that will be used in the optimal design. The minimum and recommended numbers of runs are calculated automatically by the software. If you want to use a different number of runs, select the **Custom number of runs** option and then enter a number in the input field.
  - If desired, you can modify the method for creating the design in the **Algorithm Settings** area.

    The first section in this area concerns the random elements that are used in the generation of the design.

    - 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 the resulting design.
    - Specify the **Number of random initial runs** that will be used. The software uses each random initial run to generate a design (using the algorithm you specify next), and then returns the best design that was found. Using more initial runs may produce a better design, but it will also increase the time required to generate the final design.

    The second section is used to select the algorithm that will be used to generate the design. Three algorithms are available.

	How likely to produce the optimal design?	How fast?
Federov's Method	Most likely	Slowest
Modified Federov's Method	Somewhat likely	Somewhat fast
K-Exchange Method	Least likely	Fastest

- If you want the design points (i.e., runs) in the design to cover as large of a design space as possible, select the **Distance-Based Optimal Design** option.
    - Then use the **Number of Runs in the Optimal Design** area to specify the number of runs that will be used in the optimal design. The minimum and recommended numbers of runs are calculated automatically by the software. If you want to use a different number of runs, select the **Custom number of runs** option and then enter a number in the input field.

CANDIDATE RUNS TAB

This tab is used to select which runs will be considered in the final design. By default, the table on the Candidate Runs tab shows all the runs used in the current design. If desired, you can modify the factor levels in any of the runs.

Follow the steps outlined next to choose the runs to consider and, if desired, add/remove runs.

- In the **Included in optimal design?** column, make sure the appropriate option is chosen for each run:
    - Choose **Consider** if you are not sure whether the combination should be included in the optimal design. The software will use the settings you specified on the Design Settings tab to determine whether the combination will be included.
    - Choose **Include** if want that combination to be used in the optimal design (e.g., you might be interested in examining the response at a particular combination).
    - Choose **Exclude** if you do not want to include the combination (e.g., there might be safety concerns with setting all factor levels to the highest level).
- If you want to add a new run to the table, click the **Add** button at the bottom of the window. The software will create a new row in the table with center points for all the factor values. If you do not wish to use center points, you can enter new factor values. To remove an added row, select it and click the **Delete** button.

Optimal Design Example

*The data set used in this example is available in the example database installed with the software (called "Weibull21_DOE_Examples.rsgz21"). 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 example project is "Optimal Custom Design."

In this example, a full factorial design is created that requires 24 unique factor combinations. Then, the Optimal Design utility is used to optimize the experiment so only 15 factor combinations are required. Finally, the optimized design is evaluated and modified to meet the experimenter's goals.

An experimenter wants to conduct some tests to study the effects of 4 factors. The main effects and the interaction effects AB, BC and CD will be investigated. The factors and factor levels that will be used in the experiment are shown next.

Factor	Level 1	Level 2	Level 3
A: Material Type	M1	M2	M3
B: Curing	Curing	No Curing	-
C: Tem-perature	120°C	150°C	-
D: Pressure	4 bars	5 bars	-

If a full factorial design is used, the number of required factor combinations is 24. However, due to the constraints of the test facility, only 15 unique factor combinations can be used. Moreover, because of safety concerns, the combination A = M2, B = Curing, C = 150°C and D = 5 bars cannot be used.

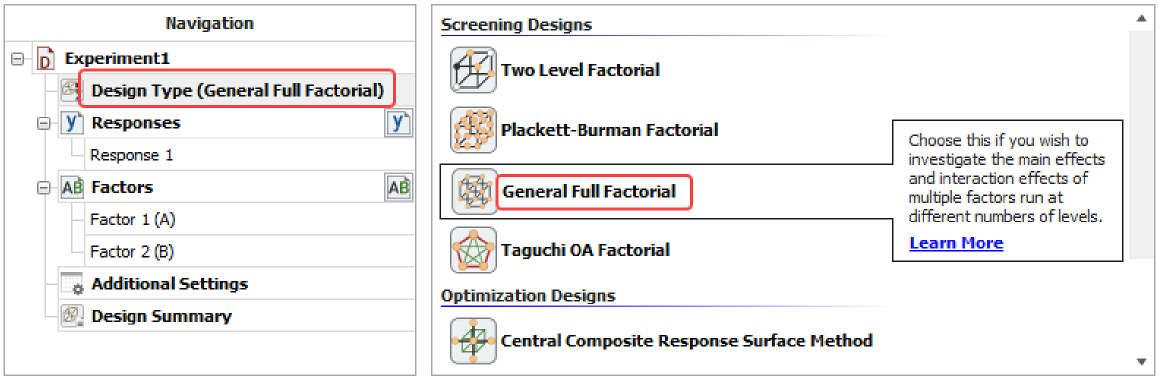
Based on the above information, the experimenters need to choose 15 runs from the 24 unique runs in the general full factorial design. The resulting design will then be evaluated and further modified if necessary.

CREATING A PRELIMINARY DESIGN

To begin, the experimenter creates a preliminary full factorial design. This will serve as a starting point for the optimal design. The design matrix is given in the "Preliminary Design" folio. The following steps describe how to create this folio on your own.

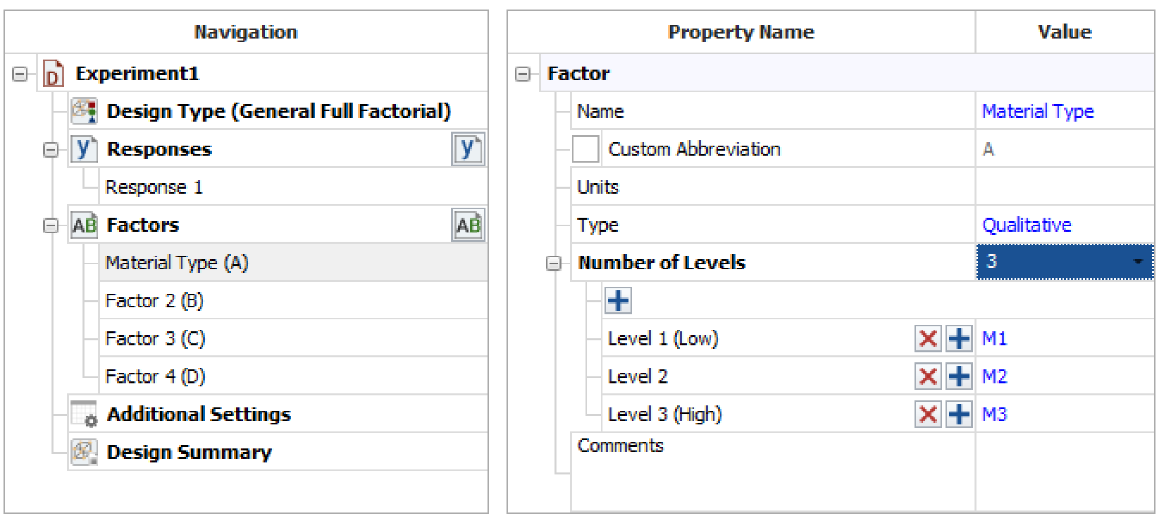
- Choose **Home > Insert > Standard Design** to add a standard design folio to the current project.

- Click **Design Type** in the folio's navigation panel, and then select **General Full Factorial** in the input panel.



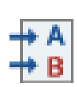
- Specify the number of factors by clicking the **Factors** heading in the navigation panel and choosing **4** from the **Number of Factors** drop-down list.

- Define each factor by clicking it in the navigation panel and editing its properties in the input panel. The properties are given in the table above. The first factor is defined as shown next. (Note that the third and fourth factors are quantitative.)



- Rename the folio by clicking the **Experiment1** heading in the navigation panel and entering **Preliminary Design** for the **Name** in the input panel.

- Next, click the **Build** icon on the control panel to create a Data tab that allows you to view the preliminary plan.

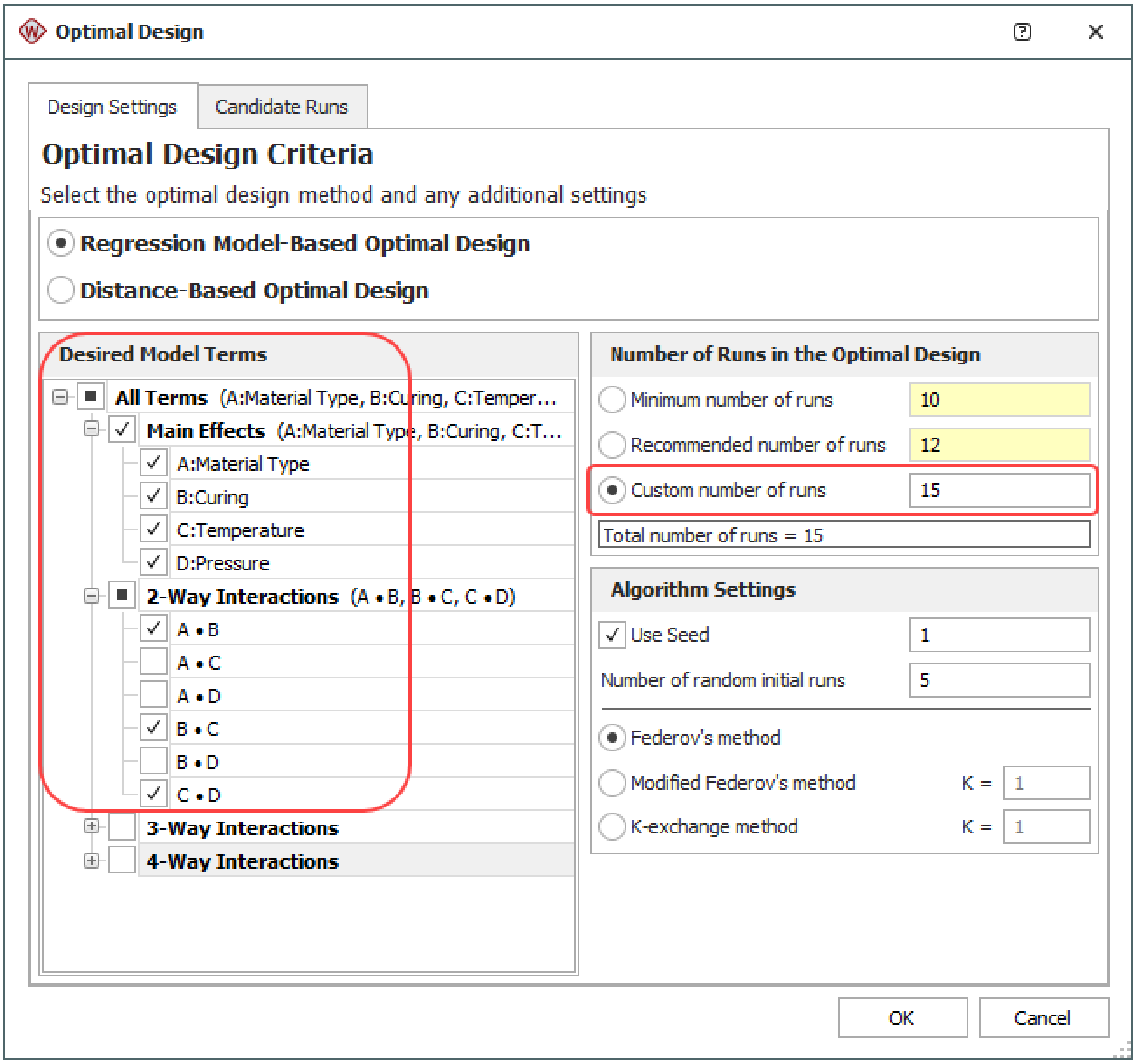


- Finally, sort the test runs in the preliminary design using the standard, non-randomized order. Sorting the test runs in this way will make it easier to find the factor level combination that you will later exclude from the design.
  - Choose **Data > Format and View > Visible Columns** and select to view the **Standard Order** column in the Visible Columns window.
  - Click a cell in the Standard Order column, and then choose **Sheet > Sheet Actions > Sort > Sort Ascending**.

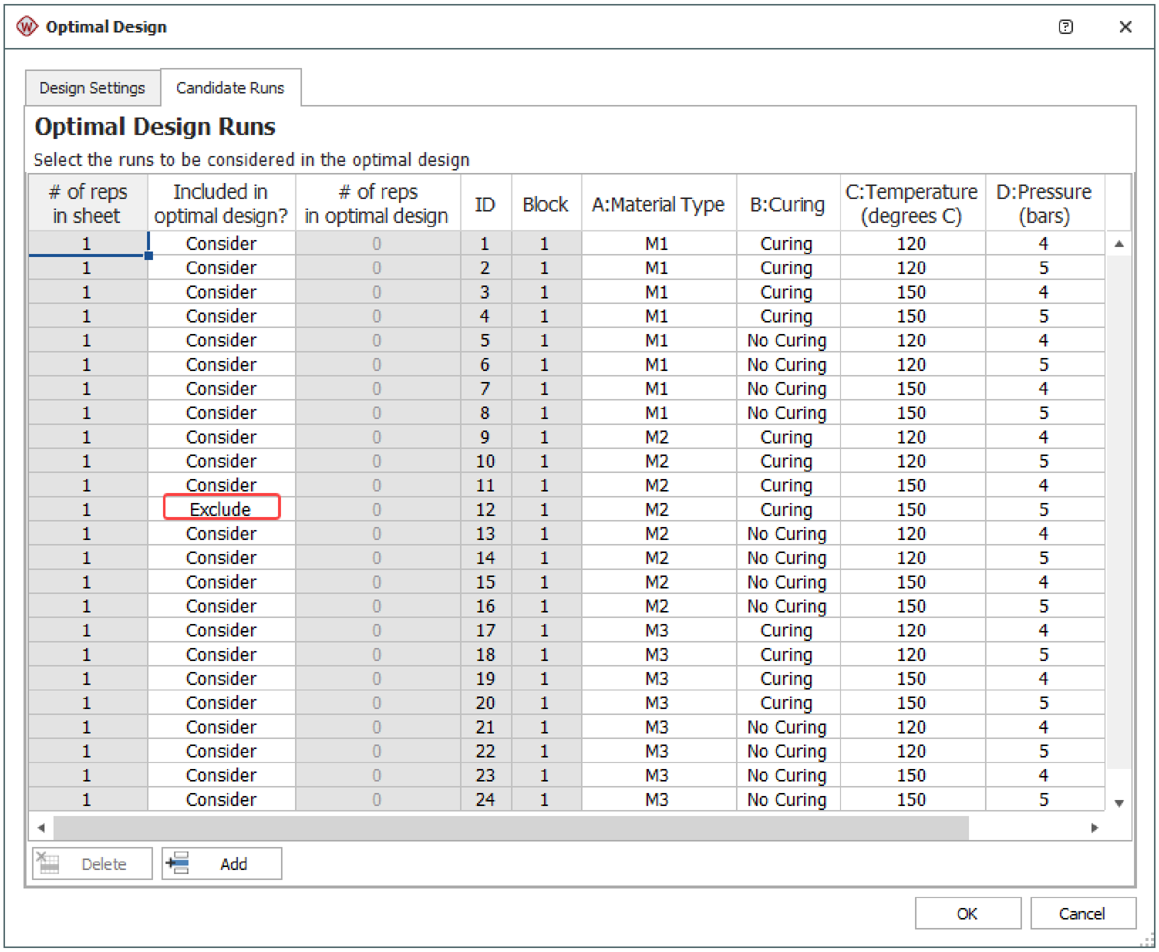
OPTIMIZING THE DESIGN

The preliminary design requires 24 factor combinations. To optimize the design so it only requires 15 unique factor combinations, view the experiment plan on the Data tab of the "Preliminary Design" folio. Then choose **Data > Design > Optimal Design**.

On the Design Settings tab of the window that appears, specify the terms that you are interested in investigating in the **Desired Model Terms** area. Also, in the **Number of Runs in the Optimal Design** area, specify how many runs you want to include in the design.



On the Candidate Runs tab, make sure the unsafe factor level combination (A = M2, B = Curing, C = 150°C and D = 5 bars) is not used by selecting **Exclude** in the second column for that combination.



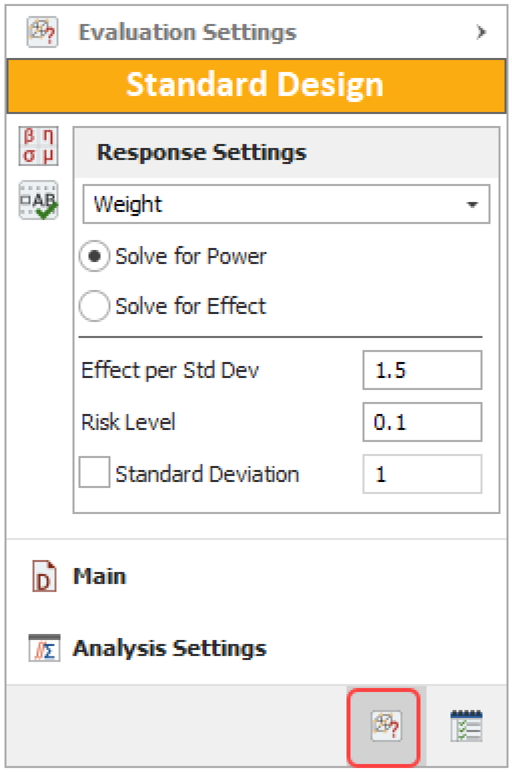
After you click **OK**, the resulting optimized design is given in the "Optimal Design" folio, as shown next. (Since the standard order is no longer relevant, you can open the Visible Columns window again to hide that column.)

	A:Material Type	B:Curing	C:Temperature (degrees C)	D:Pressure (bars)
1	M1	Curing	120	4
2	M2	No Curing	120	4
3	M3	No Curing	120	5
4	M1	Curing	150	5
5	M3	Curing	150	4
6	M2	No Curing	150	5
7	M1	No Curing	150	4
8	M1	Curing	120	5
9	M3	No Curing	150	5
10	M2	Curing	150	4
11	M2	Curing	120	4
12	M2	No Curing	150	4
13	M3	No Curing	120	4
14	M3	Curing	120	5
15	M1	No Curing	120	5

Note: To minimize the effect of unknown nuisance factors, the run order is randomly generated whenever you create a design. Therefore, if you followed these steps to create your own folio, the order of runs in your folio may be different from that of the folio in the example file. This will not affect the design evaluation results described next.

EVALUATING THE DESIGN

The experimenter wants to evaluate the power of the design (i.e., the probability that it will detect an active effect). The settings for the evaluation are specified on the Evaluation Settings page of the Data tab control panel.



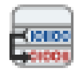
Click the **Calculate** icon to return to the Main page of the control panel. A summary of the evaluation results will be shown in the **Design Evaluation** area. Click the **Detailed Results** link in that area to view the calculated power values for each term that was included in the evaluation.

Power Study		
	Degrees of Freedom	**Power for Effect Value = 1.500000 (1.500000 * 1.000000 Std Dev)**
A:Material Type	2	0.423143
B:Curing	1	0.775168
C:Temperature	1	0.774055
D:Pressure	1	0.71044
A • B	2	0.368802
B • C	1	0.76971
C • D	1	0.644483
Residual	5	-

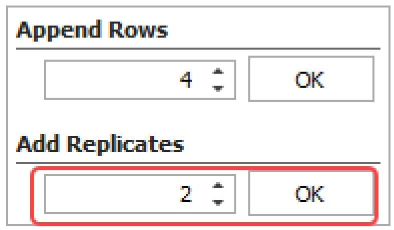
The power values for effect A and interaction AB are relatively small, which means the experiment is not likely to detect these effects if they are active. To increase the power, more runs need to be added to the experiment.

AUGMENTING THE DESIGN

Two more replicates are added to the optimal design using the Augment Design tool. To use this tool, view the experiment on the Data tab and then choose **Data > Design > Augment Design**.



In the window that appears, specify that 2 replicates will be added and click **OK**.



The new design is given in the "Optimal Design - Replicates" folio.

The power values for this design are:

Power Study		
	Degrees of Freedom	Power for Effect Value = 1.500000 (1.500000 * 1.000000 Std Dev)
A:Material Type	2	0.941619
B:Curing	1	0.999072
C:Temperature	1	0.999049
D:Pressure	1	0.996594
A • B	2	0.893016
B • C	1	0.998951
C • D	1	0.990228
Residual	35	-

The experimenter considers these values high enough. Therefore, this design is used as the final one.

Design Evaluation

The design folio control panel includes a tool that allows you to evaluate your design and compare it to other designs. By evaluating a design before you implement it, you can avoid using time or resources on an inefficient experiment. For example, if the tool shows that a design is unlikely to detect any main effects, you can increase the numbers of replicates and re-evaluate before performing the experiment. If you have already implemented a design, you can enter the response data and calculate it before performing the evaluation, which will produce more accurate results.

Specifically, the design evaluation tool allows you to:

- Calculate the design's *power* given a specified amount of effect. The power is the probability of detecting a specified amount of effect.
- Calculate the *amount of effect* given a specified power. The amount of effect is the largest difference between the response means obtained at each factor or effect level.
- Inspect the design's *alias structure*. This shows which effects are aliased with each other, and thus it helps you make sure you aren't losing any important information about interaction effects due to aliasing.
- Determine whether the design is *orthogonal* (i.e., the analysis of an effect does not depend on what other effects are included in the model). Orthogonal designs are ideal because they have minimal variance for the estimated model coefficients.
- If the design is not orthogonal, you can compare the design's *optimality* to other proposed designs to see which one is more similar to an orthogonal design.

IMPORTANT: This tool assumes that the response values obtained from the experiment will follow a normal distribution. Thus, it is not applicable to R-DOE, which is intended for life data that is not normally distributed.

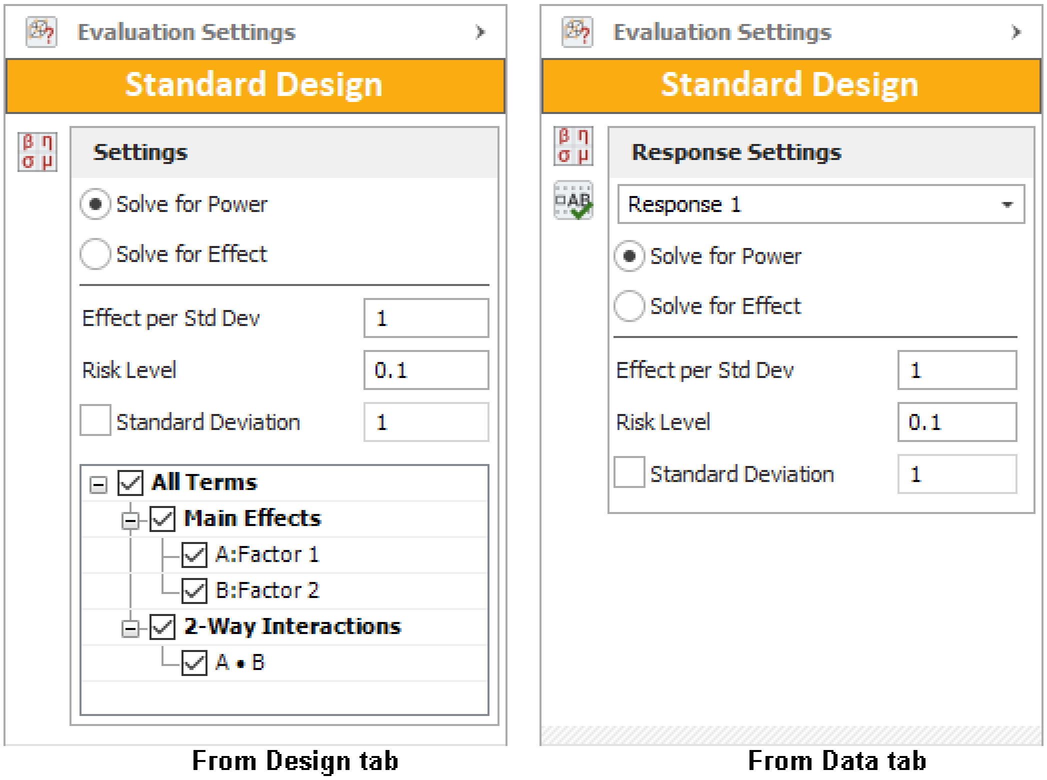
For an example of how you can use this feature to compare two designs, see Design Evaluation Example.

Selecting the Settings for the Evaluation

The Design Evaluation tool is available on both the Design tab and the Data tab.

- If you evaluate a design from the Design tab, the tool will evaluate the overall design without considering any response-specific settings.
- If you evaluate a design from the Data tab, the tool will evaluate the design for a specific response and take into account all relevant settings (and data, if entered and calculated) specific to that response.

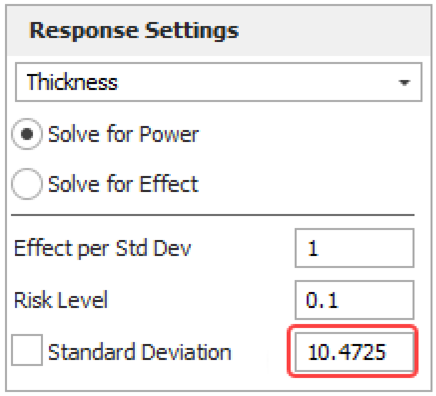
Use the Evaluation Settings page of the control panel to configure the evaluation. The page will differ slightly depending on which tab you access it from, as shown next.



Follow the steps below to define the evaluation settings.

- If you accessed the tool from the Data tab, choose a response from the **Response Settings** drop-down list. The settings you choose will apply to that response only. When accessed from the Design tab, the tool will assume all responses have the same settings.
- Choose a metric to solve for.
  - If you choose to **Solve for Power**, you will be required to specify the amount of effect in the **Effect per Std Dev** field.
    - The effect per standard deviation defines how much the effect exceeds the response variation due to noise (i.e., standard deviation). For example, if you want to know the probability of detecting an effect that is 1.5 times greater than the response's standard deviation, you'd enter **1.5**.
  - If you choose to **Solve for Effect Value**, the software will calculate an amount of effect given the **Value of Power** that you provide (e.g., if the probability of detecting an active effect is 90%, enter **0.9**).
- Enter a **Risk Level**, which is a measure of the risk that the results of analyzing the response data are incorrect (i.e., alpha = 1 - confidence level).

- The **Standard Deviation** represents the response variation that is due to noise. The default value is 1, but if you access the tool from the Data tab and then enter and calculate the response data, the default value will become the calculated standard deviation for the selected response.

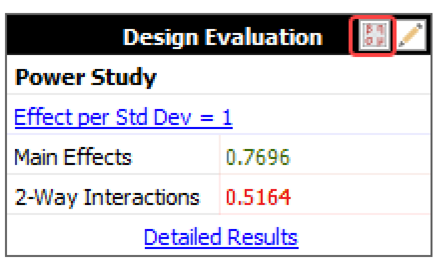


Note that the detection power is calculated in terms of the effect *per standard deviation* (i.e., the amount by which the effect exceeds the standard deviation, whatever it is), so the standard deviation itself does not affect the calculation when you solve for power. However, in this case, the total effect (Effect per Std Dev * Standard Deviation) will be shown in the Power Study table in the detailed results.

- If you access the tool from the Design tab, the area at the bottom of the page will allow you to select the terms that will be used in the evaluation. For example, if you were not interested in investigating the interaction between factors A and B, then you would you clear the **A · B** check box. When accessed from the Data tab, the tool will use the same terms you selected in the Select Terms window.

Performing the Evaluation and Interpreting the Results

After you select the desired settings, go to the Main page of the control panel and click the **Calculate** icon in the **Design Evaluation** area.



The value you selected to solve for (i.e., power or effect) will appear in the area as shown above. In this example, the probability of detecting the given amount of effect is 79.96% for main effects and 51.64% for 2-way interactions. Note that when there are multiple effects, this value refers to the effect that is the least likely to be detected. (For example, if there are two main effects A and B, and the probability of detecting A is 50% and for B it's 60%, then the Main Effects row will show 0.5000.)

To see more details concerning the evaluation (including the powers calculated for individual effects of interest), click the **Detailed Results** link at the bottom of the area to view the Evaluation Results window.

Design Evaluation Results Window

The Design Evaluation Results window shows the detailed results of the design evaluation. It is accessed by clicking the **Detailed Results** link in the control panel's **Design Evaluation** area.

All available information and tables are presented next. To view or hide any of these results in the spreadsheet, use the check boxes in the Available Report Items panel.

- The **Power Study** table shows the calculated power/effect values for each term that you selected to include in the evaluation.
    - The **Degrees of Freedom** is the number of coefficients that represent the effect of each term in the model. For qualitative factors, this is the number of levels - 1 for each term. For quantitative factors, it is always 1.
    - The last column will vary depending on whether you selected to solve for power or effect.
        - The **Power for Effect Value** is the calculated power for each term (i.e., the probability of detecting the specified amount of effect for each term). The column heading displays the total effect, which is equal to the effect per standard deviation multiplied by the standard deviation (also shown in the heading).
        - The **Effect for Power Value** is the calculated total effect for the each term (i.e., the largest difference between the response means obtained at each factor level). The column heading displays the power value that was used to calculate the effect.
- The **Alias Structure** shows which effects are aliased with each other. It takes into account only the terms you've selected to include in the evaluation. Together with your engineering knowledge, you can use this table to help determine whether any important interaction information will be lost due to aliasing. If there are no aliased effects, the sentence "All selected terms are alias free" will appear. This table is not available for one factor designs.

- The **D-Optimal Information** table shows the following values.
  - The **Determinate of X'X** is the determinant calculated for the information matrix. This value is used to compare the design to other designs and determine which is closer to being orthogonal. Larger determinant values mean you can get more information about the factors you are studying compared to a design with the same model but a smaller determinant.
  - The **D-Efficiency** is used to measure the orthogonality of a design. A value close to 1 means the design is close to being orthogonal. A value of 1 means the design is orthogonal.
  - The **Trace of X'X^-1** is the calculated trace of the variance-covariance matrix (i.e., the sum of all its diagonal elements). Like the determinate of X'X, this value is used to compare different designs. Smaller trace values mean less uncertainty for the estimated model coefficients.
- The **Regressor** table displays the regressors that were used to calculate the X'X and X'X^-1 matrices described next.
  - **X'X** is the information matrix of the design. It is used to calculate the first and second values in the D-Optimal Information table.
  - **X'X^-1** is the normalized variance-covariance matrix of the design. It is multiplied by the estimated variance of the error to obtain the variance-covariance matrix of the estimated model parameters.

The ReliaWiki resource portal has more information on design evaluation at: http://www.reliawiki.org/index.php/Design_Evaluation_and_Power_Study.

Design Evaluation Example

*The data set used in this example is available in the example database installed with the software (called "Weibull21_DOE_Examples.rsgz21"). To access this database file, choose **File > Help**, click **Open Examples Folder**, then browse for the file in the Weibull sub-folder.*

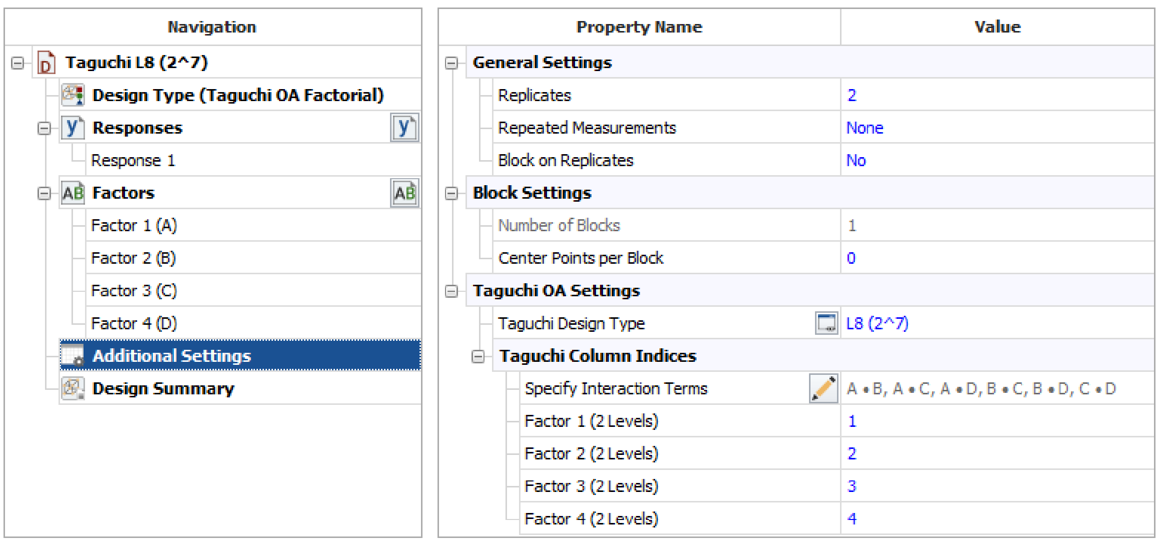
The name of the example project is "Design Evaluation - Taguchi OA vs. Two Level Full Factorial."

Experimenters often need to decide between different experiment designs. For example, a half fractional factorial design with 2 replicates will allow you to estimate the error variance more accurately than a full factorial design which uses the same number of runs, but at the cost of aliasing some effects. A full factorial design will not alias any effects, but it may be more time- and resource-intensive since it requires more factor level combinations.

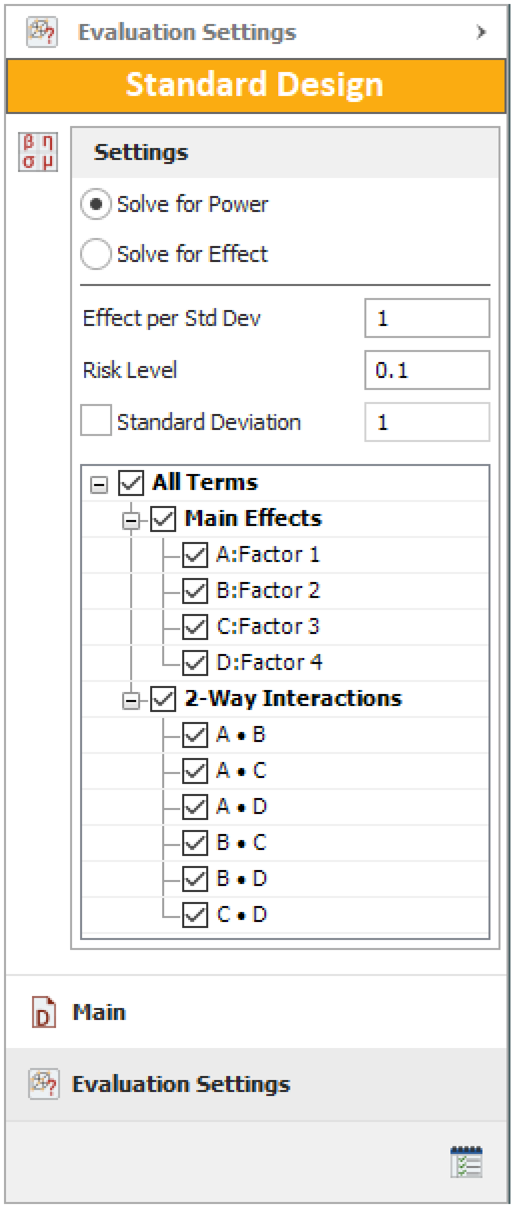
In this example, the Design Evaluation tool will be used to compare a Taguchi OA design to a two level full factorial design. The designs will be compared in terms of alias structure, detection power and resource constraints. Assume a group of engineers want to investigate the main effects and 2-way interaction effects of 4 factors run at 2 levels each. One option is to use the first 4 columns of the Taguchi L8 (2^7) with 2 replicates; the other option is to use a full factorial design. Both designs require 16 test samples. In the steps that follow, both designs will be evaluated, and the better design will be improved before the engineers agree to use it.

CREATING/EVALUATING THE TAGUCHI OA DESIGN

The Taguchi OA design is given in the "Taguchi L8 (2^7)" folio. The four factors were assigned to the first 4 columns under the Additional Settings header on the Design tab.



On the Evaluation Settings page of the control panel, specify that you will solve for power (i.e., the probability that the design will detect an active effect) and select to include all main effects and 2-way interactions in the evaluation.



Then click the **Calculate** icon to return to the Main page of the control panel and view a summary of the evaluation results in the **Design Evaluation** area.

Click the **Detailed Results** link in this area to view the calculated power values for each included term, as shown next.

Power Study		
	Degrees of Freedom	Power for Effect Value = 1.000000 (1.000000 * 1.000000 Std Dev)
A:Factor 1	1	0.57275
B:Factor 2	1	0.57275
C:Factor 3	1	0.57275
D:Factor 4	1	0.57275
A • D	1	0.57275
B • D	1	0.57275
C • D	1	0.57275
Residual	8	-

According to this table, if any of the displayed main effects or interaction effects is active, there is a 57.27% chance that the design will detect it.

Notice that some of the interactions that were selected to be included are not shown in the Power Study table. That's because they are aliased with main effects, as shown in the Alias Structure table.

Alias Structure
A:Factor 1 = A:Factor 1 + B • C
B:Factor 2 = B:Factor 2 + A • C
C:Factor 3 = C:Factor 3 + A • B

For example, according to this table, the main effect of factor A is aliased with the interaction BC. Thus, BC is not shown in the Power Study table.

CREATING/EVALUATING THE TWO LEVEL FULL FACTORIAL DESIGN

The full factorial design is given in the "2^4 Full Factorial Design - 1 Replicate" folio. On the Evaluation Settings page of the control panel, select to solve for power and include all the main effects and 2-way interaction effects.

The calculated power values for the included terms are shown next.

Power Study		
	Degrees of Freedom	Power for Effect Value = 1.000000 (1.000000 * 1.000000 Std Dev)
A:Factor 1	1	0.531763
B:Factor 2	1	0.531763
C:Factor 3	1	0.531763
D:Factor 4	1	0.531763
A • B	1	0.531763
A • C	1	0.531763
A • D	1	0.531763
B • C	1	0.531763
B • D	1	0.531763
C • D	1	0.531763
Residual	5	-

Underneath this table is an indication that "All selected terms are alias free." This means the full factorial design can be used to investigate all the main effects and 2-way interaction effects.

COMPARING THE TWO DESIGNS

- *Compare the alias structure*

  The Taguchi OA design cannot be used to investigate all the 2-way interaction effects. For example, it cannot distinguish the main effect A and the interaction effect BC. The two level full factorial design, however, does not alias any effects. Thus, the full factorial design is better in terms of aliasing.

- *Compare the detection power*

  The design evaluation shows that the power for the Taguchi OA design is 0.5727, while the power for the two level full factorial design is 0.5318. The Taguchi OA design might thereby appear better in terms of detection power. However, the two level full factorial design has more terms included in the model (because of its lack of aliasing). If we omit the terms that are aliased in the Taguchi OA design from the full factorial design as well, the power of the full factorial design will also be 0.5727. So in terms of the detection power, both designs are the same.

- *Consider other factors such as resource constraints*

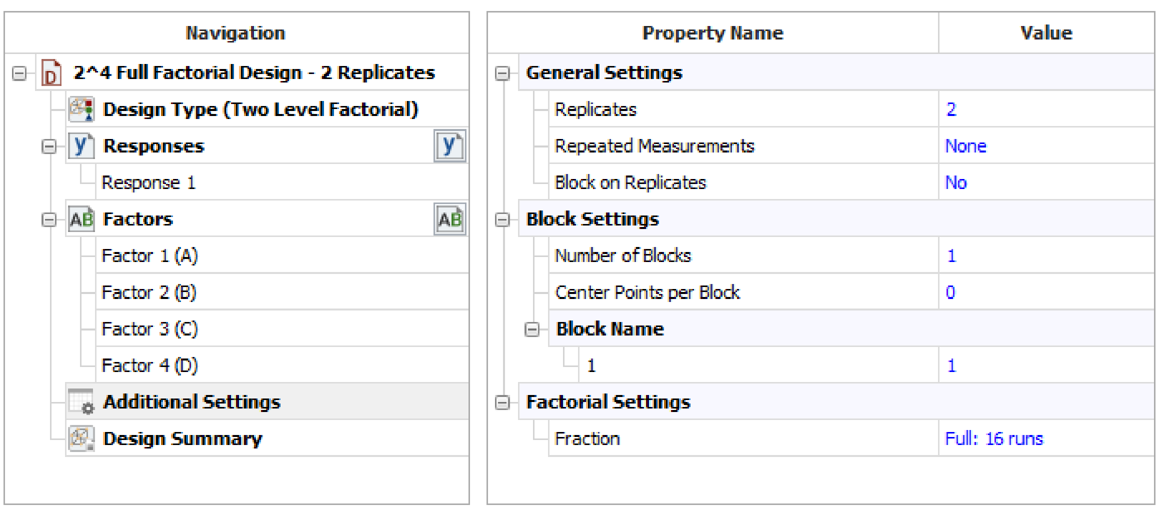
  Since there are two replicates in the Taguchi OA design, the error variance can be estimated more accurately. In the two level full factorial design, the error variance is estimated by pooling high order interactions. If some of the interactions are active, the estimated error variance may not be accurate.

CHOOSING THE RIGHT DESIGN

The experimenters decided that the most important goal is to estimate all the main effects and 2-way interactions clearly. Therefore the two level full factorial design is used. Since the power for all the main effects and 2-way interactions is only 0.5318, however, they need to increase the sample size to increase the probability of detecting active effects. So they increase the replicates and recalculate the power values. The effect of the number of replicates on the number of runs and power value is shown in the table below.

Replicates	Number of Runs	Power
1	16	0.5318
2	32	0.8623
3	48	0.9604

The experimenters decided to accept any power value above 85%, so they use 2 replicates for this experiment, as shown in the "2^4 Full Factorial Design - 2 Replicates" folio.



Factorial Designs

Factorial designs are typically used for screening factors/interactions. In other words, they help you determine which factors have a significant effect on the response and identify interactions between those factors. The following four types of factorial designs are available:

- **Two Level Factorial**: Use this design to investigate the main effects and/or interaction effects of a few factors run at two levels each. You can investigate all factors/interactions (full factorial) or only a subset of them (fractional factorial).

- **Placket-Burman Factorial**: This is a special category of two level fractional factorial design. Use this design to investigate the main effects of multiple factors run at two levels each, using few runs.

- **General Full Factorial**: Use this design to investigate the main effects and interaction effects of multiple factors run at different numbers of levels (for example, two factors at two levels each and two factors at three levels each).

- **Taguchi OA Factorial**: Use this design to investigate to investigate the main effects of multiple factors run at different numbers of levels, using few runs. This design can be thought of as a general fractional factorial design.

When selecting a factorial design type, it is important to keep these considerations in mind:

- Full factorial designs test all possible combinations of factors at each level. While such designs yield comprehensive data, they may not be feasible due to constraints on time, money and/or number of samples available for testing.

- Fractional factorial designs test a subset of the possible combinations of factors at the levels in question. This allows for the screening of larger numbers of factors/levels with less investment of time, money and/or samples. However, it also results in some loss of data for analysis, as eliminating runs naturally reduces the ability of the design to fully examine all possible effects.

  More specifically, fractionality results in some level of *aliasing* (or *confounding*), where the effect of a certain factor or factorial interaction cannot be separated from another effect. Designs in which main effects are aliased with lower-order interactions are said to be *low resolution* and are appropriate for screening (e.g., a resolution III design, in which main effects may be aliased with second-order interactions, which are interactions of two factors). Fractional factorial designs of higher resolution, along with full factorial designs, may also be useful for studying factorial effects and interactions in depth and/or for optimization. Resolution is presented in more detail in Two Level Factorial Designs.

- Using two levels per factor is generally sufficient for screening experiments. However, to test for curvature (thereby identifying factors and interaction that merit in-depth analysis) either center points or additional levels must be used. If factors have a significant curvature effect on the response, you should investigate their quadratic effects using response surface methodology.

For more information about how to use the design types, please consult the documentation on design folios.

Two Level Factorial Designs

With two level factorial designs, you can consider all combinations (for a full factorial design) or a subset of combinations (for a fractional factorial design) of up to 15 factors at 2 levels each. To design an experiment that includes factors with multiple levels and tests all factor-level combinations, use a general full factorial design instead.

Available Two Level Factorial Designs

The available two level factorial designs are shown next. This information is applicable to designs with standard response data, as well as robust parameter designs and reliability DOE.

Number of Factors

Number of Runs	2	3	4	5	6	7	8	9	10	11	12	13	14	15
4	2^2	2^{3-1}												
8		2^3	2^{4-1}	2^{5-2}	2^{6-3}	2^{7-4}								
16			2^4	2^{5-1}	2^{6-2}	2^{7-3}	2^{8-4}	2^{9-5}	2^{10-6}	2^{11-7}	2^{12-8}	2^{13-9}	2^{14-10}	2^{15-11}
32				2^5	2^{6-1}	2^{7-2}	2^{8-3}	2^{9-4}	2^{10-5}	2^{11-6}	2^{12-7}	2^{13-8}	2^{14-9}	2^{15-10}
64					2^6	2^{7-1}	2^{8-2}	2^{9-3}	2^{10-4}	2^{11-5}	2^{12-6}	2^{13-7}	2^{14-8}	2^{15-9}
128						2^7	2^{8-1}	2^{9-2}	2^{10-3}	2^{11-4}	2^{12-5}	2^{13-6}	2^{14-7}	2^{15-8}

Full Factorial Design
Resolution IV Design
Resolution V (or Higher) Design
Resolution III Design

This table provides information about the available combinations of the number of factors and the number of runs. The two level full factorial designs are displayed in white cells, and the information shown in the cells is (number of levels)^(number of factors), which yields the number of runs.

The two level fractional factorial designs are displayed in green, yellow and red cells, where:

- The information shown in the cells is the design in 2^{k-p} form. k represents the number of factors used in the design. The fractionality of the design is equal to $1/2^p$, or 2^{-p}. Thus, the design is represented by $2^k(2^{-p})$, or 2^{k-p}. Note that in such designs, the number of generators (i.e., the factorial interactions whose effects will be aliased with the main effects of particular factors) is also equal to p.

Two Level Factorial Designs: Example

*The data set used in this example is available in the example database installed with the software (called "Weibull21_DOE_Examples.rsgz21"). 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 example project is "Factorial - Two Level Full Factorial Design."

One of the initial steps in fabricating integrated circuit (IC) devices is to grow an epitaxial layer on polished silicon wafers. The wafers are mounted on a six-faceted cylinder (two wafers per facet) called a *susceptor*, which is spun inside a metal bell jar. The jar is injected with chemical vapors through nozzles at the top of the jar and heated. The process continues until the epitaxial layer grows to a desired thickness. The nominal value for thickness is 14.5 µm with specification limits of 14.5 ± 0.5 µm.

The current settings caused variations that exceeded the specification by 1.0 µm. Thus, the experimenters first need to find the factors that affect the process. Then further experiments will be conducted to optimize the process. 96 units are available for testing, and the factors and levels shown next will be used.

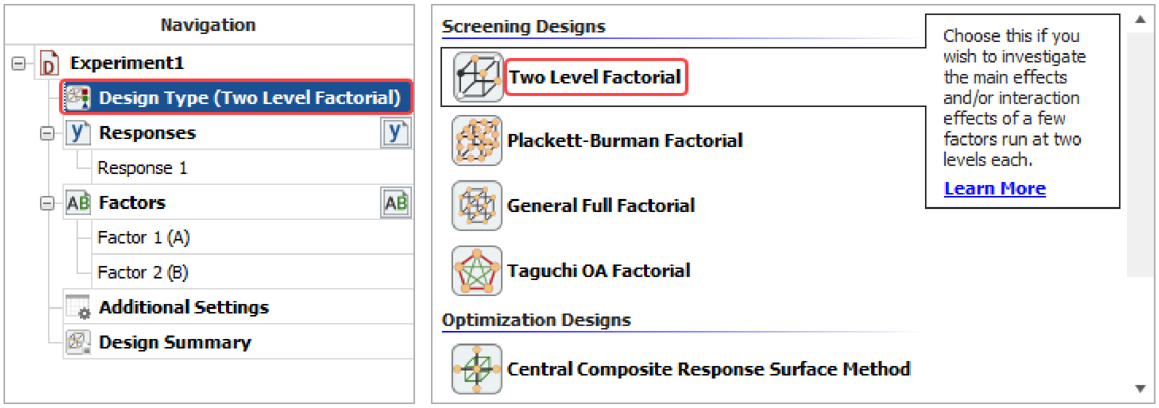
Factor	Name	Low Level	High Level
A	Susceptor-Rotation Method	Continuous	Oscillating
B	Nozzle Position	2	6
C	Deposition Temperature	1210	1220
D	Deposition Time	Low	High

DESIGNING THE EXPERIMENT

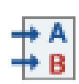
To find the important factors, the experimenters create a two level full factorial design folio, perform the experiment according to the design, and then enter the response values into the folio for analysis. The design matrix and the response data are given in the "Two Level Full Factorial Design" folio. The following steps describe how to create this folio on your own.

- Choose **Home > Insert > Standard Design** to add a standard design folio to the current project.

- Click **Design Type** in the folio's navigation panel, and then select **Two Level Factorial** in the input panel.



- Rename the response by clicking **Response 1** in the navigation panel and entering **Thickness** in the input panel.
- Specify the number of factors by clicking the **Factors** heading in the navigation panel and choosing **4** from the **Number of Factors** drop-down list.
- Define each factor by clicking it in the navigation panel and editing its properties in the input panel. The properties are given in the table above. Note that factors A and D use **Qualitative** levels; this is specified in the **Factor Type** drop-down list.
- Rename the folio by clicking the **Experiment1** heading in the navigation panel and entering **Two Level Factorial Design** for the **Name** in the input panel.
- Click the **Additional Settings** heading. In the input panel, set the number of **Replicates** to **6**. Note that by leaving **Full** selected under the **Factorial Settings** heading, you are indicating that this will be a full factorial design.
- Finally, click the **Build** icon on the control panel to create a Data tab that allows you to view the test plan and enter response data.



ANALYSIS AND RESULTS

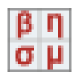
The data set for this example is given in the "Two Level Full Factorial" folio of the example project. After you enter the data from the folio, you can perform the analysis by doing the following:

Note: To minimize the effect of unknown nuisance factors, the run order is randomly generated when you create the design. Therefore, if you followed these steps to create your own folio, the order of runs on the Data tab may be different from that of the folio in the example file. This can lead to different results. To ensure that you get the very same results described next, show the Standard Order column in your folio, then click a cell in that column and choose **Sheet > Sheet Actions > Sort > Sort Ascending**. This will make the order of runs in your folio the same as that of the example file. Then copy the response data from the example file and paste it into the Data tab of your folio.

- Make sure all main effects and interactions will be considered. To do this, click the **Select Terms** icon on the control panel.

  In the window that appears, select the **All Terms** check box. Then click **OK**.

- Click the **Calculate** icon.



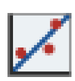
- The results in the **Analysis Summary** area on the control panel show that the nozzle position and deposition time have significant effects on the thickness, as well as the interaction of terms C and D (i.e., deposition temperature and deposition time).

Analysis Summary

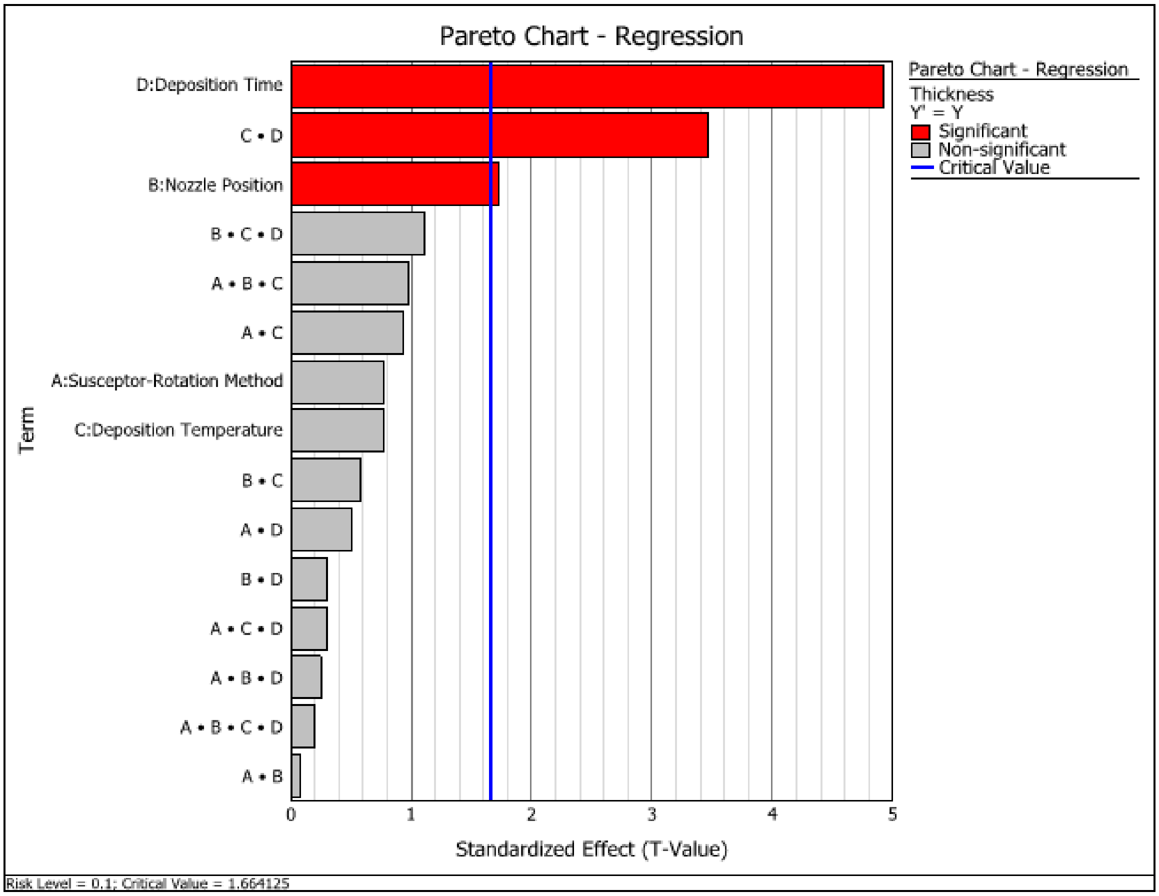
Significant Terms

Term	Coefficient
B:Nozzle Position	0.086271
D:Deposition Time	0.245021
C • D	-0.172521

- To view a plot comparing the standardized effect of each term, click the **Plot** icon.



  Then choose **Pareto Charts - Regression** from the **Plot Type** drop-down list. The following plot appears. The blue line in the plot marks the critical value determined by the risk level specified on the Analysis Settings page of the control panel. If the bar goes past the blue line, then the effect is considered significant.



OPTIMIZATION

To optimize the response, you should work with a model that only includes the significant terms. The results for the reduced model of the analysis are given in the "Reduced Model" folio of the example project. The following steps describe how to create this folio on your own.

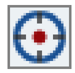
- Right-click the "Two Level Full Factorial Design" folio in the current project explorer. Then choose **Duplicate** from the shortcut menu.
- Right-click the new, duplicated folio in the current project explorer and choose **Rename** from the shortcut menu. Change the name to "Reduced Model."
- Open the new folio. Then click the **Select Terms** icon on the control panel.

To automatically select only those terms that were found to be significant, click the **Select Significant Terms** icon. Since the interaction of factors C and D is significant, C must also be included in the model. So select the **C:Deposition Temperature** check box as well. Then click **OK**.

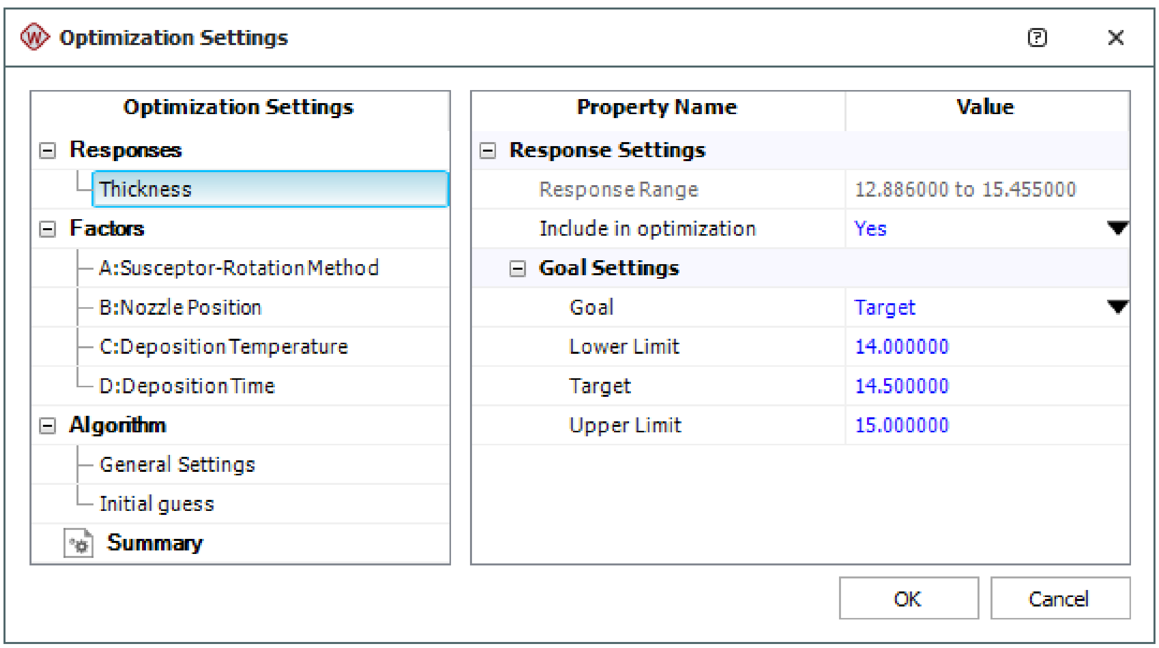
- Click the **Calculate** icon to create a reduced model using the current data.

The factor level combinations that will produce the optimum response value are given in the "Optimal Solution Plot" folio. The following steps describe how to create this folio on your own.

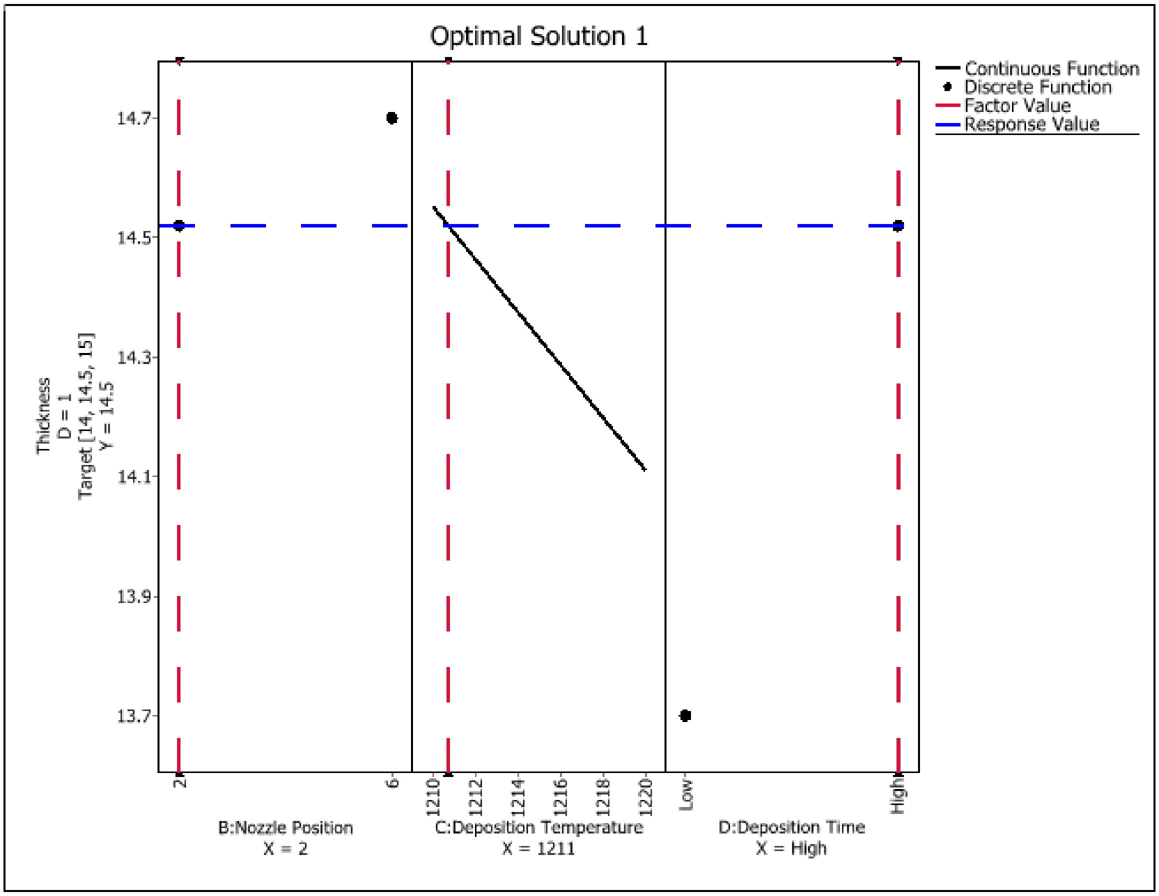
- To find the best settings of the significant factors, click the **Design - Optimization** icon on the control panel of the "Reduce Mode" folio.



- In the window that appears, use the settings shown next and click **OK**. According to these settings, the ideal thickness for the epitaxial layer is 14.5 µm, but thicknesses between 14 µm and 15 µm are still acceptable.

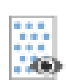


- An Optimization folio will appear with an optimal solution plot showing the first solution, displayed next. (The **Solutions** area on the control panel shows all the solutions that were found, and you can click them to view the plot representing the solution.)

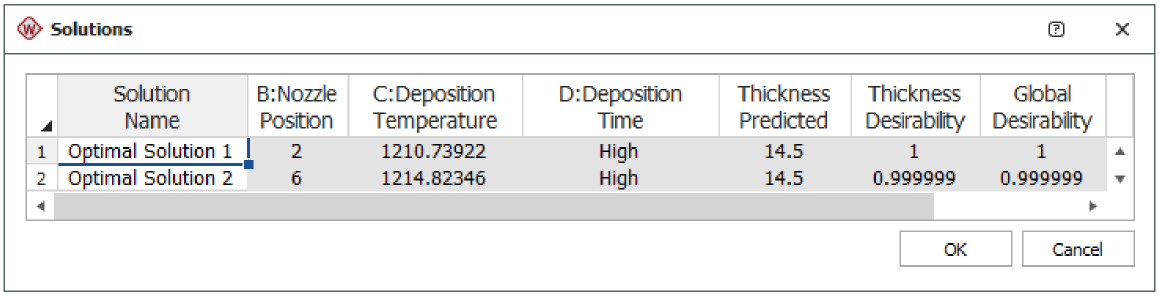


This plot shows one combination of factor settings that will produce the target response value (marked with a horizontal dotted line). Factors B and D are qualitative, so the plot uses dots (i.e., a discrete function) to show what the response level would be given either one of the two factor levels used in the experiment. Factor C is quantitative, so a continuous function is used to calculate the optimal temperature.

- You can choose **Optimization > Solutions > View Solutions** or click the icon on the control panel to see all the optimal solutions in numerical format.



Two optimal solutions for factors B, C and D were found, and they are shown next.



	Solution Name	B:Nozzle Position	C:Deposition Temperature	D:Deposition Time	Thickness Predicted	Thickness Desirability	Global Desirability
1	Optimal Solution 1	2	1210.73922	High	14.5	1	1
2	Optimal Solution 2	6	1214.82346	High	14.5	0.999999	0.999999

CONCLUSION

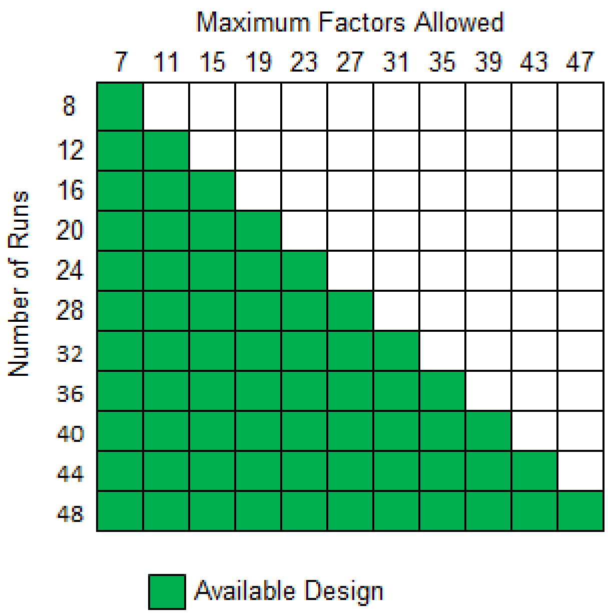
Two optimum solutions were found that give the same predicted thickness. In choosing between these two combinations of factor levels, it is important to identify which is better in terms of variability. To do this, you can perform a variability analysis, where the response is the standard deviation of the observations at each setting. (See Variability Analysis Example.)

Plackett-Burman Designs

Plackett-Burman designs are highly fractional designs that allow you to consider a subset of combinations of up to 47 factors at two levels each. Unlike two level fractional factorial designs, the number of runs in Placket-Burman designs is a multiple of four (rather than only a power of 2), and the number of factors can be up to one less than the number of runs (e.g., an experiment with 16 runs can screen up to 15 factors). Because this design is highly fractional, it is not appropriate for studying interaction effects. The design matrix was developed by Robin L. Plackett and J. P. Burman in 1946.

Available Plackett-Burman Designs

The available Plackett-Burman designs are shown next. This information is applicable to designs with standard response data, as well as robust parameter designs and reliability DOE.



This table provides information about the available combinations of the number of factors and the number of runs. The number of runs in a Plackett-Burman design is always a multiple of four. The number of factors can be up to one less than the number of runs. For example, this table shows that a design using 20 runs may use up to 19 factors.

Plackett-Burman Designs: Example

*The data set used in this example is available in the example database installed with the software (called "Weibull21_DOE_Examples.rsgz21"). 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 example project is "Factorial - Plackett-Burman Design."

Plackett-Burman design is a "screening design." Such designs are traditionally used for investigating a large number of factors to see which have a significant effect on the response. In the analysis of these designs, usually only main effects are estimated.

Consider a life test of weld-repaired castings (as described in Wu, Jeff and Hamada, Michael, *Experiments: Planning, Analysis, And Parameter Design Optimization*, John Wiley & Sons, New York, 2000). The objective of the test is to identify the important factors that affect the life and estimate the factor levels that will improve the product life. There are seven factors under investigation. A two level full factorial design would require $2^7 = 128$ runs, and thus would be too time-consuming and costly. Therefore, an eight-run Plackett-Burman experiment will be conducted instead.

For this example, the seven factors and associated levels are shown next. The response is the logged failure time of each test unit.

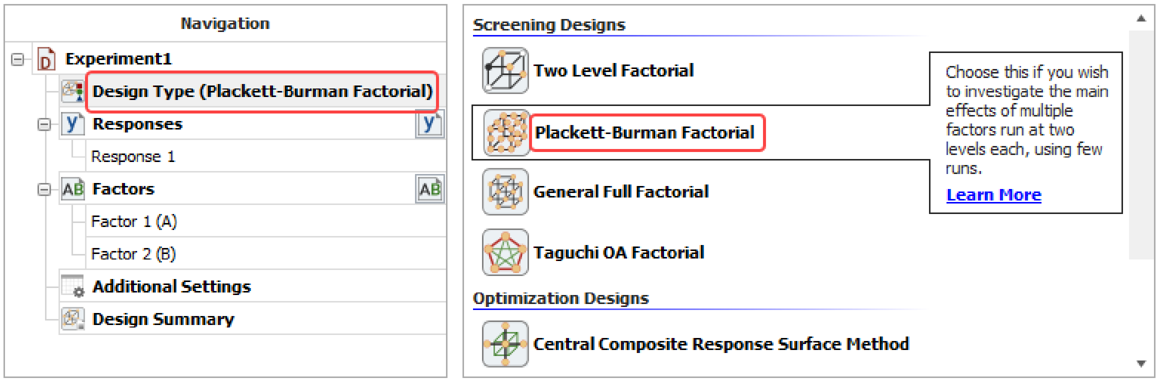
Factor	Low Level	High Level
Initial Structure	As Received	Beta Treat
Bead Size	Small	Large
Pressure Treat	None	HIP
Heat Treat	Anneal	Solution Treat/Age
Cooling Rate	Slow	Rapid
Polish	Chemical	Mechanical
Final Treat	None	Peen

DESIGN THE EXPERIMENT

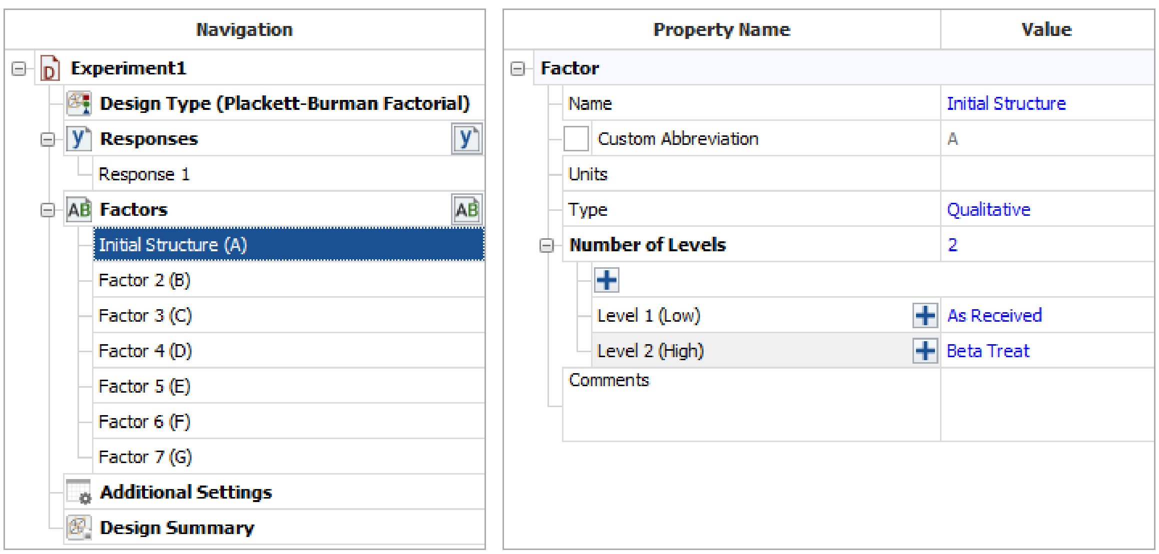
The experimenters create a Plackett-Burman design folio, perform the experiment according to the design, and enter the response values for further analysis. The design matrix and the response data are given in the "Cast Fatigue Experiment" folio of the example project. The following steps describe how to create this folio on your own.

- Choose **Home > Insert > Standard Design** to add a standard design folio to the current project.

- Click **Design Type** on the folio's navigation panel, and then select **Plackett-Burman Factorial** in the input panel.

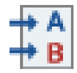


- Rename the response by clicking **Response 1** in the navigation panel and entering **Time** in the input panel.
- Specify the number of factors by clicking the **Factors** heading in the navigation panel and choosing **7** from the **Number of Factors** drop-down list.
- Define each factor by clicking it in the navigation panel and editing its properties in the input panel. The properties are given in the table above. Note that all factors use **Qualitative** levels; this is specified in the **Factor Type** drop-down list. The first factor is defined as shown next.



- Rename the folio by clicking the **Experiment1** heading in the navigation panel and entering **Cast Fatigue Experiment** for the **Name** in the input panel.
- Click the **Additional Settings** heading. On the input panel, set the number of **Replicates** to **1**. Note that by leaving **8** selected for the **Number of Runs**, you are indicating that this design will use the minimum number of runs required for a Plackett-Burman design with 7 factors.

- Finally, click the **Build** icon on the control panel to create a Data tab that allows you to view the test plan and enter response data.

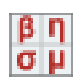


ANALYSIS AND RESULTS

The data set for this example is given in the "Cast Fatigue Experiment" folio of the example project. After you enter the data from the example folio, you can perform the analysis by doing the following:

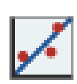
Note: To minimize the effect of unknown nuisance factors, the run order is randomly generated when you create the design. Therefore, if you followed these steps to create your own folio, the order of runs on the Data tab may be different from that of the folio in the example file. This can lead to different results. To ensure that you get the very same results described next, show the Standard Order column in your folio, then click a cell in that column and choose **Sheet > Sheet Actions > Sort > Sort Ascending**. This will make the order of runs in your folio the same as that of the example file. Then copy the response data from the example file and paste it into the Data tab of your folio.

- On the Analysis Settings page of the control panel, select to use **Individual Terms** in the analysis.
- Return to the Main page of the control panel and click the **Calculate** icon.

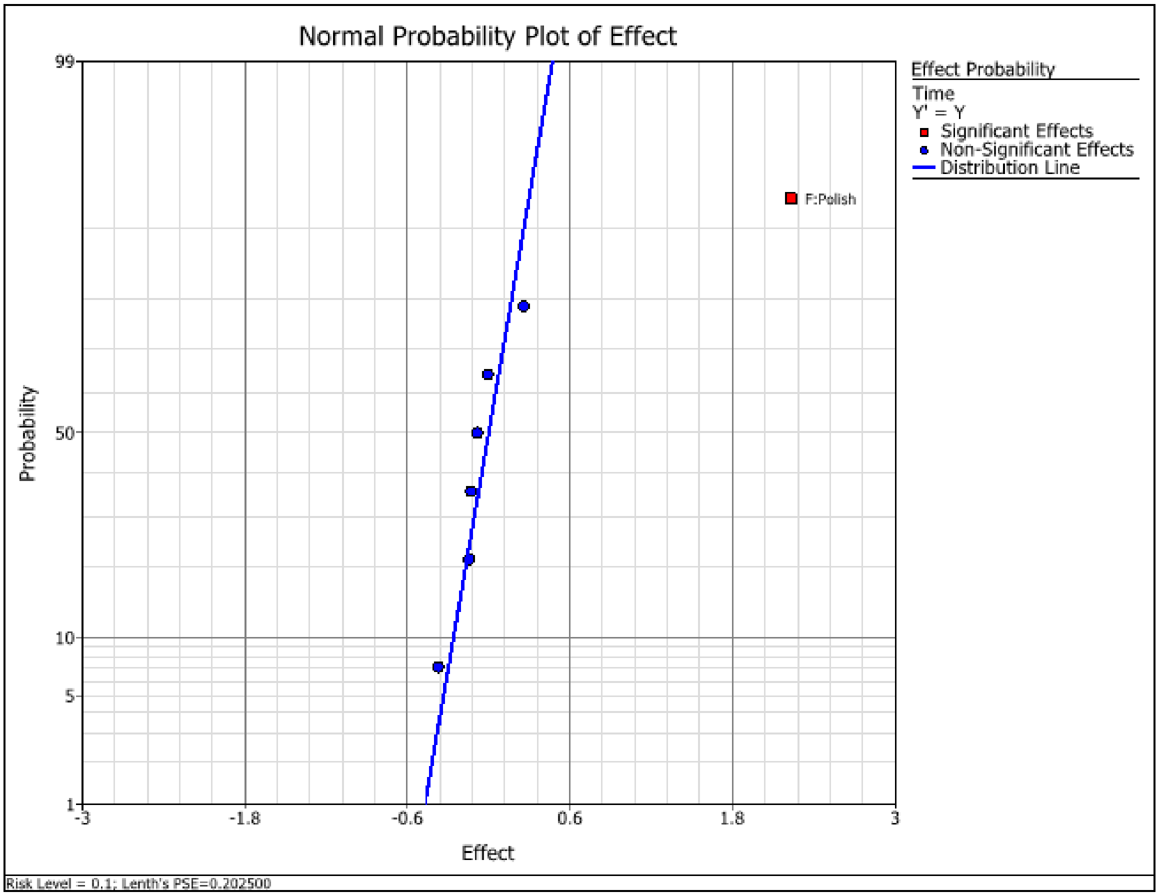


  The ANOVA table and the Regression Information table for the model are available in the Analysis Summary window after you click the **View Analysis Summary** icon on the control panel.

- Click the **Plot** icon to add the Analysis Plot tab to the folio.



  Choose the **Effect Probability** plot from the **Plot Type** drop-down list, where you can see that effect F is significant.



CONCLUSIONS

Click the **View Analysis Summary** icon on the control panel and select to the view the Regression Table. The effect values for each factor are contained in the table and shown next (some values were not calculated because the design does not use replicates):

Regression Table

Regression Information							
Term	Effect	Coefficient	Standard Error	Low Confidence	High Confidence	T Value	P Value
Intercept		5.904	-	-	-	-	-
A:Initial Structure	-0.142	-0.071	-	-	-	-	-
B:Bead Size	-0.082	-0.041	-	-	-	-	-
C:Pressure Treat	-0.368	-0.184	-	-	-	-	-
D:Heat Treat	-0.002	-0.001	-	-	-	-	-
E:Cooling Rate	-0.128	-0.064	-	-	-	-	-
F:Polish*	2.232	1.116	-	-	-	-	-
G:Final Treat	0.258	0.129	-	-	-	-	-

*: Significant terms according to Lenth's method

Factors A through E have negative effect values. F and G have positive effect values. Thus, assuming there are no interaction effects, a higher product life can be achieved by setting A, B, C, D and E at their low levels, and setting F and G at their high levels. If desired, further experiments can be conducted to study the interaction effects of these factors.

Moreover, since factor F's effect was great enough to be considered significant, it was found to be the most important factor.

General Full Factorial Designs

General full factorial designs allow you to consider all combinations of up to 7 factors at up to 20 levels each. It is not required that all factors have the same number of levels. If there are many factors with more than two levels, the number of runs for a general full factorial design may be prohibitively large. In such cases, Taguchi orthogonal array (OA) design should be used.

Available General Full Factorial Designs

The limitations on general full factorial designs are shown next. This information is applicable to designs with standard response data, R-DOE and robust parameter designs.

	Maximum Allowed
Factors	7
Levels per Factor	20
Rows per Design	65,000

This table provides information about the maximum number of factors, levels and rows possible in a general full factorial design.

General Full Factorial Designs: Example

*The data set used in this example is available in the example database installed with the software (called "Weibull21_DOE_Examples.rsgz21"). 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 example project is "Factorial - General Full Factorial Design."

In this example, a soft drink bottler is interested in obtaining more uniform fill heights in the bottles (as described in Montgomery, D. C. *Design and Analysis of Experiments*, 5th edition, John Wiley & Sons, New York, 2001). The filling machine is designed to fill each bottle to the correct target height, but in practice, there is variation around this target, and the bottler would like to examine the sources of this variability so she can reduce it. There are three control factors. The factors and associated levels that will be used in the experiment are shown next.

Factor	Unit	Level 1	Level 2	Level 3
Percent Carbonation	%	10	12	14
Operating Pressure	psi	25	30	-
Line Speed	bpm	200	250	-

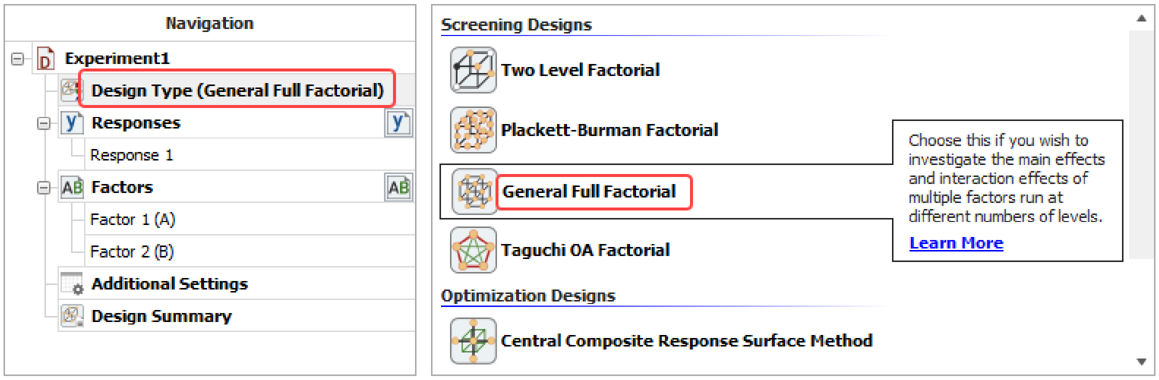
There are two replicates at each factor setting, making a total of 24 runs. The response is the average deviation from the target fill height observed in a production run of bottles. Positive deviations are fill heights above the target.

DESIGN THE EXPERIMENT

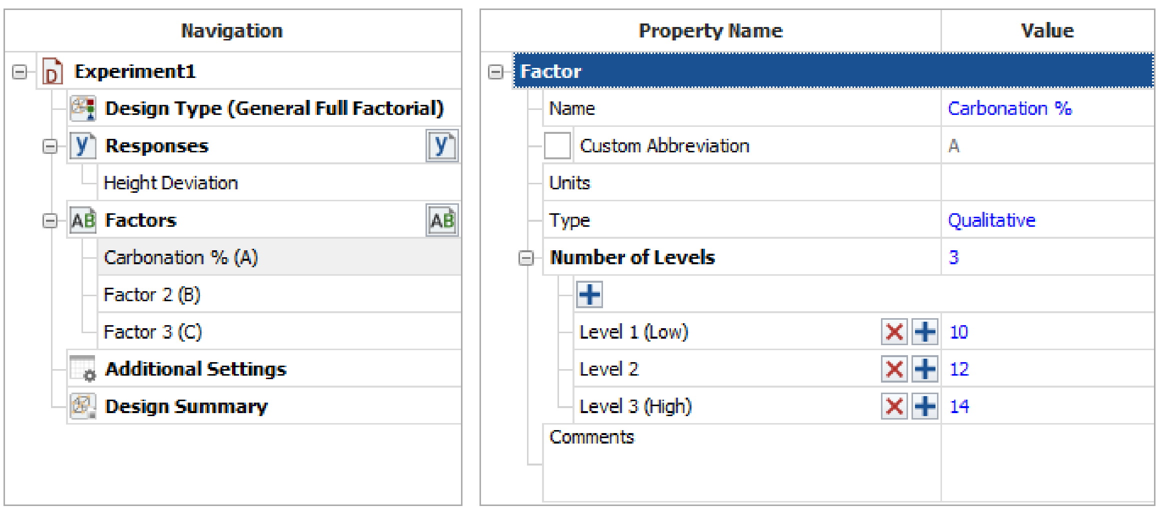
The experimenters create a general full factorial design, perform the experiment according to the design, and then enter the response values into the folio for analysis. The design matrix and the response data are given in the "Soft Drink Bottling Experiment" folio. The following steps describe how to create this folio on your own.

- Choose **Home > Insert > Standard Design** to add a standard design folio to the current project.

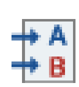
- Click **Design Type** in the folio's navigation panel, and then select **General Full Factorial** in the input panel.



- Rename the response by clicking **Response 1** in the navigation panel and entering **Height Deviation** in the input panel.
- Specify the number of factors by clicking the **Factors** heading in the navigation panel and choosing **3** from the **Number of Factors** drop-down list.
- Define each factor by clicking it in the navigation panel and editing its properties in the input panel. The properties are given in the table above. The first factor is defined as shown next.



- Rename the folio clicking the **Experiment1** heading in the navigation panel and entering **Soft Drink Bottling Experiment** for the **Name** in the input panel.
- Click the **Additional Settings** heading. In the input panel, set the number of **Replicates** to **2**.
- Finally, click the **Build** icon on the control panel to create a Data tab that allows you to view the test plan and enter response data.

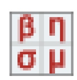


ANALYSIS AND RESULTS - PART 1

The data set for this example is given in the "Soft Drink Bottling Experiment" folio of the example project. After you enter the data from the example folio, you can perform the analysis by doing the following:

Note: To minimize the effect of unknown nuisance factors, the run order is randomly generated when you create the design. Therefore, if you followed these steps to create your own folio, the order of runs on the Data tab may be different from that of the folio in the example file. This can lead to different results. To ensure that you get the very same results described next, show the Standard Order column in your folio, then click a cell in that column and choose **Sheet > Sheet Actions > Sort > Sort Ascending**. This will make the order of runs in your folio the same as that of the example file. Then copy the response data from the example file and paste it into the Data tab of your folio.

- On the Analysis Settings page of the control panel, select to use **Individual Terms** in the analysis.
- Return to the Main page of the control panel and click the **Calculate** icon.



Click the **View Analysis Summary** icon on the control panel and select to the view the ANOVA Table.

ANOVA Table					
Source of Variation	Degrees of Freedom	Sum of Squares [Partial]	Mean Squares [Partial]	F Ratio	P Value
Model	11	328.125	29.829545	42.112299	7.417273E-8
A:Carbonation %	2	252.75	126.375	178.411765	1.186249E-9
B:Operating Pressure	1	45.375	45.375	64.058824	0.000004
C:Speed	1	22.041667	22.041667	31.117647	0.00012
A • B	2	5.25	2.625	3.705882	0.055808
A • C	2	0.583333	0.291667	0.411765	0.671494
B • C	1	1.041667	1.041667	1.470588	0.248587
A • B • C	2	1.083333	0.541667	0.764706	0.486871
Residual	12	8.5	0.708333		
Pure Error	12	8.5	0.708333		
Total	23	336.625			

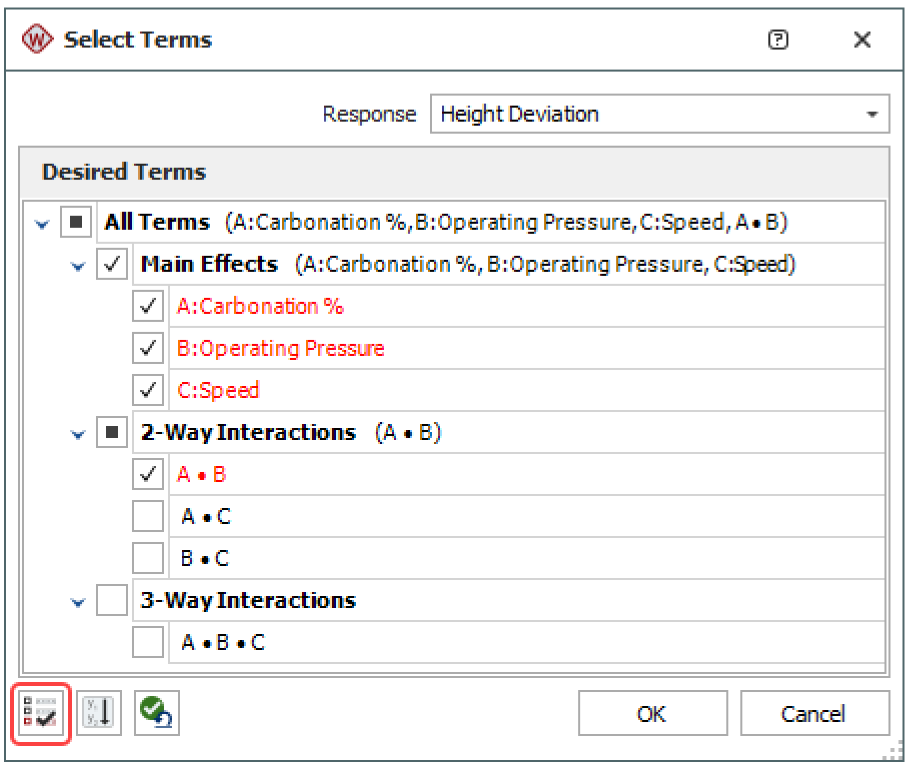
From this table, you can see that effects A, B, C and AB are significant. To simplify the analysis, the next step is to recreate the model using only the significant terms.

ANALYSIS AND RESULTS - PART 2

The results for the reduced model are given in the "Reduced Model" folio of the example project. The following steps describe how to create this folio on your own and then use the results to find the factor settings that provide the smallest deviation in height.

- Right-click the design folio in the current project explorer and select **Duplicate Item** from the shortcut menu. Rename the new folio with an appropriate name.
- Click the **Select Terms** icon on the new folio's control panel.

- In the Select Terms window that appears, click the **Select Significant Effects** button to select only the significant effects to calculate the new model, as shown next, then click **OK**.



- Click the **Calculate** icon in the new folio's control panel.
- Click the **View Analysis Summary** icon on the control panel. In the Analysis Summary window that appears, view the Diagnostic Information table. The table is shown next. In order to identify which factor settings can provide the smallest height deviation, look for the runs in the table with the Fitted Value (YF) closest to zero (highlighted).

Diagnostics									
Standard Order	Run Order	Actual Value (Y)	Fitted Value (YF)	Residual	Standardized Residual	Studentized Residual	External Studentized Residual	Leverage	Cook's Distance
24	1	10	10.208333	-0.208333	-0.256574	-0.304855	-0.296565	0.291667	0.005467
16	2	1	1.208333	-0.208333	-0.256574	-0.304855	-0.296565	0.291667	0.005467
10	3	7	6.458333	0.541667	0.667092	0.792624	0.783573	0.291667	0.036956
23	4	7	8.291667	-1.291667	-1.590759	-1.890103	-2.063231	0.291667	0.210147
12	5	11	10.208333	0.791667	0.974981	1.15845	1.171034	0.291667	0.078942
3	6	0	-0.708333	0.708333	0.872352	1.036508	1.038925	0.291667	0.063197
17	7	1	0.041667	0.958333	1.180241	1.402335	1.446713	0.291667	0.115679
7	8	3	3.041667	-0.041667	-0.051315	-0.060971	-0.059157	0.291667	0.000219
1	9	-3	-2.208333	-0.791667	-0.974981	-1.15845	-1.171034	0.291667	0.078942
18	10	2	1.958333	0.041667	0.051315	0.060971	0.059157	0.291667	0.000219
11	11	9	8.291667	0.708333	0.872352	1.036508	1.038925	0.291667	0.063197
20	12	6	4.958333	1.041667	1.28287	1.524277	1.591518	0.291667	0.136672
19	13	2	3.041667	-1.041667	-1.28287	-1.524277	-1.591518	0.291667	0.136672
2	14	0	-0.291667	0.291667	0.359204	0.426798	0.416291	0.291667	0.010715
9	15	5	4.541667	0.458333	0.564463	0.670682	0.65944	0.291667	0.02646
15	16	-1	-0.708333	-0.291667	-0.359204	-0.426798	-0.416291	0.291667	0.010715
4	17	1	1.208333	-0.208333	-0.256574	-0.304855	-0.296565	0.291667	0.005467
14	18	-1	-0.291667	-0.708333	-0.872352	-1.036508	-1.038925	0.291667	0.063197
22	19	6	6.458333	-0.458333	-0.564463	-0.670682	-0.65944	0.291667	0.02646
13	20	-1	-2.208333	1.208333	1.488129	1.768161	1.898835	0.291667	0.183906
6	21	1	1.958333	-0.958333	-1.180241	-1.402335	-1.446713	0.291667	0.115679
21	22	4	4.541667	-0.541667	-0.667092	-0.792624	-0.783573	0.291667	0.036956
5	23	0	0.041667	-0.041667	-0.051315	-0.060971	-0.059157	0.291667	0.000219
8	24	5	4.958333	0.041667	0.051315	0.060971	0.059157	0.291667	0.000219

CONCLUSIONS

You can see that the 9th and 10th runs in the standard order are at the best combination of settings. The settings for these particular runs on the Data tab are shown to be A = 12, B = 25, C = 200. Under these settings, the expected deviation is lowest.

Note that the best factor settings in this case are limited to those settings actually used in the experiment. This limitation can be avoided using response surface methodology, which may allow you to find the optimal settings for the manufacturing process.

Taguchi Orthogonal Array (OA) Factorial Designs

Taguchi orthogonal array (OA) factorial designs are highly fractional designs that allow you to consider a selected subset of combinations of multiple factors run at different numbers of levels. The number of factors that can be considered is dependent on the factor levels used.

Orthogonal arrays are balanced to ensure that all levels of all factors are considered equally. For this reason, the factors can be evaluated independently of each other despite the fractionality of the design. The available designs are based on the design matrix created by Genichi Taguchi.

Available Taguchi OA Designs

The available single level and mixed level Taguchi OA designs are shown next. This information is applicable to designs with standard response data, as well as robust parameter designs and reliability DOE.

Single Level Designs

	Factor Levels			
Number of Runs	2	3	4	5
4	L4(2^3)			
8	L8(2^7)			
9		L9(3^4)		
12	L12(2^{11})			
16	L16(2^{15})		L16(4^5)	
25				L25(5^8)
27		L27(3^{13}) L27(3^{22})		
32	L32(2^{31})			

Mixed Level Designs

Number of Runs	Factor Levels: 2 and 3	2 and 4	2 and 8	3 and 6
8		L8(2^{4} x 4^{1})		
16		L16(2^{9} x 4^{2}) L16(2^{6} x 4^{3}) L16(2^{3} x 4^{4}) L16(2^{12} x 4^{1})	L16(2^{8} x 8^{1})	
18	L18(2^{1} x 3^{7}) L18(2^{2} x 3^{6})			L18(3^{6} x 6^{1})
32		L32(2^{1} x 4^{9})		
36	L36(2^{11} x 3^{12}) L36(2^{3} x 3^{13})			
54	L54(2^{1} x 3^{25})			

These tables provide information about the available combinations of the number of factor levels and the number of runs. In the design notation, the value immediately following L represents the number of runs used in the design. Within the parentheses, each base value represents a number of levels and each exponent represents a number of factors. For example:

- The L8(2^7) design uses eight experimental runs to investigate up to 7 two-level factors.
- The L8(2^4 x 4^1) design uses eight experimental runs to investigate up to 4 two-level factors and 1 four-level factor.

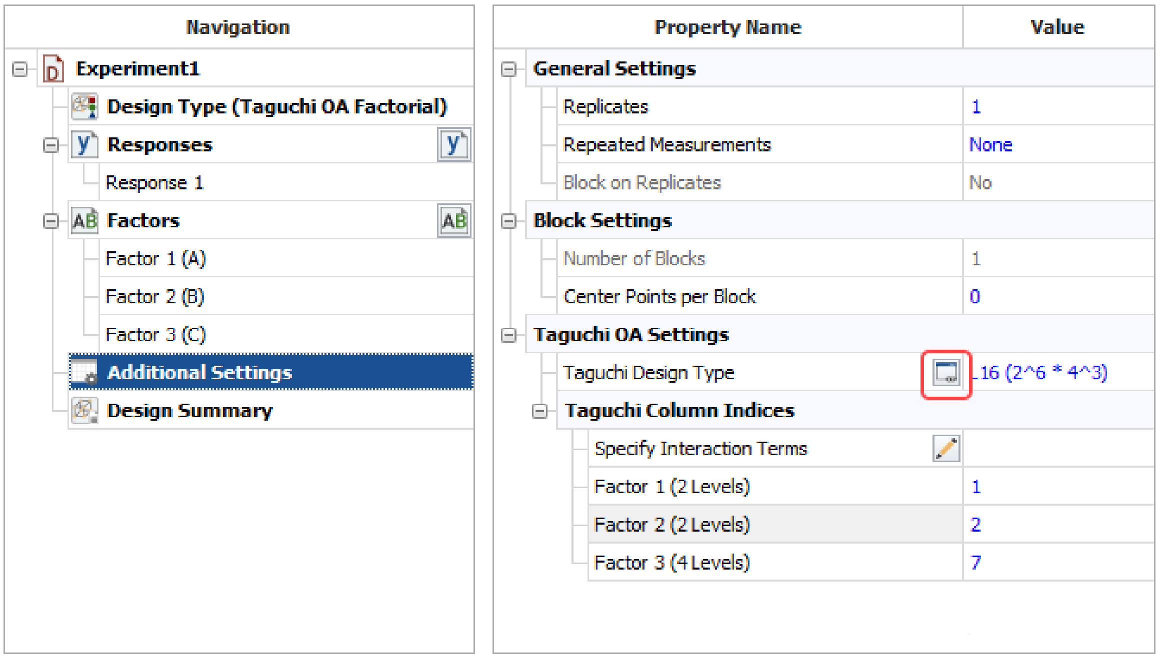
The ReliaWiki resource portal provides the design tables for all of these Taguchi OA designs at: http://www.reliawiki.org/index.php/Taguchi_Orthogonal_Arrays.

The alias relations are given at: http://www.reliawiki.org/index.php/Alias_Relations_for_Taguchi_Orthogonal_Arrays.

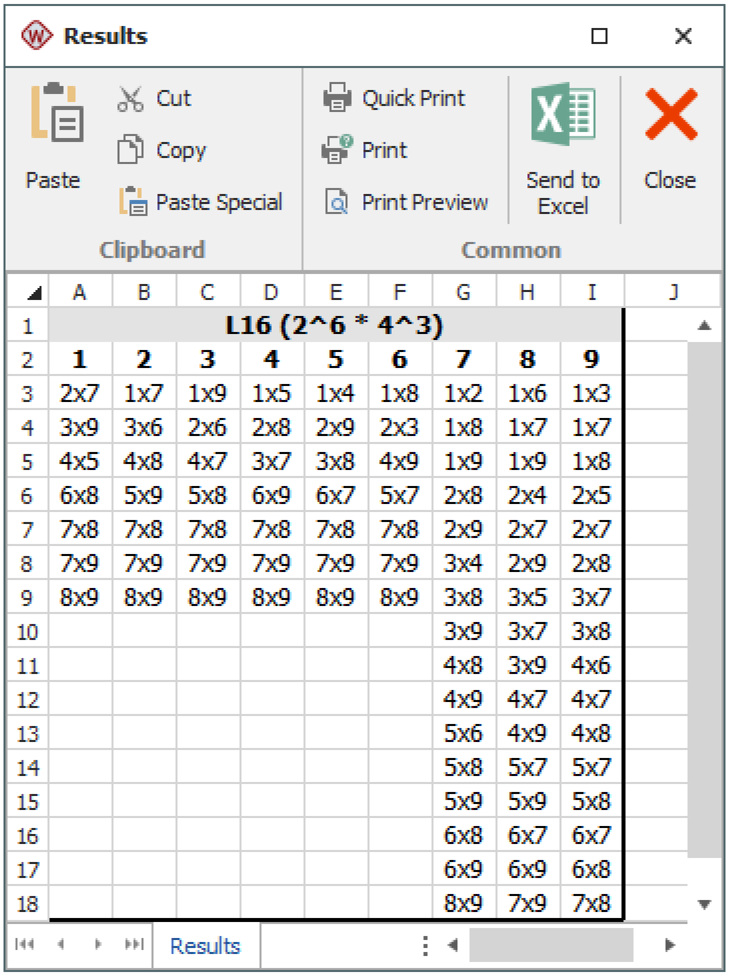
Taguchi OA Alias Table

The alias table for a Taguchi orthogonal array (OA) factorial design shows the aliased effects that would result from assigning certain factors to certain columns in the orthogonal array of the selected Taguchi design type (such as an L8 (2^7) design).

The table can be accessed from the Design tab of the standard design folio while configuring a Taguchi OA factorial design. To open it, click the **Additional Settings** heading in the navigation panel and then click the **View Alias Table** icon in the input panel.



As an example, the L16 (2^6 * 4^3) design uses 16 runs to investigate the main effects of up to 6 two-level factors and up to 3 four-level factors. The alias table for this design is shown next.



L16 (2^6 * 4^3)

1	2	3	4	5	6	7	8	9
2x7	1x7	1x9	1x5	1x4	1x8	1x2	1x6	1x3
3x9	3x6	2x6	2x8	2x9	2x3	1x8	1x7	1x7
4x5	4x8	4x7	3x7	3x8	4x9	1x9	1x9	1x8
6x8	5x9	5x8	6x9	6x7	5x7	2x8	2x4	2x5
7x8	7x8	7x8	7x8	7x8	7x8	2x9	2x7	2x7
7x9	7x9	7x9	7x9	7x9	7x9	3x4	2x9	2x8
8x9	8x9	8x9	8x9	8x9	8x9	3x8	3x5	3x7
						3x9	3x7	3x8
						4x8	3x9	4x6
						4x9	4x7	4x7
						5x6	4x9	4x8
						5x8	5x7	5x7
						5x9	5x9	5x8
						6x8	6x7	6x7
						6x9	6x9	6x8
						8x9	7x9	7x8

The numbers in Taguchi OA alias tables represent factors that are assigned to columns in the array. For instance, "1" represents the factor assigned to the first column in the array (if any). "2x3" represents the interaction effect of the factors assigned to the second and third columns. Each column in an alias table lists the 2-way interaction effects that are aliased with the main effect of the factor listed in the column heading.

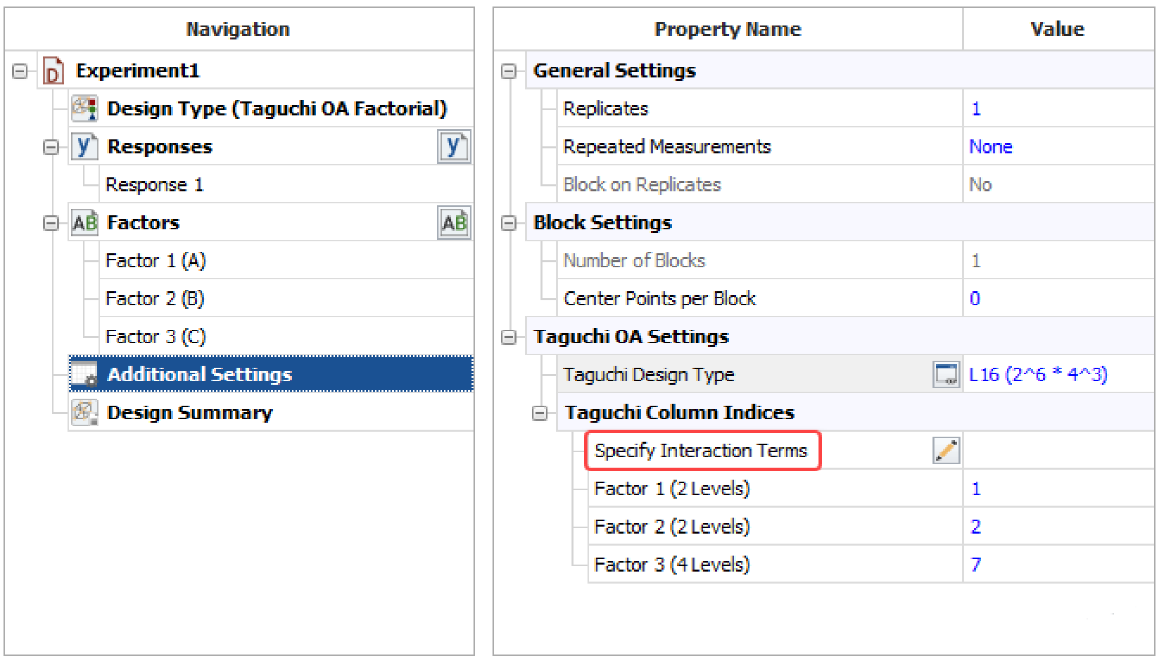
For example, suppose your design includes 2 two-level factors (A and B) and one four-level factor (C). Furthermore, suppose you used the drop-down lists under the **Taguchi Column Indices** heading of the Additional Settings to assign factor A to the first column in the orthogonal array (A = 1), B to the second column (B = 2) and C to the seventh column (C = 7). According to the above alias table, the main effect of A (1) will be aliased with the interaction effect BC (2x7).

The ReliWiki resource portal has more information on Taguchi OA alias tables at: http://www.reliawiki.org/index.php/Highly_Fractional_Factorial_Designs.

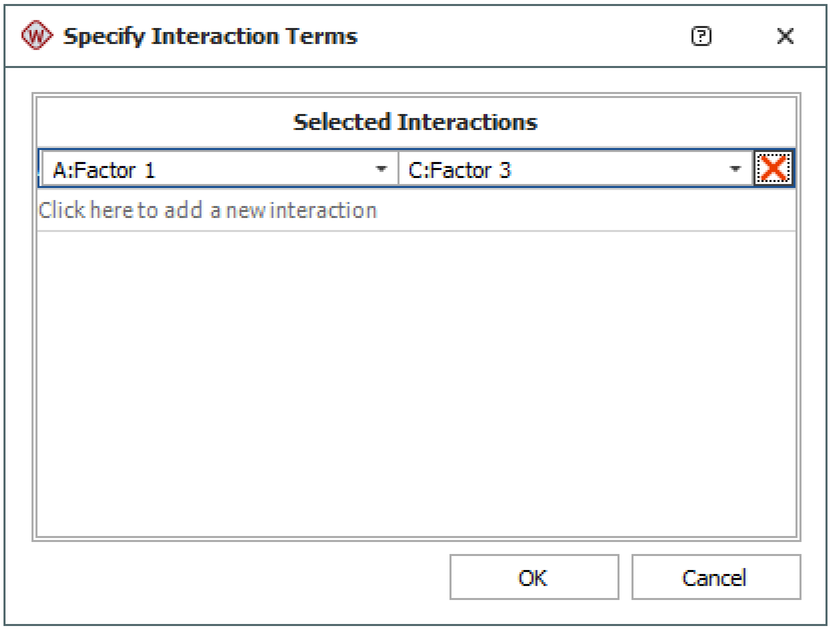
Specify Interaction Terms Window

The Specify Interaction Terms window helps you assign factors to columns in the orthogonal array in a way that will not alias main effects or any of the selected interaction terms with any other terms. In other words, it helps you configure a design that you can use to determine whether any of the main effects or selected interaction effects are significant.

The window can be accessed from the Design tab of the standard design folio while configuring a Taguchi OA factorial design. To open it, click the **Additional Settings** heading in the navigation panel and then click the **Specify Interaction Terms** link in the input panel.

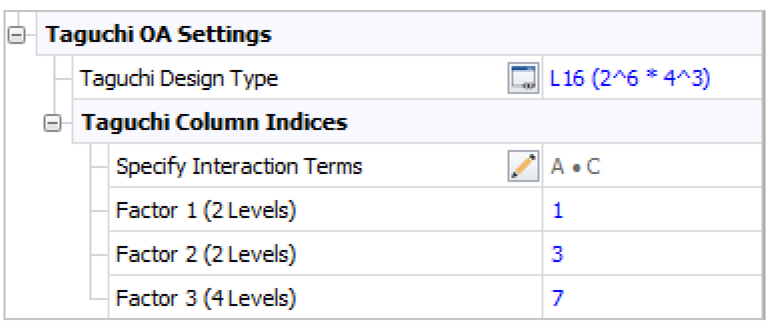


To enter an interaction effect that you wish to investigate, click the first row in the **Selected Interactions** table. Then specify the interaction using the drop-down lists in that row. For example, if you wish to include the interaction term AC, select factor A in the first drop-down list and factor C in the second, as shown next.



Once you click outside the row, the interaction term (e.g., "A·C") will appear. To remove the term, click inside its row and then click the red X.

After you click **OK**, the software will attempt to assign factors to the appropriate columns in the orthogonal array. In the example shown next, Factor 1 is now assigned to column 1, Factor 2 to column 3, and Factor 3 to column 7.



To ensure that you will be able to investigate the main effects and the interactions you entered, do not change the new factor assignments.

The ReliWiki resource portal has more information on assigning array columns to factors at: http://www.reliawiki.org/index.php/Highly_Fractional_Factorial_Designs.

Taguchi OA Designs: Example

*The data set used in this example is available in the example database installed with the software (called "Weibull21_DOE_Examples.rsgz21"). 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 example project is "Factorial - Taguchi OA Design Example."

Taguchi orthogonal array (OA) designs are often used in design experiments with factors that include more than two levels. Taguchi OA can be thought of as a general fractional factorial design.

Consider an experiment to study the effect of four three-level factors on a fine gold wire bonding process in an IC chip-package.Taguchi OA L27 (3^13) is applied to identify the critical parameters in the wire bonding process. The response is the ball size. The smaller the ball size, the better the process.

For this example, the four factors and their associated levels are:

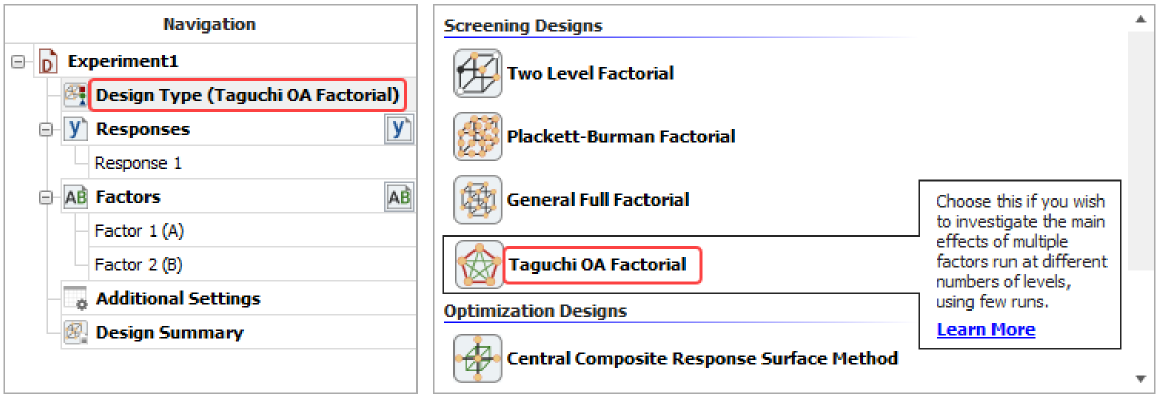
Factor	Level 1	Level 2	Level 3
Force	5	10	15
Power	40	50	60
Time	15	20	25
Temperature	155	160	165

DESIGN THE EXPERIMENT

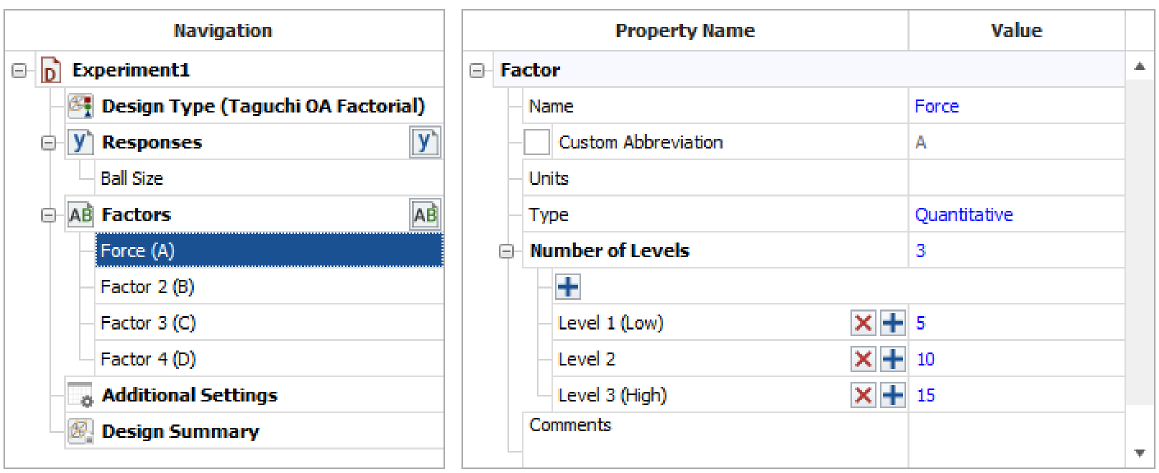
The experimenters create a Taguchi OA design folio, perform the experiment according to the design, and then enter the response values for further analysis. The design matrix and the response data are given in the "Taguchi OA L27(3^13)" folio. The following steps describe how to create this folio on your own.

- Choose **Home > Insert > Standard Design** to add a standard design folio to the current project.

- Click **Design Type** in the folio's navigation panel, and then select **Taguchi OA Factorial** in the input panel.

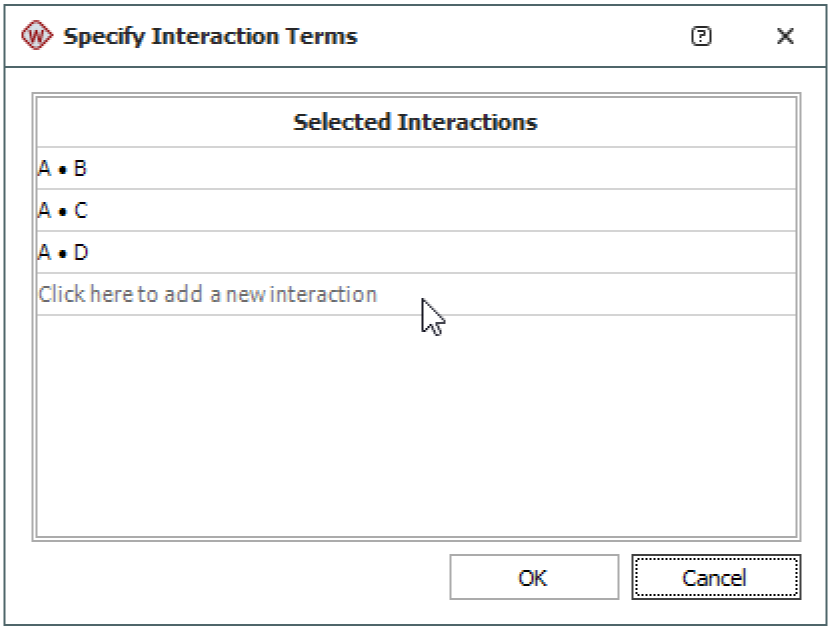


- Rename the response by clicking **Response 1** in the navigation panel and entering **Ball Size** for the **Name** in the input panel.
- Specify the number of factors by clicking the **Factors** heading in the navigation panel and choosing **4** from the **Number of Factors** drop-down list.
- Define each factor by clicking it in the navigation panel and editing its properties in the input panel. The properties are given in the table above. The first factor is defined as shown next. (Note that every factor must be configured to have three levels.)

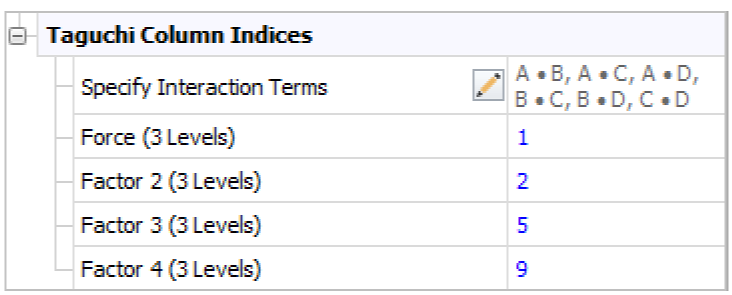


- Rename the folio by clicking the **Experiment1** heading in the navigation panel and entering **Taguchi OA L27 (3^13)** for the **Name** in the input panel.
- Click the **Additional Settings** heading to choose an orthogonal array and assign factors to columns in the array.
  - In the input panel, choose **L27 (3^13)** from the **Taguchi Design Type** drop-down list. This specifies the orthogonal array that will be used to generate the design.

- To have the software determine which columns in the array will be assigned to the factors, click the **Specify Interaction Terms** link. In the window that appears, enter all possible two-way interactions between the factors being investigated. The easiest way to do this is by repeatedly clicking the row that says "Click here to add a new interaction" until the row disappears.

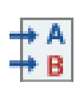


- Click **OK** to close the window. Force is now assigned to column 1 of the array, Power to 2, Time to 5 and Temperature to 9, as shown next.



These column assignments will be used to generate a design that does not alias any of the two-way interactions you entered in the Specify Interaction Terms window with any main effects.

- Finally, click the **Build** icon on the control panel to create a Data tab that allows you to view the test plan and enter response data.

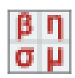


ANALYSIS AND RESULTS - PART 1

The data set for this example is given in the "Taguchi OA L27 (3^13)" folio of the example project. After you enter the data from the example folio, you can perform the analysis by doing the following:

Note: To minimize the effect of unknown nuisance factors, the run order is randomly generated when you create the design. Therefore, if you followed these steps to create your own folio, the order of runs on the Data tab may be different from that of the folio in the example file. This can lead to different results. To ensure that you get the very same results described next, show the Standard Order column in your folio, then click a cell in that column and choose **Sheet > Sheet Actions > Sort > Sort Ascending**. This will make the order of runs in your folio the same as that of the example file. Then copy the response data from the example file and paste it into the Data tab of your folio.

- On the Analysis Settings page of the control panel, select to use **Individual Terms** in the analysis.
- Return to the Main page of the control panel and click the **Calculate** icon.



Click the **View Analysis Summary** icon on the control panel and select to the view the ANOVA Table.

ANOVA Table					
Source of Variation	Degrees of Freedom	Sum of Squares [Partial]	Mean Squares [Partial]	F Ratio	P Value
Model	20	109.833333	5.491667	2.950746	0.091929
A:Force	2	22.166667	11.083333	5.955224	0.037595
B:Power	2	63.388889	31.694444	17.029851	0.00336
C:Time	2	18.666667	9.333333	5.014925	0.05244
D:Temperature	2	1.055556	0.527778	0.283582	0.762641
A • B	4	1.944444	0.486111	0.261194	0.892677
A • C	4	1.5	0.375	0.201493	0.928554
A • D	4	1.111111	0.277778	0.149254	0.956589
Residual	6	11.166667	1.861111		
Lack of Fit	6	11.166667	1.861111		
Total	26	121			

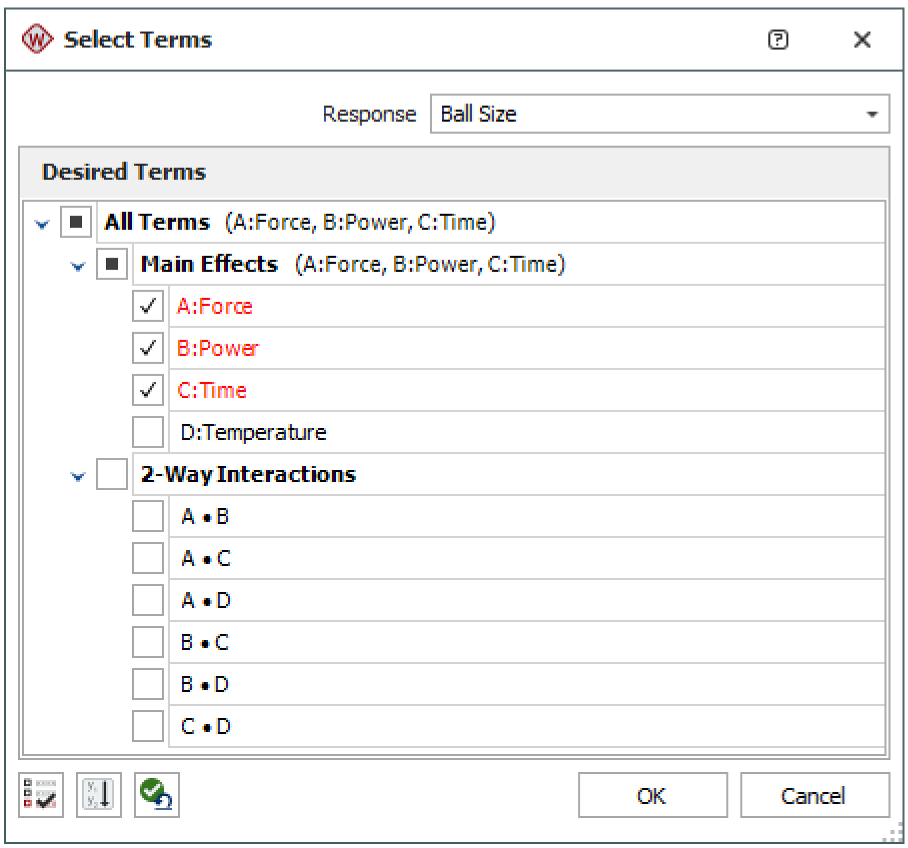
From this table, you can see that effects A, B, C are significant. To simply the analysis, the next step is to recreate the model using only the significant terms.

ANALYSIS AND RESULTS - PART 2

The results for the reduced model are given in the "Reduced Model" folio of the example project. The following steps describe how to create this folio on your own and then use the results to find the factor settings that provide the smallest deviation in height.

- Right-click the design folio in the current project explorer and select **Duplicate Item** from the shortcut menu. Rename the new folio with an appropriate name.
- Click the **Select Terms** icon on the new folio's control panel.

- In the Select Terms window that appears, click the **Select Significant Effects** button to select only the significant effects to calculate the new model, as shown next, then click **OK**.



- Click the **Calculate** icon in the new folio's control panel.
- Click the **View Analysis Summary** icon on the control panel. In the Analysis Summary window that appears, view the Diagnostic Information table. The table is shown next. In order to

identify which factor settings can provide the smallest ball size (and hence the best process), look for the runs in the table with the Fitted Value (YF) closest to zero (highlighted).

Diagnostics									
Standard Order	Run Order	Actual Value (Y)	Fitted Value (YF)	Residual	Standardized Residual	Studentized Residual	External Studentized Residual	Leverage	Cook's Distance
9	1	40	40.444444	-0.444444	-0.48525	-0.563809	-0.553953	0.259259	0.015894
11	2	38	37.833333	0.166667	0.181969	0.211428	0.206306	0.259259	0.002235
2	3	36	35.833333	0.166667	0.181969	0.211428	0.206306	0.259259	0.002235
5	4	38.5	39.055556	-0.555556	-0.606562	-0.704761	-0.695608	0.259259	0.024834
3	5	38	37.166667	0.833333	0.909843	1.057142	1.060428	0.259259	0.055877
10	6	36.5	37.166667	-0.666667	-0.727875	-0.845714	-0.839446	0.259259	0.035762
22	7	41	40.222222	0.777778	0.849187	0.986666	0.985979	0.259259	0.048675
25	8	41	40.277778	0.722222	0.788531	0.91619	0.912342	0.259259	0.04197
19	9	35	37	-2	-2.183624	-2.537141	-3.002928	0.259259	0.321854
20	10	37.5	37.666667	-0.166667	-0.181969	-0.211428	-0.206306	0.259259	0.002235
6	11	39.5	40.388889	-0.888889	-0.970499	-1.127618	-1.135763	0.259259	0.063576
14	12	41.5	41.055556	0.444444	0.48525	0.563809	0.553953	0.259259	0.015894
24	13	42.5	42.222222	0.277778	0.303281	0.352381	0.344529	0.259259	0.006209
21	14	40	39	1	1.091812	1.268571	1.289413	0.259259	0.080464
12	15	40	39.166667	0.833333	0.909843	1.057142	1.060428	0.259259	0.055877
16	16	40.5	40.444444	0.055556	0.060656	0.070476	0.0687	0.259259	0.000248
4	17	40	38.388889	1.611111	1.75903	2.043808	2.239622	0.259259	0.208858
23	18	39.5	40.888889	-1.388889	-1.516405	-1.761904	-1.868405	0.259259	0.155215
26	19	42	40.944444	1.055556	1.152468	1.339047	1.367899	0.259259	0.089652
18	20	41.5	42.444444	-0.944444	-1.031156	-1.198095	-1.212064	0.259259	0.071772
8	21	39	39.111111	-0.111111	-0.121312	-0.140952	-0.137452	0.259259	0.000993
7	22	38	38.444444	-0.444444	-0.48525	-0.563809	-0.553953	0.259259	0.015894
27	23	42	42.277778	-0.277778	-0.303281	-0.352381	-0.344529	0.259259	0.006209
15	24	42	42.388889	-0.388889	-0.424593	-0.493333	-0.483794	0.259259	0.012169
17	25	41.5	41.111111	0.388889	0.424593	0.493333	0.483794	0.259259	0.012169
1	26	35	35.166667	-0.166667	-0.181969	-0.211428	-0.206306	0.259259	0.002235
13	27	40.5	40.388889	0.111111	0.121312	0.140952	0.137452	0.259259	0.000993

CONCLUSIONS

You can see that the first run in the standard order (8th run in the run order) is at the best combination of settings. The settings for this particular run on the Data tab is shown to be A = 5, B = 40, C = 15. Under these settings, the expected ball size is lowest.

Note that the best factor settings in this case are limited to those settings actually used in the experiment. This limitation can be avoided using response surface methodology, which may allow you to find the optimal settings for the manufacturing process.

Response Surface Method Designs

In factorial design, only the linear effects of the quantitative factors are studied. Response surface methodology allows you to study the quadratic effects of the factors (i.e., effects that differ depending on the level of the factors), making it well-suited to predictive modeling and optimization. The following types of response surface method designs are available:

- **Central Composite**: A two level full or fractional factorial design is embedded in the central composite design, and additional center and axial points are also used in order to estimate curvature. This is the most commonly used response surface methodology design.
- **Box-Behnken**: Each factor must have three levels, with one level being the center point between the high and low levels. This design type is useful in cases where setting all factors at extreme values simultaneously is undesirable (e.g., if setting all factors at the high level carries a risk of equipment damage or otherwise violates constraints).

For more information about how to use the design types, please consult the documentation on design folios.

Central Composite Designs

In central composite designs, there are three kinds of points involved:

- **Factorial points**: A two level full or fractional factorial design, using up to nine factors, is embedded in the central composite design. The points associated with this embedded design are also called "corner points," and are indicated by a 1 in the Point Type column on the Data tab.
- **Axial points**: For each factor, two axial points are calculated as follows: all other factors are set to their coded 0 value (i.e., midway between the low and high — or coded -1 and 1 — values) and the factor in question is set to -alpha and +alpha, where alpha is a value specified by the user. Axial points, sometimes called "star" points, allow estimation of curvature. These points are indicated by a -1 in the Point Type column on the Data tab.
- **Center point**: All factors are set to the coded 0 value. These points are indicated by a 0 in the Point Type column on the Data tab.

Thus, for each factor, five values are used: -alpha, -1, 0, 1, +alpha.

Available Central Composite Designs

The available central composite designs are shown next. This information is applicable to designs with standard response data, as well as robust parameter designs and reliability DOE.

Factors		Full Factorial Designs	Half Fraction Designs	Quarter Fraction Designs
2	Corner Point Runs	4		
	Center Points (Default)	5		
3	Corner Point Runs	8		
	Center Points (Default)	6		
4	Corner Point Runs	16		
	Center Points (Default)	7		
5	Corner Point Runs	32	16	
	Center Points (Default)	10	6	
6	Corner Point Runs	64	32	
	Center Points (Default)	14	9	
7	Corner Point Runs	128	64	
	Center Points (Default)	10	10	
8	Corner Point Runs		128	64
	Center Points (Default)		10	10
9	Corner Point Runs			128
	Center Points (Default)			10

This table provides information about how many corner point (i.e., factorial) runs will be used in the design, based on the number of factors and the selected fractionality of the design, as well as how many center points will be used by default when a single block is used. For example, if you are using 5 factors and have selected a full factorial design, the design will use 32 corner point runs and will use 10 center points by default, provided that only a single block is used. If you are using 5 factors and have selected a half fraction design, the design will use 16 corner point runs and will use 6 center points by default, provided that only a single block is used.

Central Composite Designs: Example

*The data set used in this example is available in the example database installed with the software (called "Weibull21_DOE_Examples.rsgz21"). 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 example project is "Response Surface Method - Central Composite Design."

A chemical engineer is interested in determining the operating conditions that maximize the yield of a process.Two controllable variables influence process yield: reaction time and reaction temperature.

Name	Unit	Level -	Level +
Time	min	80	90
Temperature	F	170	180

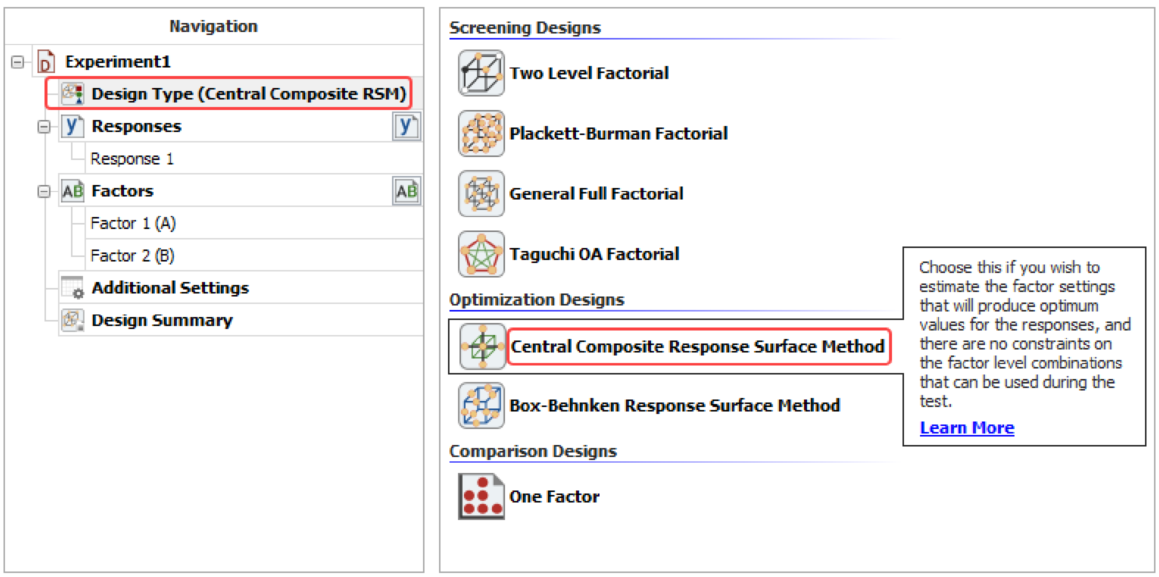
A central composite design with five center points and alpha = 1.414 is used to conduct the experiment. A full quadratic model is fitted to the data.

DESIGNING THE EXPERIMENT

The engineer creates a central composite design folio, performs the experiment according to the design, and then enters the response values into the folio for further analysis. The design matrix and the response data are given in the "Central Composite Design" folio. The following steps describe how to create this folio on your own.

- Choose **Home > Insert > Standard Design** to add a standard design folio to the current project.

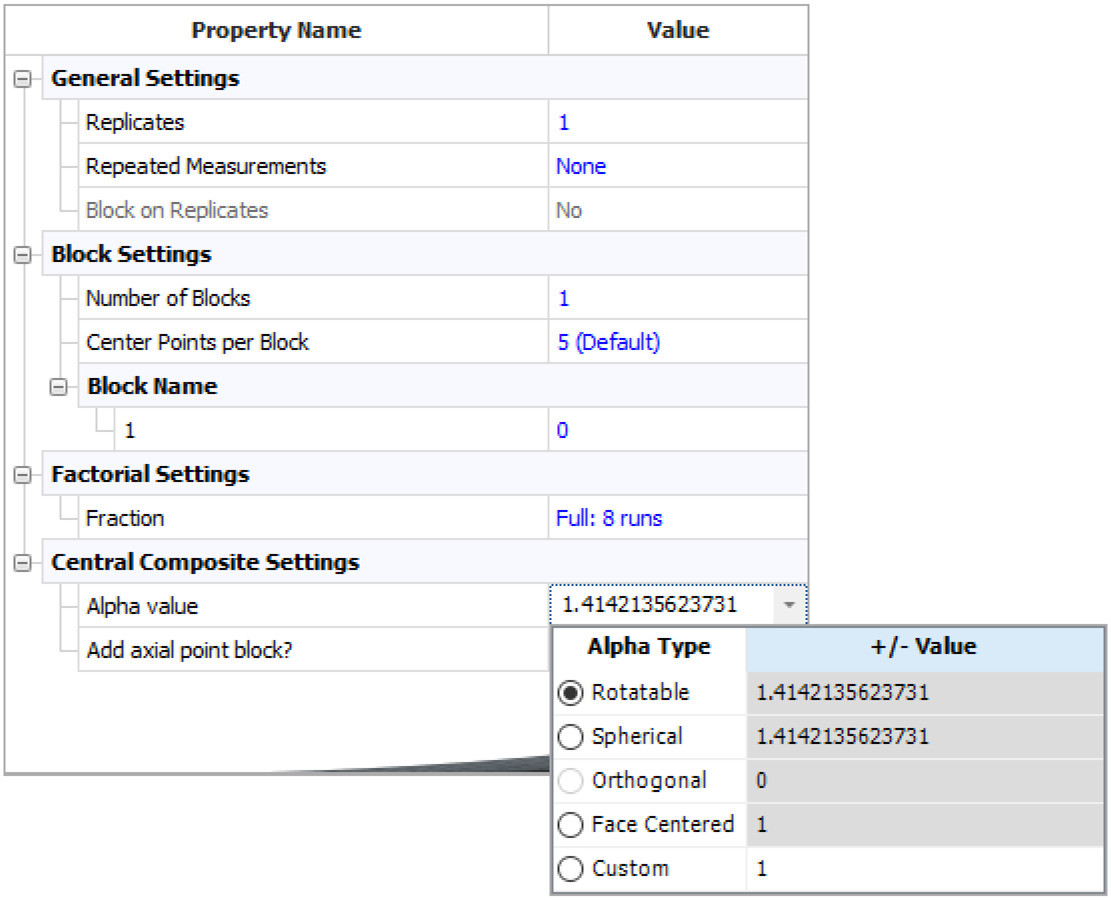
- Click **Design Type** in the folio's navigation panel, and then select **Central Composite Response Surface Method** in the input panel.



- Rename the folio by clicking the **Experiment1** heading in the navigation panel and entering **Central Composite Design** for the **Name** in the input panel.
- Rename the response by clicking **Response 1** in the navigation panel and entering **Yield** in the input panel.
- Specify the number of factors by clicking the **Factors** heading in the navigation panel and choosing **2** from the **Number of Factors** drop-down list in the input panel.
- Define each factor by clicking it in the navigation panel and editing its properties in the input panel.

- First factor:
  - Name: **Time**
  - Units: **min**
  - Factor Type: **Quantitative**
  - Level 1 (Low): **80**
  - Level 2 (High): **90**
- Second factor:
  - Name: **Temperature**
  - Units: **F**
  - Factor Type: **Quantitative**
  - Level 1 (Low): **170**
  - Level 2 (High): **180**

- Click the **Additional Settings** heading. In the input panel, set the following properties:



- Click the **Design Summary** heading to make sure that you have entered all settings correctly. The complete Design Summary is shown next.

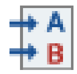
Central Composite Response Surface Method

Response Properties	
Name	**Units**
Yield	

Factor Properties								
Abbreviation	**Name**	**Units**	**Type**	**Number of Levels**	**Level 1 (Low)**	**Level 2 (High)**	**Low Alpha**	**High Alpha**
A	Time	min	Quantitative	2	80	90	77.92893219	92.07106781
B	Temperature	F	Quantitative	2	170	180	167.9289322	182.0710678

Additional Settings	
Fraction	Full
# of Unique Runs	8
# of Blocks	1
Include Axial Block	No
Alpha	1.414213562
Center Points per Block	5
Axial Center Points	0
# of Replicates	1
Block on Replicates	No
Total # of Runs	13

- Finally, click the **Build** icon on the control panel to create a Data tab that allows you to view the test plan and enter response data.



ANALYSIS AND RESULTS

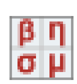
The data set for this example is given in the "Central Composite Design" folio of the example project. After you enter the data from the folio, you can specify the settings for the analysis by doing the following:

Note: To minimize the effect of unknown nuisance factors, the run order is randomly generated when you create the design. Therefore, if you followed these steps to create your own folio, the order of runs on the Data tab may be different from that of the folio in the example file. This can lead to different results. To ensure that you get the very same results described next, show the Standard Order column in your folio, then click a cell in that column and choose **Sheet > Sheet Actions > Sort > Sort Ascending**. This will make the order of runs in your folio the same as that of the example file. Then copy the response data from the example file and paste it into the Data tab of your folio.

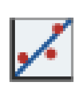
- Make sure all main effects and interactions will be considered. To do this, click the **Select Terms** icon on the control panel.

In the window that appears, select the **All Terms** check box then click **OK**.

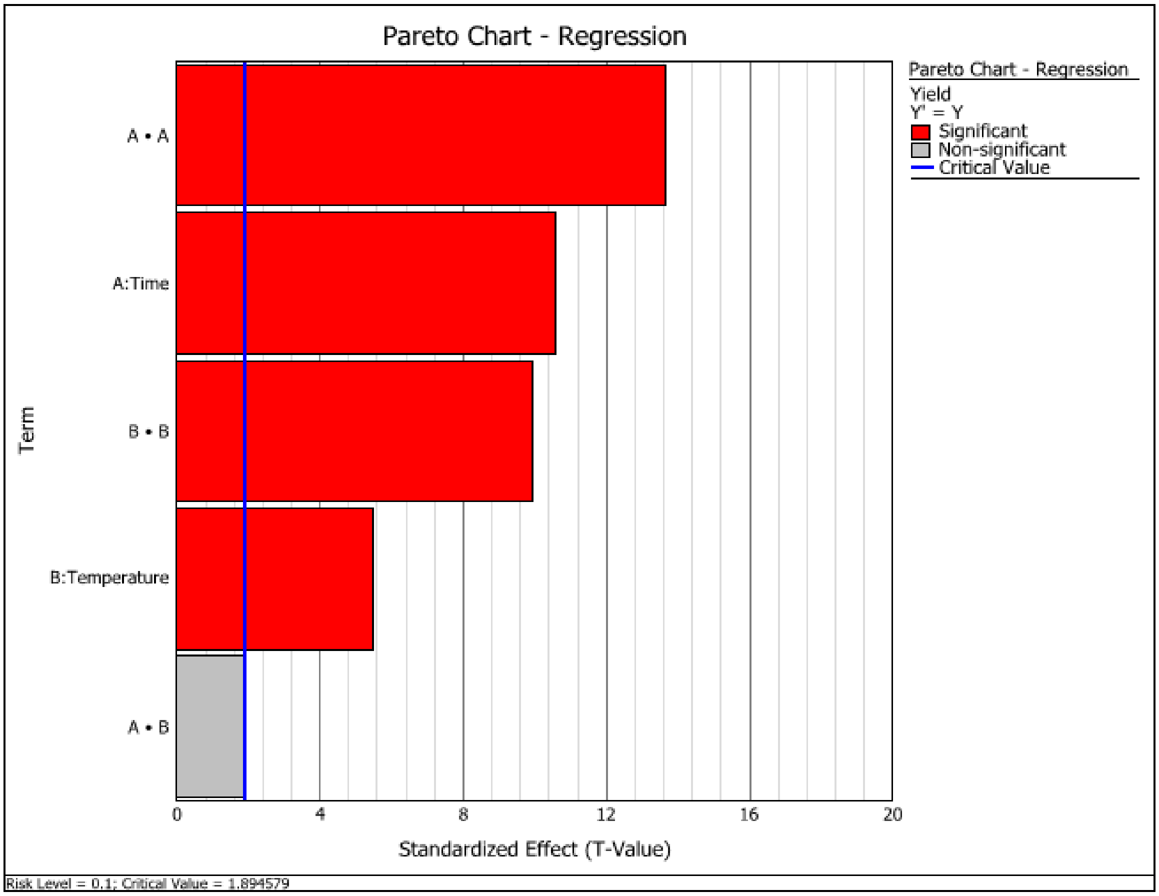
- On the Analysis Settings page of the Control Panel, select to use **Individual Terms** in the analysis.
- Click the **Calculate** icon.



- The results in the **Analysis Summary** area on the main effects A and B, as well as the quadratic effects AA and BB, are significant.
- To view a plot comparing the standardized effect of each term, click the **Plot** icon.



Then choose **Pareto Charts - Regression** from the **Plot Type** drop-down list. The following plot appears.



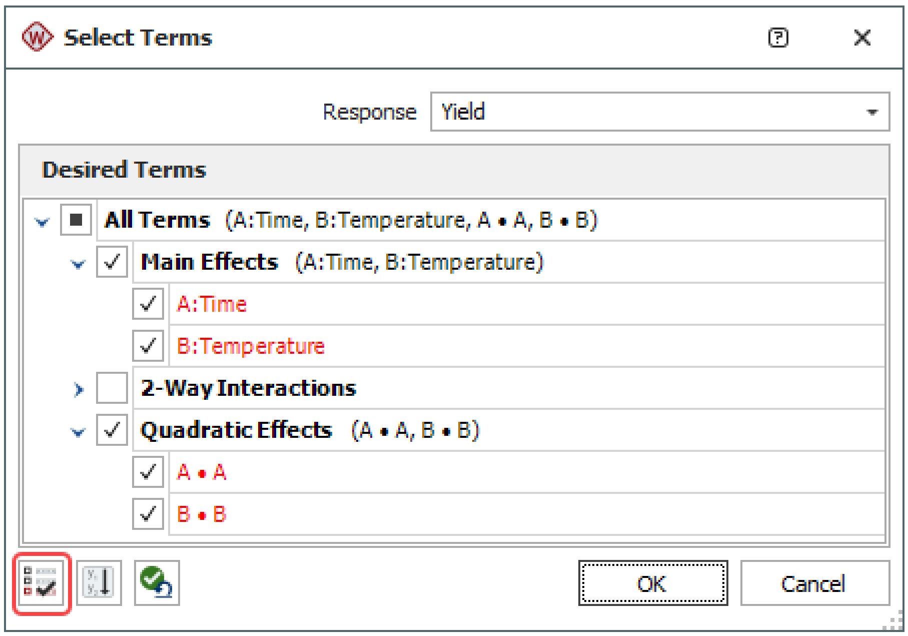
The vertical blue line in the plot marks the critical value determined by the risk level specified on the Analysis Settings page of the control panel. If the bar goes past the blue line, then the effect is considered significant.

From these results, effects A, B and AA and BB will be included in the reduced model. In fact, you could also include term AB in the model. From the Pareto chart, you can see that AB is only slightly below the critical value. The inclusion or exclusion of AB is a personal decision that should be made based on the knowledge of the experiment and the statistical results. For this example, only A, B, AA and BB will be included in the model.

OPTIMIZATION

The results for the reduced model (which only includes the terms that were found to be significant) are given in the "Reduced Model" folio. The following steps describe how to create this folio on your own.

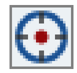
- Right-click the "Central Composite Design" folio in the current project explorer. Then choose **Duplicate** from the shortcut menu.
- Right-click the new, duplicated folio in the current project explorer and choose **Rename** from the shortcut menu. Change the name to "Reduced Model."
- Open the "Reduced Model" folio. Then click the **Select Terms** icon on the control panel.
- In the Select Terms window that appears, click the **Select Significant Effects** icon to select only the significant effects to calculate the new model, as shown next, then click **OK**.



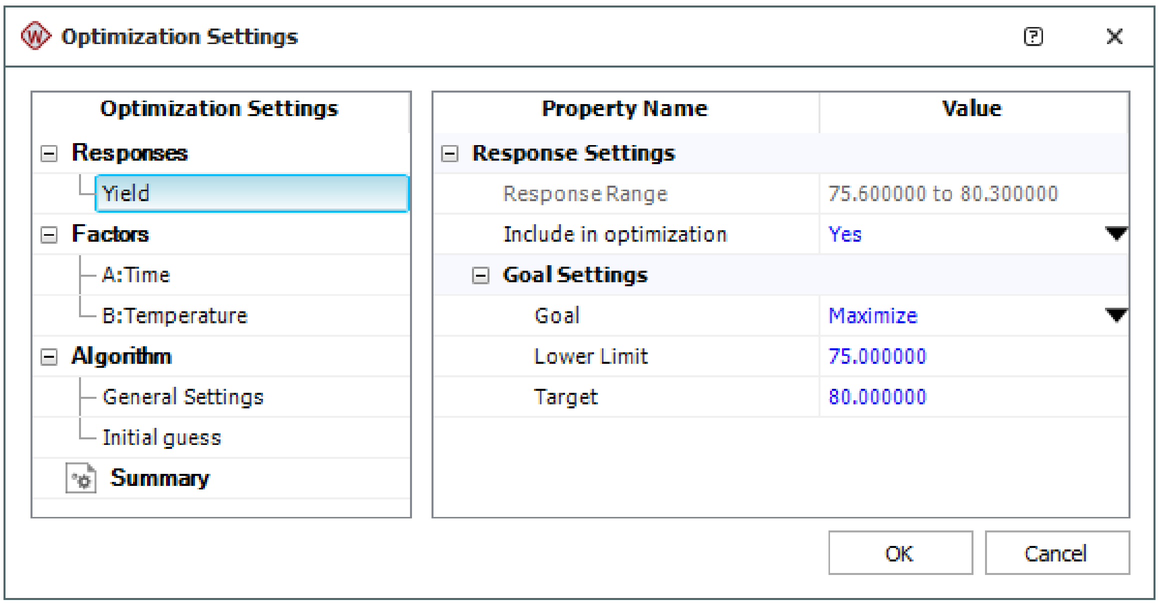
- Click the **Calculate** icon.
- In the **Analysis Summary** area on the control panel, click the **View Analysis Summary** icon to view detailed results from the analysis.
- In the Analysis Summary window, select the **Regression Table** check box on the **Available Report Items** panel. The coefficients for the model will appear in the Regression Table as shown next.

Regression Information						
Term	**Coefficient**	**Standard Error**	**Low Confidence**	**High Confidence**	**T Value**	**P Value**
Intercept	79.94	0.1365	79.686171	80.193829	585.640297	0
A:Time	0.994975	0.107913	0.794306	1.195644	9.220169	0.000016
B:Temperature	0.515165	0.107913	0.314496	0.715834	4.773899	0.001402
A • A	-1.37625	0.115724	-1.591444	-1.161056	-11.892556	0.000002
B • B	-1.00125	0.115724	-1.216444	-0.786056	-8.652078	0.000025

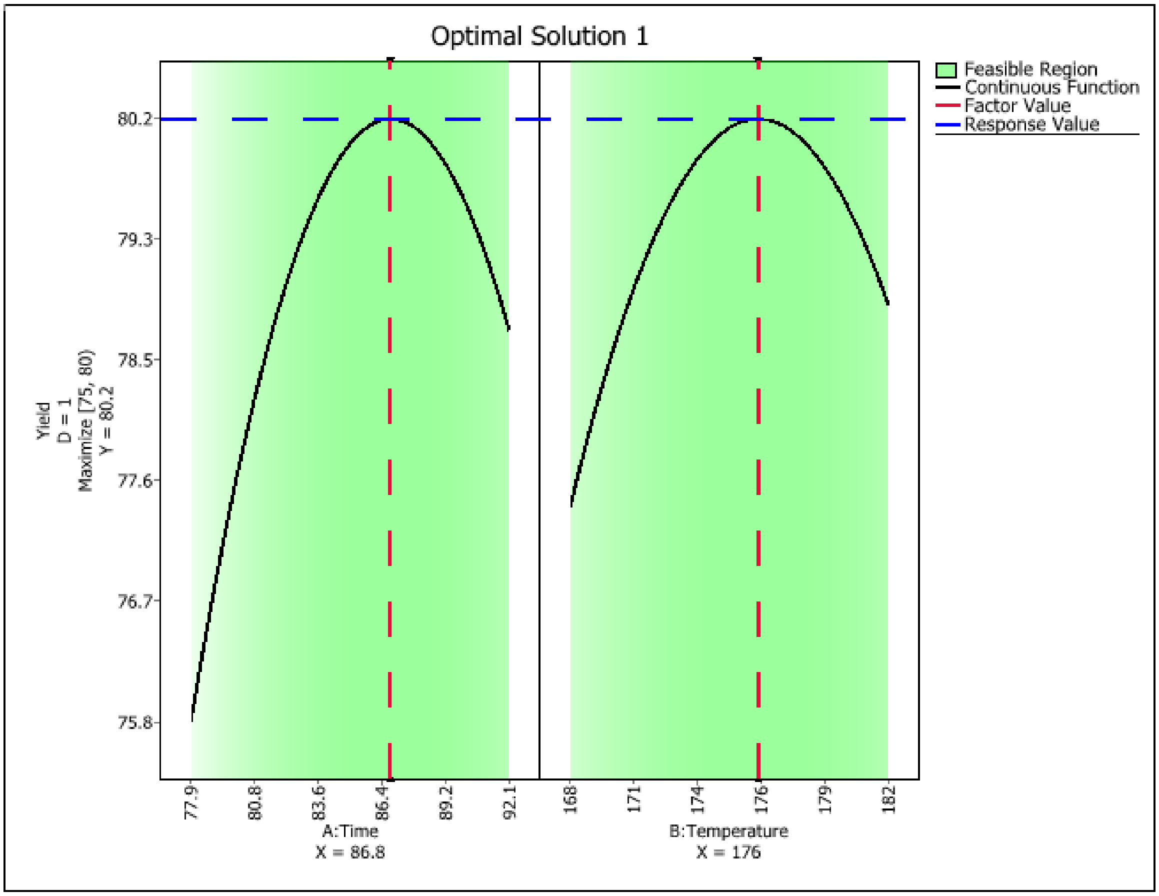
- Click the **Design - Optimization** icon on the control panel.



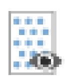
- In the Response Settings window that appears, use the settings shown next and click **OK**. These settings indicate that you want to maximize the Yield response, with values above 80 being considered 100% desirable and values below 75 being undesirable.



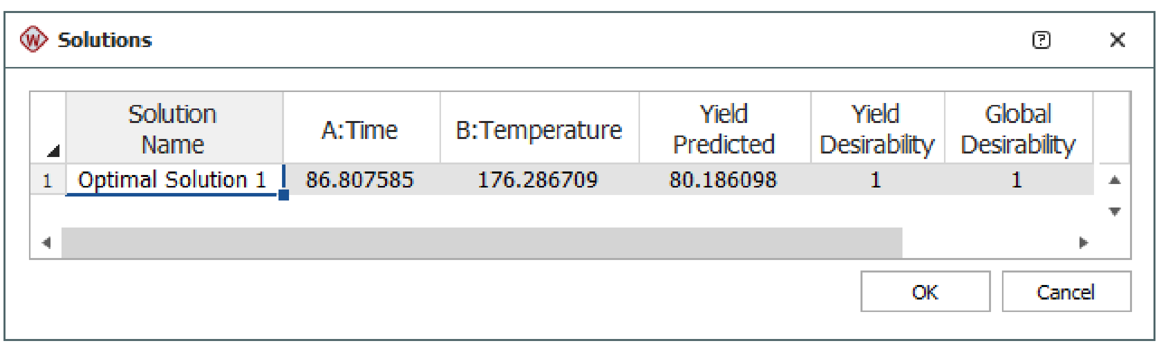
- An Optimization folio will appear with a plot showing the single solution that is found.



- You can choose **Optimization > Solutions > View Solutions** or click the icon on the control panel to see the solution in numerical format.



The optimum settings for factors A and B are shown next.



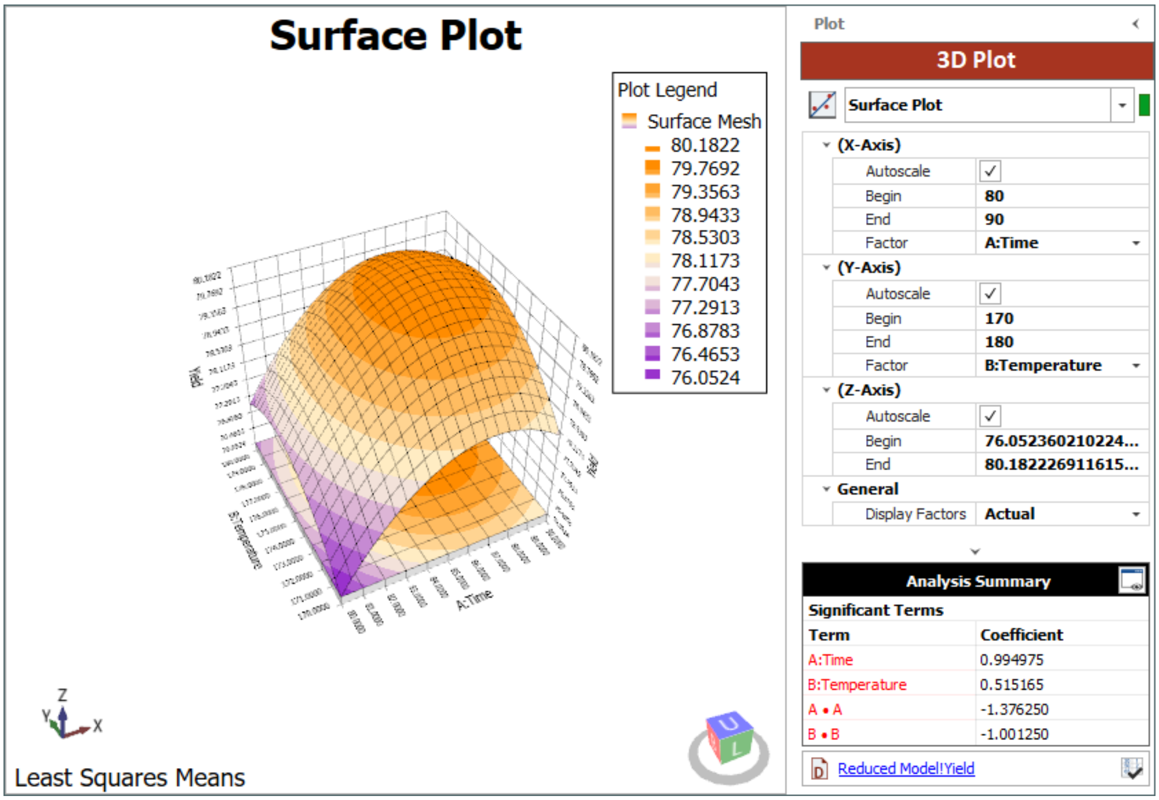
Solutions

	Solution Name	A:Time	B:Temperature	Yield Predicted	Yield Desirability	Global Desirability
1	Optimal Solution 1	86.807585	176.286709	80.186098	1	1

OK Cancel

- You also can use the surface plot to visually identify the optimal settings for factors A and B. Return to the Analysis Plot tab of the "Reduced Model" folio and click the **3D Plot** icon in the control panel.

The 3D Plot folio will open and display the surface plot, as shown next.



CONCLUSIONS

From the surface plot, you can see that the maximum yield occurs at Time = 86.8 and Temperature = 176.3° F, which is the same as the result from the optimization. The predicted maximum yield is 80.1861. Keep in mind that it is necessary to conduct an experiment using these settings to confirm this conclusion.

Box-Behnken Designs

Box-Behnken designs allow you to consider from three to nine quantitative factors at three levels each (one level being the center point between the high and low levels, which are the only levels you define when you are creating the design). The designs are spherical and rotatable or nearly rotatable. If setting all factors at the high level at the same time carries a risk of equipment damage or otherwise violates constraints, these designs provide a matrix that avoids setting all factors at extreme values simultaneously.

Available Box-Behnken Designs

The available Box-Behnken designs are shown next. This information is applicable to designs with standard response data, as well as robust parameter designs and reliability DOE.

Factors		
3	Non-center Point Runs	12
	Center Points (Default)	3
4	Non-center Point Runs	24
	Center Points (Default)	3
5	Non-center Point Runs	40
	Center Points (Default)	6
6	Non-center Point Runs	48
	Center Points (Default)	6
7	Non-center Point Runs	56
	Center Points (Default)	6
8	Non-center Point Runs	112
	Center Points (Default)	8
9	Non-center Point Runs	120
	Center Points (Default)	10

This table provides information about how many non-center point runs will be used in the design, based on the number of factors, as well as how many center points will be used by default. For example, if you are using 5 factors, the design will use 40 non-center point runs and will use 6 center points by default.

Box-Behnken Designs: Example

*The data set used in this example is available in the example database installed with the software (called "Weibull21_DOE_Examples.rsgz21"). 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 example project is "Response Surface Method - Box-Behnken Design."

Consider a UV-light system that is used to inactivate fungal spores of Aspergillus niger in corn meal.* Fungal contamination of grains during the post-harvest period has been a recurring health hazard.

The response is the $\log_{10}$ reduction of the fungal spores. Therefore, the goal is to maximize the reduction (response).

Three process parameters in the UV-light system will affect the inactivation results. They are: A) treatment time (number of pulses), B) the distance from the UV strobe and C) input voltage for the UV lamp.

Factor	Name	Unit	Level -	Level +
A	Time	s	20	100
B	Distance	cm	3	13
C	Voltage	v	2000	3800

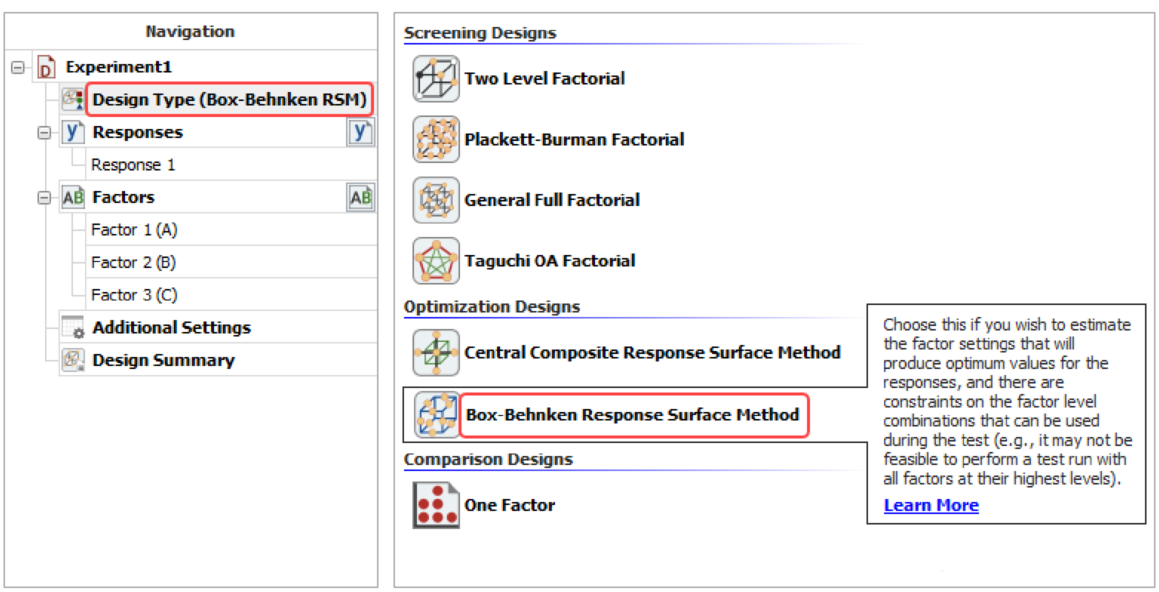
A 15 run Box-Behnken design with three center points is conducted. A full quadratic model is fitted to the data. Using this model, the optimal setting that gives the largest reduction of fungal spores is found.

DESIGNING THE EXPERIMENT

The experimenters create a Box-Behnken design folio, perform the experiment according to the design, and then enter the response values in the folio for further analysis. The design matrix and the data are given in the "UV-light Treatment" folio. The following steps describe how to create this folio on your own.

- Choose **Home > Insert > Standard Design** to add a standard design folio to the current project.

- Click **Design Type** in the folio's navigation panel, and then select **Box-Behnken Response Surface Method** in the input panel.



- Rename the folio by clicking the **Experiment1** heading in the navigation panel and entering **UV-light Treatment** for the **Name** in the input panel.
- Specify the number of factors by clicking the **Factors** heading in the navigation panel and choosing **3** from the **Number of Factors** drop-down list.
- Define each factor by clicking it in the navigation panel and editing its properties in the input panel.
  - First factor:
    - Name: **Time**
    - Units: **sec**
    - Factor Type: **Quantitative**
    - Level 1 (Low): **20**
    - Level 2 (High): **100**
  - Second factor:
    - Name: **Distance**
    - Units: **cm**
    - Factor Type: **Quantitative**
    - Level 1 (Low): **3**
    - Level 2 (High): **13**
  - Third factor:
    - Name: **Voltage**
    - Units: **V**
    - Factor Type: **Quantitative**
    - Level 1 (Low): **2,000**
    - Level 2 (High): **3,800**
- Click the **Additional Settings** heading. In the input panel, set the following properties:

Property Name	Value
General Settings	
Replicates	1
Repeated Measurements	None
Block on Replicates	No
Block Settings	
Number of Blocks	1
Center Points per Block	3 (Default)
Block Name	
1	1

- Click the **Design Summary** heading to make sure that you have entered all settings correctly. The complete Design Summary is shown next.

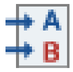
Box-Behnken Response Surface Method

Response Properties	
Name	**Units**
Response 1	

Factor Properties						
Abbreviation	**Name**	**Units**	**Type**	**Number of Levels**	**Level 1 (Low)**	**Level 2 (High)**
A	Time	s	Quantitative	2	20	100
B	Distance	cm	Quantitative	2	3	13
C	Voltage	v	Quantitative	2	2000	3800

Additional Settings	
# of Unique Runs	12
# of Blocks	1
Center Points per Block	3
# of Replicates	1
Block on Replicates	No
Total # of Runs	15

- Finally, click the **Build** icon on the control panel to create a Data tab that allows you to view the test plan and enter response data.



ANALYSIS AND RESULTS

The data set for this example is given in the "UV-light Treatment" folio of the example project. After you enter the data from the folio, you can specify the settings for the analysis by doing the following:

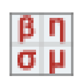
Note: To minimize the effect of unknown nuisance factors, the run order is randomly generated when you create the design. Therefore, if you followed these steps to create your own folio, the order of runs on the Data tab may be different from that of the folio in the example file. This can lead to different results. To ensure that you get the very same results described next, show the Standard Order column in your folio, then click a cell in that column and choose **Sheet > Sheet Actions > Sort > Sort Ascending**. This will make the order of runs in your folio the same as that of the example file. Then copy the response data from the example file and paste it into the Data tab of your folio.

- Make sure all main effects and interactions will be considered. To do this, click the **Select Terms** icon on the control panel.

  In the window that appears, select the **All Terms** check box. Then click **OK**.

- On the Analysis Settings page of the Control Panel, select to use **Individual Terms** in the analysis.
- Click the **Calculate** icon.

The results in the **Analysis Summary** area on the control panel show that time (A) and voltage (C) have significant effects on the reduction of the fungal spores, as well as AA (the quadratic effect of A) and the interaction of AC.

- In the **Analysis Summary** area on the control panel, click the **View Analysis Summary** icon to view detailed results from the analysis.

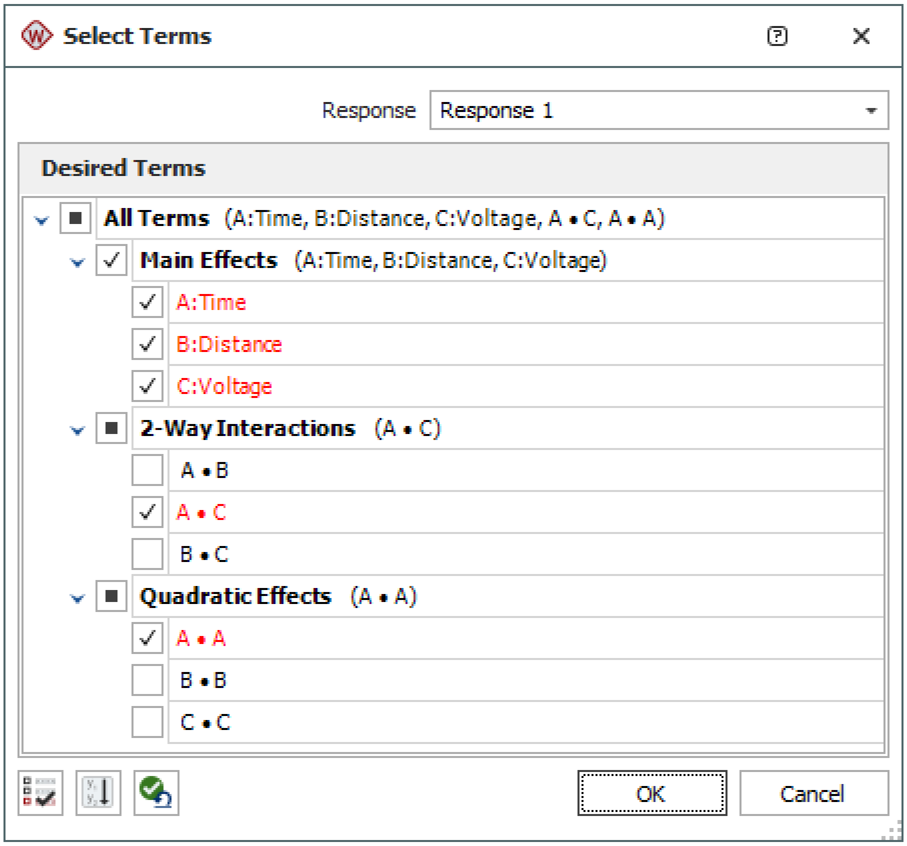
  In the ANOVA table, you can see that effects A, C, AC and AA are significant. The *p* value for factor B is 0.1481, which is close to the risk level of 0.1. Therefore, you decide to include it in the final model.

ANOVA Table					
Source of Variation	Degrees of Freedom	Sum of Squares [Partial]	Mean Squares [Partial]	F Ratio	P Value
Model	9	24.549748	2.72775	92.338855	0.000051
A:Time	1	15.29598	15.29598	517.794311	0.000003
B:Distance	1	0.08632	0.08632	2.922079	0.14807
C:Voltage	1	5.771503	5.771503	195.374954	0.000034
A • B	1	0.007482	0.007482	0.253287	0.636152
A • C	1	2.67486	2.67486	90.548456	0.000217
B • C	1	0.002116	0.002116	0.07163	0.799667
A • A	1	0.706731	0.706731	23.924009	0.004509
B • B	1	0.000144	0.000144	0.004882	0.947002
C • C	1	0.000283	0.000283	0.00957	0.925872
Residual	5	0.147703	0.029541		
Lack of Fit	3	0.147703	0.049234	-	-
Pure Error	2	0	0		
Total	14	24.697451			

OPTIMIZATION

The results for the reduced model (which only includes the terms that were found to be significant) are given in the "Reduced Model" folio. The following steps describe how to create this folio on your own.

- Right-click the "UV-light Treatment" folio in the current project explorer. Then choose **Duplicate** from the shortcut menu.
- Right-click the new, duplicated folio in the current project explorer and choose **Rename** from the shortcut menu. Change the name to "Reduced Model."
- Open the "Reduced Model" folio. Then click the **Select Terms** icon on the control panel.
- In the Select Terms window that appears, click the **Select Significant Effects** icon to select only the significant effects to calculate the new model, and also select the check box for term B, as shown next, then click **OK**.



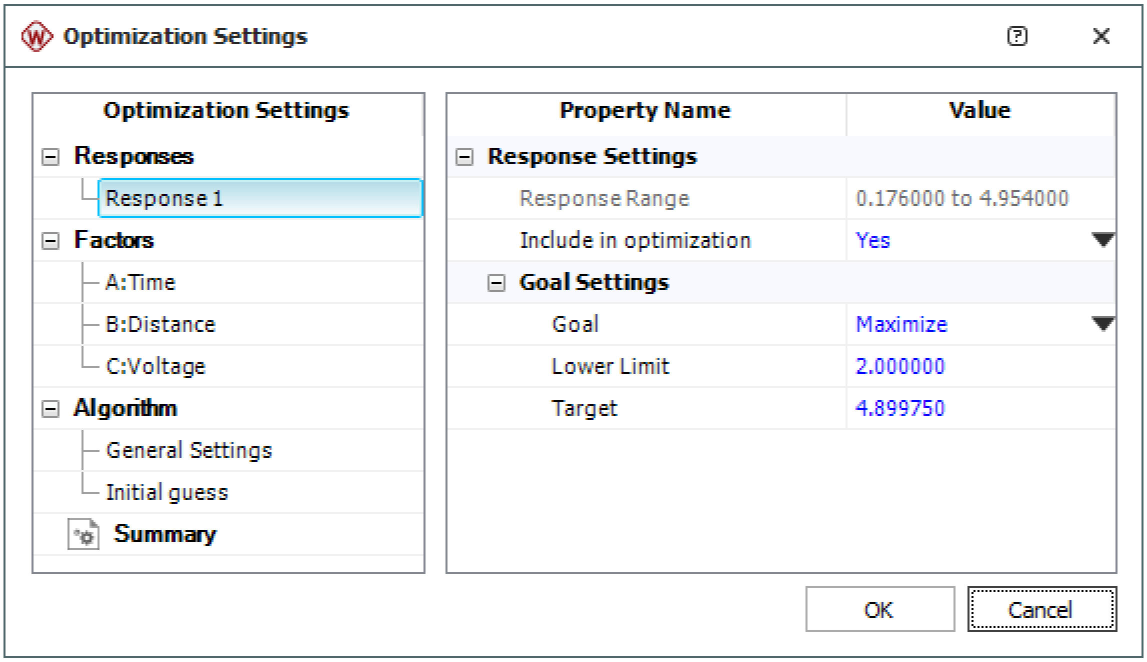
- Click the **Calculate** icon in the new folio's control panel.
- In the **Analysis Summary** area on the control panel, click the **View Analysis Summary** icon to view detailed results from the analysis.

The coefficients for the model will appear in the Regression Table, as shown next.

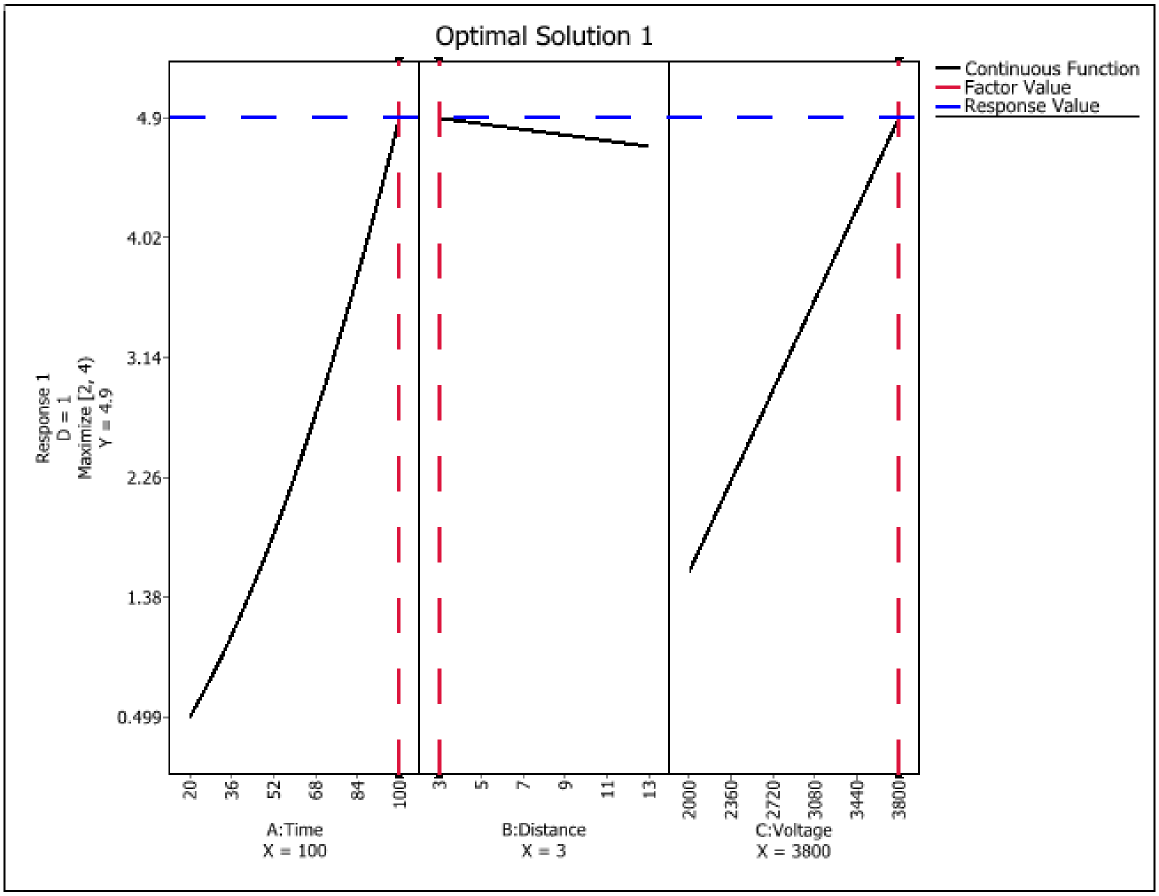
Regression Information						
Term	Coefficient	Standard Error	Low Confidence	High Confidence	T Value	P Value
Intercept	1.309571	0.050032	1.217858	1.401285	26.174839	8.370006E-10
A:Time	1.38275	0.0468	1.29696	1.46854	29.545712	2.845745E-10
B:Distance	-0.103875	0.0468	-0.189665	-0.018085	-2.219534	0.053605
C:Voltage	0.849375	0.0468	0.763585	0.935165	18.148898	2.134595E-8
A • C	0.81775	0.066186	0.696424	0.939076	12.355387	6.002732E-7
A • A	0.436429	0.068509	0.310844	0.562013	6.37041	0.00013

- Click the **Design - Optimization** icon on the control panel.

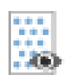
- In the Response Settings window that appears, use the settings shown next and click **OK**. These settings indicate that you want to maximize Response 1, with values above 4.89975 being considered 100% desirable and values below 2 being undesirable.



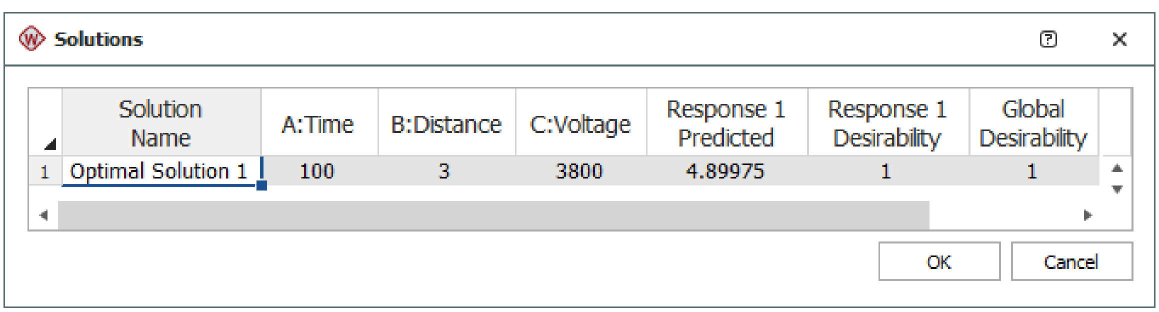
- An Optimization folio will appear with a plot showing the single solution that is found.



- You can choose **Optimization > Solutions > View Solutions** or click the icon on the control panel to see the solution in numerical format.



The optimum settings for factors A and B are shown next.



Solutions

	Solution Name	A:Time	B:Distance	C:Voltage	Response 1 Predicted	Response 1 Desirability	Global Desirability
1	Optimal Solution 1	100	3	3800	4.89975	1	1

OK Cancel

CONCLUSIONS

The optimal solution is found to be A = 100 s, B = 3 cm and C = 3800 v. Under this setting, the expected logarithmic transformation of the reduction is 4.9. Keep in mind that it is necessary to conduct an experiment using this setting to confirm this conclusion.

* S. Jun, J. Irudayaraj, A. Demirci and D. Geiser, "Pulsed UV-light treatment of corn meal for inactivation of Aspergillus niger spores," *International Journal of Food Science and Technology*, 2003, 38, 883-888.

One Factor Designs

One factor designs, also called "one factor at a time" (OFAT) or one-way analysis of variance (ANOVA), are used to determine if a particular factor has an effect on an output or response. This approach allows you to take a detailed look at the effect of the factor (using up to 255 levels), and it helps you determine whether a change in output is due to a change in the input (level) instead of random error.

The factor in a one factor design must be configured as a qualitative factor. For this reason, predictions cannot be made for factor levels that are not tested, nor can one factor designs be used for optimization.

Note: A one factor design is appropriate for planning an experiment involving a single factor. If you already have existing data that you want to examine, the one-way ANOVA folio offers this capability.

For more information about how to use the design types, please consult the documentation on design folios.

One Factor Designs: Example

*The data set used in this example is available in the example database installed with the software (called "Weibull21_DOE_Examples.rsgz21"). 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 example project is "One Factor - Operators in Pulp Mill."

Consider the following experiment, which was performed at a pulp mill. Plant performance is based on pulp brightness, as measured by a reflectance meter. Each of the four shift operators (denoted by A, B, C and D) made five pulp handsheets from unbleached pulp. Reflectance was read for each of the handsheets using a brightness tester. The data set is shown next.

Operator A	Operator B	Operator C	Operator D
59.88	59.87	60.83	61.01
60.12	60.32	60.87	60.87
60.88	60.42	60.56	60.69
60.98	59.99	61	60.53
59.9	60.12	60.5	60.63

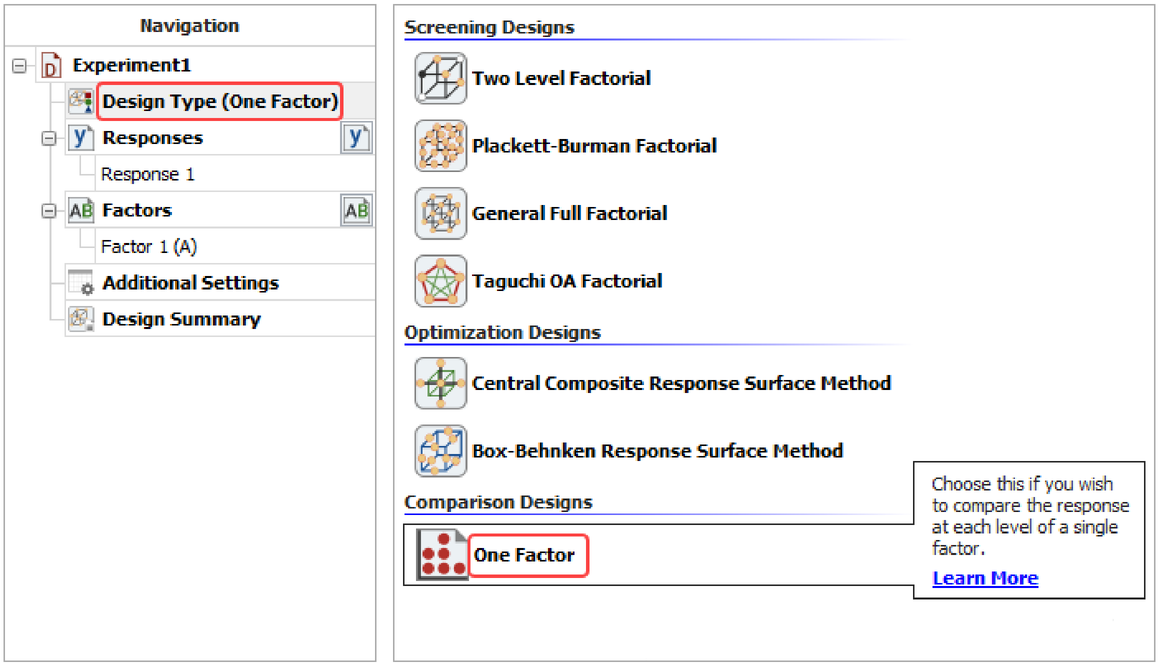
A goal of the experiment is to determine whether there are significant differences between the operators in making the handsheets and reading their brightness. If there are such differences, then the operators may not be creating the handsheets or recording their observations in a consistent manner.

Designing the Experiment

To determine whether there are differences between the operators, the experimenters create a one factor design folio and enter the response values into the folio for analysis. The design matrix and the response data are given in the "Operator Study" folio. The following steps describe how to create this folio on your own.

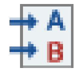
- Choose **Home > Insert > Standard Design** to add a standard design folio to the current project.

- Click **Design Type** in the folio's navigation panel, and then select **One Factor** in the input panel.



- Rename the response by clicking **Response 1** in the navigation panel and entering **Reflectance** in the input panel.
- Define the factor by clicking **Factor 1** in the navigation panel and editing its properties in the input panel.
  - Factor name: **Operator**
  - Factor type: **Qualitative**
  - Number of levels: **4**
  - Level 1: **Operator A**
  - Level 2: **Operator B**
  - Level 3: **Operator C**
  - Level 4: **Operator D**
- Rename the folio by clicking the **Experiment1** heading in the navigation panel and entering **Operator Study** for the **Name** in the input panel.
- Click the **Additional Settings** heading. In the input panel, set the number of **Replicates** to **5** (since there are 5 measurements from each operator).

- Finally, click the **Build** icon on the control panel to create a Data tab that allows you to view the test plan and enter response data.

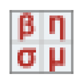


Analysis and Results

The data set for this example is given in the "Operator Study" folio of the example project. After you enter the data from the example folio, you can perform the analysis by doing the following:

Note: To minimize the effect of unknown nuisance factors, the run order is randomly generated when you create the design. Therefore, if you followed these steps to create your own folio, the order of runs on the Data tab may be different from that of the folio in the example file. This can lead to different results. To ensure that you get the very same results described next, show the Standard Order column in your folio, then click a cell in that column and choose **Sheet > Sheet Actions > Sort > Sort Ascending**. This will make the order of runs in your folio the same as that of the example file. Then copy the response data from the example file and paste it into the Data tab of your folio.

- Click the **Calculate** icon on the Data tab control panel.



- The result in the **Analysis Summary** area show that the *p* value is 0.0210. In other words, there is a 2.1% chance that the variation would be observed in the case when there is no significant difference between the response levels. Since the *p* value is less than the risk level (10%) entered on the Analysis Settings tab of the control panel, the conclusion is that the different factor levels (i.e., operators) do affect the response.

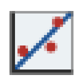
Analysis Summary	
Treatment Results	
P-Value	0.020968
The different levels affect the response	

- To see more detailed comparisons of the responses at different factor levels, click the **View Analysis Summary** icon and view the Mean Comparisons table.

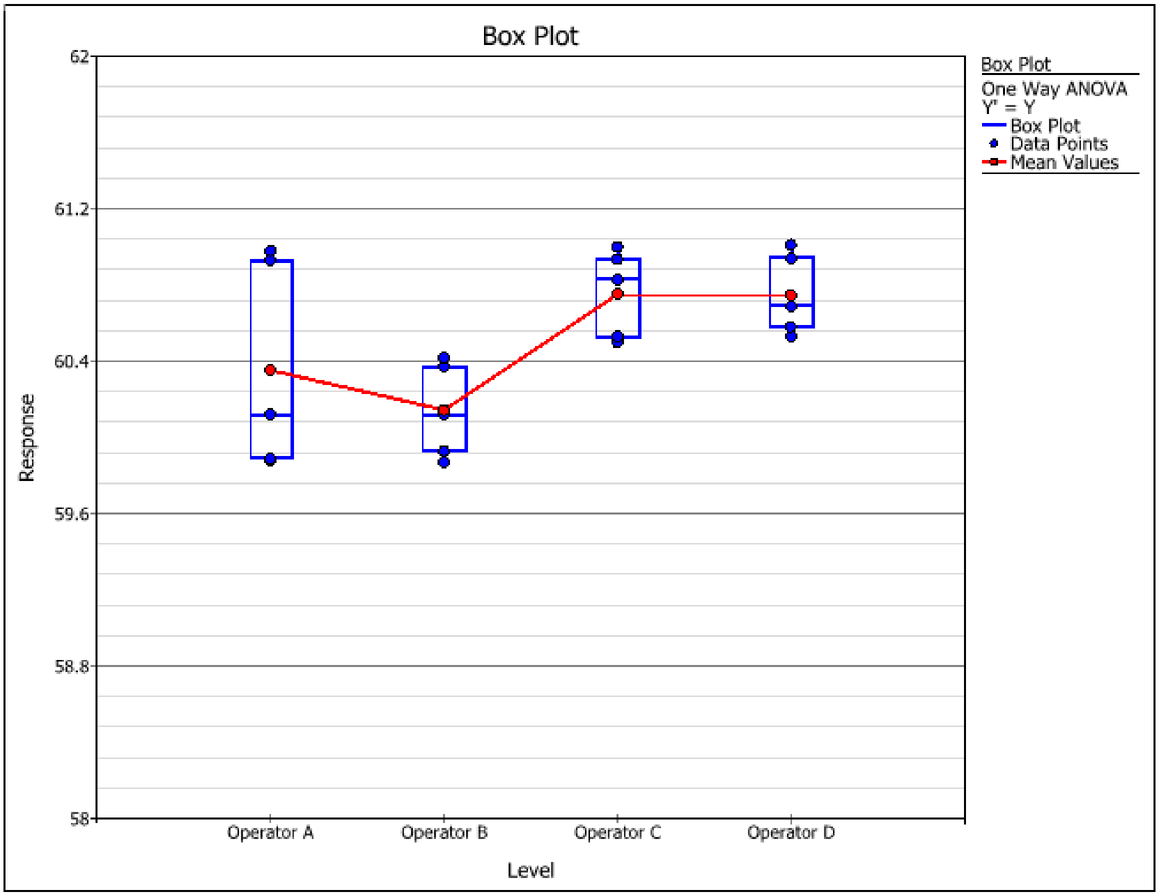
Mean Comparisons						
Contrast	Mean Difference	Pooled Standard Error	Low Confidence	High Confidence	T Value	P Value
Operator A - Operator B	0.208	0.205597	-0.150948	0.566948	1.011689	0.326742
Operator A - Operator C	-0.4	0.205597	-0.758948	-0.041052	-1.94556	0.069494
Operator A - Operator D	-0.394	0.205597	-0.752948	-0.035052	-1.91637	0.073358
Operator B - Operator C	-0.608	0.205597	-0.966948	-0.249052	-2.95725	0.00927
Operator B - Operator D	-0.602	0.205597	-0.960948	-0.243052	-2.92806	0.00985
Operator C - Operator D	0.006	0.205597	-0.352948	0.364948	0.029183	0.977079

The compared factor levels are listed in the Contrast column (e.g., "Operator A - Operator B" is the comparison of those two operators). When a *p* value is red, the compared levels are significantly different. For example, there is a significant difference between Operator A and Operator C, but there is no significant difference between Operator A and Operator B.

- To see a plot that also allows you to compare the different factor levels, click the **Plot** icon.



Then view the box plot. In this plot, you can see that see that operators C and D are nearly the same.



Conclusions

The results show that there is a significant difference between operators A and C, between A and D, and so forth. As a result, the operators receive further training to insure that they create the hand-sheets and record their observations consistently.

Robust Parameter Designs

Robust parameter designs are based on the work of Genichi Taguchi. The purpose of robust designs is to find the settings of controllable factors that will minimize variation in the response due to uncontrollable noise factors. This is done by combining an inner array of control factors with an outer array of noise factors. All control factor setting combinations specified in the inner array are tested at each noise factor settings combination specified in the outer array.

Available Robust Parameter Designs

The available designs for the inner and outer arrays of a robust parameter design depend on the design type selected for each array.

For the inner array, the following design types may be used. Click a design type to view the available designs.

- Two level factorial designs
- Plackett-Burman designs
- General full factorial designs
- Taguchi OA designs

Note that most Taguchi orthogonal arrays are saturated and therefore not effective for examining factorial interactions. If interactions of the control factors are likely, using a two level factorial design for the inner array may be desirable.

For the outer array, all of the above design types are available except Taguchi OA designs. When configuring the outer array, note the following:

- You can use a two level factorial design, a Placket-Burman design or a general full factorial design.
- Each of these design types can have 1 to 15 factors when used for the outer array.
- In addition to the settings described for configuring the inner array, you can also specify a number of replicates in the Additional Settings for the outer array.
- You cannot define blocks, center points, responses or repeated measurements for the outer array, regardless of the design type.

For more information about how to use the design types, please consult the documentation on design folios.

Using Robust Parameter Designs

The Data tab for robust parameter designs works differently from that for other design types.

In each cell in the noise condition columns, enter the response value obtained at the control factor settings specified in the current row and at the current noise condition. The noise factor settings represented by each noise condition column are available in the Noise Condition Matrix in the Design Summary on the Design tab. For example, consider the Noise Condition Matrix and Data tab shown next.

Noise Condition Matrix		
	Air Temperature	Humidity
Noise Condition 1	15	50
Noise Condition 2	30	50
Noise Condition 3	15	90
Noise Condition 4	30	90

	A:Flow Rate	B:Pressure	C:Viscosity	D:Cure Temperature	Noise Condition 1	Noise Condition 2	Noise Condition 3	Noise Condition 4	Y Mean	Y Standard Deviation	Signal Noise Ratio
1	2	2	2	120							
2	2	3	10	150							
3	30	3	2	150							
4	30	2	10	120							
5	2	2	2	150							
6	30	3	10	120							
7	30	3	10	150							
8	30	2	2	120							

In the first row of the Data tab, you would enter the response values that were measured with the control factors set as follows:

- Flow Rate: 2
- Pressure: 2
- Viscosity: 2
- Cure Temperature: 120

In the Noise Condition 1 column, you would enter the value measured at those control factor settings when Air Temperature = 15 and Humidity = 50. In the Noise Condition 2 column, you would enter the value measured when Air Temperature = 30 and Humidity = 50, and so on.

In analyzing the data set, the software can calculate results for the Y Mean, Y Standard Deviation and Signal Noise Ratio columns. You will select one or more of these columns to be used as the "responses" for the analysis by selecting the check boxes in the column headings.

- Each value in the Y Mean column is the average measured response at the specified combination of control factor settings, across all noise conditions.
- The Y Standard Deviation column provides the standard deviation of the response at the given control settings across all noise conditions. This column has the natural log transformation applied by default
- The values in the Signal Noise Ratio column indicate how robust the response is to the noise factors at the specified combination of control factor settings, where a greater value indicates that the measured response is less affected by the noise factors. This is the main point of robust parameter design, so you will almost always have this column selected.

For each column selected for inclusion in the analysis, you can specify settings on the Analysis Settings page of the control panel. For the Signal Noise Ratio column, you must specify the equation to be used in calculating the ratio. Your selection for this option is determined by the purpose of your analysis:

- Nominal (or "nominal-the-best") should be used if you have a specific target value for the measured response.
- Larger (or "larger-the-better") should be used if you want to maximize the value of the measured response.
- Smaller (or "smaller-the-better") should be used if you want to minimize the value of the measured response.

Robust Parameter Designs: Example

*The data set used in this example is available in the example database installed with the software (called "Weibull21_DOE_Examples.rsgz21"). 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 example project is "Robust Parameter Design - Method Settings."

Robust parameter design is used to find the appropriate control factor levels in the design to make the system less sensitive to variations in uncontrollable noise factors (i.e., to make the system robust).

Consider an experiment that seeks to determine a method to assemble an elastomeric connector to a nylon tube while delivering the requisite pull-off performance suitable for an automotive engineering application.

The primary design objective is to maximize the pull-off force, while secondary considerations are to minimize assembly effort and reduce the cost of the connector and assembly.

The controllable and noise factors are:

Controllable Factors	Name	Level 1	Level 2	Level 3
A	Interference	Low	Medium	High
B	Wall thickness	Thin	Medium	Thick
C	Insertion depth	Shallow	Medium	Deep
D	Percent adhesive	Low	Medium	High

Noise Factors	Name	Unit	Level -	Level +
E	Time	hour	24	120
F	Temperature	F	72	150
G	Humidity		25%	75%

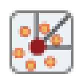
For the inner (control) array, Taguchi OA L9 (3^4) is used. For the outer (noise) array, a two level full factorial design is applied.

The measured response is the pull-off force.

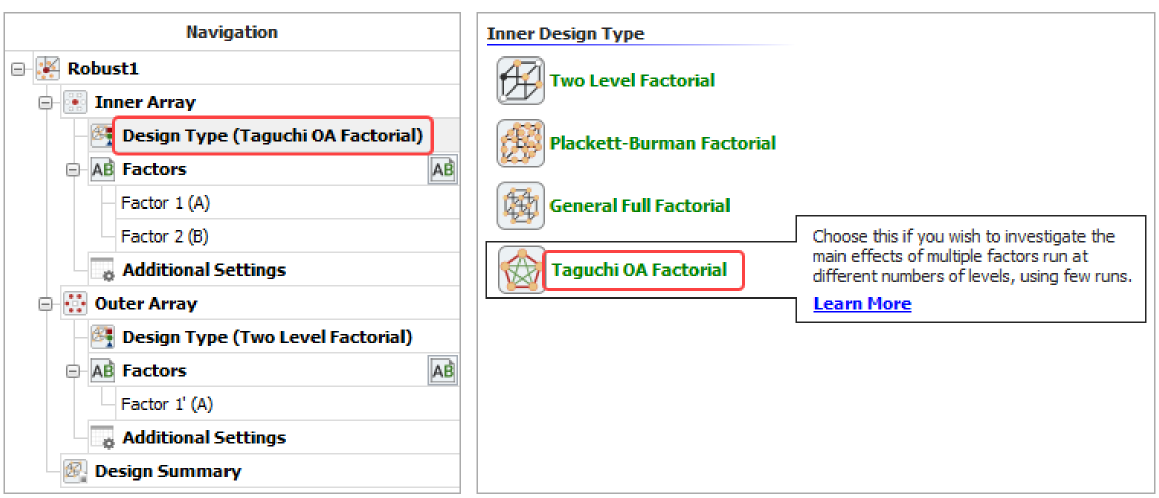
Design the Experiment

The experimenters create a robust parameter design folio, perform the experiment according to the design, and then enter the response values into the folio for further analysis. The design matrix and the response data are given in the "Method Settings" folio. The following steps describe how to create this folio on your own.

- Choose **Home > Insert > Robust Design** to add a robust parameter design folio to the current project.

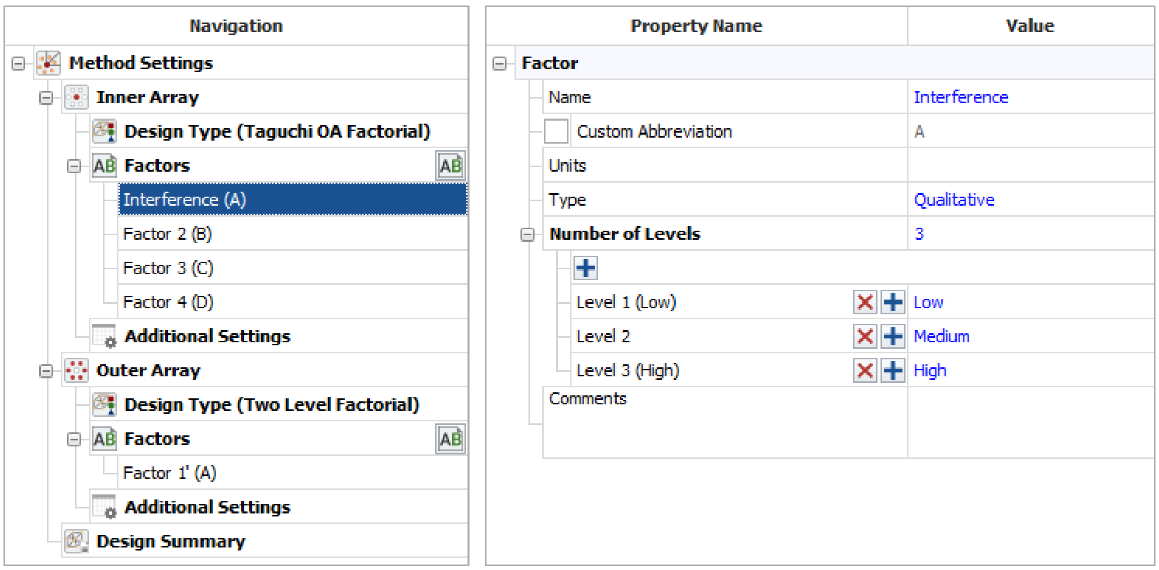


- Click **Design Type** under the **Inner Array** heading in the navigation panel, and then select **Taguchi OA Factorial** in the input panel.



- Rename the folio by clicking the **Robust1** heading in the navigation panel and entering **Method Settings** for the **Name** in the input panel.

- Specify the number of factors by clicking **Factors** under the **Inner Array** heading in the navigation panel and choosing **4** from the **Number of Factors** drop-down list.
- Define each factor by clicking it in the navigation panel and editing its properties in the input panel. The properties are given in the first table above. The first factor is defined as shown next.



- Click **Design Type** under the **Outer Array** heading in the navigation panel, and then select **Two Level Factorial** in the input panel.
- Specify the number of factors by clicking **Factors** under the **Outer Array** heading in the navigation panel and choosing **3** from the **Number of Factors** drop-down list.
- Define each factor by clicking it in the navigation panel and editing its properties in the input panel. The properties are given in the second table above.
- Click the **Design Summary** heading to make sure that you have entered all settings correctly and to view the Noise Condition Matrix. The complete Design Summary is shown next.

Taguchi Robust

Response Properties	
Name	**Units**
Y Mean	
Y Standard Deviation	
Signal Noise Ratio	

Inner Array Properties

Factor Properties								
Abbreviation	**Name**	**Units**	**Type**	**Number of Levels**	**OA Column Index**	**Level 1 (Low)**	**Level 2**	**Level 3 (High)**
A	Interference	s	Qualitative	3	1	Low	Medium	High
B	Wall Thickness	cm	Qualitative	3	2	Thin	Medium	Thick
C	Insertion Depth	v	Qualitative	3	3	Shallow	Medium	Deep
D	Percent Adhesive		Qualitative	3	4	Low	Medium	High

Additional Settings	
Design Type	Taguchi OA Factorial
Taguchi Design Type	L9 (3^4)
# of Unique Runs	9

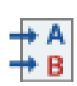
Outer Array Properties

Noise Factor Properties						
Abbreviation	**Name**	**Units**	**Type**	**Number of Levels**	**Level 1 (Low)**	**Level 2 (High)**
A	Time	hour	Quantitative	2	24	120
B	Temperature	F	Quantitative	2	72	150
C	Humidity		Quantitative	2	0.25	0.75

Additional Settings	
Design Type	Two Level Factorial
Fraction	Full
# of Unique Runs	8
# of Replicates	1
Total # of Runs	8

Noise Condition Matrix			
	Time	**Temperature**	**Humidity**
Noise Condition 1	24	72	0.25
Noise Condition 2	120	72	0.25
Noise Condition 3	24	150	0.25
Noise Condition 4	120	150	0.25
Noise Condition 5	24	72	0.75
Noise Condition 6	120	72	0.75
Noise Condition 7	24	150	0.75
Noise Condition 8	120	150	0.75

- Finally, click the **Build** icon on the control panel to create a Data tab that allows you to view the test plan and enter response data.

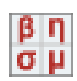


Analysis and Results

The data set for this example is given in the "Method Settings" folio of the example project. After you enter the data from the folio, you can perform the analysis by doing the following:

Note: To minimize the effect of unknown nuisance factors, the run order is randomly generated when you create the design. Therefore, if you followed these steps to create your own folio, the order of runs on the Data tab may be different from that of the folio in the example file. This can lead to different results. To ensure that you get the very same results described next, show the Standard Order column in your folio, then click a cell in that column and choose **Sheet > Sheet Actions > Sort > Sort Ascending**. This will make the order of runs in your folio the same as that of the example file. Then copy the response data from the example file and paste it into the Data tab of your folio.

- On the Analysis Settings page of the control panel, select to use **Individual Terms** for the Y Mean response, which is currently chosen in the **Response** drop-down list.
- Choose the **Y Std** response from the **Response** drop-down list and select to use **Individual Terms** for it.
- Choose the **Signal Noise Ratio** response and select **Larger** in the **Signal Noise Ratio** area, as you are aiming to maximize the pull-off force response. Select to use **Individual Terms** for this response as well.
- Click the **Calculate** icon.

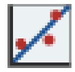


The three responses—Y Mean, Y Standard Deviation and Signal Noise Ratio—are automatically calculated and are used as the responses in the analysis. Note that the logarithmic transformation is automatically applied to the Y Standard Deviation response. To view the transformation, you can click the **Transformation** icon.

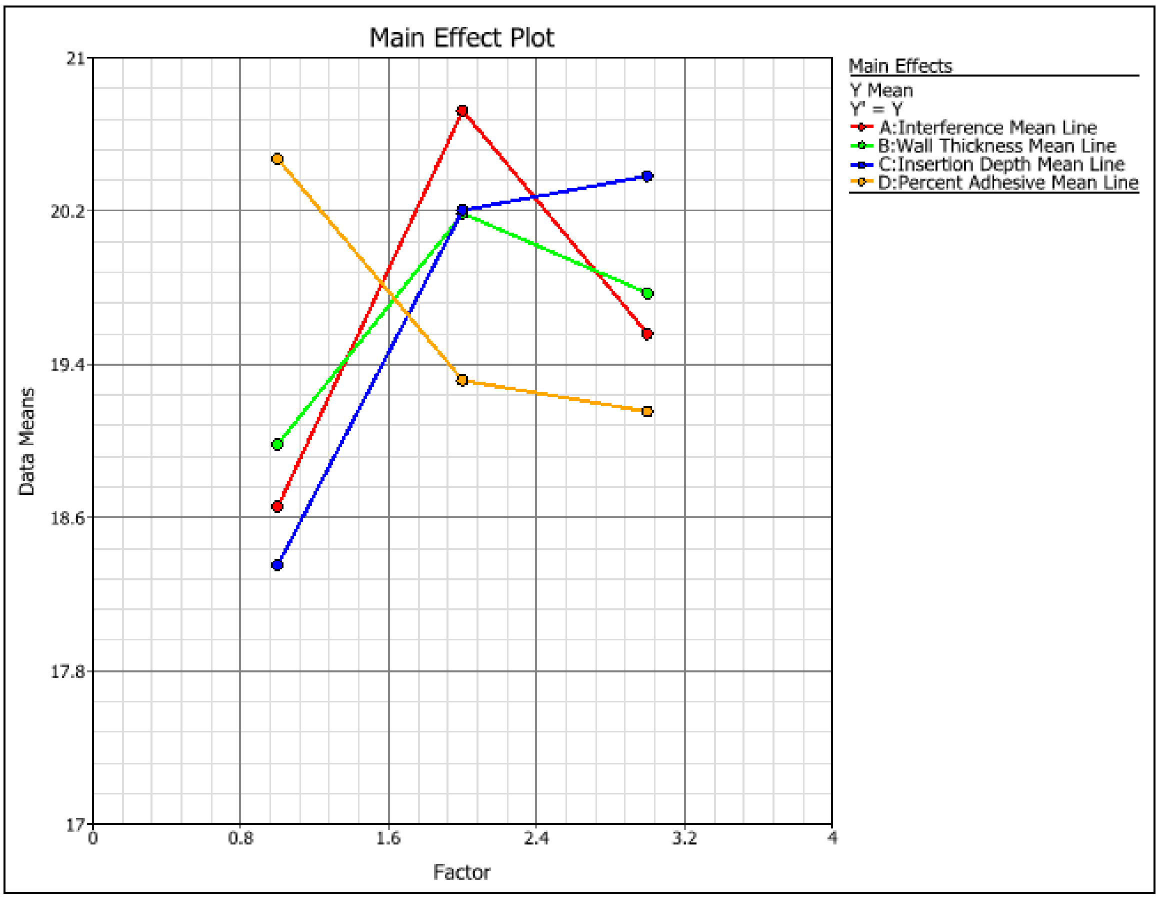
In the Select Transformation window, choose the Y Standard Deviation response from the **Response** drop-down list to see the transformation that has been applied. When you are finished viewing the transformation, click **OK** to close the window.

In robust design, you can use the effect plots to find proper factor settings to meet both mean and standard deviation requirements for the response.

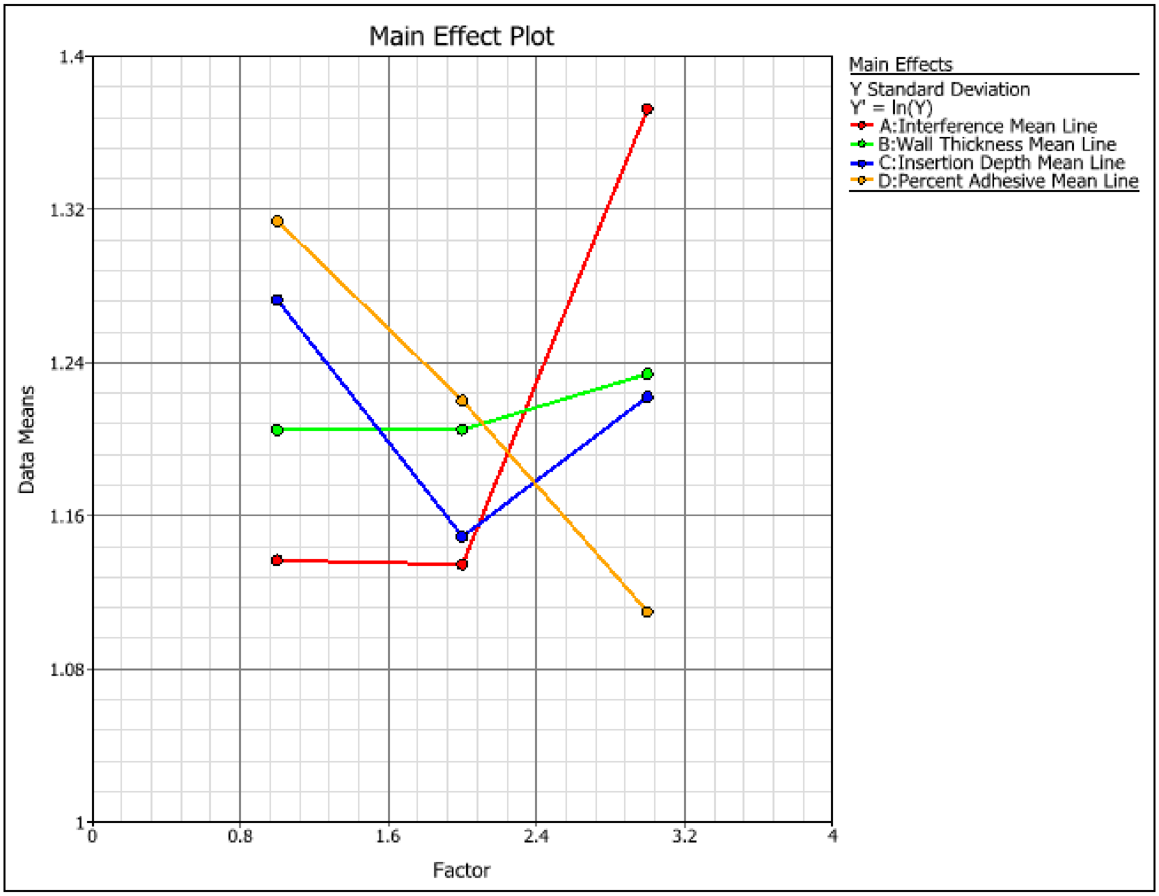
- Click the **Plot** icon.



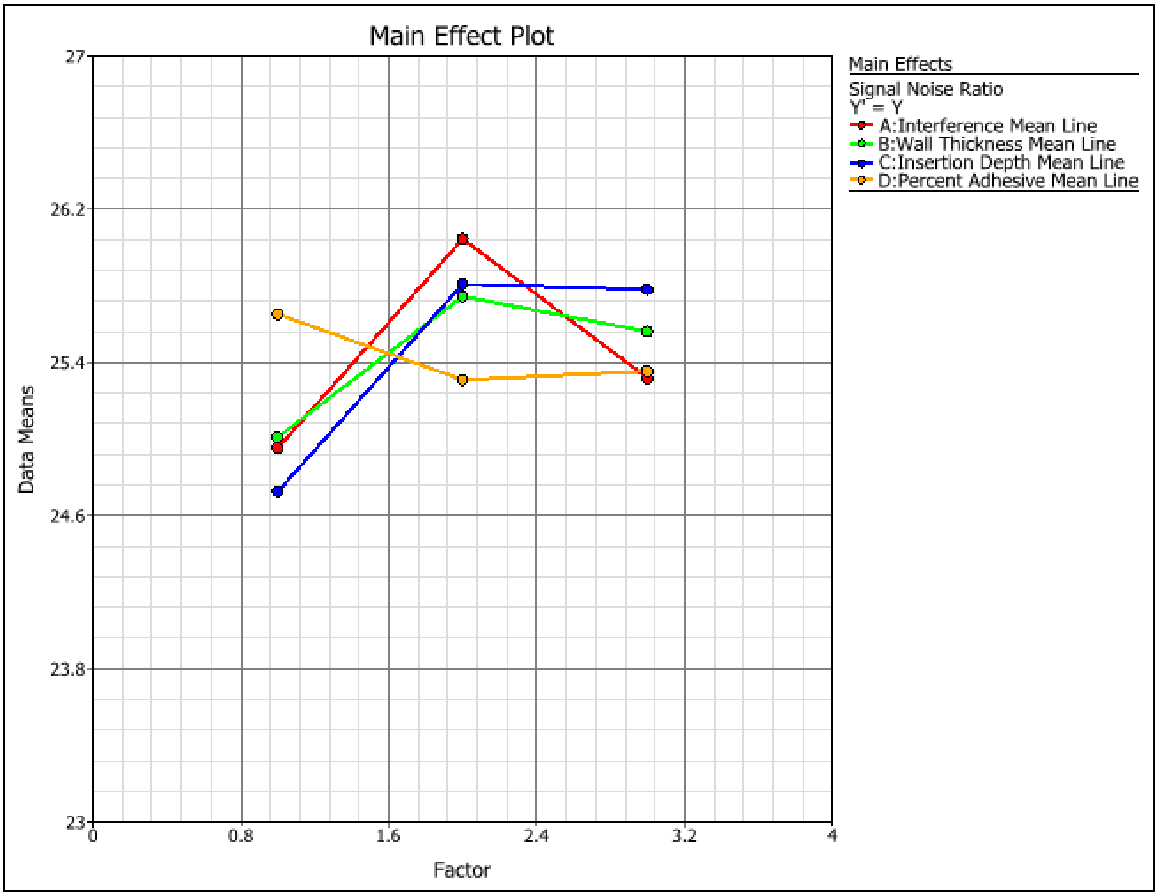
- Choose **Main Effects** from the **Plot Type** drop-down list.
- Choose **Data Means** from the drop-down list in the **Response Values** area. This allows you to view the response values based on values in the original data rather than fitted values. The main effect plot for the Y Mean (pull-off force) response is shown next.



- To view the main effect plot for the Y Standard Deviation response, choose **Y Std** in the **Response** drop-down list.



- To view the main effect plot for the Signal Noise Ratio response, choose **Signal Noise Ratio** in the **Response** drop-down list.



Conclusion

Since the goal is to maximize both signal-to-noise ratio and the pull-off force, some trade-off may need to be made in the selecting of factor settings. Examining the main effect plots for Y Mean and Signal Noise Ratio shows that the medium level for A is clearly the best choice for maximizing signal-to-noise ratio (robustness) and the average pull-off force (Y Mean). It also has a relatively low standard deviation value. For the wall thickness, B, the medium and high level are slightly better than the low level for signal-to-noise ratio; however, the medium is preferred to high level in order to maximize the average pull-off force. From the main effect plot, the better settings for factors C and D can also be determined. In the final analysis, the best settings to maximize signal-to-noise ratio are A (medium), B (medium), C (deep) and D (low), based on the experimental results for maximizing pull-off force.

Mixture Designs

Mixture designs are used when the factors in an experiment are *components* in a mixture, and when you wish to determine the best proportions to use for each component. Since the proportions of all the components in a mixture must always sum to one, mixture designs assume that all the component factors are dependent (i.e., changing one component's value in an experiment will necessarily change others).

Mixture designs can also include *process variables* as factors. For example, if the components in a mixture are different metals that are combined to produce steel, the experimenter can choose to consider process variables like temperature and pressure in order to determine the best way to process the mixture (in addition to estimating the best proportions of the metals).

Specific mixture design types include:

- Simplex designs, which assume that there are either no bounds or only lower bounds on the component amounts.
- Extreme vertex designs are more complicated designs that allow you to add linear constraints on the design and/or upper bounds on individual components.

For more information about how to use the design types, please consult the documentation on design folios.

Simplex Designs

The factor levels for components in simplex designs are subject to the following constraints:

- The proportions of all the components must sum to one.
- The amounts of each component must be at least the specified lower bound and no greater than the specified total amount.

If you want to further constrain the factor levels by adding upper bounds to components, or by adding linear constraints (i.e., bounds on combinations of components), use an extreme vertex design.

Available Simplex Designs

There are three types of simplex designs:

- **Simplex Lattice**: With this design, the blends in the experiment are determined by the specified number of levels of each component (i.e., the degree of design + 1). Since it includes all the reasonable combinations of components, this design is useful when the number of components is not large, and a higher polynomial equation is needed for optimization.
- **Simplex Centroid**: By default, this design includes single-component blends (vertices) and all centroids up to the dimension g, where g is the number of components. Users can specify the

degree of design (i.e., the dimension of centroids). Since it usually has fewer test runs than a simplex lattice design, this design is useful when the number of components is larger, but a lower polynomial equation will suffice for optimization.

- **Simplex Axial**: This design includes vertices, blends with an absent component (edge points), center points and interior points between the center point and vertices (axial points). It is mainly used for screening components.

Simplex Designs: Example

*The data set used in this example is available in the example database installed with the software (called "Weibull21_DOE_Examples.rsgz21"). 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 example project is "Mixture Design - Design with Process Variables."

A type of vinyl is used for seat covers. Several components are used to make it. A preliminary experiment is set up to study three plasticizers (X1, X2 and X3), where the percentage of their total formulation contribution is 40%. The percentage of the remaining components is to be held fixed in all blends studied, where the sum of their parts by weight make up 60% of the total. A {3, 2} simplex lattice design is used.

Two process variables, rate of extrusion (Z1) and temperature of drying (Z2), are very important to make good vinyl. A two level factorial design is used for them. The low and high for Z1 are 10 and 20; the low and high for temperature are 30 and 50. No units for them are provided.

A complete {3, 2} simplex lattice design for the components is conducted under each of the 4 combinations of Z1 and Z2.

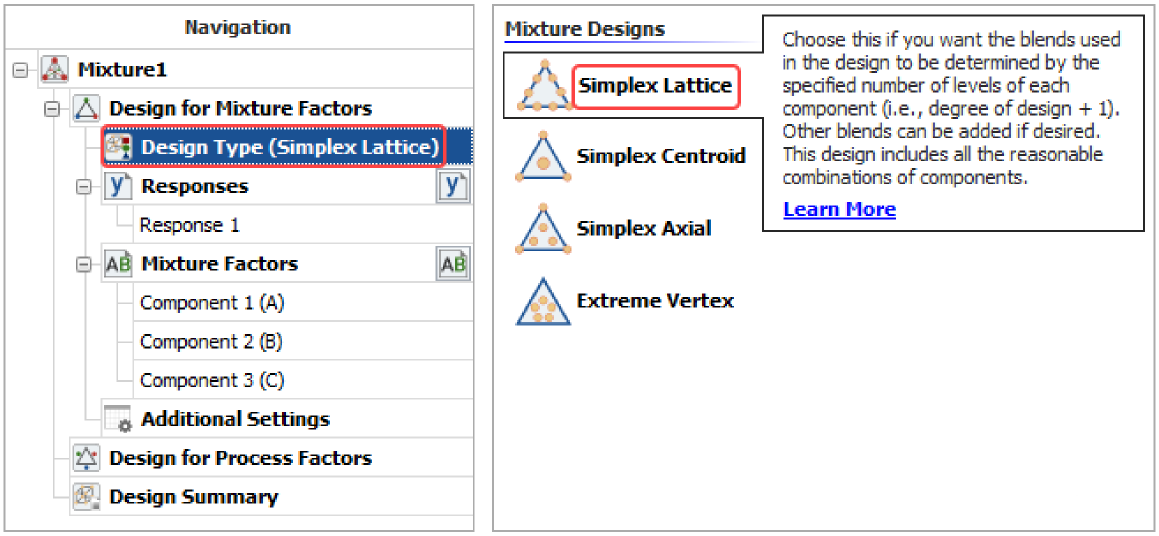
The measured response is the vinyl thickness value. The ideal thickness is 10. The acceptable range is 9 to 11.

DESIGNING THE EXPERIMENT

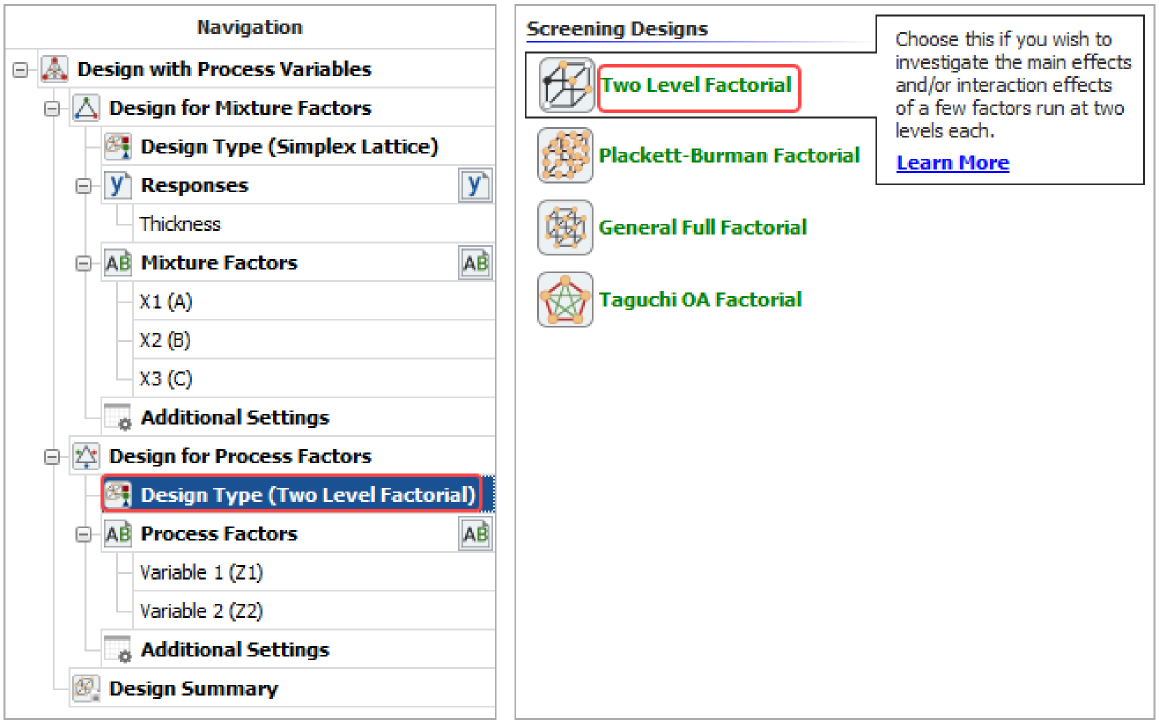
The experimenters create a simplex lattice design folio, perform the experiment according to the design, and then enter the response values into the folio for analysis. The design matrix and the response data are given in the "Design with Process Variables" folio. The following steps describe how to create this folio on your own.

- Choose **Home > Insert > Mixture Design** to add a mixture design folio to the current project.

- In the folio's navigation panel, under the **Design for Mixture Factors** heading, click **Design Type** and then select **Simplex Lattice** in the input panel.



- Rename the folio by clicking the **Mixture1** heading in the navigation panel and entering **Design with Process Variables** for the **Name** in the input panel.
- Rename the response by clicking **Response 1** in the navigation panel and entering **Thickness** in the input panel.
- Specify the number of mixture factors by clicking the **Mixture Factors** heading in the navigation panel and choosing **3** from the **Number of Factors** drop-down list.
- Rename each factor by clicking it in the navigation panel and editing its properties in the input panel.
- Click the **Additional Settings** heading to specify the sum of the components and the degree of the lattice.
  - Set the **Mixture Total** to **0.4**.
  - Set the **Degree of Design** to **2**.
- Click the **Design for Process Factors** heading in the navigation panel and choose **Yes** in the **Include process factors?** field.
- Click **Design Type** and then select **Two Level Factorial** in the input panel.



- Specify the number of process factors by clicking the **Process Factors** heading in the navigation panel and choosing **2** from the **Number of Factors** drop-down list.
- Define each factor by clicking it in the navigation panel and editing its properties in the input panel.
  - First factor:
    - Name: **Rate of Extrusion**
    - Type: **Quantitative**
    - Level 1 (Low): **10**
    - Level 2 (High): **20**
  - Second factor:
    - Name: **Temperature**
    - Type: **Quantitative**
    - Level 1 (Low): **30**
    - Level 2 (High): **50**

- Click the **Design Summary** heading to make sure that you have entered all settings correctly. The complete Design Summary is shown next.

Mixture Design

Response Properties	
Name	**Units**
Thickness	

Design for Mixture Factors

Mixture Factors						
Abbreviation	**Name**	**Units**	**Type**	**Number of Levels**	**Level 1 (Low)**	**Level 2 (High)**
A	X1		Quantitative	2	0	0.4
B	X2		Quantitative	2	0	0.4
C	X3		Quantitative	2	0	0.4

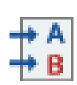
Additional Settings	
Design Type	Simplex Lattice
Mixture Total	0.4
Degree of Design	2
# of Unique Runs	6
# of Replicates	1
Total # of Runs	6

Design for Process Factors

Process Factors						
Abbreviation	**Name**	**Units**	**Type**	**Number of Levels**	**Level 1 (Low)**	**Level 2 (High)**
Z1	Rate of Extrusion		Quantitative	2	10	20
Z2	Temperature		Quantitative	2	30	50

Additional Settings	
Design Type	Two Level Factorial
Fraction	Full
# of Unique Runs	4
# of Replicates	1
Total # of Runs	4

- Finally, click the **Build** icon on the control panel to create a Data tab that allows you to view the test plan and enter response data.



ANALYSIS AND RESULTS

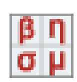
The data set for this example is given in the "Design with Process Variables" folio of the example project. After you enter the data from the folio, you can perform the analysis by doing the following:

Note: To minimize the effect of unknown nuisance factors, the run order is randomly generated when you create the design. Therefore, if you followed these steps to create your own folio, the order of runs on the Data tab may be different from that of the folio in the example file. This can lead to different results. To ensure that you get the very same results described next, show the Standard Order column in your folio, then click a cell in that column and choose **Sheet > Sheet Actions > Sort > Sort Ascending**. This will make the order of runs in your folio the same as that of the example file. Then copy the response data from the example file and paste it into the Data tab of your folio.

- Select terms for the model by clicking the **Select Terms** icon on the control panel.

  In the window that appears, select the linear effects and 2-way interaction effects of the mixture factors in the model. The linear effects and 2-way interaction effects of the process factors are crossed with each term of the mixture factors. Click **OK**.

- On the control panel, click the **Calculate** icon.



Click the **View Analysis Summary** icon on the control panel and select to the view the Regression Information table. Note that the results from this step are not visible in the example folio, which shows the reduced model results that are obtained later in this example.

Regression Information							
Term	Coefficient	Standard Error	Low Confidence	High Confidence	T Value	P Value	Variance Inflation Factor
A:X1	8.25	-	-	-	-	-	1.5
B:X2	5.5	-	-	-	-	-	1.5
C:X3	5.75	-	-	-	-	-	1.5
A • B*	10.5	-	-	-	-	-	1.5
A • C*	5	-	-	-	-	-	1.5
B • C	3.5	-	-	-	-	-	1.5
Z1 • A	-0.25	-	-	-	-	-	1.5
Z1 • B	-0.5	-	-	-	-	-	1.5
Z1 • C	1.75	-	-	-	-	-	1.5
Z1 • A • B	1.5	-	-	-	-	-	1.5
Z1 • A • C	-4	-	-	-	-	-	1.5
Z1 • B • C	1.5	-	-	-	-	-	1.5
Z2 • A	-0.25	-	-	-	-	-	1.5
Z2 • B	1	-	-	-	-	-	1.5
Z2 • C	-1.25	-	-	-	-	-	1.5
Z2 • A • B*	-5.5	-	-	-	-	-	1.5
Z2 • A • C	-4	-	-	-	-	-	1.5
Z2 • B • C	2.5	-	-	-	-	-	1.5
Z1 • Z2 • A	-1.75	-	-	-	-	-	1.5
Z1 • Z2 • B	-1	-	-	-	-	-	1.5
Z1 • Z2 • C	-0.25	-	-	-	-	-	1.5
Z1 • Z2 • A • B*	-8.5	-	-	-	-	-	1.5
Z1 • Z2 • A • C*	-5	-	-	-	-	-	1.5
Z1 • Z2 • B • C	-1.5	-	-	-	-	-	1.5

Terms with an asterisk (*) are significant terms selected by Lenth's method. Terms with coefficients (absolute value) less or equal to 1 can be removed, and coefficients close to 0 can be considered insignificant. These terms are:

- Z1*A
- Z1*B
- Z2*A
- Z2*B
- Z1*Z2*B
- Z1*Z2*C

Return to the Select Terms window and remove these terms, then recalculate the folio. The results are shown next. Note that the results from this step are not visible in the example folio, which shows the reduced model results that are obtained later in this example.

Regression Information							
Term	Coefficient	Standard Error	Low Confidence	High Confidence	T Value	P Value	Variance Inflation Factor
A:X1	8.25	0.637377	7.01146	9.48854	-	-	1.5
B:X2	5.5	0.637377	4.26146	6.73854	-	-	1.5
C:X3	5.75	0.637377	4.51146	6.98854	-	-	1.5
A • B	10.5	3.122499	4.43242	16.56758	3.362691	0.015179	1.5
A • C	5	3.122499	-1.06758	11.06758	1.601282	0.160433	1.5
B • C	3.5	3.122499	-2.56758	9.56758	1.120897	0.30518	1.5
Z1 • C	1.75	0.637377	0.51146	2.98854	2.745626	0.033485	1.5
Z1 • A • B	0	2.54951	-4.954159	4.954159	0	1	1
Z1 • A • C	-4.5	2.850439	-10.038918	1.038918	-1.578704	0.165482	1.25
Z1 • B • C	0.5	2.850439	-5.038918	6.038918	0.175412	0.866526	1.25
Z2 • C	-1.25	0.637377	-2.48854	-0.01146	-1.961161	0.097538	1.5
Z2 • A • B	-4	2.54951	-8.954159	0.954159	-1.568929	0.167714	1
Z2 • A • C	-4.5	2.850439	-10.038918	1.038918	-1.578704	0.165482	1.25
Z2 • B • C	4.5	2.850439	-1.038918	10.038918	1.578704	0.165482	1.25
Z1 • Z2 • A	-1.75	0.637377	-2.98854	-0.51146	-2.745626	0.033485	1.5
Z1 • Z2 • A • B	-10.5	2.850439	-16.038918	-4.961082	-3.683644	0.010286	1.25
Z1 • Z2 • A • C	-5.5	2.850439	-11.038918	0.038918	-1.929528	0.101911	1.25
Z1 • Z2 • B • C	-4	2.54951	-8.954159	0.954159	-1.568929	0.167714	1

Some of the terms in the above model should be removed again since their p values are relatively high (i.e., at least 0.1). These terms are:

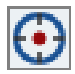
- A*C
- B*C
- Z1*A*B
- Z1*A*C
- Z1*B*C
- Z2*A*B
- Z2*A*C
- Z2*B*C
- Z1*Z2*A*C
- Z1*Z2*B*C

The results with these terms removed are shown next.

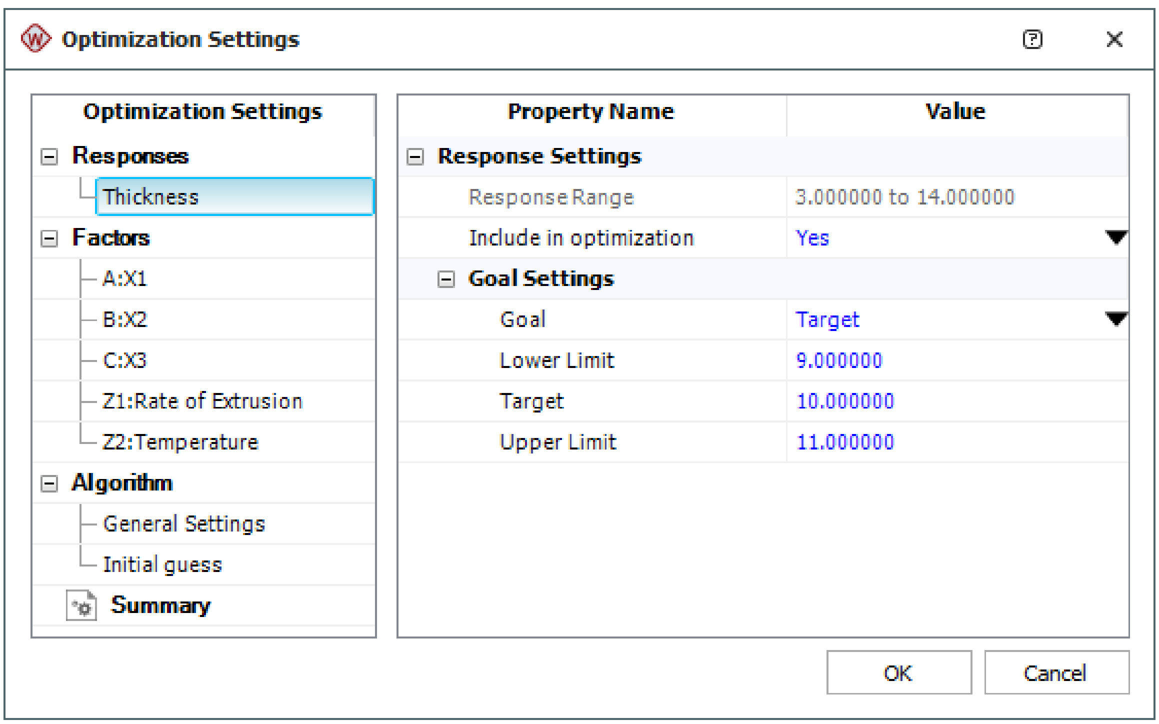
Regression Information							
Term	Coefficient	Standard Error	Low Confidence	High Confidence	T Value	P Value	Variance Inflation Factor
A:X1	8.628571	0.752763	7.314334	9.942809	-	-	1.242857
B:X2	5.728571	0.752763	4.414334	7.042809	-	-	1.242857
C:X3	6.357143	0.698923	5.136904	7.577381	-	-	1.071429
A • B	9.285714	3.953707	2.383003	16.188426	2.348609	0.032024	1.428571
Z1 • C	1.416667	0.675224	0.237804	2.595529	2.098069	0.052132	1
Z2 • C	-1.25	0.675224	-2.428862	-0.071138	-1.851237	0.082683	1
Z1 • Z2 • A	-2.3	0.739671	-3.591379	-1.008621	-3.109491	0.006744	1.2
Z1 • Z2 • A • B	-9.4	3.623633	-15.726439	-3.073561	-2.594082	0.019575	1.2

OPTIMIZATION

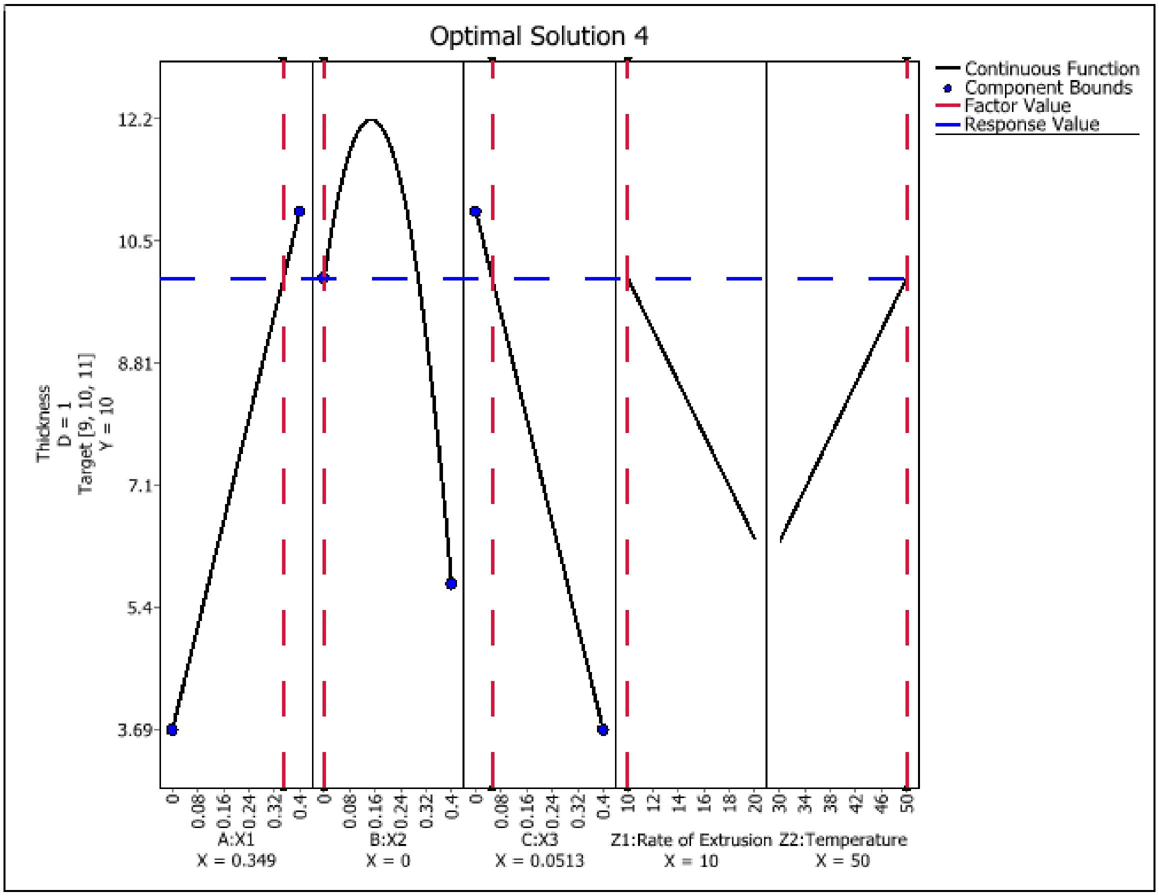
- Click the **Design - Optimization** icon on the control panel.



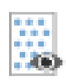
- In the Response Settings window that appears, use the settings shown next and click **OK**. These settings indicate that you want to reach a target thickness of 10, with values less than 9 or greater than 11 being undesirable.



- An Optimization folio will appear with an optimal solution plot showing the fourth solution, displayed next. (The **Solutions** area on the control panel shows all the solutions that were found, and you can click them to view the plot representing the solution.)



- You can choose **Optimization > Solutions > View Solutions** or click the icon on the control panel to see all the optimal solutions in numerical format.



Four optimum settings were found, and they are shown next.

Solutions

	Solution Name	A:X1	B:X2	C:X3	Z1:Rate of Extrusion	Z2: Temperature	Thickness Predicted	Thickness Desirability	Global Desirability
1	Optimal Solution 1	0.255119	0.0592	0.085681	10.907866	50	10	1	1
2	Optimal Solution 2	0.166109	0.154976	0.078915	11.124998	50	10	1	1
3	Optimal Solution 3	0.168858	0.141104	0.090038	10.603235	50	10	1	1
4	Optimal Solution 4	0.348684	0	0.051316	10	50	10	1	1

OK Cancel

CONCLUSIONS

The optimal solution is found to be X1 = 0.349, X2 = 0 and X3 = 0.051 with process factors Rate of Extrusion = 10.000 and Temperature = 50.000. Under this setting, the expected thickness of the vinyl is 10.00. Keep in mind that it is necessary to conduct an experiment using this setting to confirm this conclusion.

Extreme Vertex Designs

While simplex designs provide allow you to place lower bounds on individual mixture factors, there may be situations where more complicated constraints are appropriate. Extreme vertex designs allow you to impose additional limits on the component values by specifying upper bounds on components and defining linear constraints for blends.

Defining Linear Constraints

Unlike component bounds, which put limits on individual component amounts, linear constraints allow you to place limits on combinations of components. As an example, if you have three components in your experiment (C1, C2 and C3), the following two limits would be linear constraints:

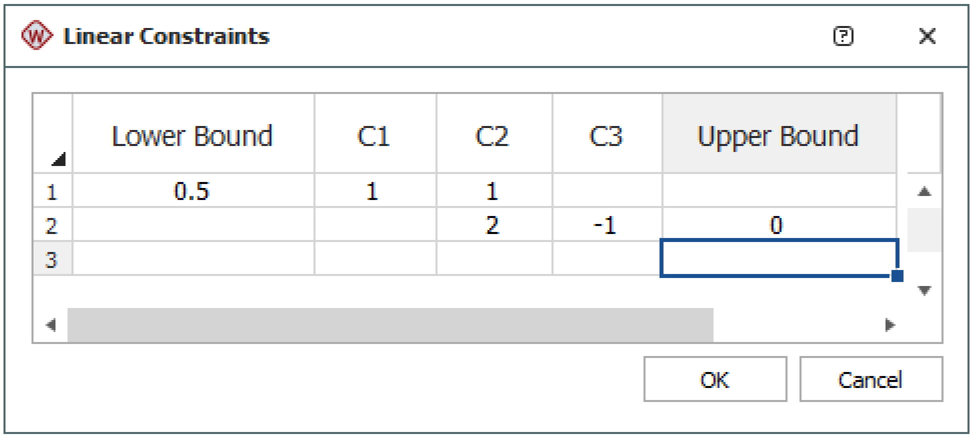
- The combined amount of C1 and C2 in any blend must be at least 0.5 grams.
- In any blend, the amount of C3 must be at least twice the amount of C2.

When you're creating an extreme vertex design, constraints like these are defined by opening the Linear Constraints window. To do this, click the **Additional Settings** heading on the navigation panel, and then click the **[+]** icon under **Linear Constraints**. To remove a constraint, click the **[-]** icon next to it.

DEFINING CONSTRAINTS

Each row in the Linear Constraints window represents an equation that defines a single constraint. The first and last columns represent lower/upper bounds of the constraint, and the middle columns represent the coefficients of each component in the blend (the column headers show the abbreviated factor names). If a constraint lacks a lower or upper bound, or if there certain components that are not considered, you can leave those cells blank.

Consider the two example constraints above. These constraints could be defined as shown next.



- According to the first constraint, the combined amount of C1 and C2 must be at least 0.5. In other words:

$$0.5 <= C1 + C2$$

  So **0.5** is entered for the lower bound, and **1** is entered for the coefficients of C1 and C2.

- The second constraint can be expressed as follows.

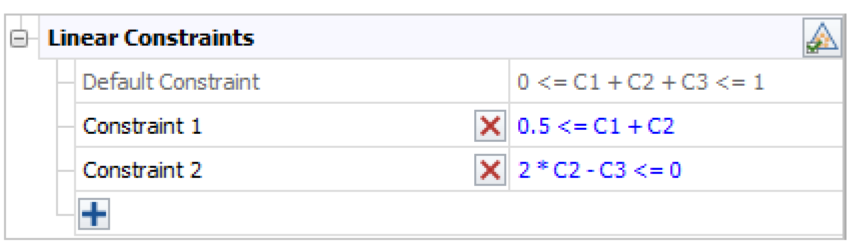
$$2*C2 <= C3$$

  In other words, C3 must be at least double the amount of C2. This is equivalent to the following:

$$2*C2 - C3 <= 0$$

  So **2** and **-1** are entered for the coefficients of C2 and C3, and **0** is entered for the upper bound.

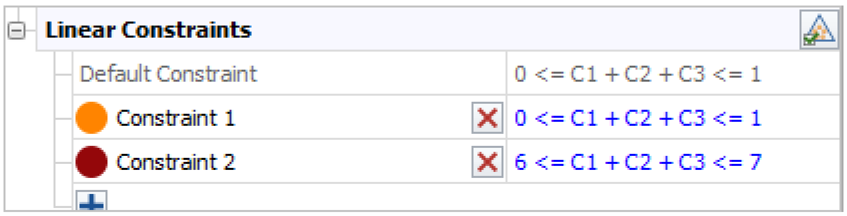
After you click **OK**, the constraints are displayed in the folio as shown next. To edit a constraint, just click the blue equation to re-open the Linear Constraints window.



VALIDATING CONSTRAINTS

To determine whether you have entered any incompatible or unnecessary linear constraints (i.e., constraints that rule out all blends, or which do not impose any new limits beyond those already defined), click the **Validate Constraints** icon in the additional settings.

Colored dots will appear next to any problematic constraints. Red dots represent incompatible constraints, and orange dots represent unnecessary ones. For example, Constraint 1 below is unnecessary because it is identical to the default constraint. And Constraint 2 is incompatible with Constraint 1 because no blend can have an amount that is both at most 1 (Constraint 1) and at least 6 (Constraint 2).



Note: Information on the number of runs in the Design Information area of the control panel and in the Design Summary will not be available until you have validated the constraints.

Extreme Vertex Designs: Example

*The data set used in this example is available in the example database installed with the software (called "Weibull21_DOE_Examples.rsgz21"). 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 example project is "Mixture Design - Extreme Vertex Design."

Four components, A, B, C and D, are used to make a polymer material. Their proportions are restricted by lower and upper bounds, and two linear constraints:

Bounds:

- $0.20 \leq A \leq 0.65$
- $0.10 \leq B \leq 0.55$
- $0.10 \leq C \leq 0.20$
- $0.15 \leq D \leq 0.35$

Linear constraints:

- $0.25 \leq A + C \leq 0.70$
- $0.25 \leq C + D \leq 0.45$

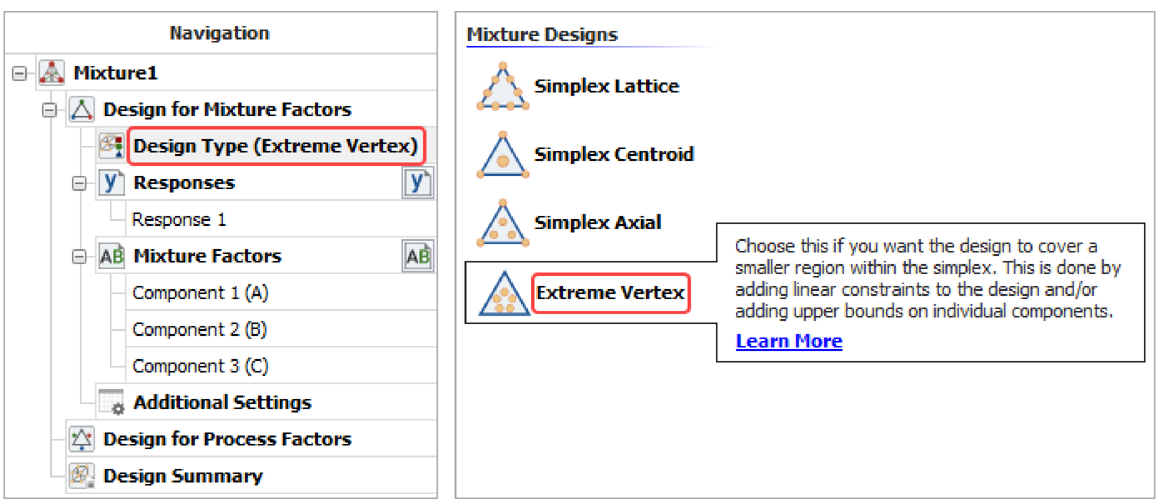
The objective of the experiment is to find the best settings of each component in order to maximize the strength of the material. An experiment within the feasible region defined by the above constraints should be conducted. The experimenters want some experiment runs to be placed at the boundary of the feasible region and some to be inside the region.

DESIGNING THE EXPERIMENT

The experimenters create an extreme vertex design folio, perform the experiment according to the design, and then enter the response values into the folio for analysis. The design matrix and the response data are given in the "Extreme Vertex Design" folio. The following steps describe how to create this folio on your own.

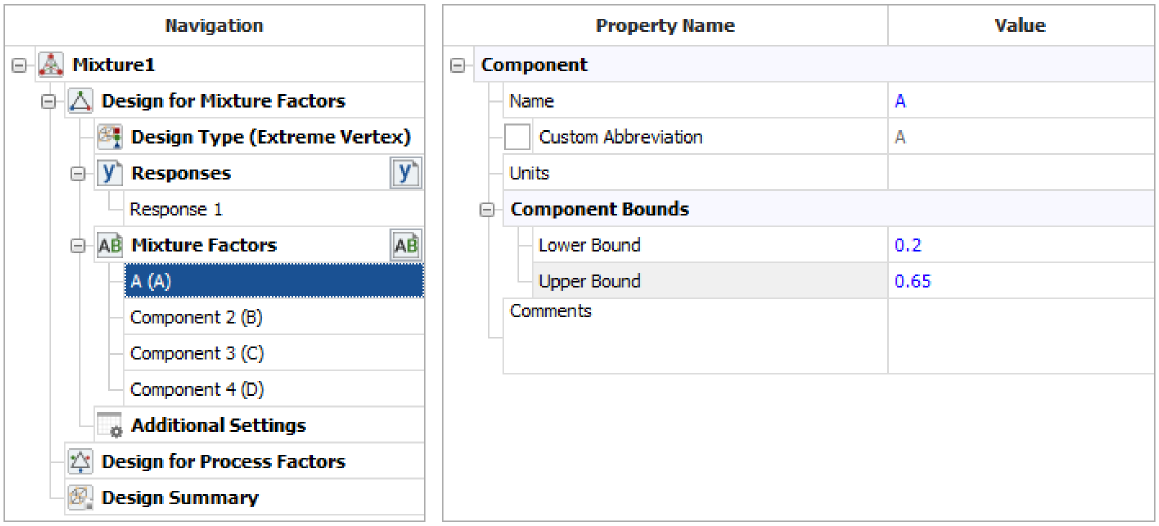
- Choose **Home > Insert > Mixture Design** to add a mixture design folio to the current project.

- In the folio's navigation panel, under the **Design for Mixture Factors** heading, click **Design Type** and then select **Extreme Vertex** in the input panel.

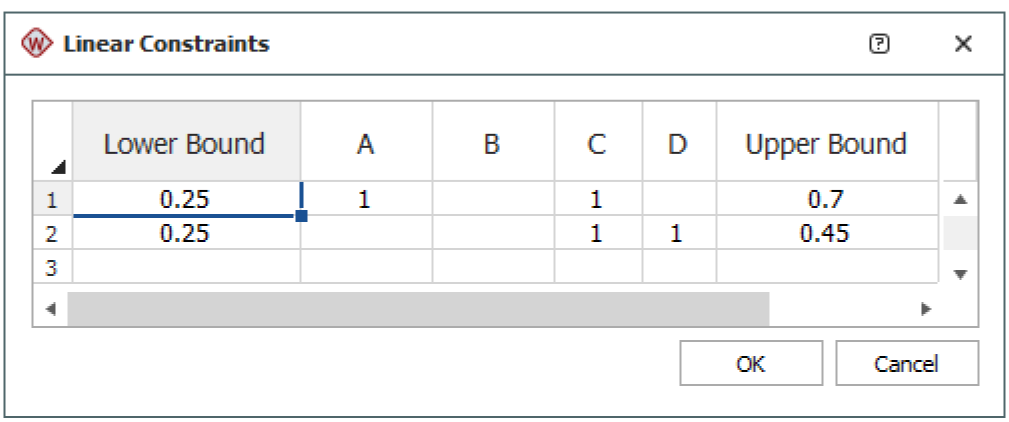


- Specify the number of mixture factors by clicking the **Mixture Factors** heading in the navigation panel and choosing **4** from the **Number of Factors** drop-down list.

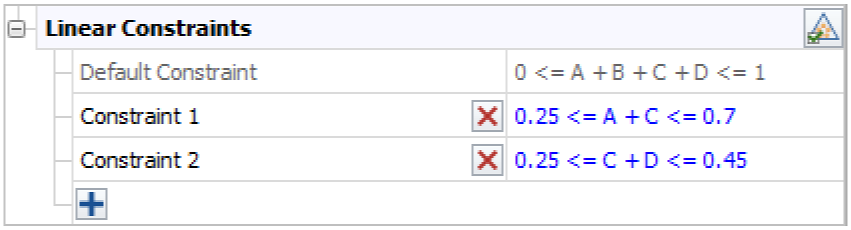
- Define each factor by clicking it in the navigation panel and editing its properties in the input panel. The bounds for each factor are given above. The first factor is defined as shown next.



- Rename the folio by clicking the **Mixture1** heading in the navigation panel and entering **Extreme Vertex Design** for the **Name** in the input panel.
- Click the **Additional Settings** heading to choose the number of center points and axial points per vertex, and to specify the linear constraints.
  - Set the number of **Center Points** to **1**.
  - Set the **Axial per Vertex** field to **1**.
  - Click the **[+]** icon under **Linear Constraints** and enter the linear constraints as shown next.



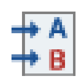
After you click **OK**, the constraints are displayed in the folio as shown next.



- To determine whether you have entered any incompatible or unnecessary linear constraints, click the **Validate Constraints** icon next to the **Linear Constraints** heading.

No constraints are marked as problematic, so you can proceed.

- Finally, click the **Build** icon on the control panel to create a Data tab that allows you to view the test plan and enter response data.



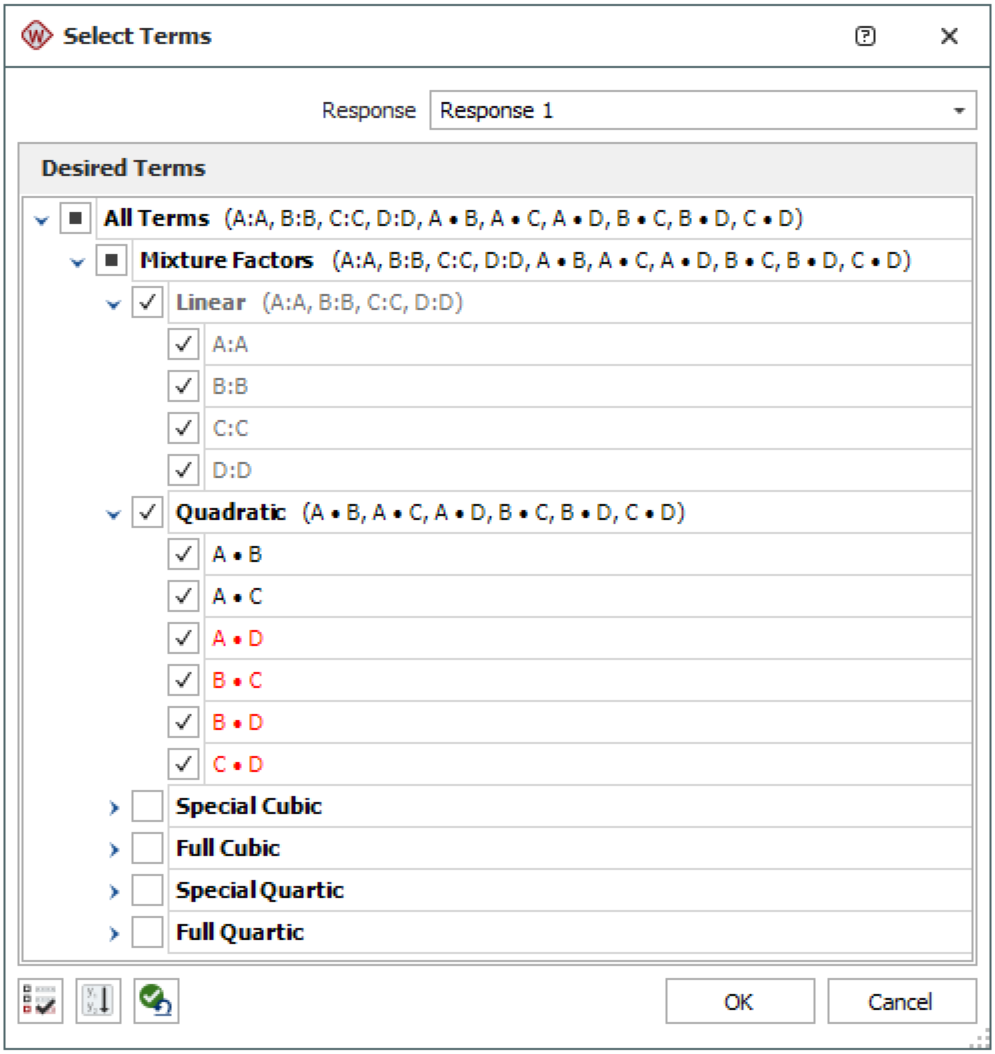
ANALYSIS AND RESULTS

The data set for this example is given in the "Extreme Vertex Design" folio of the example project. After you enter the data from the folio, you can perform the analysis by doing the following:

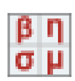
Note: To minimize the effect of unknown nuisance factors, the run order is randomly generated when you create the design. Therefore, if you followed these steps to create your own folio, the order of runs on the Data tab may be different from that of the folio in the example file. This can lead to different results. To ensure that you get the very same results described next, show the Standard Order column in your folio, then click a cell in that column and choose **Sheet > Sheet Actions > Sort > Sort Ascending**. This will make the order of runs in your folio the same as that of the example file. Then copy the response data from the example file and paste it into the Data tab of your folio.

- Make sure all terms up to 2-way interactions will be considered. To do this, click the **Select Terms** icon on the control panel.

In the window that appears, select the **Linear** and **Quadratic** check boxes. Then click **OK**.



- On the Analysis Settings page of the Control Panel, select to use **Individual Terms** in the analysis.
- Click the **Calculate** icon.



- In the Analysis Summary area on the control panel, click the **View Analysis Summary** icon to view detailed results from the analysis. Note that the results from this step are not visible in the example folio, which shows the reduced model results that are obtained later in this example.

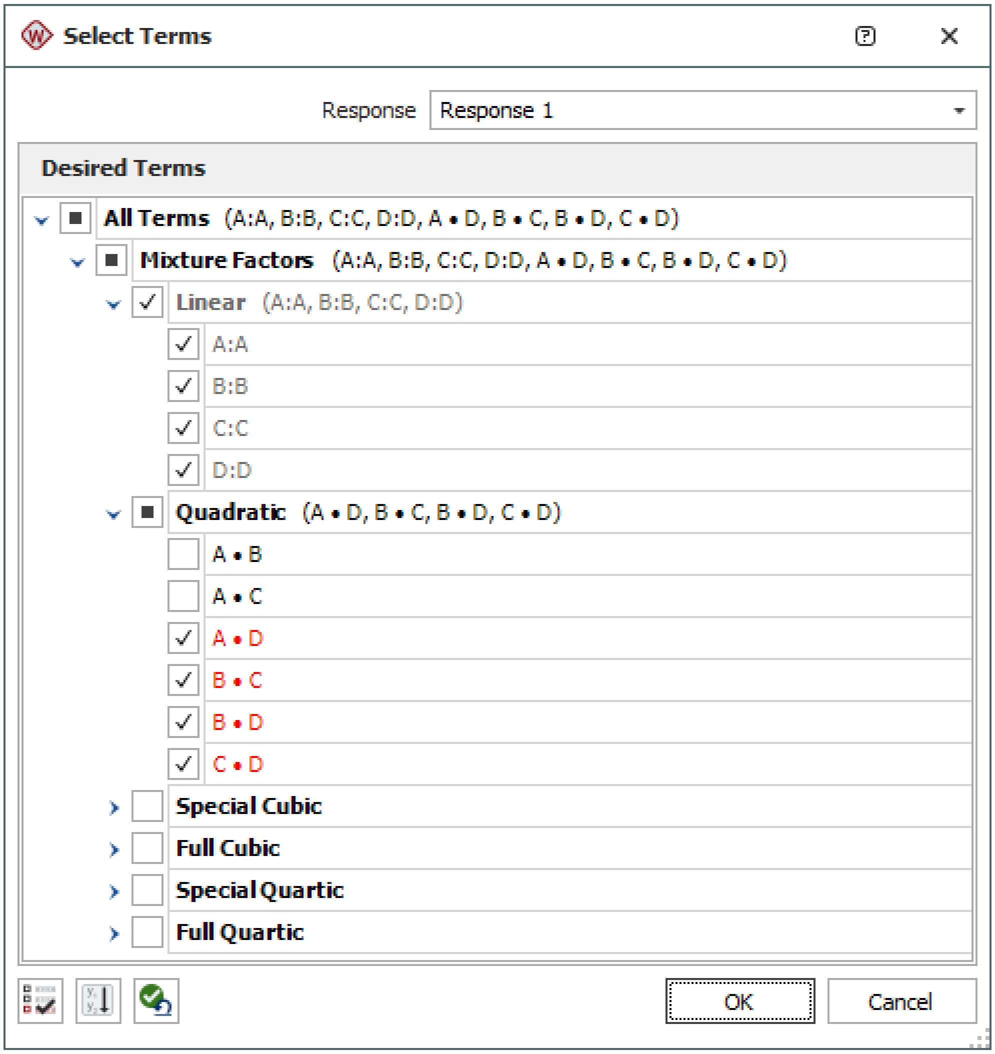
  In the ANOVA table, you can see that effects AD, BD and CD are significant. The p value for effect BC is also relatively close to the risk level of 0.1. Therefore, you decide to include it in the final model.

ANOVA Table					
Source of Variation	Degrees of Freedom	Sum of Squares [Partial]	Mean Squares [Partial]	F Ratio	P Value
Model	9	3.212292	0.356921	80.511607	9.476913E-9
Linear	3	0.391662	0.130554	29.449365	0.000015
A • B	1	0.000071	0.000071	0.015913	0.901892
A • C	1	0.000921	0.000921	0.207781	0.657379
A • D	1	0.018755	0.018755	4.230639	0.064213
B • C	1	0.005248	0.005248	1.183788	0.299851
B • D	1	0.020911	0.020911	4.717026	0.052605
C • D	1	0.032819	0.032819	7.402994	0.019901
Residual	11	0.048765	0.004433		
Lack of Fit	11	0.048765	0.004433		
Total	20	3.261057			

OPTIMIZATION

The reduced model includes only the terms that were found to be significant. The following steps describe how to obtain this model on your own.

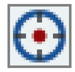
- Click the **Select Terms** icon on the control panel.
- In the Select Terms window that appears, click the **Select Significant Effects** icon to select only the significant effects to calculate the new model, and also select the check box for effect **BC**, as shown next, then click **OK**.



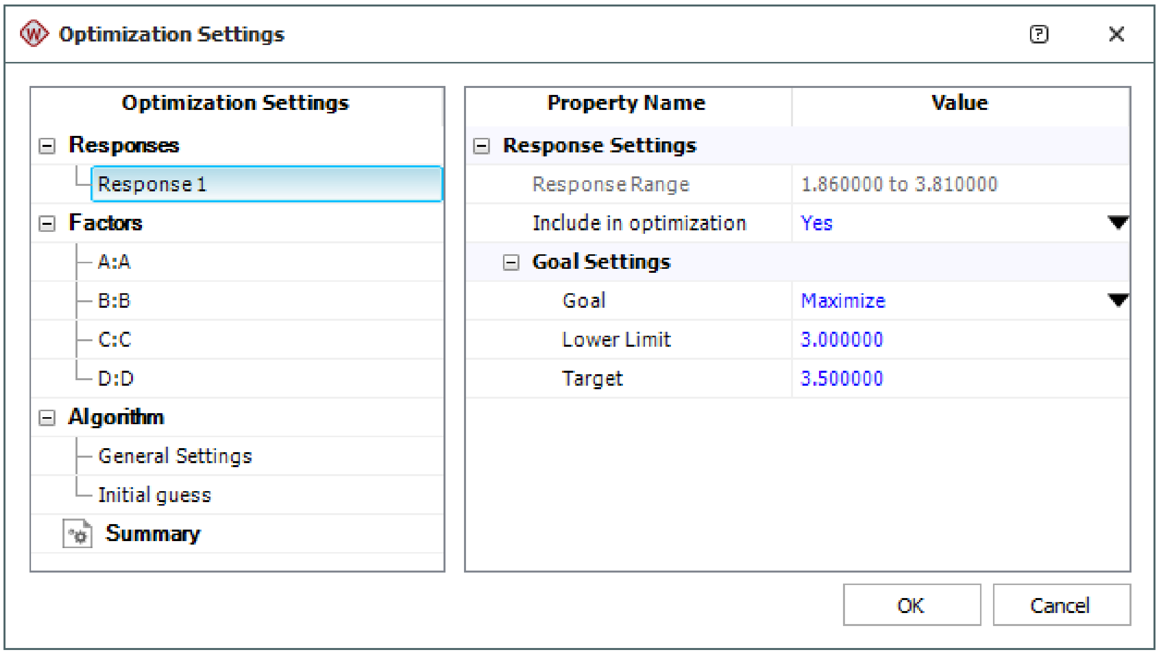
- Click the **Calculate** icon in the control panel.
- In the **Analysis Summary** area on the control panel, click the **View Analysis Summary** icon to view detailed results from the analysis.

ANOVA Table					
Source of Variation	Degrees of Freedom	Sum of Squares [Partial]	Mean Squares [Partial]	F Ratio	P Value
Model	7	3.207496	0.458214	111.214235	1.416423E-10
Linear	3	1.344613	0.448204	108.784857	1.848428E-9
A • D	1	0.04379	0.04379	10.628363	0.006206
B • C	1	1.283843	1.283843	311.604835	1.824416E-10
B • D	1	0.041538	0.041538	10.081696	0.007309
C • D	1	2.1237	2.1237	515.448782	7.628142E-12
Residual	13	0.053561	0.00412		
Lack of Fit	13	0.053561	0.00412		
Total	20	3.261057			

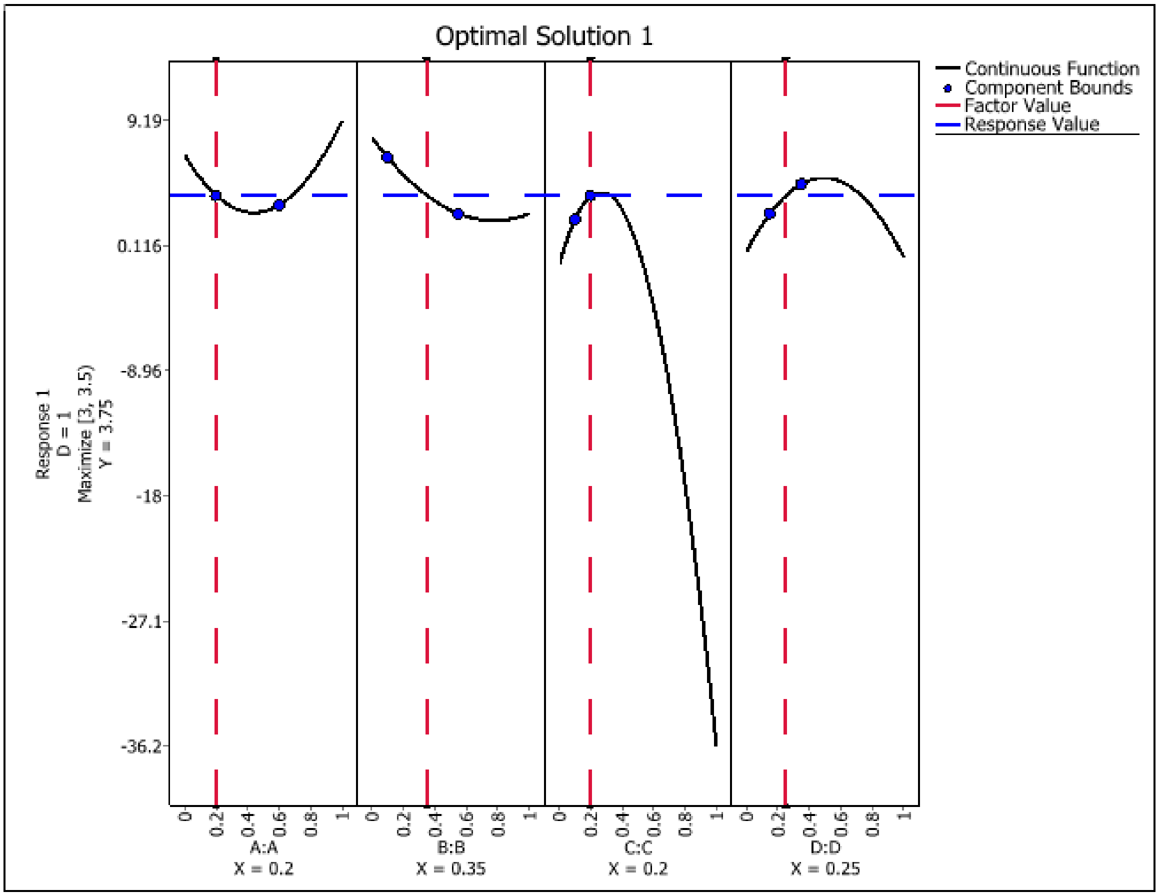
- Click the **Design - Optimization** icon on the control panel.



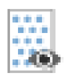
- In the Response Settings window that appears, use the settings shown next and click **OK**. These settings indicate that you want to maximize Response 1, with values above 3.5 being considered 100% desirable and values below 3.0 being undesirable.



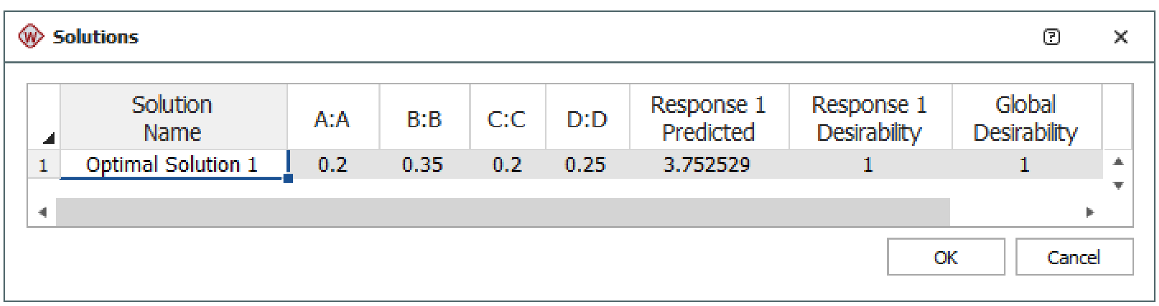
- An Optimization folio will appear with a plot showing the single solution that is found.



- You can choose **Optimization > Solutions > View Solutions** or click the icon on the control panel to see the solution in numerical format.



The optimum settings are shown next.



	Solution Name	A:A	B:B	C:C	D:D	Response 1 Predicted	Response 1 Desirability	Global Desirability
1	Optimal Solution 1	0.2	0.35	0.2	0.25	3.752529	1	1

CONCLUSIONS

The optimal solution is found to be A = 0.2, B = 0.35, C = 0.2 and D = 0.25. Under this setting, the expected strength of the material is 3.75. Keep in mind that it is necessary to conduct an experiment using this setting to confirm this conclusion.

Reliability DOE

Unlike other design types, reliability designs are specifically intended to handle life data. Only one response (typically failure time) is measured, but the designs can accommodate data sets that include suspensions (right censoring) and/or uncertainty as to when the units failed (interval and/or left censoring), in addition to complete data sets in which all of the units under test failed and the failure time for each unit is known.

In contrast to traditional DOE techniques, which assume that response values at any treatment level are normally distributed, reliability DOE (R-DOE) uses the Weibull, lognormal or exponential distribution to analyze data. You can publish the fitted distribution as a model for use in other applications, if desired. In addition, the kinds of results and plots available may differ from those that are available with DOE using standard response data.

Tip: To perform R-DOE, select any design type in a standard design folio and set the **Response Type** for one of the responses to **Life Data**. (See Adding, Removing and Editing Responses.)

This section explains how to perform reliability DOE, including:

The ReliaWiki resource portal provides more information about R-DOE at http://www.reliawiki.org/index.php/Reliability_DOE_for_Life_Tests.

Tip: If you wish to analyze life data from a prior experiment, considering using the free form folio, which does not require that you create an experiment design.

Available Reliability Designs

The available designs for reliability DOE are:

- **One factor reliability designs** allow you to test a single factor using from 2 to 20 levels to determine if the factor has an effect on the product's reliability. Note that the factor in a one factor design is treated as a qualitative factor. Therefore, predictions cannot be made for factor levels that are not tested, and the designs cannot be optimized.

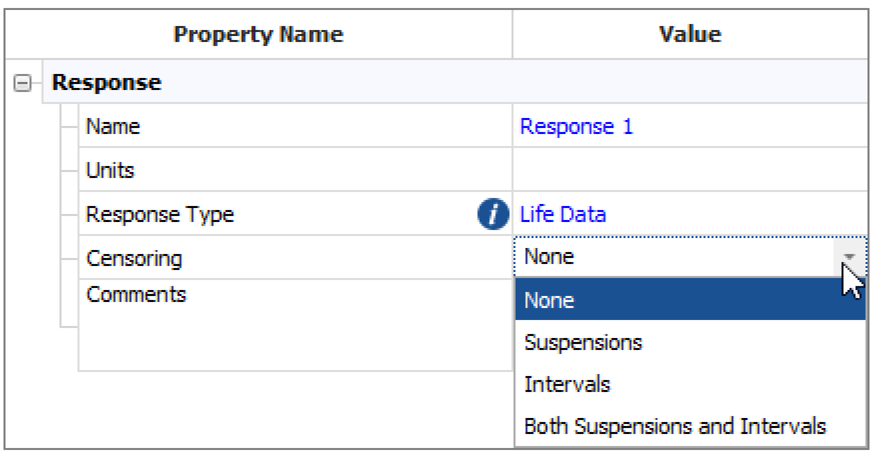
- **Factorial reliability designs** allow you to test multiple factors at two levels each to investigate the effect of the factors on the product's reliability.

- In factorial reliability design, only the linear effects of the quantitative factors are studied. **Response surface method (RSM) reliability designs** allow you to study the quadratic effects of the factors (i.e., effects that differ depending on the level of the factors), making them well-suited to predictive modeling and optimization. For example, RSM reliability designs can be used to determine the factor settings that will optimize the reliability of a component.

For more information about how to use the design types, please consult the documentation on design folios.

Reliability Design Data Types

Reliability designs allow you to enter life data (i.e., times-to-failure data) as the response. Depending on the censoring type you select, other information may also be included. When you are selecting the design settings in a design folio, you can select **Life Data** as the response type and then choose an option from the **Censoring** drop-down list on the Design tab. (See Adding, Removing and Editing Responses.)



The censoring option you choose will determine which of the following data types is used in the design:

- **Life Data with No Censoring (Complete Data):** 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. Complete data are shown next in the context of a one factor reliability design.

Time Failed	A:Materials
313	Type A
476	Type B
228	Type C
432	Type A
279	Type C
490	Type B

- **Life Data with Suspensions (Right Censored Data):** The data points for units that did not fail during the observation period are known as *suspensions* or *right censored data*, which means that the event of interest (i.e., the failure time) is to the right of our data point on the time scale. For example, if we tested six units and only four 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. Failure times with right censored data are shown next in the context of a one factor reliability design (e.g., the last data point is a suspension at time = 500).

State F or S	Time to F or S	A:Materials
F	476	Type C
F	228	Type A
F	432	Type B
S	500	Type C
F	294	Type A
S	500	Type B

- **Life Data with Intervals:** Interval censored data sets contain uncertainty as to when the units actually failed. For example, if five units under test are inspected every 100 hours, then the status of each unit (failed or still running) is known only at the time of each inspection. If a unit fails, it is known only that it failed between inspections and the exact time of failure is not known. Instead of an exact failure time, an interval of time (e.g., between 0 and 100 hours, or 100 and 200 hours) would be recorded. Interval data are shown next in the context of a factorial reliability design (e.g., the first data point indicates that a failure occurred at some time between 140 and 160 hours).

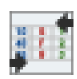
Last Inspected	State End Time	A:Temperature (K)	B:Fan Speed (rpm)
140	160	383	2000
80	100	383	2000
160	180	383	2000
180	200	333	1000
120	140	333	2000
100	120	333	1000

- **Life Data with Both Suspensions and Intervals:** This is the most general data type, as it includes all of the possible censoring types. Failure times with interval and right censored data type are shown next in the context of a factorial reliability design. (Note that the first data point is actually an exact failure time since it has an interval with identical start and end times.)

Last Inspected	State F or S	Time to F or S	A:Temperature (K)	B:Fan Speed (rpm)
140	F	140	383	2000
180	S	200	333	1000
140	F	160	333	1000
180	F	200	383	1000
80	F	100	383	1000
80	F	100	383	1000

Alter Data Type Window

The Alter Data Type window is used to specify the type of life data response (e.g., right censored data) that is used in the current experiment. It is available only for designs that are configured to accept life data. To open the window, choose **Data > Design > Alter Data Type** or by click the icon on the Data tab control panel.



The following options are available in the **Life Data Response Type** area of the Alter Data Type window.

- Select **My data set contains suspensions (right censored data)** if the response data includes times at which some test units were observed as not being failed.
- Select **My data set contains interval and/or left censored data** if the response data includes intervals during which a failure/suspension is known to have occurred.

The area at the bottom of window shows the response columns that will be included on the Data tab, given the selections in the **Life Data Response Type** area.

Publishing Models from R-DOE Results

You can publish reliability DOE (R-DOE) results as a model resource for use throughout the project. To publish a model based on R-DOE results, click the **Publish Model** icon on the Publishing page of the design folio or free form folio control panel. (Note that the Publishing page is available only when the selected response is configured as life data.)

At this point, you need to provide the factor settings that will be used to solve for the scale parameter of the published model. In the window that appears, choose the tool that will solve for the scale parameter. Two tools are available:

- The Prediction window lets you directly enter the factor level combination. Just enter the factor values in the green factor columns, perform the prediction and click the **Publish Model** button on the control panel. The resulting model will be based on the factor level combination you selected in the table. (To select a combination, click any cell in the row that contains that combination.)
- The Optimal Solution Plot helps you find the factor level combination that produces the best response value(s). After you perform the optimization, select the solution you wish to use in the **Solutions** area on the control panel, then click the **Publish Model** button on the control panel.

After you choose to publish the prediction or optimal solution, the Publish Model window will appear, allowing you to specify the model name and the model category. Click **OK** to publish the model.

All other tools for working with the published model, such as displaying its properties and tracing its usage, are on the Publishing page of the control panel for the R-DOE analysis.

One Factor Reliability Designs: Example

*The data set used in this example is available in the example database installed with the software (called "Weibull21_DOE_Examples.rsgz21"). 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 example project is "Reliability DOE - One Factor Reliability DOE."

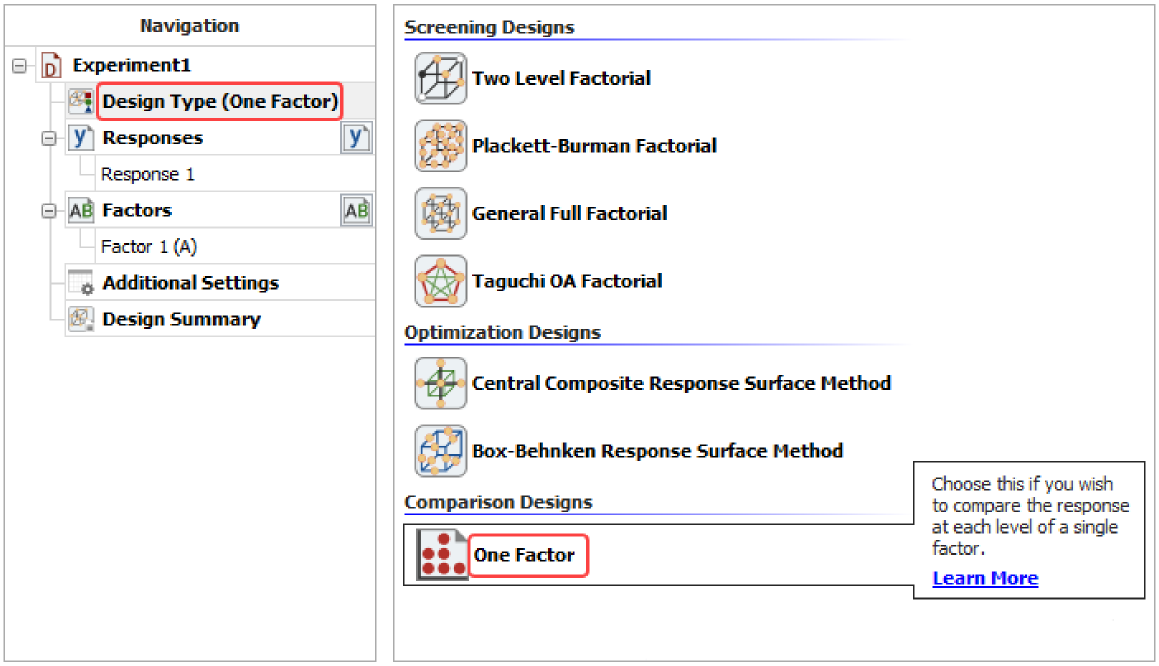
In this example, one factor reliability DOE is used to determine if there is a difference in three different materials that can be used in a product and, if there is, to determine which material is the best in terms of the product life. 10 units were tested for each material, and the test stopped at 500 hours. The product's life is assumed to follow the Weibull distribution.

Designing the Experiment

The design matrix and the response data are given in the "3 Levels One Factor" folio. The following steps describe how to create this folio on your own.

- Choose **Home > Insert > Standard Design** to add a standard design folio to the current project.

- Click **Design Type** in the folio's navigation panel, and then select **One Factor** in the input panel.



- Specify that you will be performing reliability DOE by clicking **Response 1** in the navigation panel and choosing **Life Data** from the **Response Type** drop-down list. Then choose **Suspensions** from the **Censoring** drop-down list to specify that some of the data points will be suspensions (i.e., times at which the unit was found not failed).

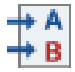
- Define the factor by clicking **Factor 1** in the navigation panel and editing its properties in the input panel.

  - Factor type: **Qualitative**
  - Number of levels: **3**
  - Level 1: **Type A**
  - Level 2: **Type B**
  - Level 3: **Type C**

- Rename the folio by clicking the **Experiment1** heading in the navigation panel and entering **3 Levels One Factor** for the **Name** in the input panel.

- Click the **Additional Settings** heading. In the input panel, set the number of **Replicates** to **10** (since there are ten units tested for each material).

- Finally, click the **Build** icon on the control panel to create a Data tab that allows you to view the test plan and enter response data.

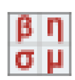


Analysis and Results

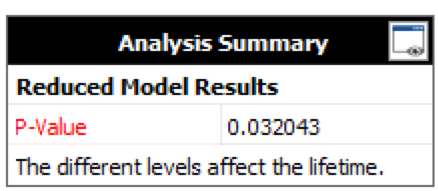
The data set for this example is given in the "3 Levels One factor" folio of the example project. After you enter the data from the example folio, you can perform the analysis by doing the following:

Note: To minimize the effect of unknown nuisance factors, the run order is randomly generated when you create the design. Therefore, if you followed these steps to create your own folio, the order of runs on the Data tab may be different from that of the folio in the example file. This can lead to different results. To ensure that you get the very same results described next, show the Standard Order column in your folio, then click a cell in that column and choose **Sheet > Sheet Actions > Sort > Sort Ascending**. This will make the order of runs in your folio the same as that of the example file. Then copy the response data from the example file and paste it into the Data tab of your folio.

- In the **Distribution** drop-down list on the Data tab control panel, select to use the **Weibull** distribution for the analysis.
- Click the **Calculate** icon on the Data tab control panel.



- The result in the **Analysis Summary** area show that the *p* value is 0.032. In other words, there is a 3.2% chance that the variation would be observed in the case when there is no significant difference between the response levels. Since the *p* value is less than the risk level (10%) entered on the Analysis Settings tab of the control panel, the conclusion is that the different factor levels (i.e., materials) do affect the response.

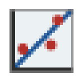


- To see more detailed comparisons of the responses at different factor levels, click the **View Analysis Summary** icon and view the Life Comparisons table.

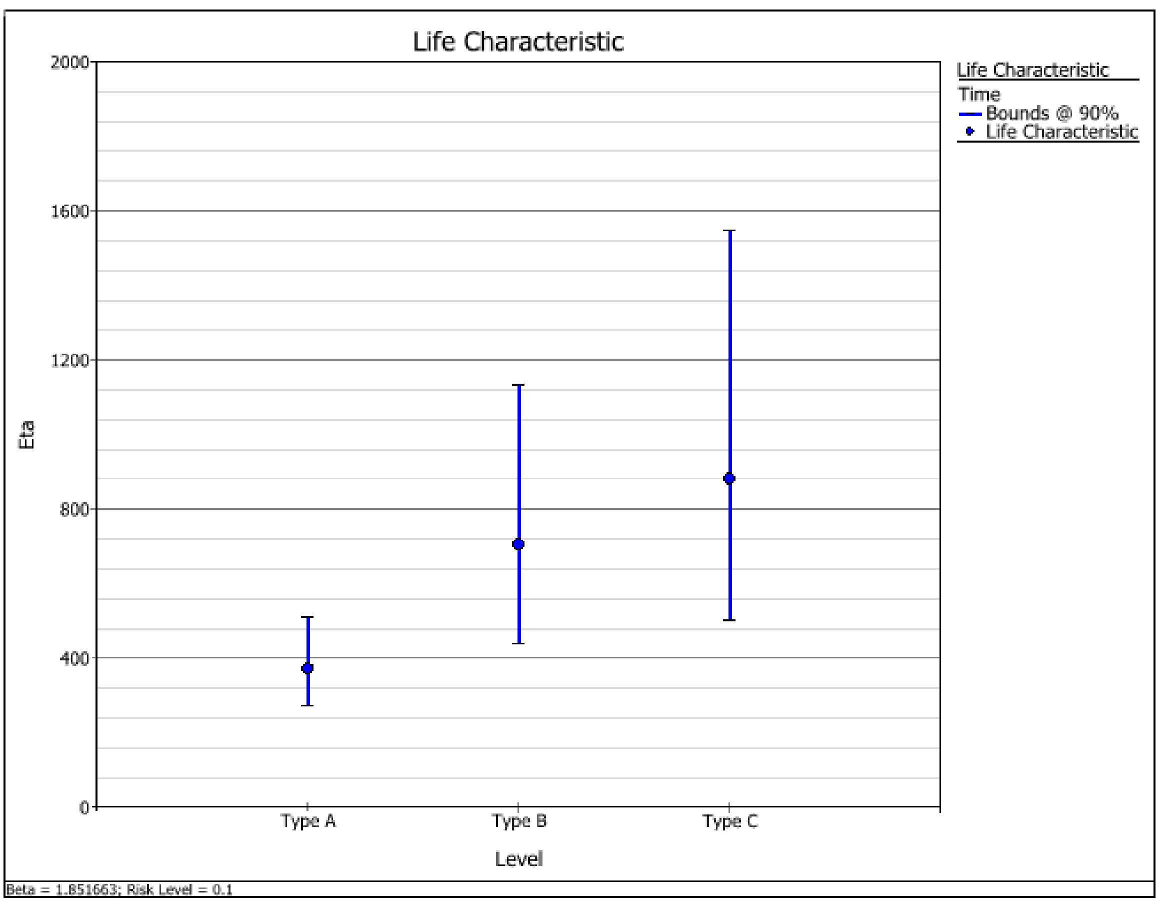
Life Comparisons						
Contrast	Ln(Eta) Difference	Pooled Standard Error	Low Confidence	High Confidence	Z Value	P Value
Type A - Type B	-0.636789	0.347428	-1.208381	-0.065196	-1.832866	0.066823
Type A - Type C	-0.858691	0.393927	-1.506785	-0.210597	-2.179819	0.029271
Type B - Type C	-0.221902	0.414426	-0.90372	0.459916	-0.535445	0.592342

The compared factor levels are listed in the Contrast column (e.g., "Type A - Type B" is the comparison of those two materials). When a *p* value is red, the compared levels are significantly different. For example, there is a significant difference between the Type A and Type B materials, but there is no significant difference between Type B and Type C.

- To see a plot that shows how the product life differs for each material, click the **Plot** icon.



Then view the life characteristic plot.



This plot shows the calculated life characteristic for each type of material. The tick marks at the top and bottom of the vertical lines mark the two-sided confidence bounds on the life characteristic. Since the Weibull distribution was used to calculate the data, the life characteristic is eta (i.e., the time at which unreliability = 63.2%).

Conclusions

The analysis showed that the type of material used does affect the product life. From the life characteristic plot, you can see that the Type C material has the largest eta value and therefore the longest expected life.

Factorial Reliability Designs: Example

*The data set used in this example is available in the example database installed with the software (called "Weibull21_DOE_Examples.rsgz21"). 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 example project is "Reliability DOE - Life Time Test of Fluorescent Lights."

In this example, two level factorial reliability DOE is used to determine the best factor settings to improve the reliability of fluorescent light bulbs. Five two-level factors were used in a fractional design to investigate the main effects of the factors and the interaction effect of the first two factors. It is assumed that none of the other interaction effects are significant and that the life of the bulb follows a lognormal distribution.

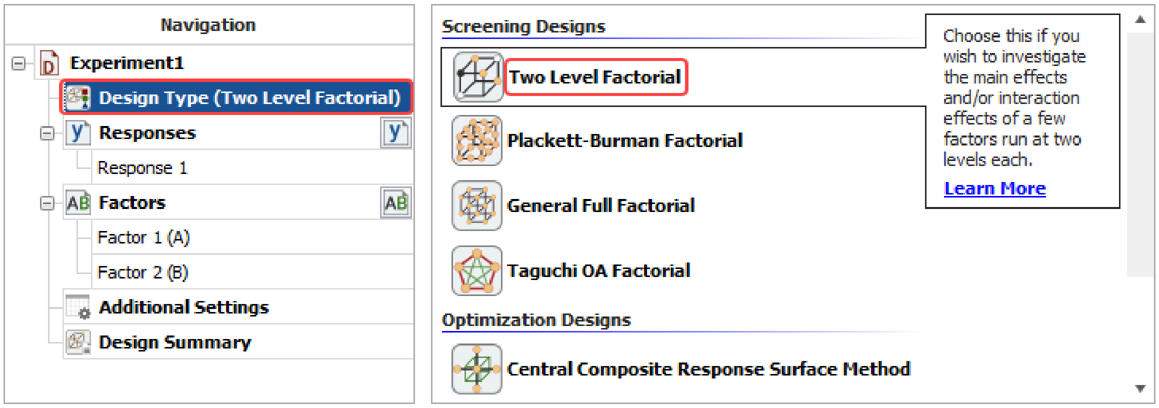
Each treatment in the design has two replicates (i.e., two bulbs were tested at each factor level combination), and the experiment was conducted over 20 days with inspections every two days. (The failure times were short because the lights were subjected to an accelerating factor that stressed the lights at higher than normal conditions.) Some of the units were still not failed at the end of the 20-day test.

Designing the Experiment

The design matrix and the response data are given in the "Fluorescent Light Life Test" folio. The following steps describe how to create this folio on your own.

- Choose **Home > Insert > Standard Design** to add a standard design folio to the current project.

- Click **Design Type** in the folio's navigation panel, and then select **Two Level Factorial** in the input panel.



- Specify that you will be performing reliability DOE by clicking **Response 1** in the navigation panel and choosing **Life Data** from the **Response Type** drop-down list. Then choose **Both Suspensions and Intervals** from the **Censoring** drop-down list to specify that some of the data points will be suspensions (i.e., times at which the unit was found not failed) and all of the times-to-failure data will be entered as intervals (since the units are inspected only once every two days).

- Specify the number of factors by clicking the **Factors** heading in the navigation panel and choosing **5** from the **Number of Factors** drop-down list.

- Define each factor by clicking it in the navigation panel and editing its properties in the input panel. In this example, the default labels are used for all the factors. (See Adding, Removing and Editing Factors.)

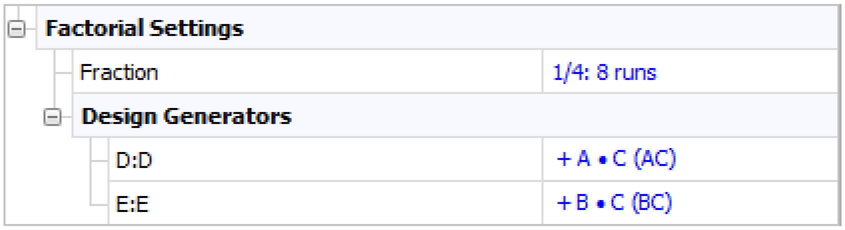
- Rename the folio by clicking the **Experiment1** heading in the navigation panel and entering **Fluorescent Night Life Test** for the **Name** in the input panel.

- Click the **Additional Settings** heading to choose the number of replicates, specify the kind of fractional design that will be used and define the generators (i.e., aliased effects).

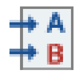
  - Set the number of **Replicates** to **2** (since there are 2 units tested for each factor level combination).

  - Under the **Factorial Settings** heading, choose **1/4: 8 runs** from the **Fraction** drop-down list. This means 8 runs will be used for each replicate of the design, instead of the 32 runs that would be required for a full factorial design.

  - Since a fractional design is being used, you must choose **Design Generators** for the last two factors. Since it is assumed that the interaction effects AC and BC are not significant, you choose to alias the main effect of D with the interaction effect AC, and the main effect of E with the interaction effect BC, as shown next.



- Finally, click the **Build** icon on the control panel to create a Data tab that allows you to view the test plan and enter response data.



Analysis and Results

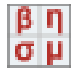
The data set for this example is given in the "Fluorescent Light Life Test" folio of the example project. After you enter the data from the folio, you can perform the analysis by doing the following:

Note: To minimize the effect of unknown nuisance factors, the run order is randomly generated when you create the design. Therefore, if you followed these steps to create your own folio, the order of runs on the Data tab may be different from that of the folio in the example file. This can lead to different results. To ensure that you get the very same results described next, show the Standard Order column in your folio, then click a cell in that column and choose **Sheet > Sheet Actions > Sort > Sort Ascending**. This will make the order of runs in your folio the same as that of the example file. Then copy the response data from the example file and paste it into the Data tab of your folio.

- In the **Distribution** drop-down list on the Data tab control panel, select to use the **Lognormal** distribution for the analysis.
- Make sure all main effects and the interaction effect AB will be considered. To do this, click the **Select Terms** icon on the control panel.

In the window that appears, select the **Main Effects** and **A·B** check boxes. Then click **OK**.

- On the Analysis Settings page of the control panel, enter a **Risk Level** of **0.5** and choose to analyze **Individual Terms**.
- Return to the Main page of the control panel and click the **Calculate** icon.



- The results in the **Analysis Summary** area show that second, fourth and fifth factors have a significant effect on the bulb's life.

Analysis Summary	
Significant Terms	
Term	**Coefficient**
B:B	-0.201466
D:D	-0.272858
E:E	0.152712

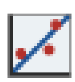
- To see the coefficients for all the terms in the regression model, click the **View Analysis Summary** icon and view the MLE Information table.

MLE Information							
Term	**Effect**	**Coefficient**	**Standard Error**	**Low Confidence**	**High Confidence**	**Z Value**	**P Value**
Std		0.15887	0.042521	0.09401	0.268481		
Intercept		2.939208	0.064308	2.81314	3.065277	45.705279	0
A:A	0.233575	0.116788	0.061844	-0.004451	0.238027	1.88842	0.05897
B:B	-0.402933	-0.201466	0.060374	-0.319823	-0.08311	-3.336974	0.000847
C:C	-0.102032	-0.051016	0.061642	-0.171859	0.069827	-0.827615	0.407889
D:D	-0.545716	-0.272858	0.062129	-0.394656	-0.151061	-4.391791	0.000011
E:E	0.305423	0.152712	0.062603	0.029984	0.275439	2.439348	0.014714
A • B	-0.097511	-0.048755	0.061848	-0.170001	0.072491	-0.788312	0.430514

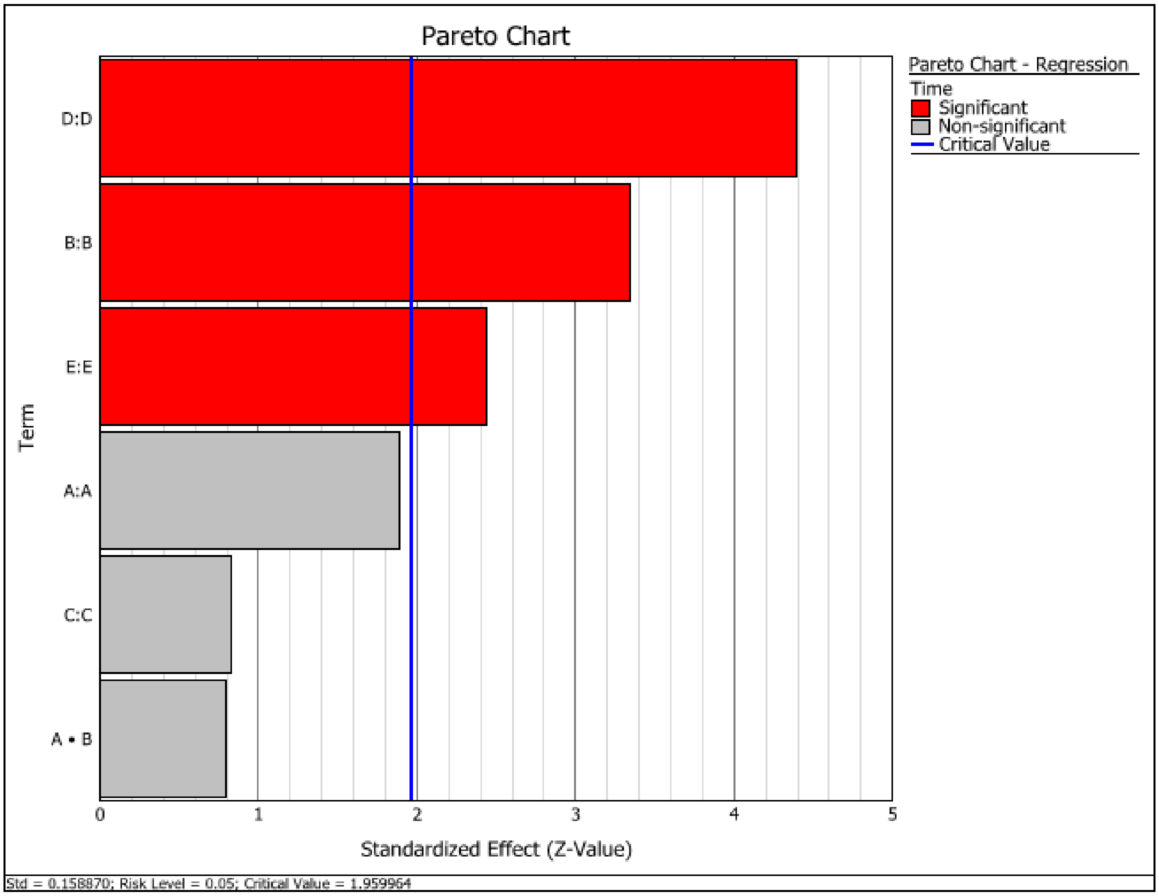
From the MLE Information table, you can see the model for the ln-mean or the scale parameter in the lognormal distribution is:

$$\mu = 2.9392 + 0.1168A - 0.2015B - 0.051\mathrm{C} - 0.2729D + 0.1527E - 0.0488AB$$

- To view a plot comparing the standardized effect of each term, click the **Plot** icon.



Then choose **Pareto Charts - Regression** from the **Plot Type** drop-down list. The following plot appears.



The horizontal blue line in the plot marks the critical value determined by the risk level specified on the Analysis Settings page of the control panel. If the bar goes past the blue line, then the effect is considered significant.

Conclusion

From the regression model, we can determine that in order to improve the reliability, factors B, C and D should be set to their low (-1) level, indicated by their negative coefficients, while A and E should be set at their high (+1) level, indicated by their positive coefficients. Under this setting, the predicted scale parameter in the lognormal distribution is 3.7829. Therefore, the life distribution under this factor setting is a lognormal distribution with a standard deviation of 0.1589 and ln-mean of 3.7829.

To do more advanced analysis, you can enter the data into an Accelerated Life Testing folio for accelerated life data analysis to further investigate the life stress relationship. Since factors A and C are not shown to be significant, they can be removed in further analysis.

Response Surface Method Reliability Designs: Example

*The data set used in this example is available in the example database installed with the software (called "Weibull21_DOE_Examples.rsgz21"). 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 example project is "Reliability DOE - Optimization for a Single Response."

In this example, two factors in a manufacturing process, curing temperature and curing time, have a significant effect on the life of a component. A central composite design is used to investigate the relationship between these factors and the life of the component. Life is defined as cycles-to-failure under normal use conditions. The warranty period is 1,000 cycles.

5 units are available for testing, and the factors and levels shown next will be used.

Factor	Low Level	High Level
Temperature (degrees C)	80	100
Time (min)	300	500

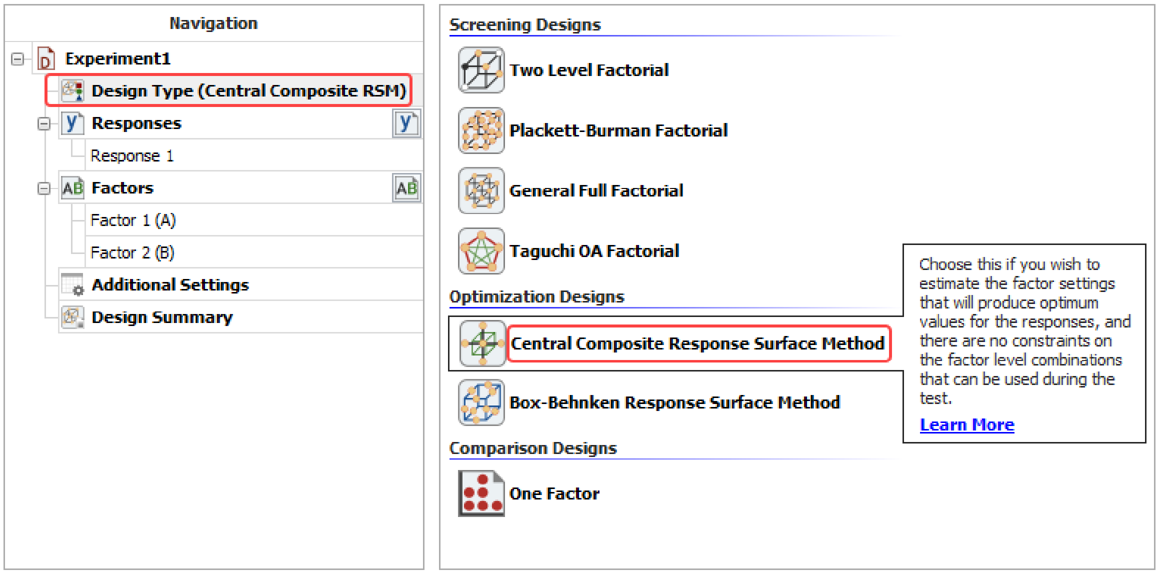
A reasonable reliability should be achieved for the warranty period. If the reliability is too low, the warranty cost will be high. If the product is overdesigned or too reliable, then customers will not replace their old components, which will negatively affect sales. The target reliability is decided to be 0.95 and values below 0.92 or above 0.98 are considered unacceptable. The goal of the experiment is to find the optimal settings of temperature and time.

Design the Experiment

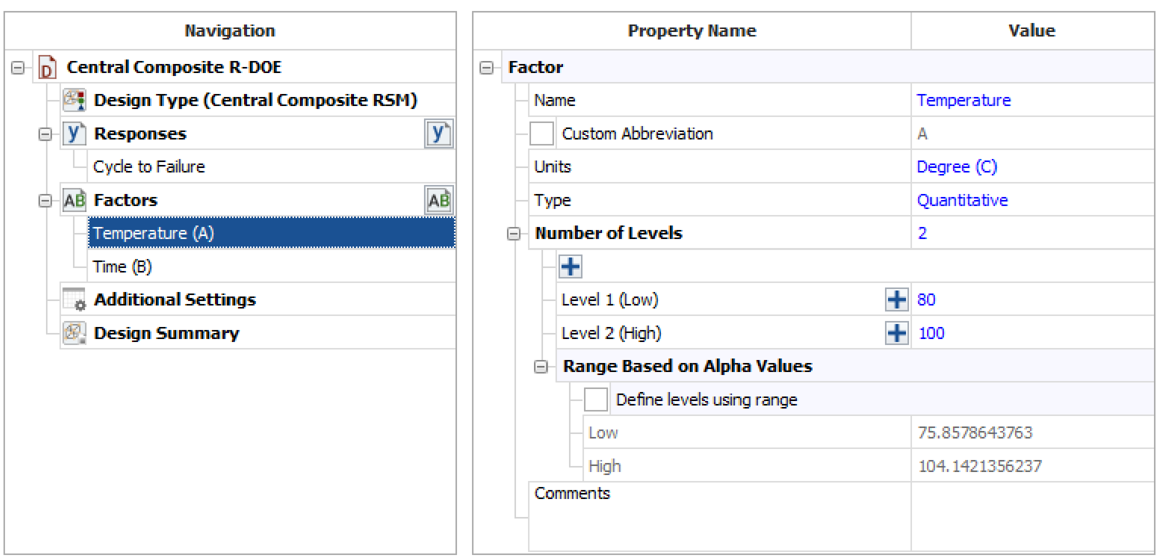
The design matrix and the response data are given in the "Central Composite R-DOE" folio. The following steps describe how to create this folio on your own.

- Choose **Home > Insert > Standard Design** to add a standard design folio to the current project.

- Click **Design Type** in the folio's navigation panel, and then select **Central Composite Response Surface Method** in the input panel.

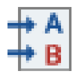


- Rename the folio by clicking the **Experiment1** heading in the navigation panel and entering **Central Composite R-DOE** for the **Name** in the input panel.
- Specify that you will be performing reliability DOE by clicking **Response 1** in the navigation panel and choosing **Life Data** from the **Response Type** drop-down list. Then choose **Suspensions** from the **Censoring** drop-down list to specify that some of the data points will be suspensions (i.e., times at which the unit was found not failed). Rename the response to **Cycle to Failure**.
- Define each factor by clicking it in the navigation panel and editing its properties in the input panel. The first factor is shown next.



See Adding, Removing and Editing Factors.

- Click the **Additional Settings** heading to choose the number of replicates, specify the kind of fractional design that will be used and define the generators (i.e., aliased effects).
  - Set the number of **Replicates** to **5** (since there are 5 units tested for each factor level combination).
  - Choose **Yes** from the **Block on Replicates** drop-down list. This specifies that each replicate will be assigned its own block in the design.
- Finally, click the **Build** icon on the control panel to create a Data tab that allows you to view the test plan and enter response data.

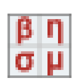


Analysis and Results - Part 1

The data set for this example is given in the "Central Composite R-DOE" folio of the example project. After you enter the data from the folio, you can perform the analysis by doing the following:

Note: To minimize the effect of unknown nuisance factors, the run order is randomly generated when you create the design. Therefore, if you followed these steps to create your own folio, the order of runs on the Data tab may be different from that of the folio in the example file. This can lead to different results. To ensure that you get the very same results described next, show the Standard Order column in your folio, then click a cell in that column and choose **Sheet > Sheet Actions > Sort > Sort Ascending**. This will make the order of runs in your folio the same as that of the example file. Then copy the response data from the example file and paste it into the Data tab of your folio.

- On the Analysis Settings page of the control panel, select to use **Individual Terms** in the analysis.
- Return to the Main page of the control panel and click the **Calculate** icon.



Click the **View Analysis Summary** icon on the control panel and select to the view the MLE Information table.

MLE Information							
Term	Effect	Coefficient	Standard Error	Low Confidence	High Confidence	Z Value	P Value
Beta		2.040932	0.190486	1.750418	2.379663		
Intercept		5.28804	0.101905	5.120385	5.455695	51.891877	0
Block[1]		0.038387	0.133357	-0.181013	0.257788	0.287854	0.773459
Block[2]		-0.012788	0.127015	-0.221755	0.196179	-0.100682	0.919803
Block[3]		-0.1563	0.126028	-0.363642	0.051042	-1.240201	0.214901
Block[4]		0.128926	0.12446	-0.075836	0.333688	1.035885	0.300256
A:Temperature	-1.612815	-0.806408	0.080138	-0.938251	-0.674564	-10.06275	0
B:Time	1.399946	0.699973	0.080782	0.56707	0.832876	8.664969	0
A • B	-0.122902	-0.061451	0.11033	-0.242968	0.120066	-0.556973	0.577546
A • A	-0.03002	-0.01501	0.090778	-0.164359	0.134339	-0.165346	0.868671
B • B	1.927736	0.963868	0.091625	0.813126	1.11461	10.519748	0

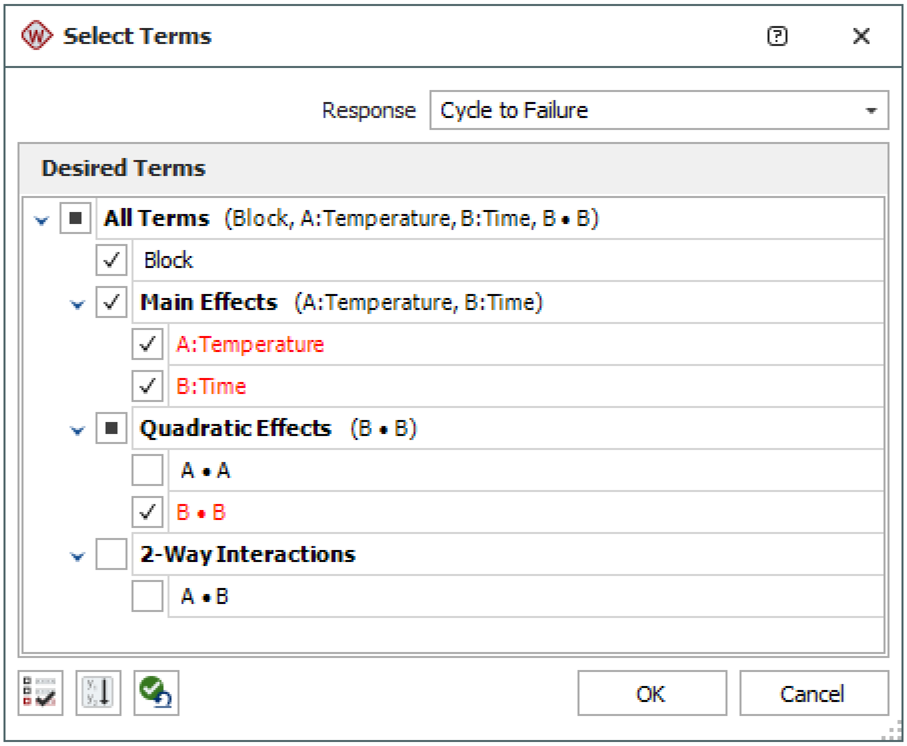
From this table, you can see that effects A and B, as well as the quadratic effect of B (i.e., B·B), are significant. To simply the analysis, the next step is to recreate the model using only the significant terms.

Analysis and Results - Part 2

The results for the reduced model are given in the "Reduced Model" folio of the example project. The following steps describe how to create this folio on your own in order to generate the regression model that will be used for the optimization.

- Right-click the design folio in the current project explorer and select **Duplicate Item** from the shortcut menu. Rename the new folio with an appropriate name.
- Click the **Select Terms** icon on the new folio's control panel.

- In the Select Terms window that appears, click the **Select Significant Effects** button to select only the significant effects to calculate the new model, as shown next, then click **OK**.



- Click the **Calculate** icon in the new folio's control panel.
- To see the details of the reduced model, view the MLE Information table.

MLE Information							
Term	Effect	Coefficient	Standard Error	Low Confidence	High Confidence	Z Value	P Value
Beta		2.020303	0.191407	1.728713	2.361076		
Intercept		5.283772	0.084937	5.144034	5.423511	62.208313	0
A:Temperature	-1.612258	-0.806129	0.077849	-0.934207	-0.678051	-10.355009	0
B:Time	1.417919	0.70896	0.079131	0.578773	0.839146	8.959351	0
B • B	1.936327	0.968163	0.09004	0.820028	1.116299	10.752535	0

Optimization

The optimal factor settings are available in the "Optimization" folio of the example project. Follow the steps outlined below to create the folio from scratch and solve for the optimal settings.

- On the control panel of the "Reduced Model" standard design folio, click the **Optimization** icon.

- The Optimization Settings window will appear. Click the cell in the Goal column and choose **Target** from the drop-down list. Then, enter the target response value, as well as the lower and upper limits on the value.

  When you enter target values to optimize a life data response, the values must be entered in terms of ln(eta) for the Weibull distribution. From the MLE Information table, we know that beta = 2.0203. If you assume that beta is the same under different factor values and that only the scale parameter is affected by the factor values, then you can use the required reliability value to calculate the required eta.

  The reliability function for a Weibull distribution is:

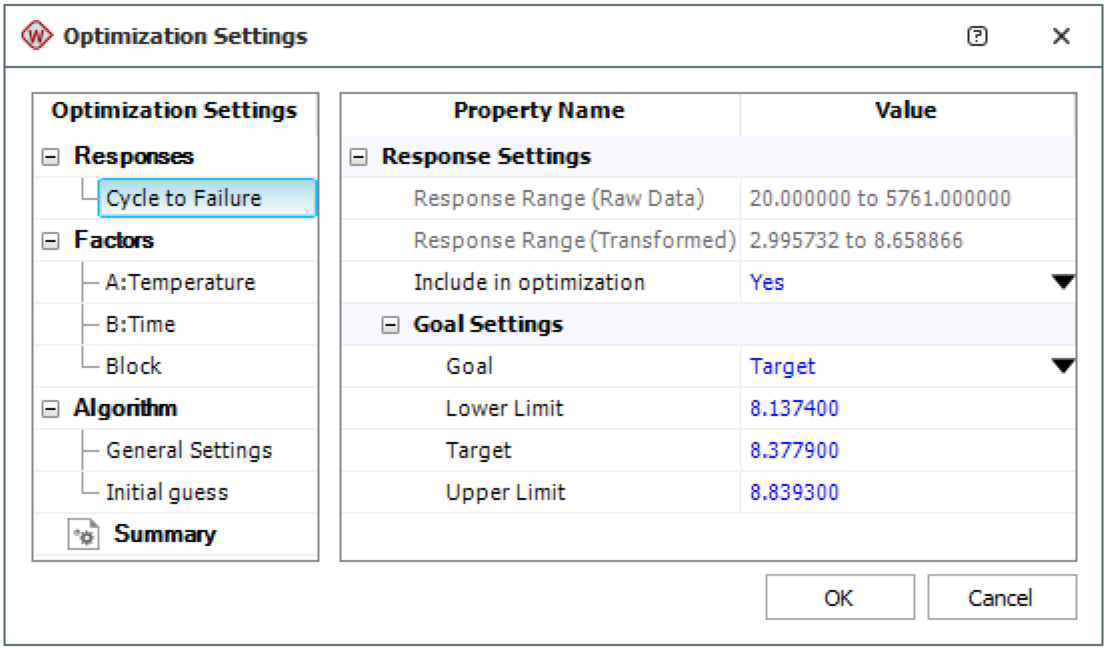
$$R(t) = e^{-\left(\frac{t}{\eta}\right)^{\beta}}$$

  Therefore, since the target reliability is 0.95, the target eta is 4,350. Similarly, the lower (0.92) and upper (0.98) limits on the reliability are equivalent to eta values of 3,420 and 6,900, respectively.

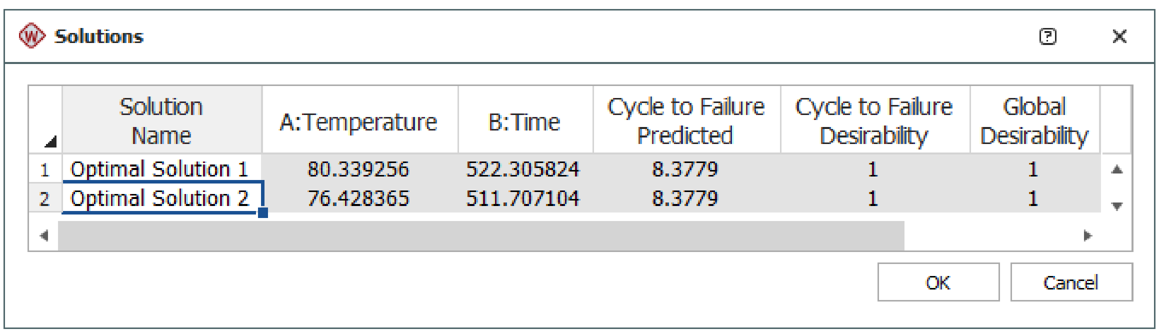
  In the software, we assume the logarithmic transform of eta is a linear function of all the effects. For this example, it is:

$$\ln(\eta) = \alpha_0 + \alpha_1 A + \alpha_2 B + \alpha_{22} B^2$$

  So the logarithmic transform is used in the optimization. In this case the lower limit is 8.1374, the target value is 8.3779 and the upper limit is 8.8393. After entering the values, the window will appear as shown next.



- Click **OK** to calculate the optimal solutions. The second solution will be shown in the plot. To view all the numerical solutions at once, click the **View Solutions** icon () at the bottom of the control panel. Two optimal solutions will be shown.



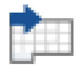
In this example, optimal solution 2 uses the shortest curing time and lowest temperature among both solutions. Therefore, it can be used as the final recommended factor setting.

Free Form Folio

The free form folio allows you to analyze data for traditional and reliability DOE by manually entering the factor level combinations and observed response values from a prior experiment. After you designate columns for factors and responses, you can then analyze the data in much the same way that it is analyzed on the Data tab of a design folio. You can also use this folio to perform predictions and optimize responses.

Note: In general, the free form folio is useful for analyzing existing data, rather than planning an experiment. To plan an experiment, use the design folio.

To add a free form folio to your project, choose **Home > Insert > Free Form**, or right-click the **DOE** folder on the current project explorer and choose **Add Free Form** on the shortcut menu.



Free Form Folio: Control Panel

The free form folio control panel allows you to control the settings for analyzing response data and displays the results of the analysis. The panel includes the following pages:

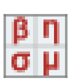
- The **Main Page** contains most of the tools you will need to analyze response data.
- The **Analysis Settings Page** allows you to set the risk level for the analysis.
- The **Comments Page** allows you to enter notes or other text that will be saved with the folio.
- If you selected to analyze a life data response, the **Identifiers** and **Publishing** pages will also be available. These pages allow you to publish the results as a model.

Main Page

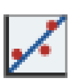
The Main page of the control panel includes the following tools and areas.

FOLIO TOOLS

The folio tools are arranged on the left side of the control panel. Use these tools to configure your analysis and experiment with the analysis results. Depending on the type of design you are working with, the control panel may contain some or all of the following tools:



Calculate analyzes the data for each response that is selected to be included in the analysis. To exclude a response from the analysis, clear the check box in its column heading.


Plot creates a plot based on the analysis. If you click this icon before the current data set has been analyzed, an analysis will be performed automatically.


Modify Design opens a window that allows you to specify the properties for the factors and responses in the analysis.

Select Terms opens a window that allows you to specify the terms (i.e., factors, factorial interactions, blocks and quadratic effects) to be considered for each response.

View Analysis Summary opens a window that contains detailed information about analysis results.

Optimization adds an optimal solution plot to the Optimization folio that is linked to the design folio. If no linked Optimization folio exists, the software will create one.

Overlaid Contour Plot adds an overlaid contour plot to the Optimization folio that is linked to the folio. If no linked Optimization folio exists, the software will create one.

Select Transformation opens a window that allows you to select a transformation to apply to each response.

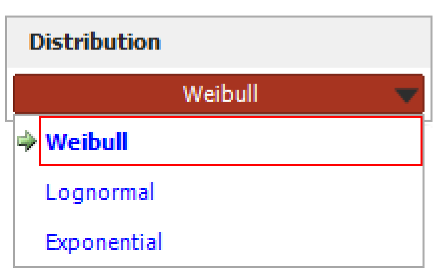
Prediction opens a window that allows you to enter values for each factor and see predicted results for the selected response.

SETTINGS AREA

The Main page of the control panel may also contain some or all of the following settings.

- The **Response** drop-down list is used to select the response that all the displayed settings and analysis results will apply to.

- The **Distribution** area is used to select a life distribution for reliability DOE. It is available when the selected response is configured as life data on the Data tab.



- The **Settings** area is used to view the available analysis settings. An example is shown next.

Settings	
Partial SS	Grouped Terms
Risk Level: 0.1	
Transformation: Y' = Y	
Analyzed	
12 Observations	

In this example:

- The partial sum of squares ("Partial SS") and grouped terms are used.
- The specified risk level is 10%.
- No transformation is used (i.e., the "transformed" response, *Y'*, is equal to the non-transformed response, *Y*). To configure this setting, you can click the text or the Select Transformation icon in the folio tools.
- "Analyzed" is shown in green, which indicates that the current response data has been analyzed using the current settings. Otherwise, this would show "Modified" in red.
- The last row indicates that the analyzed data consists of 12 observations.

To configure the settings used in the analysis, click the blue text. These settings are also accessible on the Analysis Settings page of the control panel.

ANALYSIS SUMMARY AREA

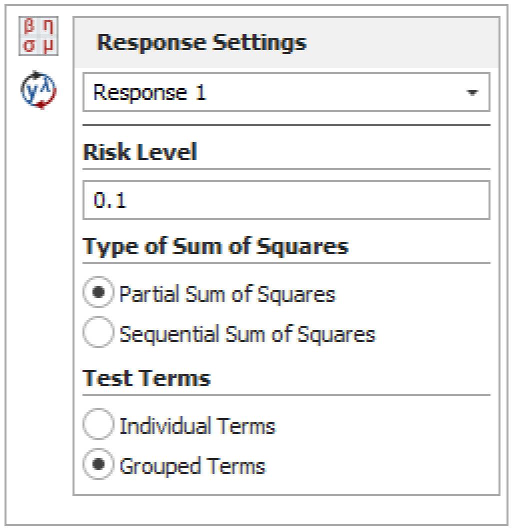
The **Analysis Summary** area is shown when the current response data has been analyzed using the current settings. This area displays the terms that were found to be significant, as well as their associated regression coefficients.

Analysis Summary	
Significant Terms	
Term	Coefficient
B:Nozzle Position	0.086271
D:Deposition Time	0.245021
C • D	-0.172521

Click the **View Analysis Summary** icon in this area to view the details of the analysis results (including an ANOVA table).

Analysis Settings Page

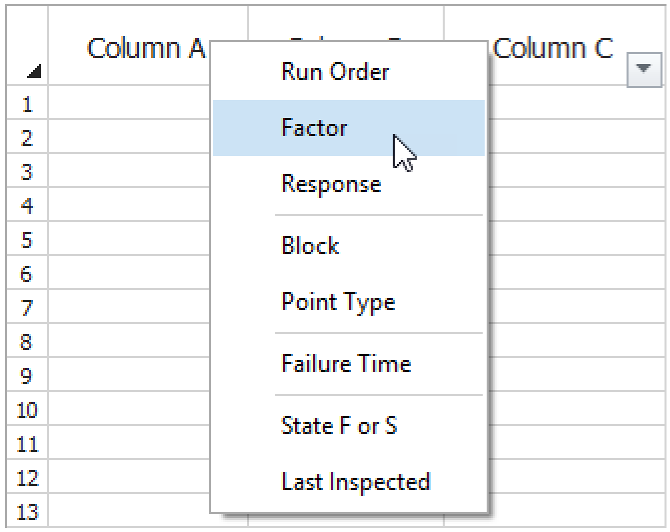
The Analysis Settings page of the free form folio control panel includes additional settings that may be applicable for the current analysis. It may contain some or all of the following options (which can also be changed on the Main page of the control panel).



- The **Response Settings** drop-down list is used to choose the response that all the visible settings and analysis results will apply to. When you choose a response here, it will automatically be chosen when you return to the Main page.
- The **Risk Level**, or alpha value, is a measure of the risk that the analysis results are incorrect (i.e., alpha = 1 - confidence level).
- The next area allows you to specify the **Type of Sum of Squares** to use in the analysis. Select the **Partial Sum of Squares** option to test if a term is significant given that all other terms are considered in the model. Select the **Sequential Sum of Squares** option to test if an additional term is significant given that the terms before it are already in the model.
- The **Test Terms** area specifies what the information in the ANOVA table in the free form folio analysis results applies to. If you select **Individual Terms**, the software will separately examine the effect of each individual predictor. If you select **Grouped Terms**, it will treat all the predictors as a single group.

Free Form Folio: Data Tab

The Data tab in the free form folio includes columns that can be configured to record different kinds of information. To configure a column, click the arrow in the column heading and choose an option from the menu that appears.



The following kinds of information can be recorded in each column. To calculate, there must be at least one Factor column and at least one Response or Failure Time column. All the possible columns are described next.

- **Run Order** columns are used to record the order in which each test run is performed.
- **Factor** columns are used to record the factor levels for each factor used in the test.
- **Response** columns are used to record the response values obtained during each test run. These columns are used for standard response data (i.e., they are not used for the purposes of reliability DOE).
- **Block** columns are used to identify the block that each test run belongs to.
- **Point Type** columns are used to identify the kind of data point that is obtained at the given test run. Enter 0 if the data point is a center point and 1 if it is not.
- **Failure Time** columns are used for reliability DOE. This column records the test unit's exact failure/suspension time or, when a Last Inspected column is included, the time at which a unit was found to be failed/suspended since its last inspection.
- **State F or S** columns are used for reliability DOE, when the times in the Failure Time column include suspensions. Enter F if the time is a failure and S if it is a suspension.
- **Last Inspected** columns are used when the test runs include interval data for reliability DOE. The Last Inspected time is the beginning of the interval and the Failure Time column is used for the observation time. For example, if a test unit was found failed at 100 hours and was last inspected at 80 hours, then you'd enter 80 in the Last Inspected column and 100 in the Failure Time column.

Free Form Folio: Analysis Results

When accessed from a free form folio, the Analysis Summary window will contain detailed information about analysis results, including information that describes how each factor and factorial interaction affects the response that is currently selected on the control panel.

If the current response data has been analyzed, you can open the window by clicking the **View Analysis Summary** icon on the control panel.

If the current response data has not been analyzed, the icon will still be available so you can view the folio's analysis history.

Select an item in the Available Report Items panel to display it on the spreadsheet. Each item is described next.

Analysis Results

The **Analysis of Variance (ANOVA) table** and the **Likelihood table** provide general information about the effects of the factors and factorial interactions on the selected response. This information may be presented for individual factors and interactions or for groups of factors and interactions, depending on your analysis setting on the control panel.

ANOVA Table Columns

- **Source of Variation** is the source that caused the difference in the observed output values. This can be a factor, factorial interaction, curvature, block, error, etc. If your design includes more than one factor and you have selected to use grouped terms in the analysis (specified on the Analysis Settings page of the control panel), the effects will be grouped by order (i.e., main effects, two-way interactions, etc.). Sources displayed in red are considered to be significant.

- The number of **Degrees of Freedom** for the **Model** is the number of regression coefficients for the effects included in the analysis (e.g., two coefficients might be included in the regression table for a given main effect). The number of degrees of freedom for the **Residual** is the total number of observations minus the number of parameters being estimated.

- **Sum of Squares** is the amount of difference in observed output values caused by this source of variation.

- **Mean Squares** is the average amount of difference caused by this source of variation. This is equal to Sum of Squares/Degrees of Freedom.

- **F Ratio** is the ratio of Mean Squares of this source of variation and Mean Squares of pure error. A large value in this column indicates that the difference in the output caused by this source of variation is greater than the difference caused by noise (i.e., this source affects the output).

- **P Value** (alpha error or type I error) is the probability that an equal amount of variation in the output would be observed in the case that this source does not affect the output. This value is compared to the risk level (alpha) that you specify on the Analysis Settings page of the control panel. If the *p* value is less than alpha, this source of variation is considered to have a significant effect on the output. In this case, the term and its *p* value will be displayed in red.

- The following values are shown underneath the ANOVA table, and they indicate how well the model fits the data:

  - **S** is the standard error of the noise. It represents the magnitude of the response variation that is caused by noise. Lower values indicate better fit.

  - **R-sq** is the percentage of total difference that is attributable to the factors under consideration. It is equal to Sum of Squares(factor)/Total Sum of Squares. Higher values usually indicate better fit.

  - **R-sq(adj)** is an R-sq value that is adjusted for the number of parameters in the model. Higher values indicate better fit.

  - **PRESS** is the prediction error sum of squares, which provides a measure of the model's validity. The lower the PRESS value, the better the model's predictive ability.

  - **R-sq(pred)** is a measure of how well the model predicts new observations. It is equal to 1-PRESS/Total Sum of Squares. The larger the value, the more accurate the model's predictions are likely to be.

Likelihood Table Columns (for reliability DOE)

- **Model** displays the model for which the results apply.

  - **Reduced** assumes that the product life is the same at different levels of the factor.

  - **Full** assumes that the product life is different at different levels of the factor.

- **Degrees of Freedom** is the degrees of freedom of this source of variation. This is also the number of parameters in the model for this source.
- **Ln(Likelihood Value)** is the logarithm transformation of the likelihood value for this source of variation.
- **Likelihood Ratio** is the likelihood ratio value for this source of variation.
- **P Value** is the probability that LR is from a chi-squared distribution, which would indicate that the factor has no effect on product life. This value is compared to the risk level (alpha) that you specify on the Analysis Settings page of the control panel. If the *p* value is less than alpha, then the factor is considered to have a significant effect on product life. In this case, the effect and its *p* value will be displayed in red.

The **Regression table** and the **MLE Information table** provide specific information on the contribution of each predictor to the variation in the response and an analysis of the significance of this contribution.

Regression Table Columns

- **Term** is the factor, factorial interaction, curvature, block, etc. under consideration. Terms displayed in red are considered to be significant. In cases where there is no error in the model, significant effects are determined according to Lenth's method and the term names are displayed in red and followed by an asterisk (*).
- **Coefficient** is the regression coefficient of the term, which represents the contribution of the term to the variation in the response.
- **Standard Error** is the standard deviation of the regression coefficient.
- **Low Confidence** and **High Confidence** are the lower and upper confidence bounds on the regression coefficient.
- **T Value** is the normalized regression coefficient, which is equal to Coefficient/Standard Error.
- **P Value** (alpha error or type I error) is the probability that an equal amount of variation in the output would be observed in the case that this term does not affect the output. This value is compared to the risk level (alpha) that you specify on the Anaysis Settings page of the control panel. If the *p* value is less than alpha, this source of variation is considered to have a significant effect on the output. In this case, the term and its *p* value will be displayed in red.

MLE Information Table Columns (for reliability DOE)

- **Term** is the factor, factorial interaction, etc. under consideration. Terms displayed in red are considered to be significant.
- **Effect** is a measure of how much the response value (Y) changes when the value of the corresponding term in the model (using coded values) changes from -1 to 1.
- **Coefficient** is the regression coefficient of the term, which represents the contribution of the term to the variation in the response.
- **Standard Error** is the standard deviation of the regression coefficient.
- **Low Confidence** is the lower confidence bound of the regression coefficient, using Fisher bounds.
- **High Confidence** is the upper confidence bound of the regression coefficient, using Fisher bounds.
- **Z Value** is the normalized regression coefficient, which is equal to Coefficient/Standard Error.
- **P Value** is the probability that an equal amount of variation in the output would be observed in the case that this source does not affect the output. This value is compared to the risk level (alpha) that you specify on the Analysis Settings page of the control panel. If the *p* value is less than alpha, this source of variation is considered to have a significant effect on the output. In this case, the term and its *p* value will be displayed in red.

The **Regression Equation** information is presented using multiple tables.

Regression Equation

- The **Response** table displays the response that the regression equation applies to and the units of measurement that were entered for the response (if any).
- The **Additional Settings** table shows the transformation and risk level you entered for the response.
- The **Significant Terms** table is applicable only when at least one term was found to be significant. It shows the significant terms in the Name column and the associated regression coefficients in the Coefficient column.

- The **Equation** tables show the regression coefficients for the model of the selected response. For example, consider this table:

Equation (Coded values)	
Response 1 =	
9.3903	
-0.8467	A:Factor 1
+1.2813	B:Factor 2
+0.7307	A • B

The corresponding model for this table is $y = 9.3903 - 0.8467x_1 + 1.2813x_2 + 0.7303x_1x_2$.

- The first Equation table shows the coefficients for a model that uses coded values for the factors.
- The second Equation table shows the coefficients for a model that uses the actual factor values.

Additional Results

All of the following tables provide information that was generated from the main calculations. The available tables will vary depending on the design type you are working with. The results that could be available include:

Alias Structure

This item is available for all designs with at least two factors. It describes the alias structure for the design, taking into account only the terms you've selected to include in the analysis. Together with your engineering knowledge, you can use this information to help determine whether any important interaction information was lost due to aliasing. When aliased terms exist, the following areas will be shown:

- **Terms selected to be in the model** lists all the terms that are considered for inclusion in the regression model (i.e., the selections in the Select Terms window).
- **Terms included in the model** lists all the selected terms that are included in the model. The alias structure determines which terms are excluded.
- **Alias Structure** lists the aliased effects based on the selected terms. For example, A • B = A • B + C • D means the interaction effect A • B is aliased because it is indistinguishable from effect C • D. Therefore, the model cannot include both interaction terms; it will include only one (e.g., A • B).

Alias Summary

The terms in the first column of this table are aliased with the terms shown in the second column. Only the terms in the first column are included in the model.

Var/Cov Matrix

This shows the variance/covariance matrix, which is available for one factor R-DOE designs and all other designs with two or more factors. The diagonal elements in this matrix are used to calculate the coefficients in the MLE or Regression Information table.

Diagnostic Information

This table is available for one factor R-DOE designs and all other designs with two or more factors. It displays various analysis results for each run and highlights significant values. The following columns are included:

- **Run Order** is the randomized order, generated by the software, in which it is recommended to perform the runs to avoid biased results. Note that any changes made to the Run Order column on the Data tab will be reflected here.
- **Standard Order** is the basic order of runs, as specified in the design type, without randomization. Note that any changes made to the Standard Order column on the Data tab will be reflected here.
- **Actual Value (Y)** is the observed response value for the run, as entered in the response column on the Data tab.
- **Predicted Value (YF)** is the response value predicted by the model given the factor settings used in the run.
- **Residual** (or "regular residual") is the difference between the actual value (Y) and the predicted value (YF) for the run.
- **Standardized Residual** is the regular residual for the run divided by the constant standard deviation across all runs.
- **Studentized Residual** is the regular residual for the run divided by an estimate of its standard deviation.
- **External Studentized Residual** is the regular residual for the run divided by an estimate of its standard deviation, where the run in question is omitted from the estimation.
- **Leverage** is a measure of how much the run influences the predicted values of the model, stated as a value between 0 and 1, where 1 indicates that the actual response value of the run is exactly

equal to the predicted value (i.e. the predicted value is completely dependent upon the observed value).

- **Cook's Distance** is a measure of how much the output is predicted to change if the run is deleted from the analysis.

Values that are considered to be significant, or outliers, are displayed in red. For the residual columns, significant or critical values are those that fall outside the residual's upper or lower bounds, calculated based on the specified alpha (risk) value.

The ReliaWiki resource portal has more information on how significant values are determined for the Leverage and Cook's Distance columns at: http://www.reliawiki.org/index.php/Multiple_Linear_Regression_Analysis.

Least Squares Means

This table shows the predicted response values for the given factor levels. It includes the following columns:

- **Effect** is the main effect or interaction used to predict the response. The coefficients for effects not used in the prediction are set to zero.
- **Level** is the combination of factor levels used to predict the response.
- **Mean** is the predicted response value.

Free Form Folio: Plots

The following plots may be available for free form folios, depending on the type of data you analyzed. For general information on working with plots, see Plot Utilities.

Effect Plots

Effect plots allow you to visually evaluate the effects of factors and factorial interactions on the selected response.

- The **Pareto Chart - Regression** plot shows the standardized effects of the selected terms (i.e., factor or combination of factors). The vertical blue line is the threshold value. If a bar is beyond the blue line, it will be red, indicating that the effect is significant.
- The **Pareto Chart - ANOVA*** plot shows the inverse p value (1 - p) of each selected term. The vertical blue line is the threshold value. If a bar is beyond the blue line, it will be red, indicating that the term is significant.

- The **Pareto Chart - LRT** shows the inverse *p* value for the reduced model in the likelihood table. The vertical blue line is the threshold value. If the bar is beyond the blue line, it will be red, indicating that the factor has a significant effect on reliability.

- The **Effect Probability** plot is a linear representation of probability versus the standardized effect (i.e., the probability that any term's standardized effect will be lower than the given value). The points on this plot represent the values for each term in the T Value column of the Regression Table in the detailed analysis results. If there is no error in the design, then the probability versus the effect is shown and the points on this plot represent the values for each term in the Effect column of the Regression Table in the analysis results.

  - Select **Normal** in the **Scale Type** area to display the negative and positive values of the effects (coefficients). The negative effects will appear to the left of the probability line.

  - Select **Half-normal** to display the absolute values of all the effects, which allows you to compare the size of each effect. All the effects will appear to the right of the probability line.

- The **Main Effects** plot shows the mean effect of the selected factor(s). The points are the observed Y values at the low and high level for each factor. The line connects the mean value at each factor level, and you can specify how the means are calculated in the **Calculation Options** area. Note that if you are using actual factor values in the plot, you can plot only one factor at a time. If you are using coded values, you can plot multiple factors simultaneously. For mixture designs, this plot applies only to process factors (i.e., process variables).

- The **Interactions** plot shows the mean effect of a selected factor versus another selected factor at each level. If the green and red mean effect lines are parallel, there is no interaction between the two factors. You can specify how the means are calculated in the **Calculation Options** area. For mixture designs, this plot applies only to process factors (i.e., process variables).

- The **Interaction Matrix** shows multiple Interactions plots. The plots shown depends on the factors you select. For example, if you select factors A and B, then two interactions plots will be shown: one showing A versus B and another showing B versus A.

- The **Term Effect Plot** shows the fitted means for all combinations of all factor levels for each selected term. You can specify how the means are calculated in the **Calculation Options** area.

- The **Cube Plot** shows the mean values of the selected response for the combinations of the low and high levels of three selected factors. You can specify how the means are calculated in the **Calculation Options** area. Note that you have the option of selecting "none" for the third factor, generating a square (2-dimensional) plot. Only two level factors can be included in the

Cube plot, and at least two quantitative factors (each run at two levels) must be included in the model for the Cube plot to be available.

- The **Scatter Plot** shows the observed values of the currently selected response plotted against the levels of the selected factor. A 3-dimensional version of this plot is available in the 3D plot folio.

- The **Contour Plot** shows how varying two selected factors affects the predicted response values, which are represented as colors. See Contour Plots. A 3-dimensional version of this plot ("Surface Plot") is available in the 3D plot folio. For mixture designs, this plot applies only to process factors (i.e., process variables).

Residual Plots

Residuals are the differences between the observed response values and the response values predicted by the model at each combination of factor values. Residual plots help to determine the validity of the model for the currently selected response. When applicable, a residual plot allows the user to select the type of residual to be used:

- **Regular Residual** is the difference between the observed Y and the predicted Y.

- **Standardized Residual** is the regular residual divided by the constant standard deviation.

- **Studentized Residual** is the regular residual divided by an estimate of its standard deviation.

- **External Studentized Residual** is the regular residual divided by an estimate of its standard deviation, where the observation in question is omitted from the estimation.

The plots are described next.

- The **Residual Probability*** plot is the normal probability plot of the residuals. If all points fall on the line, the model fits the data well (i.e., the residuals follow a normal distribution). Some scatter is to be expected, but noticeable patterns may indicate that a transformation should be used for further analysis. Two additional measures of how well the normal distribution fits the data are provided by default in the lower title of this plot. Smaller values for the Anderson-Darling test indicate a better fit. Smaller *p* values indicate a worse fit.

- The **Residual vs. Fitted*** plot shows the residuals plotted against the fitted, or predicted, values of the selected response. If the points are randomly distributed around the "0" line in the plot, the model fits the data well. If a pattern or trend is apparent, it can mean either that the model does not provide a good fit or that Y is not normally distributed, in which case a transformation should be used for further analysis. Points outside the critical value lines, which are calculated based on the specified alpha (risk) value, may be outliers and should be examined to determine the cause of their variation.

- The **Residual vs. Order*** plot shows the residuals plotted against the order of runs used in the design. If the points are randomly distributed in the plot, it means that the test sequence of the experiment has no effect. If a pattern or trend is apparent, this indicates that a time-related variable may be affecting the experiment and should be addressed by randomization and/or blocking. Points outside the critical value lines, which are calculated based on the specified alpha (risk) value, may be outliers and should be examined to determine the cause of their variation.

- The **Residual vs. Factor*** plot shows the residuals plotted against values of the factor selected in the **Residual Factor** area. It is used to determine whether the residuals are equally distributed around the "0" value line and whether the spread and pattern of the points are the same at different levels. If the size of the residuals changes as a function of the factor's settings (i.e., the plot displays a noticeable curvature), the model does not appropriately account for the contribution of the selected factor. Points outside the critical value lines, which are calculated based on the specified alpha (risk) value, may be outliers and should be examined to determine the cause of their variation.

- The **Residual Histogram*** is used to demonstrate whether the residual is normally distributed by dividing the residuals into equally spaced groups and plotting the frequency of the groups. The **Residual Histogram Settings** area allows you to:

  - Select **Custom Bins** to specify the number of groups, or bins, into which the residuals will be divided. Otherwise, the software will automatically select a default number of bins based on the number of observations.

  - Select **Superimpose pdf** to display the probability density function line on top of the bins.

- The **Residual Autocorrelation*** plot shows a measure of the correlation between the residual values for the series of runs (sorted by run order) and one or more lagged versions of the series of runs. The default number of lags is the number of observations, *n*, divided by 4. If you select **Custom Lags** in the **Auto-Correlation Options** area, you can specify up to *n* -1 lags. The correlation is calculated as follows:

$$r_k = \frac{\sum_{i=1}^{N-k}\left(Y_i - \overline{Y}\right)\left(Y_{i+k} - \overline{Y}\right)}{\sum_{i=1}^{N}\left(Y_i - \overline{Y}\right)^2}$$

where:

- k is the lag.
- $\overline{Y}$ is the mean value of the original series of runs.

  For example, lag 1 shows the autocorrelation of the residuals when run 1 is compared with run 2, run 2 is compared with run 3 and so on. Lag 3 shows the autocorrelation of the residuals when run 1 is compared with run 4, run 2 is compared with run 5 and so on. Any lag that is displayed in red is considered to be significant; in other words, there is a correlation within the data set at that lag. This could be caused by a factor that is not included in the model or design, and may warrant further investigation.

The **Fitted vs. Actual** plot shows the fitted, or predicted, values of the currently selected response plotted against the observed values of the response. If the model fits the data well, the points will cluster around the line.

Diagnostic Plots

- The **Leverage vs. Order** plot shows *leverage* plotted against the order of runs used in the design. Leverage is a measure (between 0 and 1) of how much a given run influences the predicted values of the model, where 1 indicates that the actual response value of the run is exactly equal to the predicted value (i.e. the predicted value is completely dependent upon the observed value). Points that differ greatly from the rest of the runs are considered outliers and may distort the analysis.
- The **Cook's Distance*** plot can show Cook's distance (i.e., a measure of how much the output is predicted to change if each run is deleted from the analysis) plotted against either the run order or the standard order for the currently selected response. Points that differ greatly from the rest of the runs are considered outliers and may distort the analysis.
- The **Box-Cox Transformation*** plot can help determine, for the currently selected response and model, what transformation, if any, should be applied. The plot shows the sum of squares of the residuals plotted against lambda. The value of lambda at the minimum point of this curve is considered the "best value" of lambda, and indicates the appropriate transformation, which is also noted by default in the lower title of the plot.

* These plots are available only when there is error in the design, indicated by a positive value for sum of squares for Residual in the ANOVA table of the analysis results.

Free Form Folio: Example

*The data set used in this example is available in the example database installed with the software (called "Weibull21_DOE_Examples.rsgz21"). 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 example project is "Free Form Folio - Optimization."

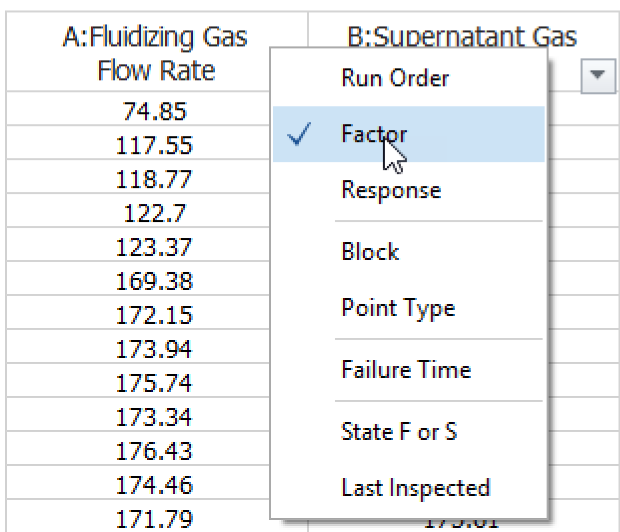
A chemical engineering experiment involving heat transfer was performed several weeks ago. There were some difficulties during the experiment (the factor values were difficult to control because of the testing equipment), and some of the test runs had to be discarded. While the testing facility will not be used for future experiments, the engineers would still like to analyze the old data and find the possible optimal values for each factor. They could then use the optimal values as a starting point for future experiments. The factors are:

- Fluidizing gas flow rate (lb/hr)
- Supernatant gas flow rate (lb/hr)
- Nozzle opening (mm)
- Gas temperature (F)

There are two responses: heat transfer efficiency and thermal efficiency. The objective of the analysis is to maximize both responses.

Data Entry

The data set (without the discarded test runs) is shown in the "Historical Data" free form folio. Each column of data was defined as a factor or response column by clicking the arrow in the column heading.

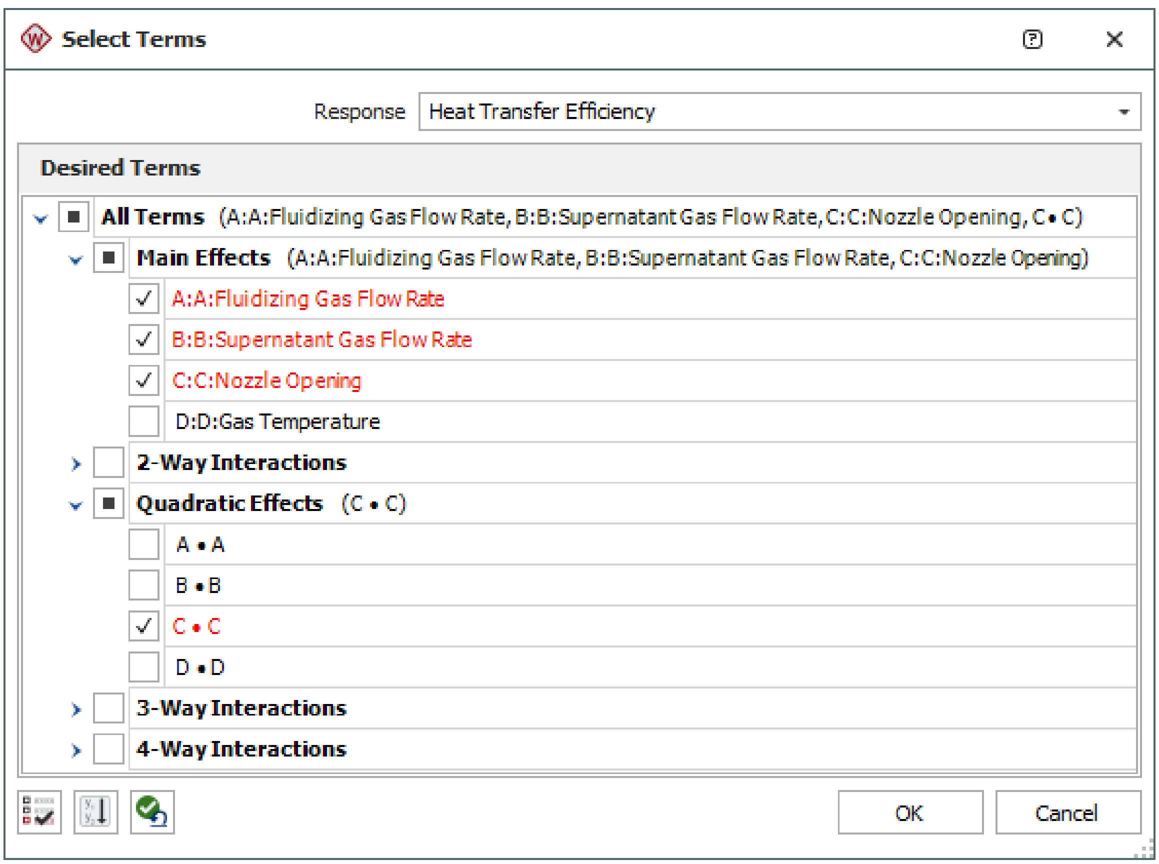


The factors and responses were then defined (e.g., the engineers named the factors/responses and specified that all the factors are quantitative) by clicking the **Modify Design** icon on the control panel.

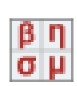
Analysis and Results

For terms to be included in the analysis of the heat transfer efficiency response were selected by clicking the **Select Terms** icon.

The **Heat Transfer Efficiency** was chosen from the drop-down list at the top of the window, and the following terms were selected.



For the second response, the main effects of factors A, B and D were selected; the interaction effects AB and BD were selected; and the quadratic effects AA and DD were selected. Click the **Calculate** icon to analyze both responses.



The regression table for heat transfer efficiency is shown next:

Regression Information								
Term	Effect	Coefficient	Standard Error	Low Confidence	High Confidence	T Value	P Value	Variance Inflation Factor
Intercept		96.73931	4.606396	88.664066	104.814554	21.001083	1.553091E-12	
A:A:Fluidizing GasFlow Rate	97.150489	48.575244	6.380095	37.390617	59.759872	7.613561	0.000002	1.008083
B:B:Supernatant GasFlow Rate	136.447531	68.223765	7.098754	55.779292	80.668239	9.610667	8.404083E-8	1.007477
C:C:NozzleOpening	-96.100427	-48.050213	6.680103	-59.76077	-36.339657	-7.193035	0.000003	1.001055
C • C	85.253387	42.626693	11.340602	22.746048	62.507339	3.758768	0.001897	1.001335

Next is the regression table for thermal efficiency:

Regression Information								
Term	Effect	Coefficient	Standard Error	Low Confidence	High Confidence	T Value	P Value	Variance Inflation Factor
Intercept		60.443203	2.361782	56.302881	64.583526	25.5922	8.637535E-14	
A:A:Fluidizing GasFlow Rate	11.617396	5.808698	3.183011	0.22872	11.388676	1.824907	0.087993	1.139416
B:B:Supernatant GasFlow Rate	97.019316	48.509658	14.308163	23.426729	73.592587	3.390348	0.004037	18.586712
D:D:GasTemperature	82.426428	41.213214	11.375616	21.271188	61.15524	3.622944	0.002505	18.794616
A • A	-24.658227	-12.329114	4.848383	-20.828573	-3.829655	-2.54293	0.022511	1.236587

Now that models have been created for both responses, you can use the optimization folio to investigate how different factor level combinations affect the response values. (See Optimization Example.)

Optimization Folio

The optimization folio provides three tools that allow you to examine factor level combinations and evaluate how well they meet your response goal(s). These tools can be accessed by choosing **Home > Insert > Optimization**.

The Select Optimization Tool window will appear. In this window, you will select a folio to optimize and then select one of the following:

- The **optimal solution plot** helps you determine the factor level combination that will produce the user-specified optimum value for one or more responses. This plot is available for all designs with at least two factors.
- The **overlaid contour plot** displays all of the factor level combinations for two quantitative factors that will produce response values that fall within the user-specified goal. This plot is available for all designs with at least two quantitative factors.
- The **dynamic overlaid contour plot** displays a series of overlaid contour plots side by side. The x- and y-axes represent two of the quantitative factors. The level of one or more additional

factors is varied across the plots. This plot is available for all designs with at least three factors, where at least two of the factors are quantitative.

Note that each optimization folio can have only one optimization tool of each type.

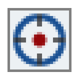
Optimal Solution Plot

The optimal solution plot provides a graphical analysis of the factor settings that will produce the user-specified optimum value for one or more responses. This plot is available for all designs with at least two factors.

Creating the Optimal Solution Plot

There are three different ways to create an optimal solution plot:

- On the Home tab, click the **Optimization** icon in the Insert gallery.



  In the Select Optimization Tool window that appears, select a folio to optimize and then select to create an optimal solution plot.

- You can create an optimal solution plot directly from a calculated design folio by clicking the **Optimization** icon on the control panel. An optimization folio containing an optimal solution plot for the folio will be added to the DOE folder in the current project explorer. If there is an optimization folio already associated with the design folio, clicking the **Optimization** icon on the control panel for the design folio will add the optimal solution plot as a sheet within that folio.
- Open an existing optimization folio and add additional optimization tools based on the same design folio by choosing **Optimization > Tools > Add Optimization Tool**.

Note that each optimization folio can have only one optimization tool of each type.

When you first create an optimal solution plot, you will need to specify the settings that the plot is based on. The Optimization Settings window will appear automatically. The navigation panel on the left of this window lists all of the settings pages.

The pages listed under the **Responses** heading allow you to select which response(s) you want to optimize, and to specify the optimal settings for each response. The settings on these pages are required in order to create the plot.

To specify the response settings:

Click each response in the navigation panel and use the **Include in optimization** drop-down list to specify whether the response will be optimized. Note that if a response is missing all data at a given level for a qualitative factor or a block (which is essentially a type of qualitative factor), that level cannot be included in the optimization of the response. Therefore, any optimal solutions found will not use factor levels that do not have data for all selected responses.

The options under the **Goal Settings** heading allow you to specify whether you want to minimize the response, maximize it or bring it as close as possible to a particular target value. You will then specify a target value for the response, as well as a lower limit and/or an upper limit, depending on the type of goal. These settings affect the desirability rankings of the solutions as follows:

- Target
  - Lower Limit: Responses below this value will not be considered desirable at all.
  - Target: The closer to this value the response is, the more desirable it is.
  - Upper Limit: Responses above this value will not be considered desirable at all.
- Minimize
  - Target: Responses below this value will be considered 100% desirable.
  - Upper Limit: Responses above this value will not be considered desirable at all.
- Maximize
  - Lower Limit: Responses below this value will not be considered desirable at all.
  - Target: Responses above this value will be considered 100% desirable.

Note that if a transformation has been applied to any response that you are optimizing, the transformed response values are used in optimization. For this reason, the values that you enter for Lower Limit, Target and/or Upper Limit must also be transformed values. For example, life data responses always have the lognormal transformation applied; therefore, if you are maximizing a life data response, the Lower Limit and Target values should be in terms of Ln(failure time).

If you are including more than one response in the optimization, the **Desirability** fields will be available. These are used in determining how desirable each optimal solution is.

- Use the **Weight** field to specify how much the target influences the local desirability of solutions (i.e., how well each solution optimizes that particular response).
- Use the **Importance** field to specify, on a scale of 0.1 to 10, how much the local desirability of the solution for a given response influences the global desirability of the solution (i.e., how

well each solution meets the optimization requirements for all selected responses). This is a way of specifying which response(s) it is most important to optimize (e.g., if one solution does a better job of optimizing Response 1 and another solution does a better job of optimizing Response 2, your inputs in this column will determine which is the more desirable solution).

Inputs for the Weight and Importance are on a scale from 0.1 to 10, where 0.1 is least important and 10 is most important. In other words, if you specify a Weight value of 0.1, response values that are relatively far from the target could still be highly desirable, whereas if you specify a Weight value of 10, only response values extremely close to the target would be highly desirable.

The pages listed under the **Factors** heading allow you to select which factors will be varied in the optimization, and to specify boundaries or constraints on any quantitative factors to be used in the optimization process, thereby excluding solutions that call for factor settings outside the range(s) you specify. The factor values are expressed in terms of actual or coded values. To change the setting, close the Optimization Settings window and choose **Plot > Display > [Value Type]**.

To specify the factor settings:

Click each factor in the navigation panel and use the **Treatment in Plot** drop-down list to specify whether the factor's value will be varied or held constant during the optimization.

Then use the **Lower Limit** and **Upper Limit** fields to specify the bounds for each quantitative factor.

The pages listed under the **Algorithm** heading allow you to specify how the optimization calculations are performed.

To specify the calculation settings:

The **General Settings** page includes the following settings.

- **Design Runs** specifies the number of runs in the design that will be checked for optimal solutions.
- **Random Runs** specifies the number of random start points that will be analyzed.
- **Epsilon** is the convergence criterion that controls how long the software attempts to find optimal solutions. This is for informational purposes and cannot be changed.

- **Fractions** determines the width of the range from each start point that will be checked for optimal solutions. This is for informational purposes and cannot be changed.
- **Use Seed** allows the user to specify a start point for the random number generation used in the optimization process, providing repeatability of results.

The **Initial Guess** page allows you to enter your own estimates of the value for each factor as a starting point for the optimization process. If you do not specify a starting point, random values between the limits specified under the factor settings will be used or, in the case of qualitative factors, every level specified for the qualitative factor is tried for each of the random values of the other factor(s) tried.

To enter a starting point, choose **Yes** from the **Use Initial Guess** drop-down list, then enter a value for each factor. For quantitative factors, this will be a numeric value. For qualitative factors and blocks, click inside the field and select the desired option from the drop-down list. If you are supplying initial guesses, you must specify a value for each row.

Note: If optimization does not yield any solutions with a desirability (either global or local) greater than zero, a plot displaying the initial guess value for each factor will be displayed. This not an optimal solution; rather, it is intended to be used as a starting point for manual editing.

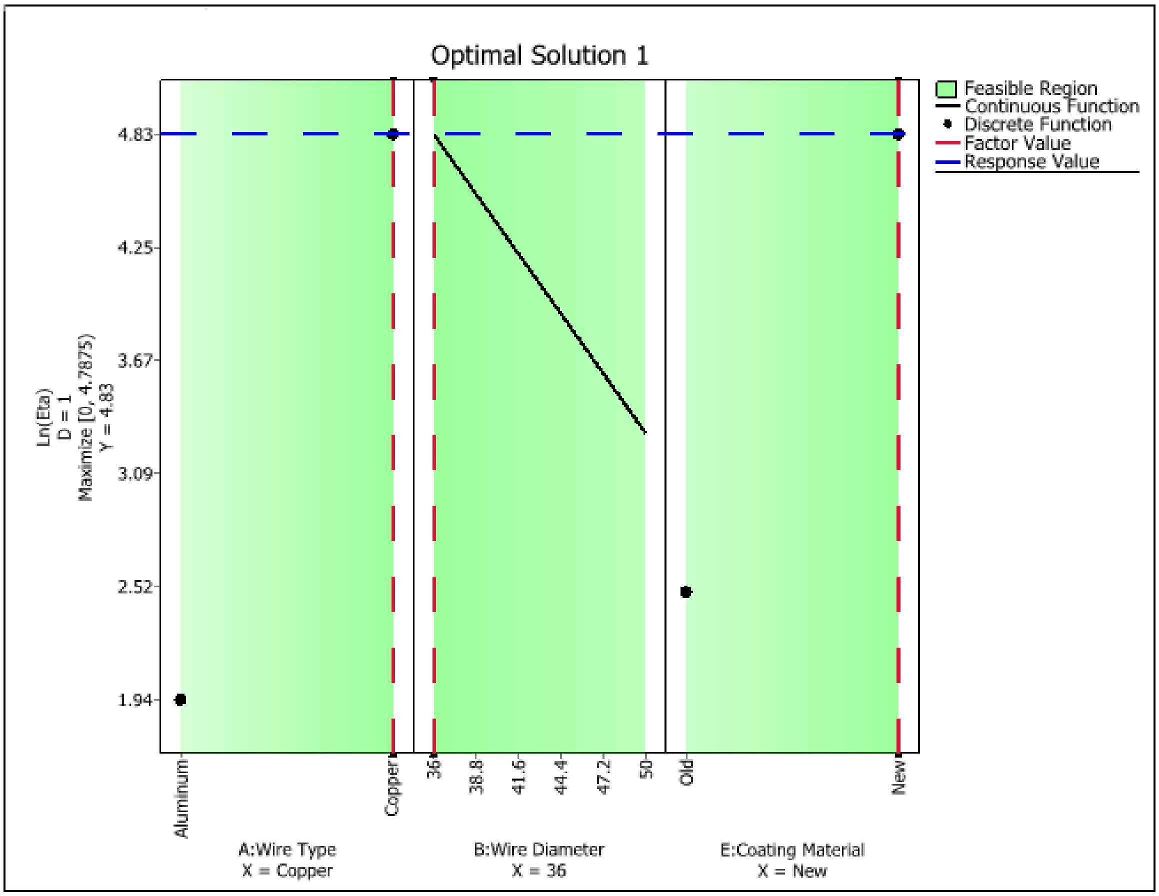
The **Summary** page allows you to review all your optimization settings.

Once you have specified and reviewed the settings, click **OK**. The software will compute up to five optimal solutions (i.e., the combinations of factor settings that will produce the most desirable response values, based on your specifications in the Optimization Settings window) and will create a plot of the first optimal solution. Other solutions will be displayed in the **Solutions** area of the control panel; click a solution to view its plot. Note that the order in which the solutions are displayed is random and does not indicate their desirability.

Note: You can specify new settings and re-compute the optimal solutions at any time by clicking the **Calculate** icon ( ) to re-open the Optimization Settings window. You can also change the folio that the optimal solution plot is based on by clicking the **Select Data Source** icon ( ).

Viewing an Optimal Solution Plot

In the plot of a solution, each response selected for inclusion is presented in a separate row and each factor selected for inclusion is presented in a separate column. In addition, a column can be displayed for blocks. By default, the response value is marked with a blue dashed line and the factor settings that yield that response value are marked with red dashed lines. For example, in the plot shown next, the response is being maximized, with a target value of 4.5 or greater. Three factors are included in the optimization. In this solution, the actual response value of 4.826 is achieved by setting factor A (a qualitative factor) to "copper," factor C (a quantitative factor) to 36.000 and factor E (a qualitative factor) to "new."



As you can see, possible settings for qualitative factors (and blocks) are displayed as points, while possible settings for quantitative factors are displayed as lines. To change the responses and/or factors that are shown in the plot, click the **Select Responses & Factors** icon.

In addition, you can choose to display the feasible region for the factor settings (i.e., the region between the limits specified in the Optimization Settings window) by selecting the **Display Feasible Region** check box on the control panel. In the plot shown here, the feasible region is indicated by green shading, where reduced desirability is indicated by lighter shading.

Working with Solutions

You can create your own custom solutions by clicking the **Create Solution** icon ( ) in the **Solutions** area on the control panel. In the window that appears, enter a name for the solution and specify the factor values to be used for the solution. Your new solution will be added to the solutions list.

Custom solutions can be edited by selecting the solution in the list and clicking the **Edit Solution** icon ( ) or by dragging the factor lines directly on the plot. Note that if you edit an optimal solution generated by the software, your changes will be used to create a new custom solution and the original solution will remain unchanged. You can also duplicate a solution by selecting the solution and clicking the **Duplicate Solution** icon ( ).

To remove a solution from the solutions list, select the solution and click the **Delete Solution** icon ( ). This cannot be undone.

You can view all of the optimal and custom solutions in a numeric format by clicking the **View Solutions** icon ( ). The Solutions window shows each solution, including factor settings, the associated block (if any) and the predicted response(s) at those settings. The response desirability columns rate the desirability of each solution from the perspective of optimizing each response. If more than one solution has been optimized, the global desirability column rates the overall desirability of each solution (i.e., how well its relative optimization of each response meets the requirements specified by the Weight and Importance settings on the Response Settings tab of the Optimization Settings window).

The **Transfer Solution to Overlaid** icon ( ) creates an overlaid contour plot using the factor values in the currently selected solution.

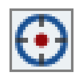
Overlaid Contour Plot

The overlaid contour plot displays all of the factor level combinations for two quantitative factors that will produce response values that fall within the user-specified goal. This plot is available for all designs with at least two quantitative factors.

Creating the Overlaid Contour Plot

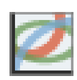
There are three different ways to create an overlaid contour plot:

- On the Home tab, click the **Optimization** icon in the Insert gallery.

  In the Select Optimization Tool window that appears, select a folio to optimize and then select to create an overlaid contour plot.

- You can create an overlaid contour plot directly from a calculated design folio by clicking the **Overlaid Contour Plot** icon on the control panel.

  An optimization folio containing an overlaid contour plot for the folio will be added to the DOE folder in the current project explorer. If there is an optimization folio already associated with the design folio, clicking the **Overlaid Contour Plot** icon on the control panel for the design folio will add the overlaid contour plot as a sheet within that folio.

- Open an existing optimization folio and add additional optimization tools based on the same design folio by choosing **Optimization > Tools > Add Optimization Tool**.

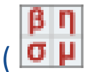
Note that each optimization folio can have only one optimization tool of each type.

When you first create an overlaid contour plot, you will need to specify the settings that the plot is based on. The Overlaid Contour Plot Settings window will appear automatically. Click each response under the **Responses** heading and use the **Include in optimization** drop-down to specify whether the response will be optimized.

Then use the **Goal** field to specify whether you want to minimize the response, maximize it or get it within a range of values. You will then specify a lower limit and/or an upper limit for the response, depending on which action you have selected.

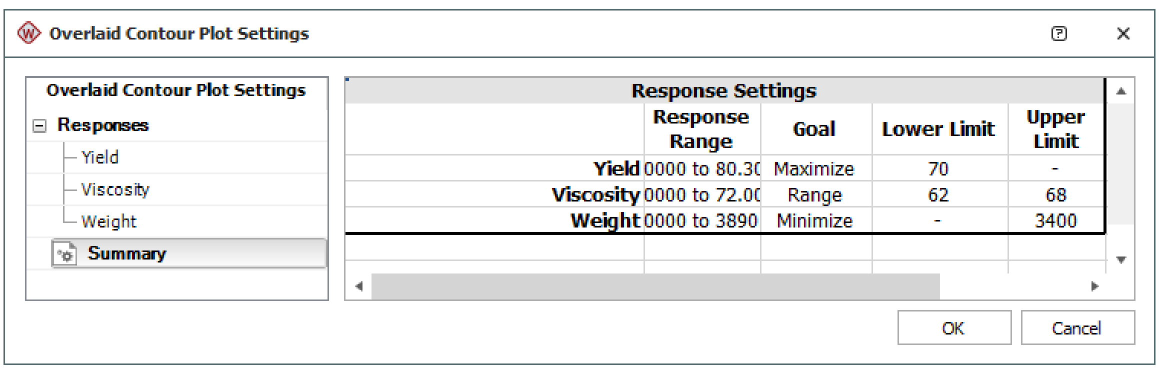
Note that if a transformation has been applied to any response that you are optimizing, the transformed response values are used in optimization. For this reason, the values that you enter for Lower Limit and/or Upper Limit must also be transformed values. For example, life data responses always have the lognormal transformation applied; therefore, if you are maximizing a life data response, the Lower Limit value should be in terms of Ln(failure time).

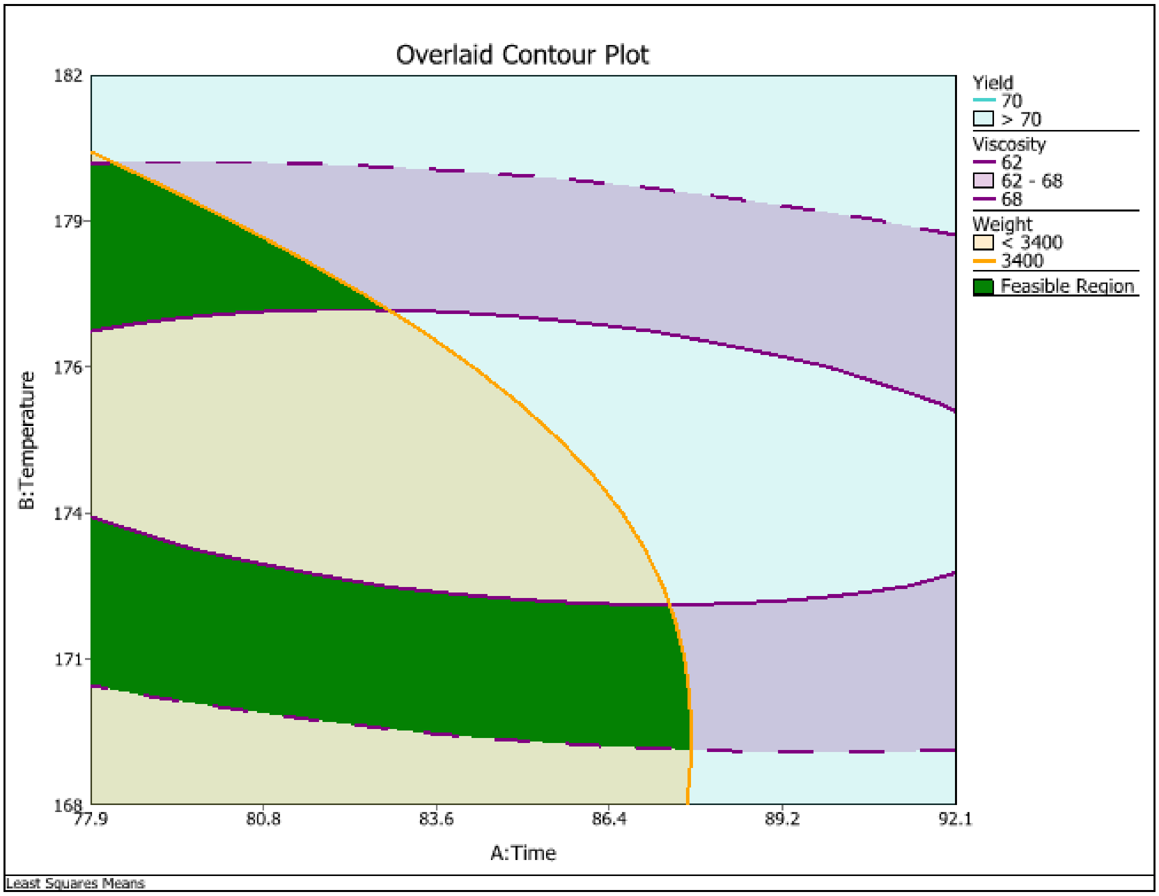
Once you have specified the settings, click **OK**. The overlaid contour plot will be created.

Note: You can specify new settings at any time by clicking the **Calculate** icon () to re-open the Overlaid Contour Plot Settings window. You can also change the folio that the overlaid contour plot is based on by clicking the **Select Data Source** icon ().

Viewing the Overlaid Contour Plot

For each response that is included in the plot, a shaded region shows the combinations of factor settings that result in the response values that you specified in the Overlaid Contour Plot Settings window. For example, consider the settings and plot shown next.





In the plot, the area shown in blue represents all of the combinations of factor settings for the time and temperature factors that will cause the yield to be greater than 70. The areas shown in purple represent the factor setting combinations that will result in a viscosity between 62 and 68. The area in beige represents the factor setting combinations that will give a weight of less than 3,400. Where all three of these areas overlap (i.e., the areas shown in green), the factor settings will give desirable values for all three results.

You can specify which factor is shown on each axis using the drop-down lists in the **Factors** area on the control panel.

The overlaid contour plot looks at the interaction between two factors. If there are more than two factors used in the experiment, the factors that are not shown on the plot axes will be held at a constant value. In the **Factor Values** area on the control panel, you can choose to use least squares means to determine the value used for each factor or to specify your own custom values. If you select **Use Custom Values**, then click **Set** to open the Set Custom Factor Values window and specify the custom values that you want to use. Note that these values will be used only when the factors are not shown on the plot axes. Note that if a response is missing all data at a given level for a qualitative factor or a block (which is essentially a type of qualitative factor), that level cannot be included in the optimization of the response. Therefore, any level you specify that does not have data for a given response will instead be treated as equal to 0 in the optimization of that response.

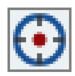
Dynamic Overlaid Contour Plot

The dynamic overlaid contour plot displays a series of overlaid contour plots side by side. The x- and y-axes represent two of the quantitative factors. The level of one or more additional factors is varied across the plots. This plot is available for all designs with at least three factors, where at least two of the factors are quantitative.

Creating the Dynamic Overlaid Contour Plot

There are two ways to create a dynamic overlaid contour plot:

- On the Home tab, click the **Optimization** icon in the Insert gallery.

  In the Select Optimization Tool window that appears, select a folio to optimize and then select to create a dynamic overlaid contour plot.

- Open an existing optimization folio and add additional optimization tools based on the same design folio by choosing **Optimization > Tools > Add Optimization Tool**.

Note that each optimization folio can have only one optimization tool of each type.

When you first create a dynamic overlaid contour plot, you will need to specify the settings that the plot is based on. The Dynamic Overlaid Contour Plot Settings window will appear automatically. The navigation panel on the left of this windows lists all of the settings pages.

The pages listed under the **Responses** heading allow you to select which response(s) you want to optimize, and to specify the optimal settings for each response.

To specify the response settings:

Click each response under the **Responses** heading and use the **Include in optimization** drop-down to specify whether the response will be optimized.

Then use the **Goal** field to specify whether you want to minimize the response, maximize it or get it within a range of values. You will then specify a lower limit and/or an upper limit for the response, depending on which action you have selected.

Note that if a transformation has been applied to any response that you are optimizing, the transformed response values are used in optimization. For this reason, the values that you enter for Lower Limit and/or Upper Limit must also be transformed values. For example, life data responses always have the lognormal transformation applied; therefore, if you are maximizing a life data response, the Lower Limit value should be in terms of Ln(failure time).

The pages listed under the **Factor** heading allows you to specify which factors will be shown on the x- and y-axes and whether to vary or hold constant each factor not shown on the axes, as well as the value ranges to use for the factors. The factor values are expressed in terms of actual or coded values. To change the setting, close the Optimization Settings window and choose **Plot > Display > [Value Type]**.

To specify the factor settings:

Click each factor and use the **Treatment in Plot** field to specify how it should be used in the plot. You should set one quantitative factor to be displayed on the x-axis and another quantitative factor to be displayed on the y-axis. The remaining factors may each be either held constant or varied.

For factors displayed on the axes, use the **Lower Limit** and **Upper Limit** fields to specify the bounds for each factor.

For factors that are held constant, specify the setting used for the factor in the **Constant Valu**e field. If the factor is qualitative, a drop-down list of the available factor settings will be provided.

For quantitative factors that are varied, use the **Lower Limit** and **Upper Limit** fields to specify the bounds for each factor. Then enter the number of different settings within that range to use for the factor in the **Number of Intervals** field. Qualitative factors will be varied across all possible settings. Note that if a response is missing all data at a given level for a qualitative factor or a block (which is essentially a type of qualitative factor), that level cannot be included in the optimization of the response. Therefore, any level that does not have data for a given response will instead be treated as equal to 0 in the optimization of that response.

The settings that you specify for the factors that are varied determine the number of possible solutions (i.e., plots) that will be generated and evaluated to determine if they contain a feasible region.

The **Summary** page allows you to review all your optimization settings.

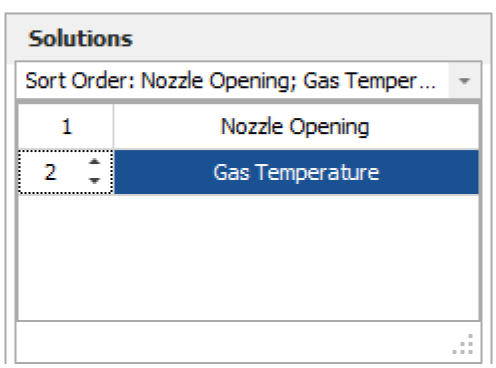
Once you have specified and reviewed the settings, click **OK**. The dynamic overlaid contour plot will be created. As the range of solutions are examined, the plots for the solutions are displayed. Once all solutions have been considered, the first overlaid contour plot is displayed. Additional solutions will be displayed in the **Sorted Solutions** area on the control panel.

Note: You can specify new settings at any time by clicking the **Calculate** icon ( ) to re-open the Dynamic Overlaid Contour Plot Settings window. You can also change the folio that the dynamic overlaid contour plot is based on by clicking the **Select Data Source** icon ( ).

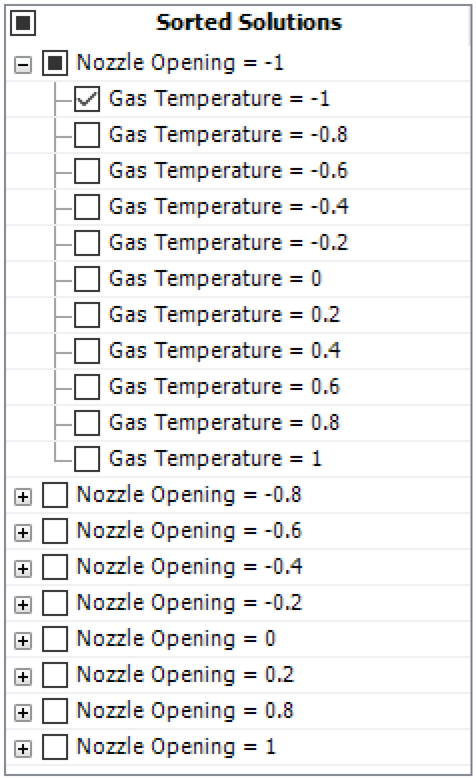
Viewing the Dynamic Overlaid Contour Plot

When the dynamic overlaid contour plot is created, all of the factor setting combinations that are possible based on your settings in the Dynamic Overlaid Contour Plot Settings window are generated and examined to determine whether they contain a feasible region (i.e., a region where response values fall within the specified range). All factor setting combinations that contain a feasible region are listed in the **Sorted Solutions** area on the control panel. You can select multiple solutions to be displayed side by side; up to 99 overlaid contour plots can be displayed simultaneously.

To facilitate finding and displaying solutions of interest to you, the order in which the solutions are sorted can be changed. Click the drop-down in the **Solutions** area to view a list of the factors that are either varied or held constant. Use the Priority Up and Priority Down arrows that appear when you click a priority number in the first column to move the factors up and down the list. In the example shown next, the solutions are being sorted first by the Nozzle Opening factor setting, then by the Gas Temperature factor setting.



This results in the following sorted list, where solutions are sorted by Gas Temperature within Nozzle Opening settings:



You can then select or clear entire groups of solutions for display.

Optimization Example

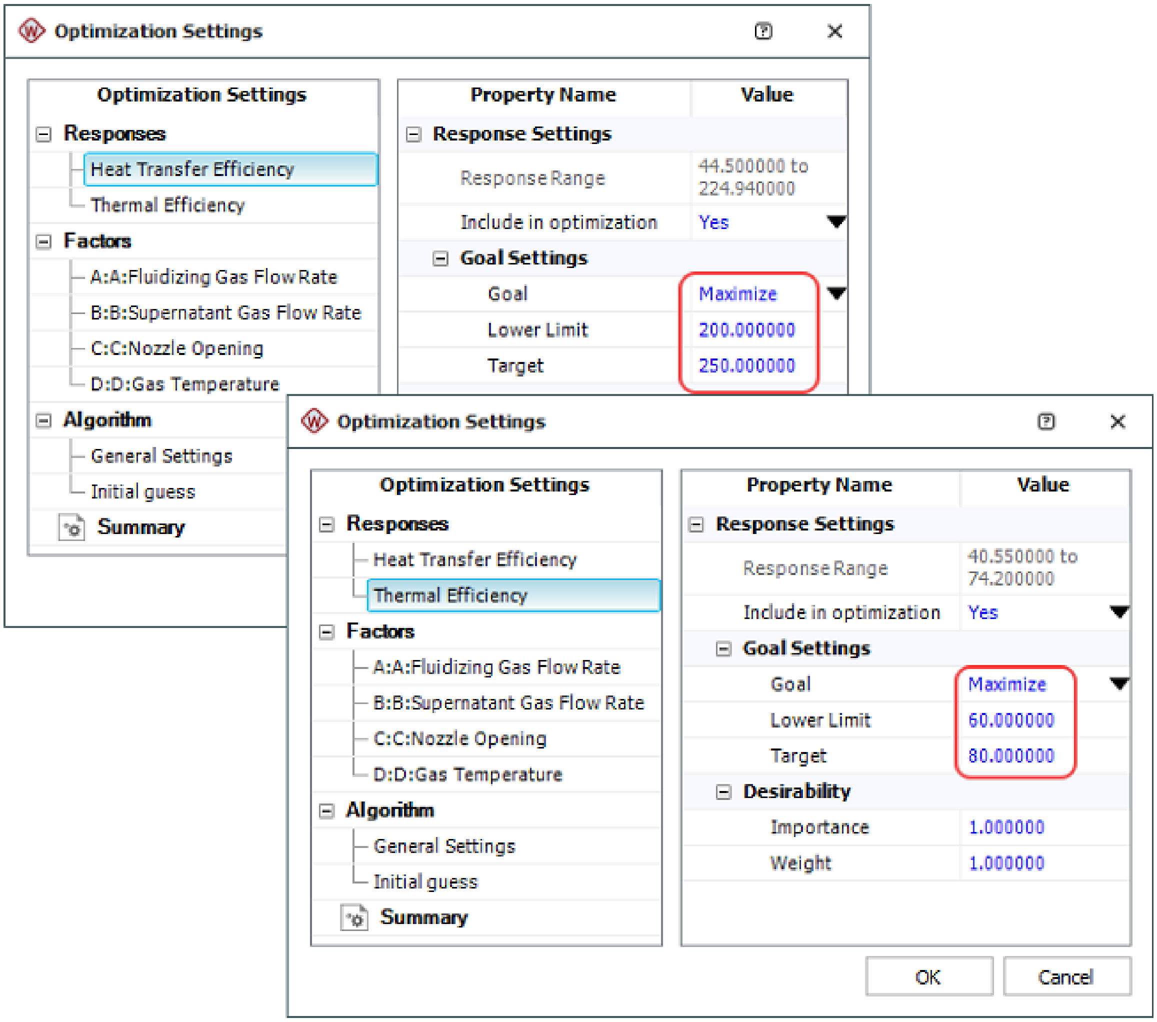
*The data set used in this example is available in the example database installed with the software (called "Weibull21_DOE_Examples.rsgz21"). To access this database file, choose **File > Help**, click **Open Examples Folder**, then browse for the file in the Weibull sub-folder.*

The name of the example project is "Free Form Folio - Optimization."

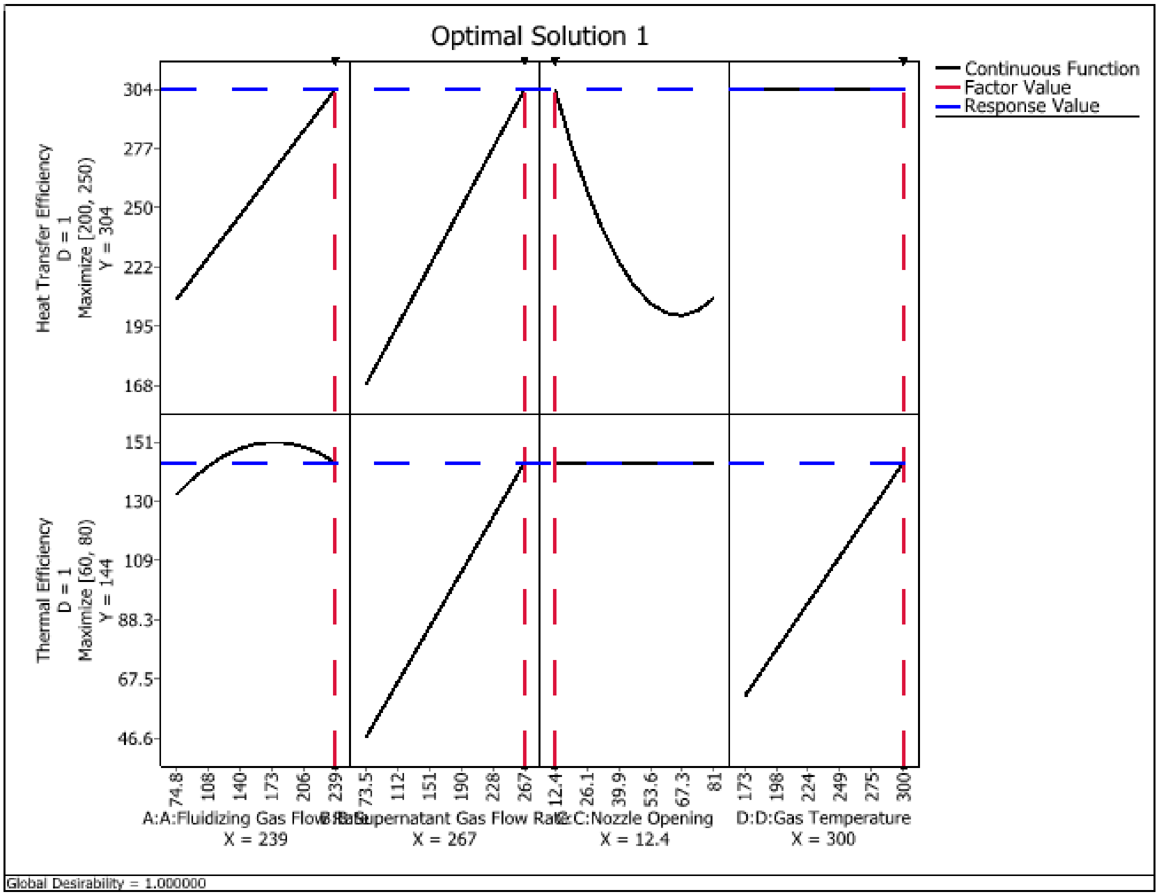
In a previous example, data from a chemical engineering experiment was analyzed using the free form folio, with the goal of maximizing both response values. The results of the analysis are given in the "Historical Data" folio.

The next step is to find the best factor level combination using the models created in the free form folio. The "Optimal Solution Plot" in the example project provides a graphical analysis of the factor settings that will produce optimum values for the responses.

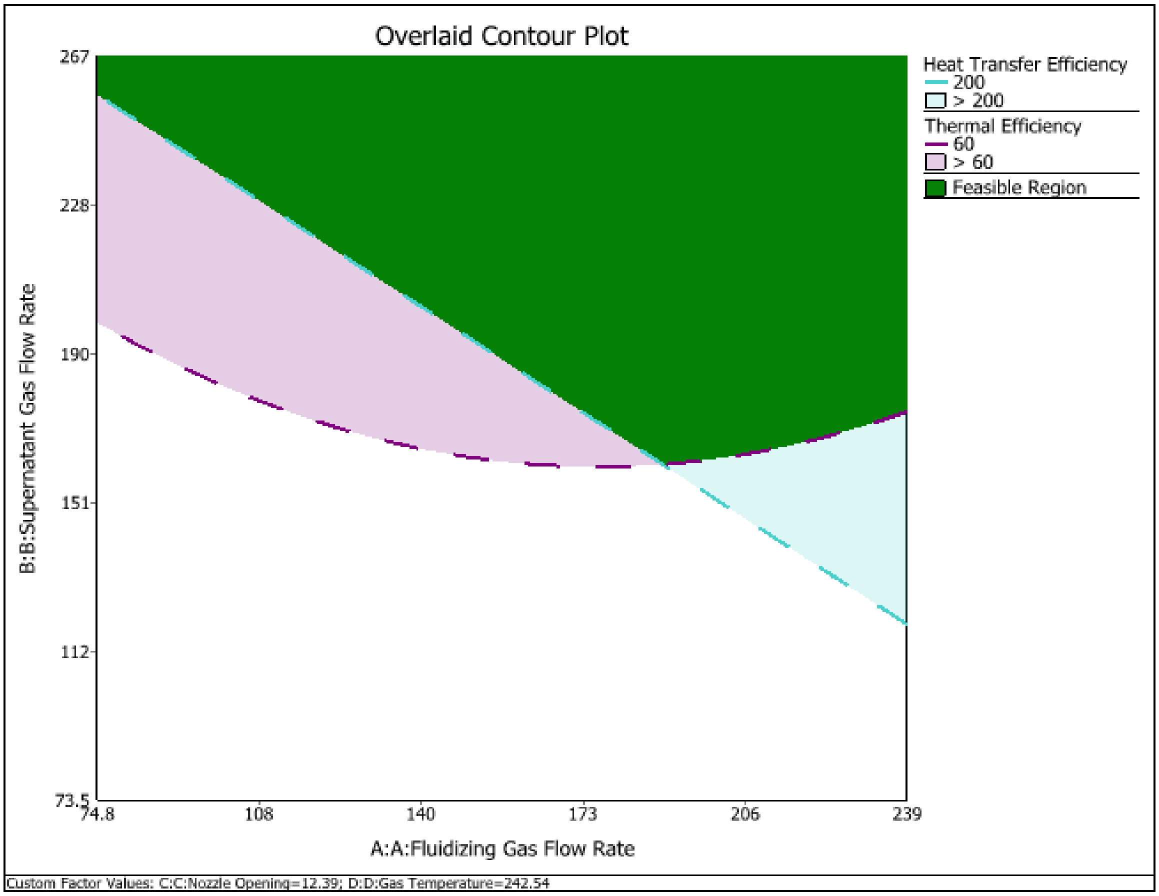
Click the **Calculate** icon to see the optimum settings for the responses. In this case, any values less than 60 for the heat transfer efficiency and less than 200 for the thermal efficiency are considered unacceptable. The Target values are the highest values that are expected given the process requirements.



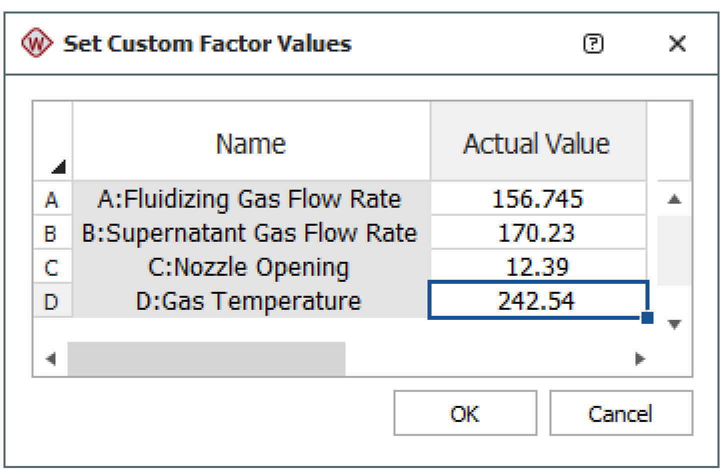
The plot is shown next. You can see the optimal solution in a numeric format by clicking the **View Solutions** icon () on the control panel.



The overlaid contour plots of these two responses are shown in the "Overlaid Contour Plot" folio. This folio shows the ranges of values for two factors (selected on the control panel) that produce the desired response values.

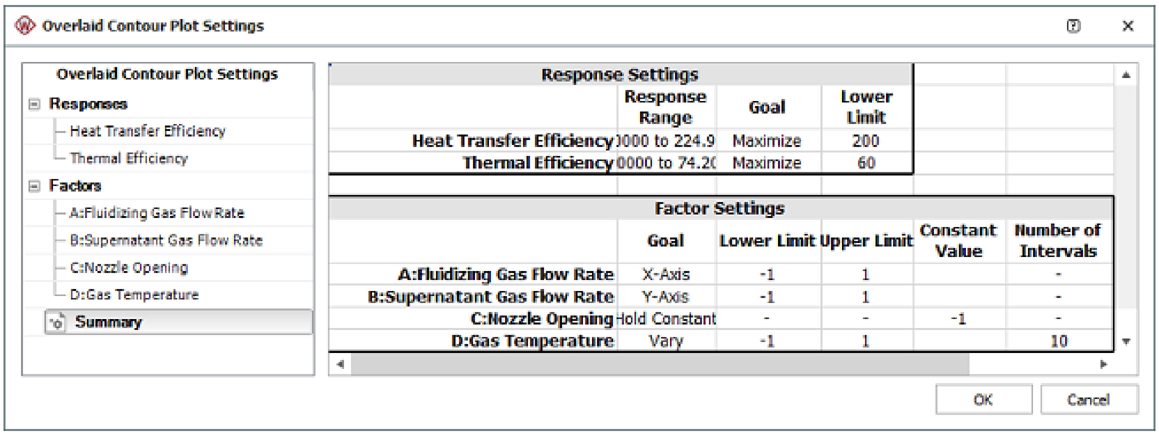


The green "feasible" area is the region where factors A and B produce the desired response values. In this example, factors C and D are fixed at custom values specified by clicking the **Set** link on the control panel.



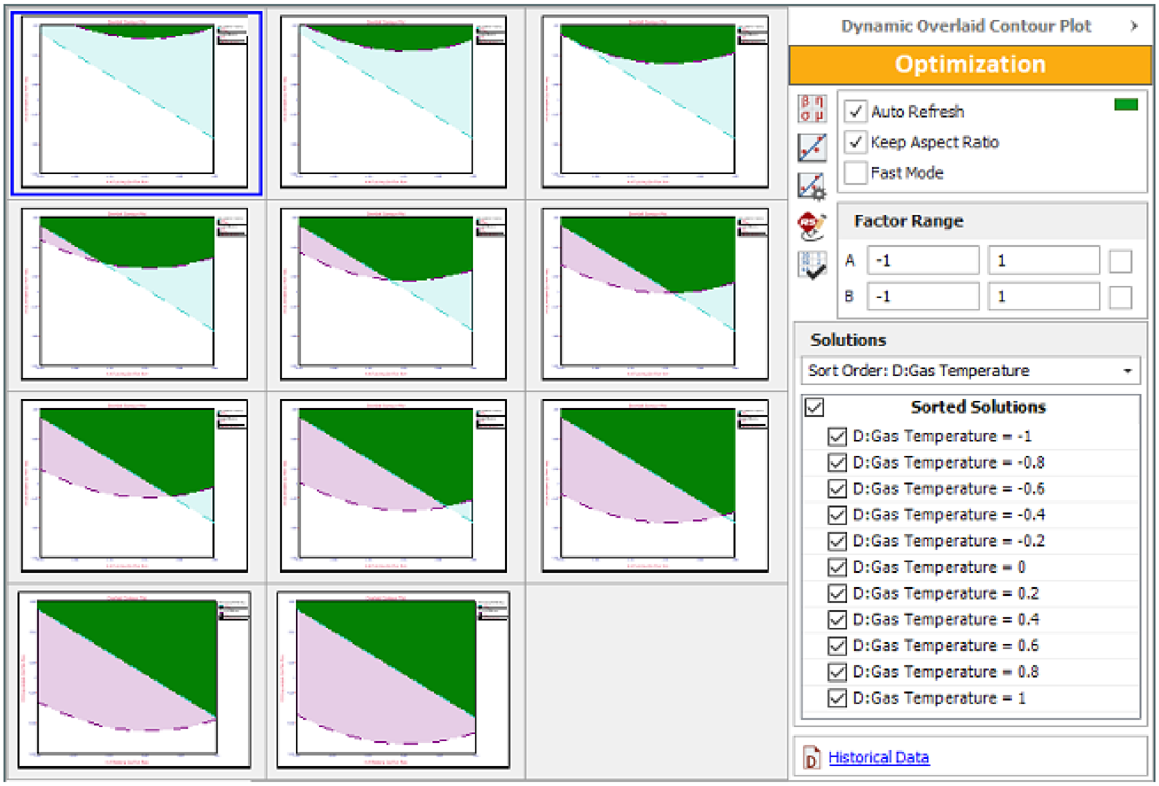
Are there any other feasible regions if we vary the values of C and D, or if we only hold C constant but vary the value of D? The "Dynamic Overlaid Contour Plot" folio includes a tool that can help you answer this question.

The engineers decided to make factor C (nozzle opening) constant, but vary factor D (gas temperature) along with A (fluidizing gas flow rate) and B (supernatant gas flow rate). This is specified on the **Factor** pages of the window that appears when you click the **Calculate** icon. A summary of the settings is shown next.



With these settings, the tool creates multiple overlaid contour plots where A and B are varied along the x- and y-axes. In each plot, C is held constant at its lowest factor value. To vary D, ten plots are created, and each uses a different value for factor D. For example, the first plot uses D's lowest value, and the last plot uses its highest value.

The plots are shown next, and each is identified in the **Sorted Solutions** area using the value for gas temperature that it uses. Double-click a plot to zoom in on it, and double-click it again to zoom out and view all the plots.



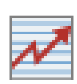
Conclusion

In this example, we obtained useful results from historical data. Different optimization tools were used to explore different factor level combinations. The easiest method is to use the optimal solution plot to quickly find the optimal solution. However, the overlaid contour plot can also be used to display the feasible region of two factors for multiple responses, while all other factors are held constant. And the dynamic overlaid plot can show the feasible region of two selected factors under different values for all other factors.

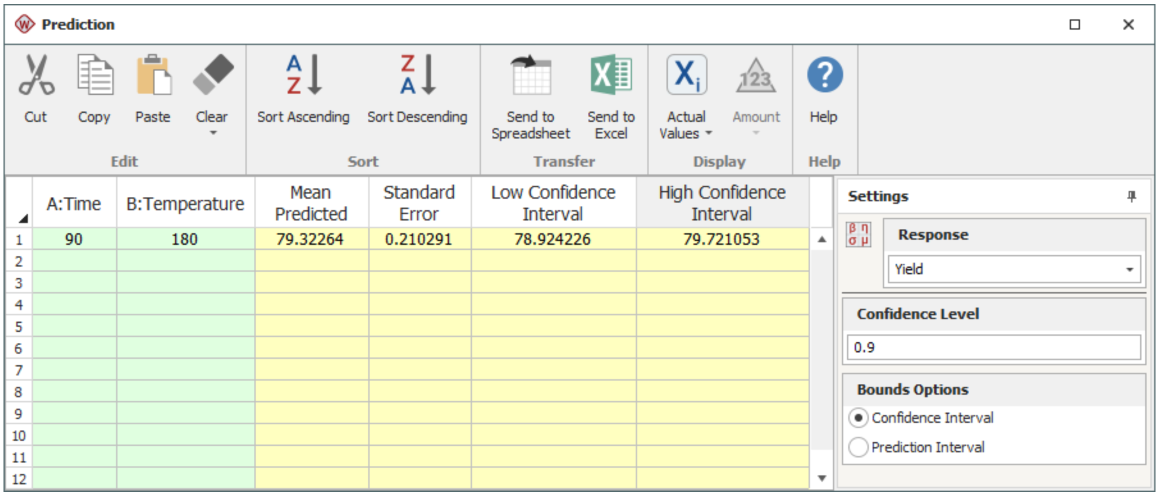
Prediction Window

The Prediction window is available after you analyze response data with at least one quantitative factor. The window allows you to enter values for each factor and see predicted results for the selected response.

The tool can be accessed from design folios, free form folios and multiple linear regression folios. To open it choose **Data** > **Analysis** > **Prediction** or click the icon on the control panel.



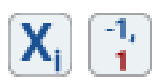
The values available for prediction in the Prediction window vary depending on whether your response values contain standard or life data. The Prediction window for standard data is shown next. The window for R-DOE looks similar, except it lacks the **Bounds Options** area.



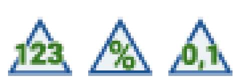
Note: If the prediction is based on a reliability DOE analysis, the **Publish Model** button will be available on the control panel. When you click this button, the selected prediction will be used to publish the analysis results as a model. (See Publishing Models from R-DOE.)

To use the Prediction window, follow the steps outlined next:

- First, in the green factor columns, enter the factor values that will be used to generate the prediction.
  - If the design uses blocks, choose values in the Block column as well.
  - You can choose to use either actual or coded values by choosing **Display** > **[Data Type]**.



  - For mixture designs, you can display the component factor values as amounts (e.g., 2 pounds), proportions (e.g., 0.2 = 20% of the total mixture used in the run) or pseudovalues (scaled values between 0 and 1, where 0 is the minimum possible value for the factor and 1 is the maximum possible value) by choosing **Display** > **[Value Type]**.



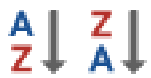
  - Note that any factors that you omit from the analysis will be hidden in this window.

- Choose the response you wish to generate predictions for in the **Response** drop-down list.
- In the **Confidence Level** field enter the level (as a decimal) that will be used to generate two-sided confidence bounds on the prediction.
- If you're working with standard response data, use the **Bounds Options** area to specify the type of bounds to be displayed for the predicted value. If you are working with R-DOE, this tool will always use confidence intervals.
  - If you select **Confidence Interval**, the value in the Mean Predicted column will be treated as a predicted mean value and bounds will be calculated accordingly.
  - If you select **Prediction Interval**, the value in the Mean Predicted column will be treated as a predicted new observation and bounds will be calculated accordingly. The prediction interval is wider than the confidence interval because the uncertainly of the error is considered in addition to the uncertainty of the parameter(s).
- Once you have specified all required values, click **Calculate**. The software will use the existing model to predict results for the currently selected response. The following columns will be populated:
  - The first column shows the predicted value. The name of this columns depends on the type of design you are working with.
    - For a standard design, this is the **Mean Predicted** column. It may be treated as a predicted mean value or a predicted new observation value, depending on the type of bounds selected in the control panel.
    - For R-DOE, the first column will represent the characteristic life of the selected distribution:
      - For Weibull, it is the **Eta Predicted** column. This is the predicted time at which unreliability = 63.2%.
      - For lognormal, it is the **Ln-Mean Predicted** column. This is the predicted time at which unreliability = 50%.
      - For exponential, it is the **MTTF Predicted** columns. This is the predicted mean life.
  - **Standard Error** is the standard error of the predicted value (i.e., the value in the preceding column).

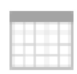
- The last two columns will vary depending on the design type and/or the selected bounds option.
  - The **Low Confidence Interval** and **High Confidence Interval** columns are always used for R-DOE. They are optional for other designs.
  - The **Low Prediction Interval** and **High Prediction Interval** columns are not available for R-DOE. They are optional for other designs.

All entered and calculated values are retained in the Prediction window after you close it; the next time you open the window, your inputs and results will appear. To clear the entire window, click the **Clear** icon in the window toolbar.

The Prediction window can be sorted by any column. Select a cell in the column you wish to sort by and then choose **Sort > Sort [Ascending/Descending]**.



The information in the Prediction window can be transferred to a spreadsheet by choosing **Transfer > Send to Spreadsheet**.



The information can also be saved as a Microsoft Excel® file (*.xls or *.xlsx) by choosing **Transfer > Send to Excel**.

Tests and Planning Folios

IN THIS CHAPTER

Test Design Folios

Test Design Assistant

Weibull++ includes 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 **Home > Insert > Test Design**.

The Test Design Assistant will appear and ask you to select one of the following:

- **Reliability Demonstration Test** solves for values associated with a specified test design such as required test time, required sample size, demonstrated reliability and confidence level.
- **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.
- **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 life data or life-stress data folio.

Note that in addition to the tools discussed in these topics, the software includes two other tools that can assist you in designing reliability life tests:

- **ALTA Test Plan** 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.
- **SimuMatic** 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 Designing Reliability Tests with SimuMatic.)

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. (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.)

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.

Control Panel

- **Test Design Method** allows you select a test design option. The method you select will determine the required inputs on the RDT sheet.
  - **Parametric Binomial** 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.

- **Non-Parametric Binomial** solves for the demonstrated reliability, confidence level or sample size that is associated with a specified test design. It does not require a life distribution. 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.
- **Exponential Chi-Squared** 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.

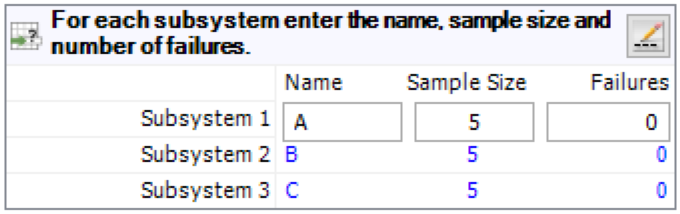
  This option is based on the chi-squared distribution, and it is only for products that have an assumed exponential life distribution (i.e., products with a constant failure rate).

- **Non-Parametric Bayesian** 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.

  In the RDT data sheet, 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:

  - 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, choose **Prior tests at the subsystem level**, and then specify the number of subsystems that you will enter data for. You can enter the information for each subsystem in the RDT sheet or click the **Edit Bayesian Subsystems** icon to open a window you can use to enter the prior test data.



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. The ReliaWiki resource portal has more information on these methods at: http://www.reliawiki.org/index.php/Reliability_Test_Design.

	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

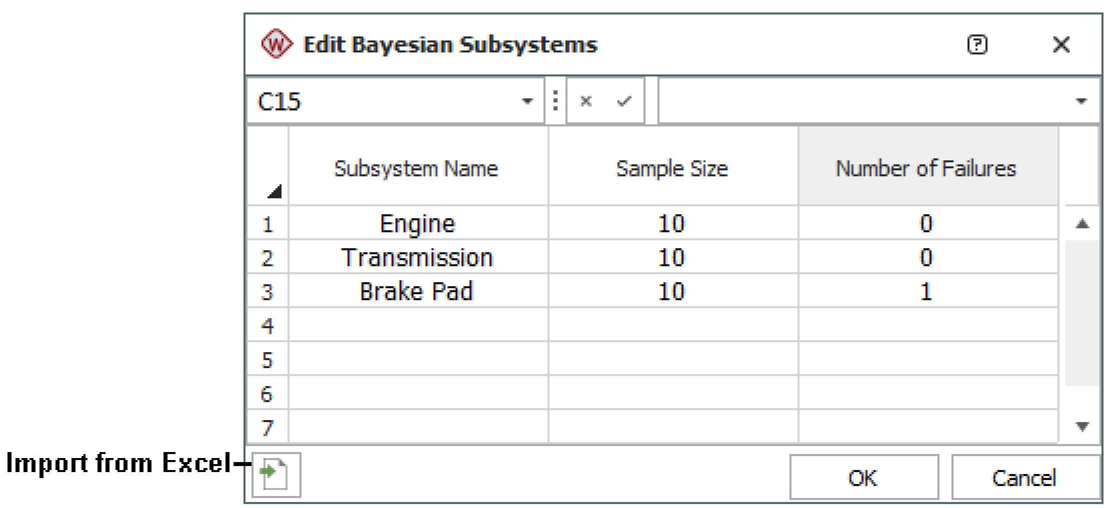
Non-Para-metric Bayesian	Reliability at test time (if acceleration factor = 1)	The demonstrated reliability or The associated confidence level or The necessary sample size

- **Units** in the control panel allows you to define the units of time that will be used in the test.
- **Display Options** is available when you select to solve for sample size. Select the check box to display the sample sizes as integers.
- **Acceleration Factor** allows you to 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 the acceleration factor is 2. If you are testing units under normal operating conditions, then the acceleration factor is 1.

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.

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.



- **Subsystem Name** is the name or description of the subsystem that was tested.
- **Sample Size** indicates how many units were used in the subsystem's test.
- **Number of Failures** indicates how many units failed during the test.
- **Import from Excel** command allows you to import the first three columns of an Excel sheet into the table.

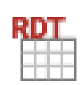
The software assumes a series configuration of these subsystems (i.e., the failure of any one subsystem is sufficient for the whole system to fail) and uses this subsystem-level information to estimate the expected value and variance of the reliability of the entire system.

Parametric Binomial Table and Plot

When you use the parametric binomial option of the RDT tool, you can 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.

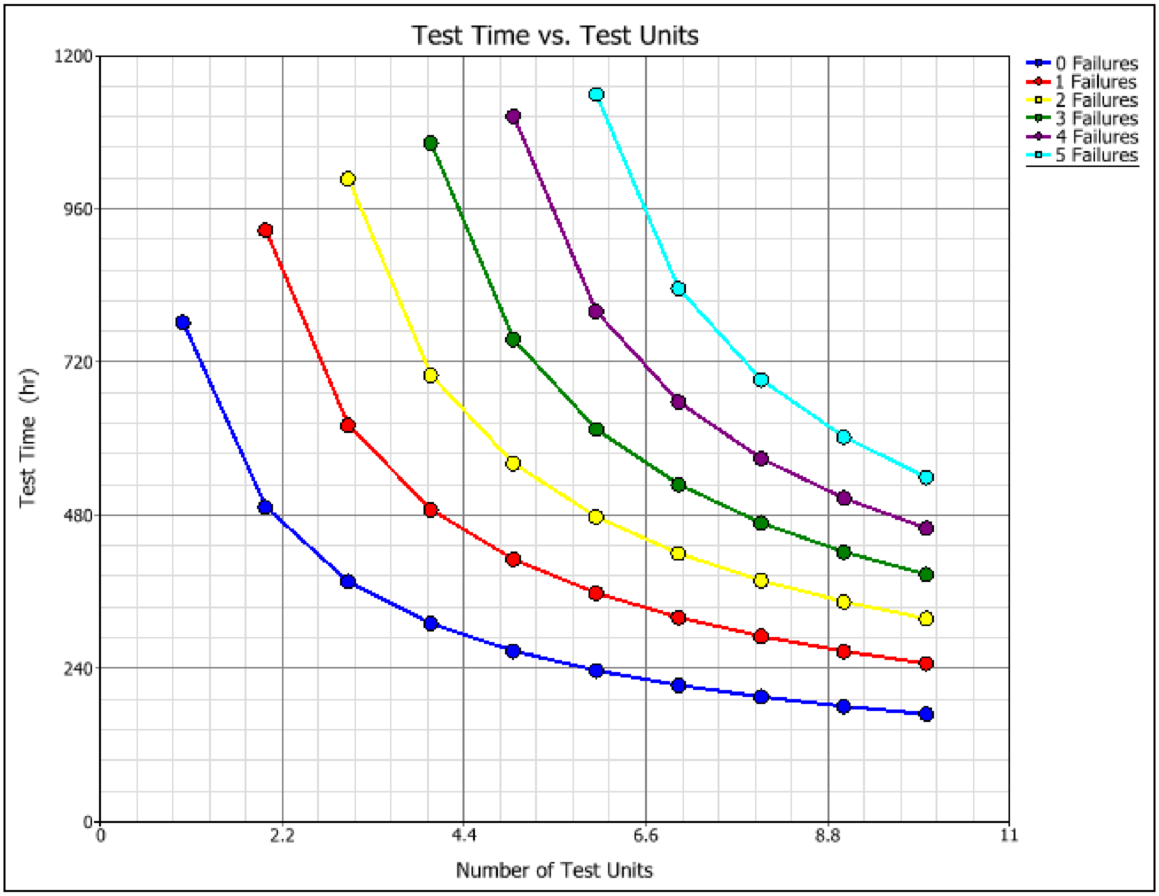
- **Sample size for given test time** solves for sample size given a range of test times. When you select this option, the **Test Time Range** area will require that you enter a starting test time, an ending test time and an increment value by which the test time will increase.
- For either option, you 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.



Reliability Demonstration Test Design Example

The following example demonstrates 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.

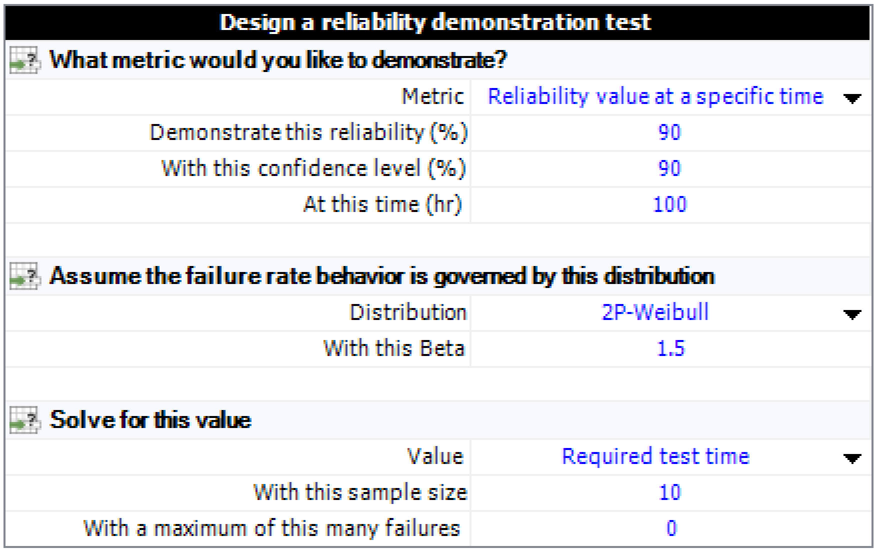
A reliability engineer is asked to plan a test for a component whose failure behavior is governed by a Weibull distribution, where beta = 1.5. The goal of the test is to show, with 90% confidence, that the component has a reliability of at least 90% at 100 hours. The component will be tested at normal 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.

DESIGN A ZERO-FAILURE TEST

1. Open the tool by choosing **Home > Insert > 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. The result shows that 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.

GENERATE A TABLE OF POSSIBLE TEST PLAN SCENARIOS

6. Click the **Show RDT Table** icon on the control panel.

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. A table showing a range of test duration values as a function of sample size and number of allowable failures appears, 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

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. The ReliaWiki resource portal has more information on the Expected Failure Time Plot at: http://www.reliawiki.org/index.php/Reliability Test Design.

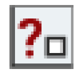
Tip: You can also use this plot to determine whether a specified reliability demonstration test will exceed the allowable number of failures. For example, if you have designed a zero-failure test and wish to determine whether zero failures will occur during the test, you could examine the lower confidence 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.

Generating an Expected Failure Time Plot

Follow the steps outlined below to generate an Expected Failure Time 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 that reduces its life to 50 hours, then the acceleration factor is 2. 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.

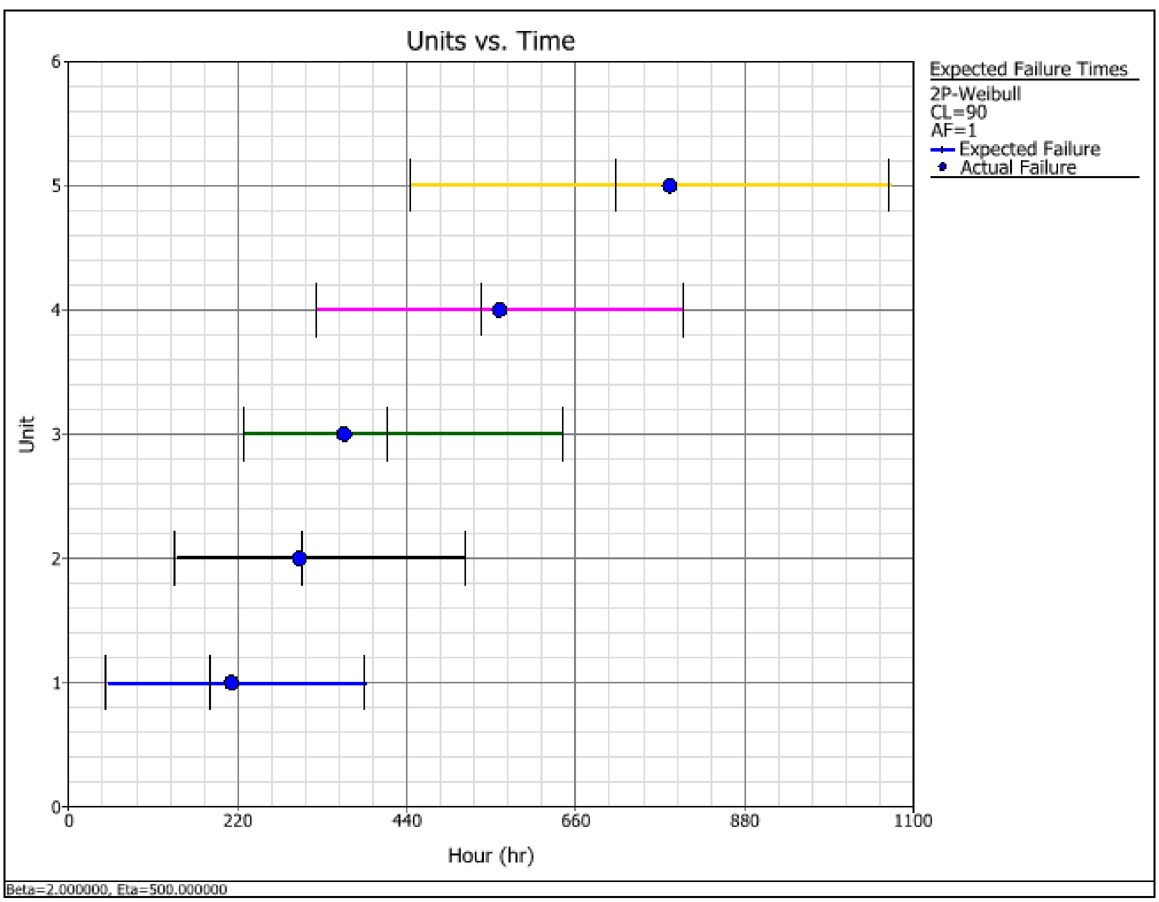
- A window will appear where you can select a life distribution and enter its parameters. If the estimated values of the distribution parameters are not available, you can use the Quick Parameter Estimator (QPE) to solve for them.



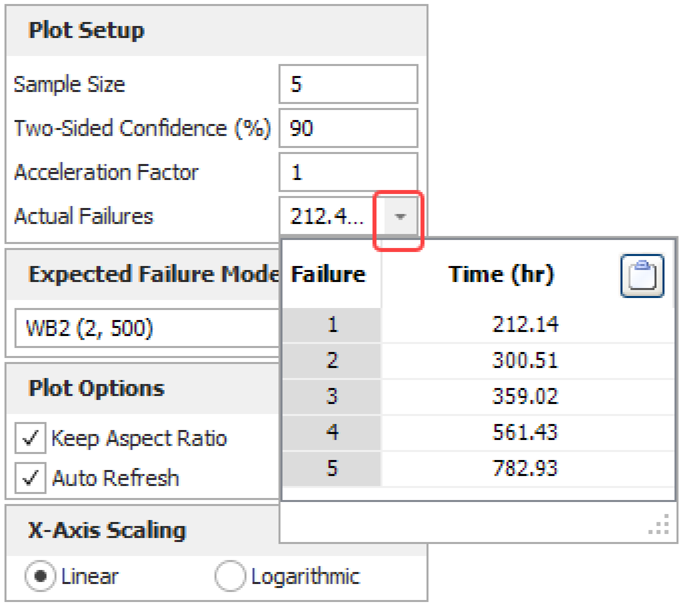
- In the same window, select the time units (e.g., hours) that will be used in the test. Then click **OK** to close the window and generate the plot.

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.

- 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:

- **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.

- **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.
- **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.
- **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.
- **RS Draw** icon launches ReliaSoft Draw, which allows you to annotate your plot and view your plot in greater detail.
- **Export Plot** icon opens the Save As window, which allows you to save the current plot graphic in *.jpg, *.gif, *.png or *.wmf format.

Expected Failure Time Plot 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 90% confidence bounds for the failure times of each unit in the test.

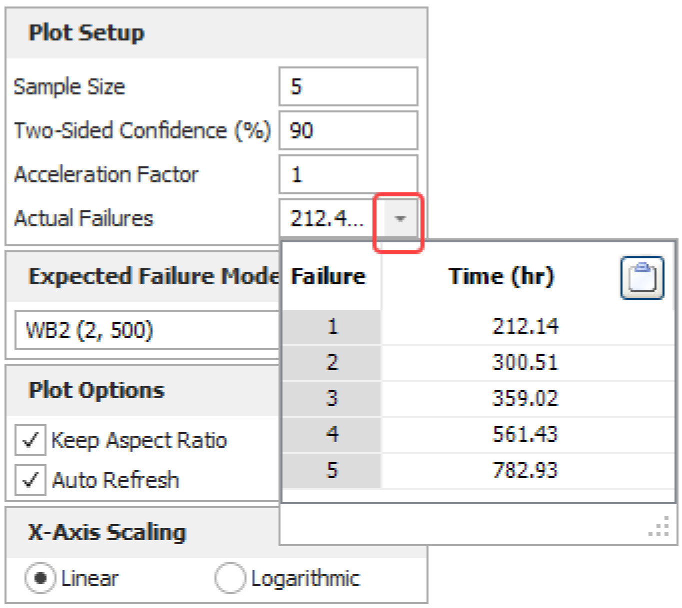
1. On the Home tab, click the **Test Design** icon in the Insert gallery.

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 **90** for the two-sided confidence level. Because the units will be tested under normal operating conditions, enter **1** for the acceleration factor.

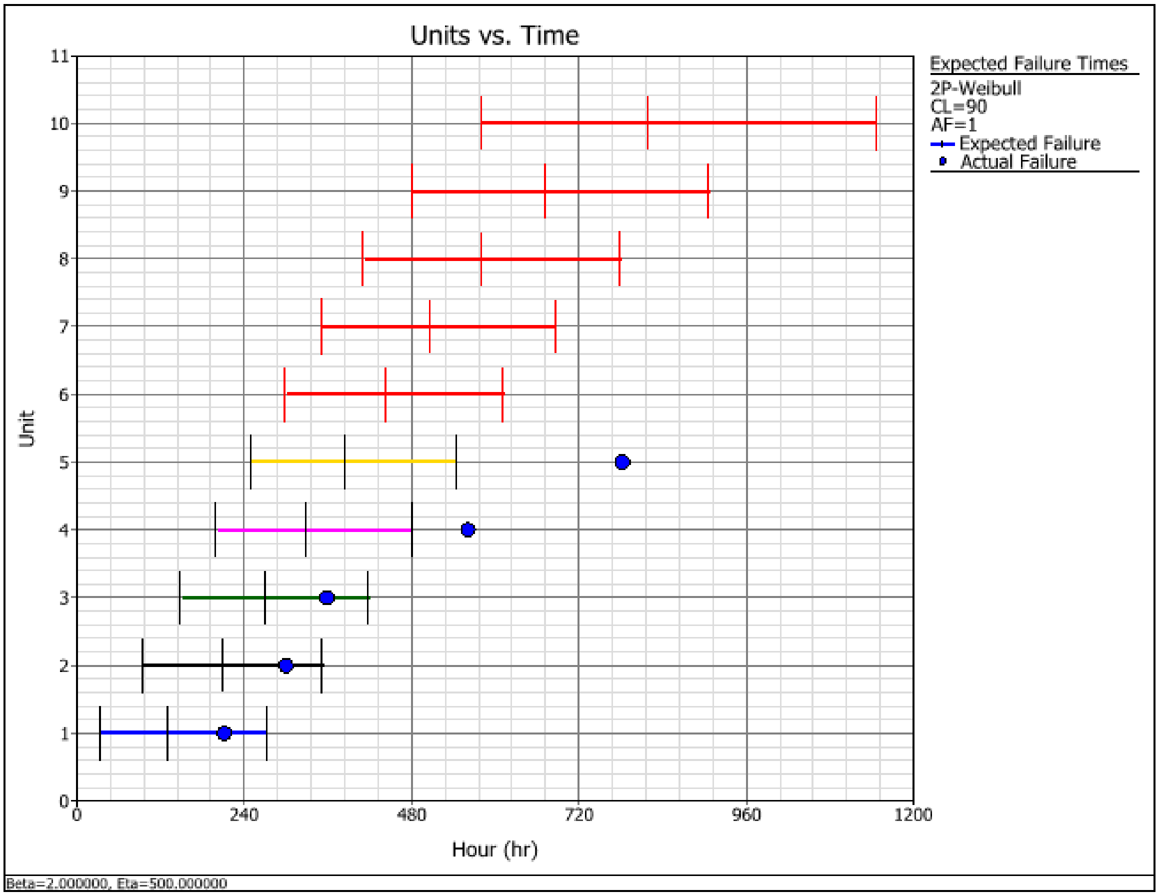
4. Next, enter the life distribution of the product by clicking the arrow in the **Expected Failure Model** area of the control panel. In the window that appears, select **2P-Weibull** for the distribution. Then enter the distribution's parameters and select **Hour** for the time unit.

5. Then click the **Redraw Plot** icon to recalculate the expected failure times. The plot will show the predicted failure times for the ten units, with 90% 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.



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 (e.g., B10 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.

The ReliaWiki resource portal has more information on the Difference Detection Matrix at: http://www.reliawiki.org/index.php/Reliability Test Design.

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	3	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 headings represent possible mean life values for Design 1, and the row headings 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.)

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

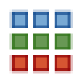
you enter **500** as one of the test times, then the matrix will determine whether 500 hours (assuming your selected time unit is hours) is sufficient to detect a difference in the selected life characteristic. The number of rows available in this table is determined by what you entered in the **Number 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. See Reading the Detection Matrix for information on how to read the results.

  - 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.

Difference Detection Matrix 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. Choose **Home > Insert > 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 default 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 1 Mean Life				
Design 2 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

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. The ReliaWiki resource portal has more information on the Target Reliability tool at: http://www.reliawiki.org/index.php/Target_Reliability_Tool.

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.

Generating a Plot

Follow the steps outlined below to generate a plot.

- Open the tool by choosing **Home > Insert > 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 in 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:
  - **Cost vs. Reliability** 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.
  - **Profit vs. Reliability** 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.
  - **R3OI vs. Reliability** 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.

Control Panel

The Target Reliability Tool control panel contains the following options:

- The **Analysis Details (...)** 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.
- The **RS Draw** icon launches ReliaSoft Draw, which allows you to annotate your plot and view your plot in greater detail.
- The **Export Plot** icon opens the Save As window, which allows you to save the current plot graphic in *.jpg, *.gif, *.png or *.wmf format.

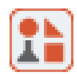
ALTA Test Plan Utility

For life-stress data only.

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.

Follow the steps outlined below to generate a recommended test plan.

1. Open the tool by choosing **Home** > **Insert** > **ALTA Test Plan**.



2. 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).
3. From the **Test Plan Type** drop-down list, choose an appropriate test plan. 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. See ALTA Test Plan Types for details on each test plan.
4. 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**).
5. 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.)
6. 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.

7. 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.

8. 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.

9. 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.

10. Click **Calculate** to generate the recommended test plan. The Test Plan Results sheet will appear. See Test Plan Results Sheet for information on how to read the results and evaluate the plan. You can further evaluate the recommended plan by using the utility's control panel.

ALTA Test Plan Types

For life-stress data only.

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.

ALTA Test Plan Results Sheet

For life-stress data only.

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 the **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 two 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.

26	**Recommended Test Plan**					
27	**Stress Level**	**Stress 1**	**Stress 2**	**Unit Allocation (%)**	**Unit Allocation (Qty)**	**Probability of Failure**
28						
29	First Stress Level	400	25	19.3	9.65	0.987188
30	Second Stress Level	314.515474	10	28.1013	14.05065	0.22706
31	Third Stress Level	275	12.175197	52.5987	26.29935	0.22706
32						
33	**BX% Life Estimate**					
34	Time at Which Unreliability (Tp)=5%	68.085121				
35	Standard Deviation of Tp	10.478742				

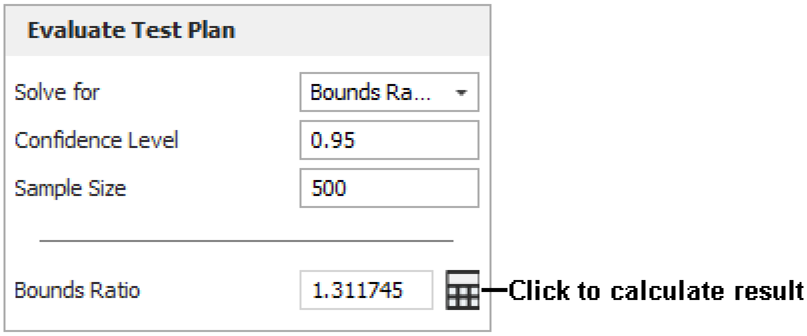
The row labeled "First Stress Level" recommends that 19.3% of the available units be tested at a level of 400 for Stress 1 and 25 for Stress 2. The **Unit Allocation (Qty)** column is visible when the **Show Allocations as Qty** option was chosen from the **Unit Allocation** drop-down list on the Test Plan Inputs sheet. In this example, it shows that 19.3% of the number of available units is 9.65.

- **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 Inputs sheet.
- **Standard Deviation of Tp** is calculated only if you choose **Show Allocations as Qty** on the Test Plan 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.

ALTA Test Plan Control Panel

For life-stress data only.

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: Weibull++ 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.

To solve for one of these criteria, follow the steps outlined below:

- From the **Solve for** drop-down list in 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.
- Click the **Calculate Result** icon in 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.

ALTA Test Plan Utility Example

For life-stress data only.

This topic 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:

1. On the Home tab, click **ALTA Test Plan** icon in the Insert gallery.

2. Select to design a two-stress test by choosing **2** from> the **Number of Simultaneous Stresses** drop-down list.

3. 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.

4. Enter **10** in the **BX% Life Estimate Sought** field to specify that the test will be used to estimate the B10 life.

5. Enter **600** in the **Available Test Time** field.

6. From the **Unit Allocation** drop-down list, choose **Show Allocations as % and Qty**. Then enter **150** in the **Number of Units Available** field.

7. Choose **Weibull** from the **Lifetime Distribution** drop-down list. Then enter **3** in the **Beta** field.

8. 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.

9. 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.

10. 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.

11. Click the **Generate Test Plan** button. The Test Plan Results sheet will appear.

12. 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. 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.

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	357.095521	4	32.682537	49.023805	0.361058
31	Third Stress Level	300	7.285183	47.917463	71.876195	0.361058
32						
33	**BX% Life Estimate**					
34	Time at Which Unreliability (Tp)=10%	1040.496656				
35	Standard Deviation of Tp	96.666847				

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.

1. From the **Solve for** drop-down list in the **Input** area of the control panel, select **Bounds Ratio**.
2. In the **Confidence Level** input field, enter **0.90**.
3. 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.
4. Click the **Calculate Result** icon inside 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.

Repairable Systems Test Design Folio

For reliability growth data analysis only.

In a zero-failure reliability demonstration test (RDT), the engineer aims to demonstrate a specified target metric (e.g., an MTBF of at least 500 hours with 90% confidence) by testing a specified number of systems 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, so 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.

The repairable systems test design folio can assist you in using the NHPP model to design a demonstration test for repairable systems by solving for either the test time required per system or the number of systems that must be tested. To create a test design folio, choose **Home > Insert > Repairable Systems Test Design**.

IMPORTANT: The repairable systems test design folio in Weibull++ (known as the "Repairable Systems - Design of Reliability Tests" tool in prior versions of RGA) is based on the non-homogeneous Poisson process (NHPP), so it is suitable only for designing tests involving repairable systems. For tests involving non-repairable items, use the RDT utility, which provides four test design methods that are suitable for non-repairable systems: parametric binomial, non-parametric binomial, exponential chi-squared and non-parametric Bayesian.

Follow the steps outlined next to use the utility:

- In the **Calculation Options** area on the control panel, specify the **Units** of measurement for the calculation. Note that "Assumes Beta = 1" is displayed in this area, which means that the utility assumes the failure intensity is constant during the test.
- Select the **Display systems as integers** option on the control panel if you will be solving for the number of systems to test and want to round the calculation up to the nearest whole number (e.g., any number greater than 4 would both be displayed as 5).
- In the **What metric would you like to demonstrate?** area on the RDT sheet, specify the metric that you intend to demonstrate with the test you are designing.

    - In the **Metric** drop-down list, specify whether you will demonstrate the system's MTBF or failure intensity.
    - The **Demonstrate this [metric]** field allows you specify the value of the metric you wish to demonstrate. For example, if you wish to demonstrate an MTBF of 1,000 hours, you would enter **1,000** in this field.
    - The **With this confidence level (%)** field allows you to specify, as a percentage, the confidence level at which the target metric will be demonstrated. For example, if you wish to demonstrate with 90% confidence that the MTBF is at least 1,000 hours, enter **90** in this field.
- In the **Solve for this value** area, specify what value you are using the RDT utility to solve for. The **Value** drop-down list allows you to choose whether you are solving for the test time (per system) that will be required to demonstrate the target metric, or the number of systems that you'll need to test.
    - If you select **Required test time**, you'll enter the number of systems that will be tested in the next input field.
    - If you select **Required number of systems**, you'll enter the test time in the next input field.

  In the last input field, enter the maximum number of failures that you will allow to occur during the test. For example, if you want to demonstrate the target metric with 2 failures or less (e.g. two systems fail once, but no other failures occur), then you would enter **2** in this field.

After you have made all of the required inputs, click the **Calculate** icon to solve for the specified value. To examine other possible test scenarios, you can also create a table and plot based on the inputs.

See Repairable Systems RDT Example.

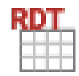
RDT Table and Plot

For reliability growth data analysis only.

In addition to calculating a single value (i.e., required test time per system or number of systems to test), the repairable systems test design folio also allows you to create a table and plot that provide quick ways to consider many possible test plan scenarios without having to perform each calculation individually. 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 plot will graphically display the results with one line for each number of allowable failures.

Follow the steps outlined below to create the table.

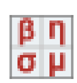
- Open the repairable systems test design folio and solve for required test time or number of systems.
- Click the **Create RDT Table** icon on the control panel.



- The Table of RDT sheet will appear. In the **Calculation Options** area, select which value you wish to have the table solve for.
  - **Test time given number of systems** solves for the test time given different numbers of systems that could be tested. When you select this option, the **Number of Systems Range** area will require that you enter the smallest and largest integer that you want to consider, along with an increment to determine how many values in between will also be calculated. For example, if you specify 10 to 20 with a increment of 2, the table will show results for 10, 12, 14, etc.
  - **Number of systems given test time** solves for the number of systems that must be tested given a different 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 metric if the number of failures exceeds this number. Thus, if you are designing a zero-failure test, then the test will demonstrate the target metric only if no failures occur.

- Click **Calculate** to create the table.

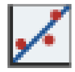


An example table is shown next.

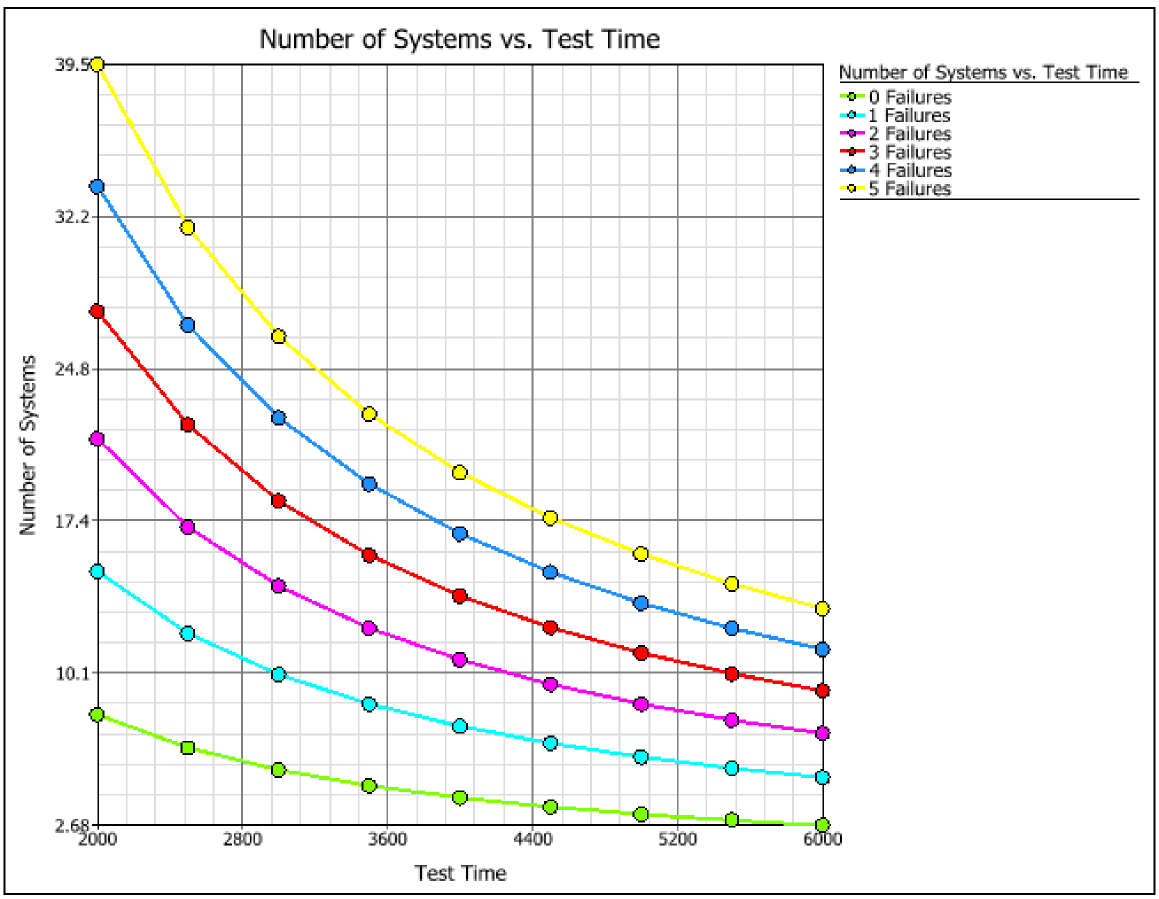
	Amount of Test Time	# of Systems for 0 Failures	# of Systems for 1 Failures	# of Systems for 2 Failures	# of Systems for 3 Failures	# of Systems for 4 Failures	# of Systems for 5 Failures
1	2000	8.04719	14.971542	21.395149	27.575226	33.604898	39.529963
2	2500	6.437752	11.977234	17.116119	22.060181	26.883919	31.62397
3	3000	5.364793	9.981028	14.263433	18.383484	22.403266	26.353308
4	3500	4.598394	8.555167	12.2258	15.757272	19.202799	22.58855
5	4000	4.023595	7.485771	10.697575	13.787613	16.802449	19.764981
6	4500	3.576529	6.654019	9.508955	12.255656	14.93551	17.568872
7	5000	3.218876	5.988617	8.55806	11.03009	13.441959	15.811985
8	5500	2.926251	5.444197	7.780054	10.027355	12.219963	14.374532
9	6000	2.682397	4.990514	7.131716	9.191742	11.201633	13.176654

You can use this table (and the plot of the results shown below) to consider different test scenarios. For example, according to the fifth cell in the column called "# of Systems for 0 Failures," if you tested 5 systems (i.e., 4.0236 rounded up to the nearest whole number) for 4,000 hours each and no failures occurred, then the test would demonstrate the target metric.

- To view a plot of the table results, click the **Plot** icon.



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. To choose which lines to display in the plot, use the options in the **Select Failures** area on the control panel. In the following plot, lines for all numbers of allowable failures considered in the table (i.e., 0 to 5) are shown.



Repairable Systems Test Design Folio Example

For reliability growth data analysis only.

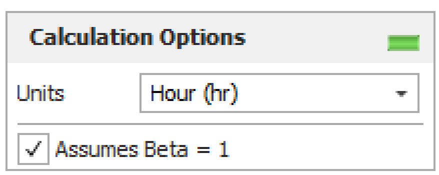
This topic presents an example of how to use the repairable systems test design folio to calculate the number of repairable systems needed to show a specified MTBF at a specified confidence level in a zero-failure demonstration test. An RDT table and plot will also be created in order to examine a variety of different test scenarios.

At the end of a reliability growth testing program for a repairable system, a manufacturer wants to demonstrate that a new product has achieved an MTBF of 10,000 hours with 80% confidence. The available time for the demonstration test is 4,000 hours for each system under test. Assuming zero failures are allowed, determine the required number of systems to be tested in order to demonstrate the desired MTBF.

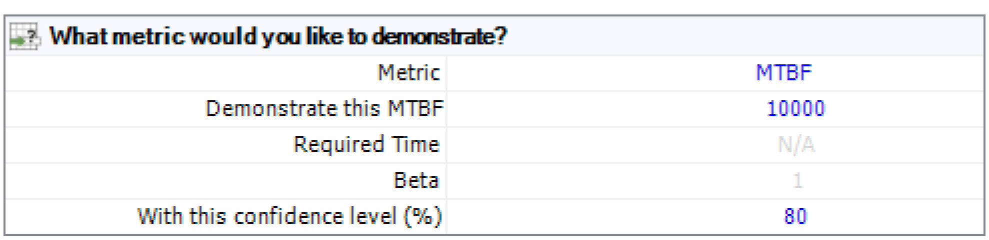
Follow the steps outlined next to solve for the required number of systems.

- Create a new test design folio by choosing **Home > Insert > Repairable Systems Test Design**.

- On the control panel, specify that time values will be in hours, and select to round up the number of systems to the next whole number (e.g., any number above 4 will be displayed as 5), as shown next.



- On the input sheet, specify that you wish to demonstrate an **MTBF** of **10,000** hours with a confidence level of **80**%, as shown next.



What metric would you like to demonstrate?	
Metric	MTBF
Demonstrate this MTBF	10000
Required Time	N/A
Beta	1
With this confidence level (%)	80

- Then specify that you wish to use the folio to solve for the **Required number of systems** that must be tested to demonstrate this value, given an available test time of **4,000** hours and with **0** failures allowed, as shown next.

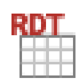
Solve for this value	
Value	Required number of systems
With this test time	4000
With a maximum of this many failures	0

- After you select the **Display systems as integers** option on the control panel, click the **Calculate** icon to view the result.

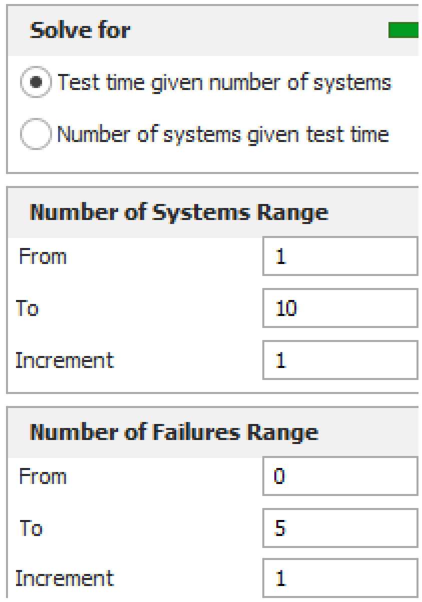
Results	
Number of Systems	**5.000000**

According to the folio, if you test 5 systems for 4,000 hours and no failures occur, then you will demonstrate an MTBF of at least 10,000 hours with 80% confidence.

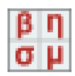
- You can use the RDT table to compare the required test times given different numbers of systems and different numbers of allowable failures. To generate an RDT table using these inputs, click the **Create RDT Table** icon.



- Use the default settings on the control panel of the Table of RDT sheet. According to these settings, the table will calculate the test time required to demonstrate the target metric. The tool will consider different numbers of systems that could be tested, from 1 to 10 in increments of 1. And the number of allowable failures will range from 0 to 5, also in increments of one.



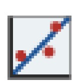
- Click the **Calculate** icon to recreate the table.



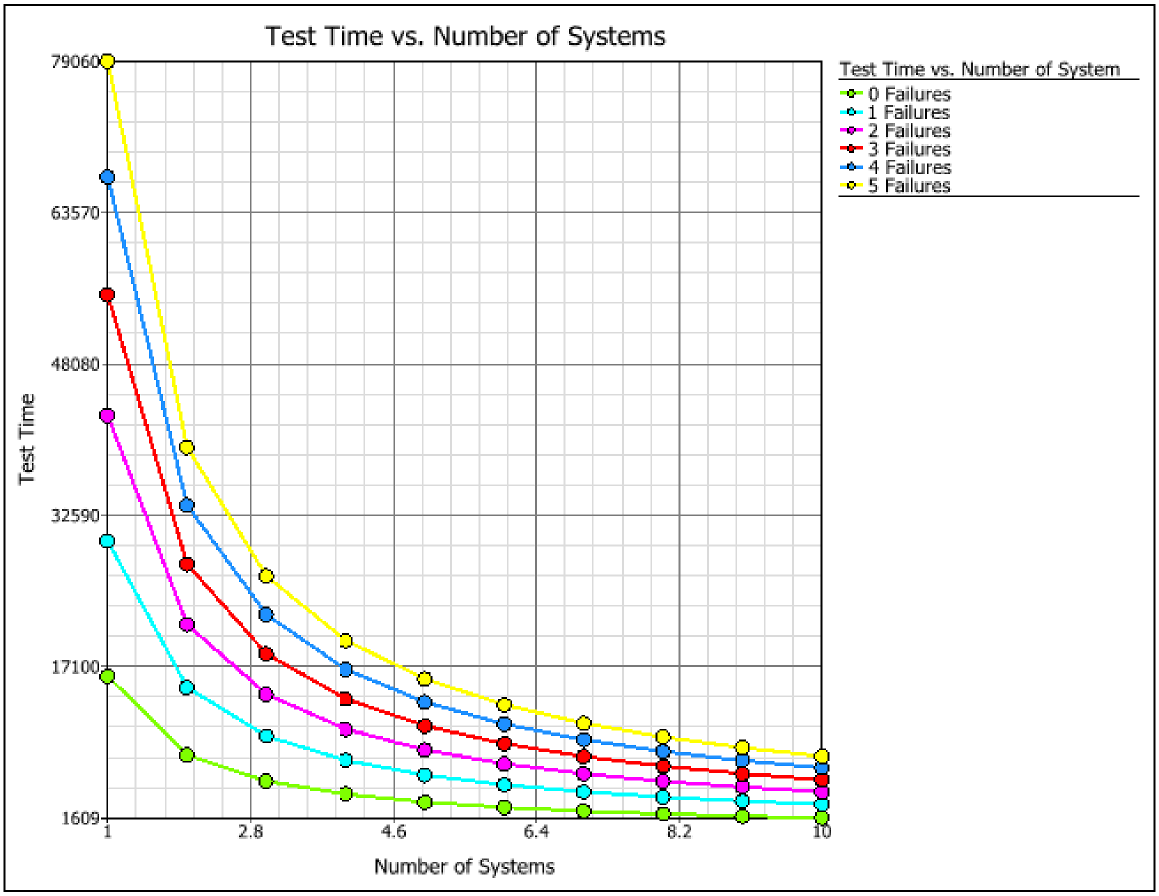
The table appears next. For example, by looking in the second column, you can investigate the test times required in a zero-failure demonstration test, given different numbers of systems being tested.

	Number of Systems	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	16094.37943	29943.08472	42790.29846	55150.45166	67209.79691	79059.92508
2	2	8047.189713	14971.54236	21395.14923	27575.22583	33604.89845	39529.96254
3	3	5364.793142	9981.028239	14263.43282	18383.48389	22403.26564	26353.30836
4	4	4023.594856	7485.771179	10697.57462	13787.61292	16802.44923	19764.98127
5	5	3218.875885	5988.616943	8558.059692	11030.09033	13441.95938	15811.98502
6	6	2682.396571	4990.514119	7131.71641	9191.741943	11201.63282	13176.65418
7	7	2299.197061	4277.583531	6112.89978	7878.635951	9601.399558	11294.27501
8	8	2011.797428	3742.88559	5348.787308	6893.806458	8401.224613	9882.490635
9	9	1788.264381	3327.009413	4754.477607	6127.827962	7467.755212	8784.43612
10	10	1609.437943	2994.308472	4279.029846	5515.045166	6720.979691	7905.992508

- To show the table's results in a plot, click the **Plot** icon.



Like the RDT table, the plot shows the relationship between number of systems and test time, given different numbers of allowable failures. To choose which lines to display in the plot, use the options in the **Select Failures** area on the control panel. In the following plot, lines for all numbers of allowable failures considered in the table (i.e., 0 to 5) are shown.



Measurement Systems Analysis

In order to successfully conduct design of experiments, it is important to use an adequate system for recording measurements (where a "measurement system" includes the operators who perform the measurements, the measuring device that they used and their method for using the device). Weibull++'s DOE feature includes three tools for evaluating and comparing measurement systems. You can use these tools to:

- Help you determine whether you should use a particular measuring device.
- Compare one measuring device to another.
- Compare a measuring device's performance before and after a repair.
- Investigate the sources of variation in the measurement results (e.g., variation could result from differences in how operators use a measuring device).

The three tools for measurement systems analysis are described next:

- The linearity and bias folio is used to estimate a measuring device's *accuracy* (i.e., the extent to which the device underestimates or overestimates the measured values, across the device's expected operating range).
- The gage R&R folio is used to estimate a measurement system's *precision* (i.e., the variation in measurements that can be expected when the same part is measured repeatedly using the same measuring device).
- The gage agreement folio is used to compare the accuracy and precision of two systems.

To add one of these folios to your project, right-click the **Test and Planning** folder in the current project explorer or choose **Home > Insert** then choose the folio type (Linearity and Bias, Gage R&R, or Gage Agreement).

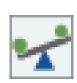
The ReliaWiki resource portal provides more information on measurement systems analysis at: http://www.reliawiki.org/index.php/Measurement_System_Analysis.

Linearity and Bias Folio

The linearity and bias folio is used to estimate the accuracy of a measuring device, or *gage*. Accuracy is defined in terms of *bias* and *linearity*.

- Bias is the difference between the "true" value of a measured part and the observed average. For example, if the reading of a scale is always much higher than the actual weight of the measured object, then the scale has a bias issue.
- Linearity is the relationship between a gage's bias and the "true" value of the measured part. For example, if the bias of a scale increases rapidly as the weight of the measured object increases, then the scale has a linearity issue.

To add a linearity and bias folio to your project, choose **Home > Insert > Linearity and Bias**, or right-click the **Test and Planning** folder on the current project explorer and choose **Add Measurement Systems Analysis > Linearity and Bias** on the shortcut menu.



Linearity and Bias Folio Control Panel

The linearity and bias folio control panel allows you to control the settings for analyzing the measurements obtained from the gage under investigation. It also displays the results of the analysis. This topic focuses on the Main page and Analysis page of the linearity and bias folio control panel, which contains most of the tools you will need to perform the analysis. For more information about the control panel in general, see Control Panels.

CONTROL PANEL MAIN PAGE

- The **Settings** area on the control panel is used to view the settings and status of the analysis. To configure the settings used in the analysis, click the blue text. The risk level setting is accessible on the Analysis Settings page of the control panel.

Settings
Risk Level: 0.1
Analyzed
10 Observations

  In this example:

  - The specified risk level is 10%.
  - "Analyzed" is shown in green, which indicates that the current response data has been analyzed using the current settings. Otherwise, this would show "Modified" in red.
  - The last row indicates that the analyzed data consists of 10 observations.

- The **Analysis Summary** area is shown when the current measurement data has been analyzed using the current settings. This area can display the following values:

Analysis Summary	
% Linearity	5.300%
Linearity	0.0636
Bias	0.0376
% Bias	3.1373%

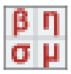
  - **% Linearity** is the percentage of increase of the process variation due to gage linearity. For example, a % Linearity of 5.3 means that the observed variation in measurements will be 5.3% greater than the variation of the parts themselves (i.e., the "true" process variation).
  - **Linearity** represents the variation of the measurements due to the gage linearity. To calculate this value, you must specify the process standard deviation on the Analysis Settings page of the control panel.
  - **Bias** is the difference between the "true" value of a measured part and the observed average. A positive bias means the gage overestimates. A negative bias means the gage underestimates.

- **% Bias** is the ratio of the bias to the process variation. To calculate this value, you must specify the process standard deviation on the Analysis Settings page of the control panel.

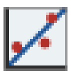
Click the **View Analysis Summary** icon in this area to view the details of the analysis results.

Folio Tools

The folio tools are arranged on the left side of the control panel. Use these tools to configure your analysis and experiment with the analysis results.



Calculate analyzes the measurements and displays a summary of the results on the control panel.


Plot creates a plot based on the analysis. If you click this icon before the current data set has been analyzed, an analysis will be performed automatically. (See Linearity and Bias Folio Plots.)

View Analysis Summary opens a window that contains detailed information about current and past analysis results.

CONTROL PANEL ANALYSIS SETTINGS PAGE

The Analysis page of the linearity and bias folio control panel includes the risk level and process standard deviation (if any) that will be used for the analysis.

- The **Risk Level**, or alpha value, is a measure of the risk that the analysis results are incorrect (i.e., alpha = 1 - confidence level).

- The **Process Standard Deviation** area allows you to specify the standard deviation of the manufacturing production process (i.e., the standard deviation of the parts). When the check box is selected, the software will multiply the value by 6 to obtain the process variation (e.g., if the diameter in centimeters is being measured and the standard deviation is 1, then the process variation is 6 centimeters). When this value is provided, the analysis results will include Linearity and % Bias values.

Linearity and Bias Folio Data Tab

The Data tab of the linearity and bias folio contains a control panel and a sheet with the following four columns for entering data:

- **Part** is the identifier for the part that was measured.
- **Reference** is the "true" value of the measured part (also called the *master* value). If the true value is unknown, the reference value can be the average of several measurements obtained via the most accurate measuring equipment available.
- **Reading** is the measurement obtained from the gage under investigation.
- **Trial** is used to help you keep track of multiple trials that are used in a particular study. For example, the first measurement of every part would be considered trial #1, the second measurement trial #2, and so on (as shown in the example data set below). These values are not used in the calculations.

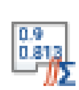
As an example, suppose a gage was used to measure the diameter of two objects. The first object is 2 inches in diameter, and the second is 4 inches. Each object was measured five times. The resulting data set is shown next. For example, the first row shows that Part 1 has an actual diameter of 2 inches, but the gage produced a measurement of 1.95 inches.

	Part	Reference	Reading	Trial
1	1	2	1.95	1
2	1	2	2.1	2
3	1	2	2	3
4	1	2	1.92	4
5	1	2	1.97	5
6	2	4	4.09	1
7	2	4	4.16	2
8	2	4	4.16	3
9	2	4	4.1	4
10	2	4	4.06	5

Linearity and Bias Folio Analysis Results

When accessed from a linearity and bias folio, the Analysis Summary window will contain detailed information about the gage's bias and linearity.

If the current response data has been analyzed, you can open the window by clicking the **View Analysis Summary** icon on the control panel.



If the current response data has not been analyzed, the icon will still be available so you can view the folio's analysis history.

Select an item in the Available Report Items panel to display it on the spreadsheet. Each item is described next.

ANALYSIS RESULTS

The **Analysis of Variance (ANOVA) table** provides general information about the effects of the reference value on the bias.

ANOVA Table Columns

- **Source of Variation** is the source that caused the difference in the measurements. Sources displayed in red are considered to be significant.
- The number of **Degrees of Freedom** for the **Reference** is the number of regression coefficients for the reference value in the model. The number of degrees of freedom for the **Residual** is the total number of observations minus the number of parameters being estimated.
- **Sum of Squares** is the amount of difference in measurements caused by this source of variation.
- **Mean Squares** is the average amount of difference caused by this source of variation. This is equal to Sum of Squares/Degrees of Freedom.
- **F Ratio** is the ratio of Mean Squares of this source of variation and Mean Squares of pure error. A large value in this column indicates that the difference in the output caused by this source of variation is greater than the difference caused by noise (i.e., this source affects the output).
- **P Value** (alpha error or type I error) is the probability that an equal amount of variation in the measurements would be observed in the case that this source does not affect the output. This value is compared to the risk level (alpha) that you specify on the Analysis Settings page of the control panel. If the *p* value is less than alpha, this source of variation is considered to have a significant effect on the output (e.g., if Reference has a significant effect, then the gage has a linearity issue). In this case, the term and its *p* value will be displayed in red.

- The following values are shown underneath the ANOVA table:
  - **S** is the standard error of the noise. It represents the magnitude of the difference caused by noise.
  - **R-sq** is the percentage of total difference in the measurements that is attributable to differences in the reference values. It is equal to Sum of Squares(Reference)/Total Sum of Squares.
  - **R-sq(adj)** is an R-sq value that is adjusted for the number of parameters in the model.

The **Linearity Analysis table** provides specific information about the linearity of the measurement device.

Linearity Analysis Table Columns

- **Term** is the intercept or slope of the fitted regression model. The slope represents the measurement device's linearity. Terms displayed in red are considered to be significant (e.g., if "Slope" is displayed in red, then the device has a linearity issue).
- **Coefficient** is the regression coefficient of the term, which represents the contribution of the term to the variation in the response.
- **Standard Error** is the standard deviation of the regression coefficient.
- **Low Confidence** and **High Confidence** are the lower and upper confidence bounds on the regression coefficient.
- **T Value** is the normalized regression coefficient, which is equal to Coefficient/Standard Error.
- **P Value** (alpha error or type I error) is the probability that an equal amount of variation in the output would be observed in the case that this term does not affect the output. This value is compared to the risk level (alpha) that you specify on the Anaysis Settings page of the control panel. If the *p* value is less than alpha, this source of variation is considered to have a significant effect on the output. In this case, the term and its *p* value will be displayed in red.
- Additional values will be shown underneath the Linearity Analysis table:
  - If you specified the process standard deviation on the Analysis Settings page of the control panel, these values will be shown.

    - **Process Variation** is the total variation of the parts' reference values (i.e., the "true" process variation). It is equal to the specified process standard deviation * 6.
    - **Linearity** is the variation in measurements due to gage linearity.
  - The **% Linearity** value is always shown. It is the percentage of the variation in measurements that is due to gage linearity.

The **Bias Analysis table** provides specific information about the bias of the measurement device.

Bias Analysis Table Columns

- **Reference** is the reference value entered in the data sheet. It represents the "true" value of the part when it was measured. If a reference value is displayed in red, then there is a significant bias for measurement taken at that value.
- **Bias** is the average bias for measurements at the given reference value.
- **% Bias** is the ratio of the bias to the process variation at the given reference value. This column is available only when the process standard deviation is specified on the Analysis Settings page of the Data tab control panel.
- **Std. of Mean** is the standard deviation of the bias at each reference value. If there are multiple parts with the same reference value, it is the pooled standard deviation of all the parts.
- **T Value** is the ratio of the absolute value of the Bias column and the Std. of Mean column. It is used to calculate the *p* value.
- **P Value** is calculated from the *T* value and the corresponding degree of freedom for each reference value. If the *p* value is smaller than the risk level specified on the Analysis Settings page of the control panel, then the corresponding row has significant bias. In this case, the *p* value will be shown in red.
- **Average** shows the mean of all the values in the given column. If "Average" is displayed in red, then the gage's average bias is significant.

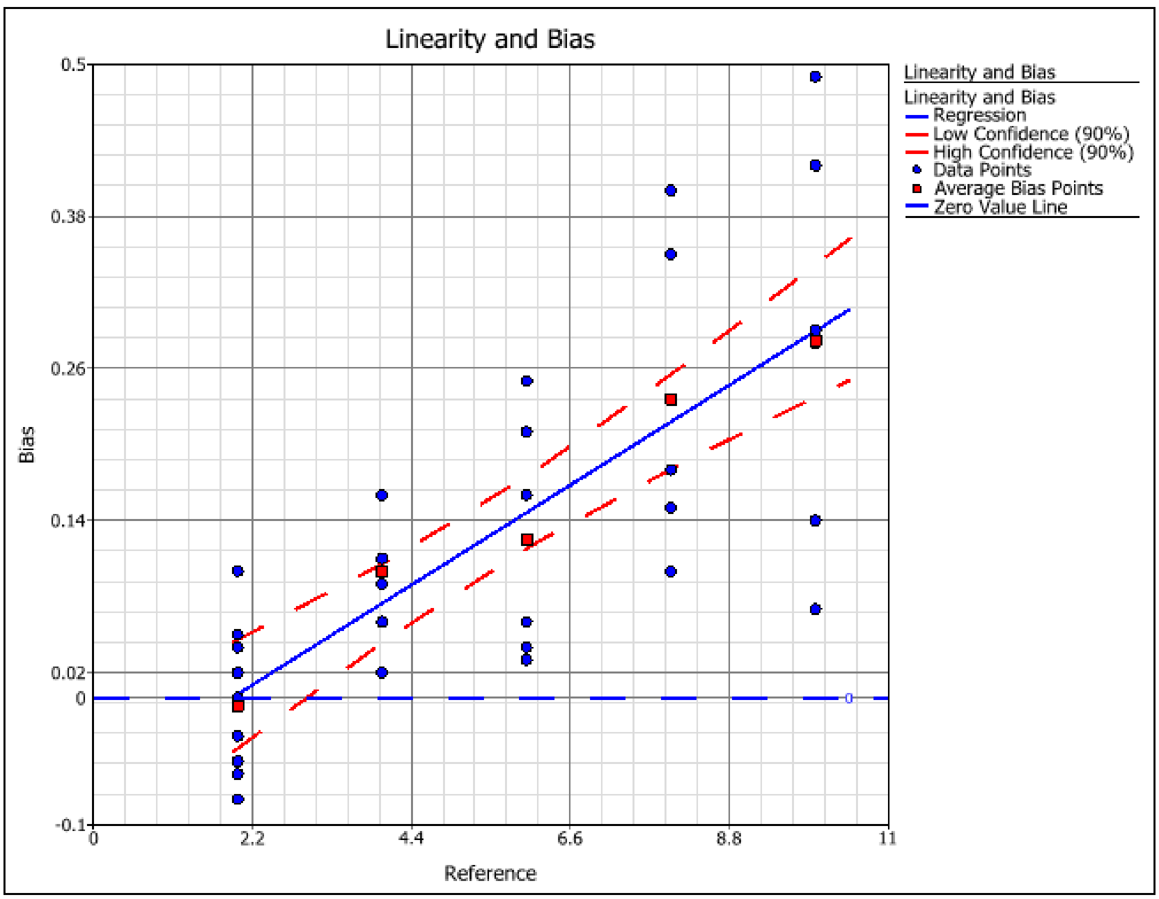
Linearity and Bias Folio Plots

The following plots are available in the linearity and bias folio. They allow you to visually assess the accuracy of the measurement system. For general information on working with plots, see Plot Utilities.

- The **Process Variation** bar chart is available only when you specify the process standard deviation on the Settings page of the Data tab control panel. This chart shows the percentage of the observed process variation (i.e., total variation in the parts plus variation due to linearity/bias) that is due to linearity and the percentage due to bias.

- The **Bias vs. Part** plot shows the individual and average bias values sorted by the measured parts. Individual bias values (i.e., the differences between the reference values and particular measurements) are plotted as blue points, and average values (i.e., the differences between the reference values and the observed averages) are plotted as red squares. A blue dotted Zero Value Line marks the point at which there is no bias. You can use this plot to examine how the bias of the system (and the variation of the observed values) varies depending on the part.
- The **Linearity and Bias** plot shows the individual and average bias values versus the reference values. The blue dotted Zero Value Line marks the point at which there is no bias. In addition, a regression line is fitted to the average bias points, showing the relationship between bias and reference (i.e., linearity). Confidence bounds are also drawn on the fitted line. The confidence level of the bounds is based on the risk level entered on the Settings page of the Data tab control panel (e.g., if the risk level is 0.1, then 90% two-sided upper and lower bounds are drawn on the line).

  For example, the following plot shows that, as the reference value increases, the bias increases (i.e., there is linearity in the gage). In addition, you can see that when the reference value is 2, there is almost no bias (i.e., the average bias is almost zero).



Linearity and Bias Folio Example

*The data set used in this example is available in the example database installed with the software (called "Weibull21_DOE_Examples.rsgz21"). 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 example project is "Gage Linearity and Bias Study."

In this example, a gage linearity and bias study was conducted to evaluate a gage that is used to measure the diameter of holes. The study will assess the gage's *linearity*, which represents its accuracy across the expected range of measurements, and its *bias*, which represents the difference between the gage's measurements and the actual value of the measured part. This will be done by using the gage to measure holes in an NIST standard coupon, where the true diameter of each hole is known. If the gage is found to be unacceptable, a new gage will be evaluated.

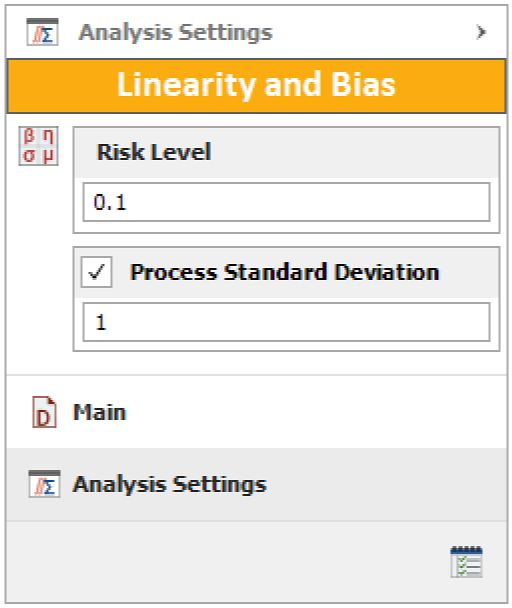
The data set or the study is given below and has been entered in the example folio. The first thirteen rows of the data sheet are shown next.

	Part	Reference	Reading	Trial
1	1	2	1.95	1
2	1	2	2.1	2
3	1	2	2	3
4	1	2	1.92	4
5	1	2	1.97	5
6	1	2	1.94	6
7	1	2	2.02	7
8	1	2	2.05	8
9	1	2	1.95	9
10	1	2	2.04	10
11	2	4	4.09	1
12	2			
13	2			

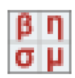
Note: Complete data set is not shown.

In the first row, you can see that the first part (Part 1) has an actual value of 2, and the first measurement of the part was 1.95. The Trial column (which is not used in the calculations) was used to keep track of how many times each part was measured, and in this case you can see that Part 1 was measured a total of 10 times. If you examine all the data points in the example project, you will see that a total of five parts were measured, and some were measured fewer than 10 times. To avoid any variation in measurement that could be due to different operators, all measurements in this study were performed by the same operator.

On the Analysis Settings page of the control panel, you can see that a risk level of 0.1 (10%) will be used to determine whether there is a statistically significant bias or linearity in the gage. In addition, you can see that a process standard deviation of 1 was specified.



To analyze the data, click the **Calculate** icon.



The following summary of results will appear on the control panel.

Analysis Summary	
% Linearity	3.581320%
Linearity	0.214879
Bias	0.125294
% Bias	2.088235%

- **% Linearity** is the percentage of increase of the process variation due to gage linearity. In this example, the observed variation in measurements will be 3.5813% greater than the variation of the parts themselves (i.e., the "true" process variation). Because this is a relatively small value, linearity is not considered a problem for this gage.
- **Linearity** represents the variation of the measurements due to the gage linearity. To calculate this value, you must specify the process standard deviation on the Analysis Settings page of the control panel.

- **Bias** is the difference between the "true" value of a measured part and the observed average. A positive bias means the gage overestimates. A negative bias means the gage underestimates.

- **% Bias** is the ratio of the bias to the process variation. To calculate this value, you must specify the process standard deviation. The lower the % Bias, the better the gage. In this case, the % Bias is 2.0882%, which is also small enough to consider the bias acceptable.

The bias values for each reference and the overall study can be seen by clicking the **View Analysis Summary** icon on the control panel. The results of the bias analysis are shown next.

Bias Analysis

Bias Analysis					
Reference	Bias	% Bias	Std. Mean	T Value	P Value
2	-0.006	0.1	0.01827	0.328415	0.75011
4	0.1	1.666667	0.019149	5.22233	0.001972
6	0.125	2.083333	0.038536	3.243746	0.022854
8	0.236	3.933333	0.058703	4.020262	0.015861
10	0.281667	4.694444	0.065188	4.320852	0.007565
Average:	0.125294	2.088235	0.017043	7.351698	4.242048E-8
% Bias =	2.088235%				

The *p* values show that some of the biases are significant when compared to the variation of the measured standard coupon. This is understandable since the variation of the standard coupon is usually very small. However, when compared to process variation, the average bias of this gage (i.e., average % bias) is still relatively small, so the gage is considered acceptable.

Gage R&R Folio

The gage R&R folio is used to estimate the precision of a measurement system, or the system's ability to produce the same value when used to measure the same part multiple times. Precision is defined in terms of *repeatability* and *reproducibility* (R&R).

- Repeatability is the variation in measurements when the same operator measures the same part multiple times with the same gage. If this variation is too high, then the gage may be unacceptable.

- Reproducibility is the variation in measurements when different operators measure the same part multiple times with the same gage. If this variation is too high, then the operators are not using the gage in the same way.

To add a gage R&R folio to your project, choose **Home > Insert > Gage R&R**, or right-click the **Test and Planning** folder on the current project explorer and choose **Add Measurement Systems Analysis > Gage R&R** on the shortcut menu.

Gage R&R Folio Data Tab

The Data tab of the gage R&R folio contains a control panel and a sheet with the following four columns for entering data:

- **Part** is the identifier for the part that was measured.
- **Operator** is the identifier for the person who performed the measurement.
- **Reading** is the measurement obtained from the gage under investigation.
- **Trial** is used to help you keep track of multiple trials that are used in a particular study. For example, in a crossed gage R&R study, the first measurement of a part by an operator would be trial #1, the second measurement by the same operator trial #2, and so on. These values are not used in the calculations.

As an example, suppose a scale used to weigh two parts (in pounds). Part 1 is measured four times: twice by Operator A and twice by Operator B. Part 2 is also measured twice by each operator. The resulting data is shown next. For example, the first row shows that Part 1 was measured by Operator A, who obtained a measurement of 9.856 pounds.

	Part	Operator	Reading	Trial
1	1	A	9.856	1
2	1	A	8.278	2
3	1	B	10.72	1
4	1	B	7.477	2
5	2	A	8.642	1
6	2	A	7.754	2
7	2	B	9.018	1
8	2	B	7.454	2

Gage R&R Folio Control Panel

The gage R&R folio control panel allows you to control the settings for analyzing the measurement system data. It also displays the results of the analysis. This topic focuses on the Main page and Analysis page of the gage R&R folio control panel, which contains most of the tools you will need to perform the analysis. For more information about the control panel in general, see Control Panels.

CONTROL PANEL MAIN PAGE

- The **Settings** area of the control panel displays the settings and status of the analysis. To configure the settings used in the analysis, click the blue text. The risk level and design type settings are accessible on the Analysis Settings page of the control panel.

Settings
Risk Level: 0.1
CrossedDesign
Analyzed
27 Observations

In this example:

- The specified risk level is 10%.
- A crossed (rather than nested) gage R&R design type will be used. This means each operator will measure the same part multiple times.
- "Analyzed" is shown in green, which indicates that the current response data has been analyzed using the current settings. Otherwise, this would show "Modified" in red.
- The last row indicates that the analyzed data consists of 27 observations.
- The **Analysis Summary** area is shown when the current measurement data has been analyzed using the current settings. This area breaks down the percentage of measurement variance due to repeatability, reproducibility and the measured part.

Analysis Summary	
Variance Results	
Repeatability	23.159522%
Reproducibility	61.227832%
Operator	57.426911%
Operator * Part	3.800921%
Part	15.612646%
Total Gage RR	84.387354%
Total Variation	100.000000%

- **Repeatability** is the variance due to properties of the gage.
- **Reproducibility** is the variance due to the operator or the interaction between the operator and the measured parts. It is the sum of these variance values.

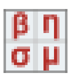
- **Operator** is the variance due to the operators' different ways of performing measurements on the same parts.
- **Operator * Part** is the variance due to the operators' different ways of performing measurements on different parts. To calculate this result, you must go to the Analysis Settings page of the control panel and select to include it in the analysis.

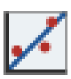
- **Part** is the variance due to differences in the part being measured (i.e., the part-to-part variance).
- **Total Gage R&R** is the total of repeatability and reproducibility.
- **Total Variation** is the total of the part-to-part variance, repeatability and reproducibility.

Click the **View Analysis Summary** icon in this area to view the details of the analysis results.

Folio Tools

The folio tools are arranged on the left side of the control panel. Use these tools to configure your analysis and experiment with the analysis results.


Calculate analyzes the measurement data and displays a summary of the results on the control panel.


Plot creates a plot based on the analysis. If you click this icon before the current data set has been analyzed, an analysis will be performed automatically.


View Analysis Summary opens a window that contains detailed information about current and past analysis results.

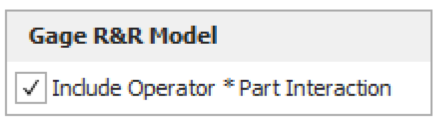
CONTROL PANEL ANALYSIS SETTINGS PAGE

The Analysis Settings page of the gage R&R folio control panel includes the following settings.

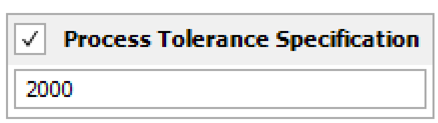
- The **Risk Level**, or alpha value, is a measure of the risk that the analysis results are incorrect (i.e., alpha = 1 - confidence level).

- The **Gage R&R Design Type** area allows you to select which type of gage R&R design you will use.

  - With the **Crossed** design type, each operator measures the same part multiple times. For example, if Operator A measures Part 1 twice and Part 2 twice, and Operator B does the same, then the design is crossed. This design type is preferable when individual parts can be measured repeatedly.
  - With the **Nested** design type, each part is measured multiple times by only one operator. For example, if Operator A measures Part 1 twice and Part 2 twice, and Operator B measures Part 3 twice and Part 4 twice, then the design is nested. In this case of destructive testing, this type of design assumes that all the measured parts of the same type (e.g., Part 1) are similar enough to be considered identical.
- The **Gage R&R Model** area allows you select whether the analysis model will include terms that represent the interaction between operators and parts. This option is available for crossed designs only.



- **Process Tolerance Specification** area allows you to specify the range of values that are acceptable for the measured product. For example, if the specification for a part is 14.5 ± 0.5 µm, then the process tolerance specification is 1 µm. When this value is provided, the calculation results will include the ratio of measurement variation to the process tolerance.



Gage R&R Folio Analysis Results

When accessed from a gage R&R folio, the Analysis Summary window will contain detailed information about the gage's precision.

If the current response data has been analyzed, you can open the window by clicking the **View Analysis Summary** icon on the control panel.

If the current response data has not been analyzed, the icon will still be available so you can view the folio's analysis history.

Select an item in the Available Report Items panel to display it on the spreadsheet. Each item is described next.

ANALYSIS RESULTS

The **Analysis of Variance (ANOVA) table** provides general information about the effects of the operator on the measurements.

ANOVA Table Columns

- **Source of Variation** is the source that caused the difference in the measurements. Part (Operator) is the part variation within operators, and it is shown only for nested designs.
- The number of **Degrees of Freedom** is the number of regression coefficients in the regression model for this source of variation.
- **Sum of Squares** is the amount of difference in measurements caused by this source of variation.
- **Mean Squares** is the average amount of difference caused by this source of variation. This is equal to Sum of Squares/Degrees of Freedom.
- **F Ratio** is the ratio of Mean Squares of this source of variation and Mean Squares of pure error. A large value in this column indicates that the difference in the output caused by this source of variation is greater than the difference caused by noise (i.e., this source affects the output).
- **P Value** (alpha error or type I error) is the probability that an equal amount of variation in the measurements would be observed in the case that this source does not affect the output. This value is compared to the risk level (alpha) that you specify on the Analysis Settings page of the control panel. If the *p* value is less than alpha, this source of variation is considered to have a significant effect on the output. In this case, the term and its *p* value will be displayed in red.

The **Gage Variance Results table** shows the variance and standard deviation of variance components.

Gage Variance Results Table Columns

- **Source** is the source of variation in the measurements. Total Gage R&R is the sum of Repeatability and Reproducibility, and Total Variation is the sum of Total Gage R&R and Part. Part(Operator) is the part variation within operators and is shown for nested designs.
- The **Variance Results** section includes these columns:
  - **Variance** is a measure of how far the measurements are spread out due to this source of variation.
  - **Variance Contribution** is the percentage of variance that is due to this source of variation.
- The **Standard Deviation** section includes these columns:
  - **Standard Deviation** is the standard deviation that is due to this source.
  - **Standard Deviation Ratio (%)** is the percentage of the total standard deviation that is due to this source.
  - **Study Variation** represents the spread of values that could result from this source of variation. It is equal to 6 * the standard deviation.
  - **Study Variation to Tolerance Ratio** is the Study Variation divided by the process tolerance specification entered on the Analysis Settings page of the control panel. Thus, a ratio of 100% means the variation in measurements is equal to the acceptable range of part values.

Gage R&R Folio Plots

The following plots are available in gage R&R folios, depending on whether you are using a crossed or nested design. They allow you to visually display the data and examine the precision of the measurement system. For general information on working with plots, see Plot Utilities.

- The **Variation Pie** includes several pie charts that allow you to visually examine the sources of variation in the measurements.
  - The **Total Variation Component** chart displays the ratio of each source of variance to the total variance.

- The **Gage and Part Variation** chart displays the ratio of the total gage variance (reproducibility + repeatability) to the total variance, and the ratio of the part variance to the total variance.
- The **Gage R&R Variation** chart displays the percentage of repeatability and reproducibility to the total gage variance.
- The **Gage Reproducibility** chart is available only for crossed designs with the operator * part interaction included in the model. It further decomposes reproducibility to operator variance and operator-part interaction variance.

- The **Run Chart** is available only for crossed designs. Each column shows the measurements that were taken of one part, and the measurements are grouped and color-coded according to the operator who performed the measurements. The blue dotted Mean line is the mean of all the measurements. You can use this plot to look for trends that depend on the operators. For example, this chart could show you that one operator's average measurement is biased to low, or that one part was measured in inconsistent ways.
- The **Variation Components** bar chart shows the contribution of different sources of variation to the total measurement variation. You can use this chart to compare the Part variation to variation that is due to the gage (Repeatability), the operators (Reproducibility) or the sum of both (Total Gage R&R).
- The **Measurement by Part** plot displays the measurements grouped by part. (With nested designs, the x-axis label "Part(Operator)" indicates that one operator measured each part.) The connected red dots in each group are the mean measurements for each part. You can use this plot to study how the variation and mean values depend on the parts. For example, you might conclude from this plot that measurements for one part have less variation than that of the other parts.
- The **Measurement by Operator** plot displays the measurements grouped by operator. The connected red dots in each group are the mean measurements from each operator. You can use this plot to study how the variation and mean values depend on the operator. For example, you might conclude from this plot that one operator tends to record higher values than the others.
- The following charts are used to assess the system's reproducibility. The control limits are marked with red dotted lines near the top and bottom of the chart. If one operator is outside the control limits, then that operator's method differs from the others. If all the operators have at least one measurement outside the limits, then the measurement system is overly sensitive to operator technique and has a reproducibility issue.

- The **R-Chart by Operator** ("R" stands for "Range") is available when 10 or fewer measurements for each part are included. The blue dotted Mean line is the average range of measurements for all operators. Each blue dot represents the range of measurements for a particular part when measured by a particular operator.
- The **S-Chart by Operator** ("S" stands for "Standard Deviation") is available when more than 10 measurements of each part per operator are included. The horizontal blue line is the average standard deviation of measurements for all operators. Each blue dot represents the standard deviation of measurements for a particular part when measured by a particular operator.

- The **X-Bar by Operator** plot shows how the mean reading changes across the parts. The mean readings are grouped by operator. The blue dotted Mean line is the overall mean of all the measurements. Red dotted lines mark the control limits. Since the control limits are calculated using the repeatability variation, most data points should fall outside the control limits. If they do not, then the system has a repeatability issue.
- The **Operator * Part Interaction** plot is similar to the X-bar by operator plot, except the lines for each operator are color-coded and placed on top of each other. This plot is useful for assessing whether the operator * part interaction has a significant effect on the measurements. If the operator lines are the same shape (i.e., the segments are parallel to each other), the effect is not significant.

Gage R&R Folio Example

*The data set used in this example is available in the example database installed with the software (called "Weibull21_DOE_Examples.rsgz21"). 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 example project is "Gage R and R Study."

A gage R&R study was conducted using a crossed experiment, where each operator measures each part multiple times. The goal of the study is to determine whether the operators are using the gage in a consistent manner, and whether the gage itself is sufficiently precise. The data set is given in the "Gage R and R Crossed Design" folio. The product tolerance is 2,000 (i.e., the range of acceptable values for the product is 2,000). We want to evaluate the precision of this gage using the following three ratios:

- The *precision-to-tolerance ratio* (or P/T ratio) is the variation due to the measuring system (i.e., the operators and the gage) divided by the product tolerance.
- The *gage-to-part variation ratio* is the variation due to the measuring system divided by the variation in the parts themselves (i.e., part-to-part variation).

- The *gage-to-total variation ratio* is the variation due to the measuring system divided by the total variation in measurements.

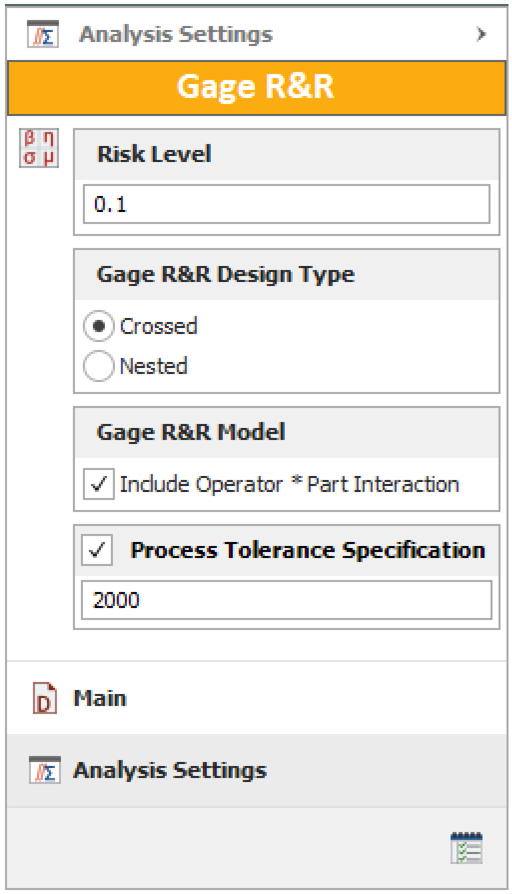
The data set for this example has been entered on the Data tab of the folio. There are 3 parts and 3 operators in the experiment. Each operator measured each part 3 times. The first 15 rows are shown next.

	Part	Operator	Reading	Trial
1	1	A	405	1
2	1	A	232	2
3	1	A	476	3
4	1	B	389	1
5	1	B	234	2
6	1	B	456	3
7	1	C	684	1
8	1	C	674	2
9	1	C	634	3
10	2	A	409	1
11	2	A	609	2
12	2	A	444	3
13	2	B	506	1
14	2			
15	2			

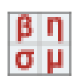
Note: Complete data set is not shown.

For example, you can see that Operator A measured Part 1 three times and measured Part 2 three times. Each measurement is shown in the Reading column.

On the Analysis Settings page of the control panel, the risk level and design type are specified. In addition, the operator * part interaction is selected to be included in the analysis, and the process tolerance specification is specified.



Click the **Calculate** icon to analyze the data.



A summary of the results will appear on the Main page of the control panel. It shows a breakdown of the variance for different sources of measurement variation.

Analysis Summary	
Variance Results	
Repeatability	23.159522%
Reproducibility	61.227832%
Operator	57.426911%
Operator * Part	3.800921%
Part	15.612646%
Total Gage RR	84.387354%
Total Variation	100.000000%

Click the **View Analysis Summary** icon to view the results in detail. The ANOVA table indicates that both the part and operator have a significant influence on the measurement variation.

ANOVA Table					
Source of Variation	Degrees of Freedom	Sum of Squares	Mean Squares	F Ratio	P Value
Part	2	105544.5185	52772.25926	5.065525	0.080126
Operator	2	332413.8519	166206.9259	15.953936	0.012409
Part • Operator	4	41671.7037	10417.92593	1.492357	0.246187
Repeatability	18	125655.3333	6980.851852		
Total	26	605285.4074			

The P/T ratio in the Gage Variance Results table is in the last column in the **Total Gage R&R** row. The calculated value is 47.85%, which is much higher than the recommended 10% maximum.

Gage Variance Results						
Source	Variance Results		Standard Deviation			
	Variance	Variation Contribution (%)	Standard Deviation	Standard Deviation Ratio (%)	Study Variation	Study Variation to Tolerance Ratio
Repeatability	6980.851852	23.16%	83.551492	48.12%	501.308953	25.07%
Reproducibility	18455.58025	61.23%	135.851317	78.25%	815.1079	40.76%
Operator	17309.88889	57.43%	131.567051	75.78%	789.402306	39.47%
Operator * Part	1145.691358	3.80%	33.848063	19.50%	203.088377	10.15%
Part	4706.037037	15.61%	68.600561	39.51%	411.603369	20.58%
Total Gage R&R	25436.4321	84.39%	159.488031	91.86%	956.928187	47.85%
Total Variation	30142.46914	100.00%	173.615867	100.00%	1041.6952	52.08%

The gage-to-part variation ratio is the standard deviation for the Total Gage R&R (i.e., total variation due to the measurement system) divided by the standard deviation for the Part (i.e., part-to-part variation).

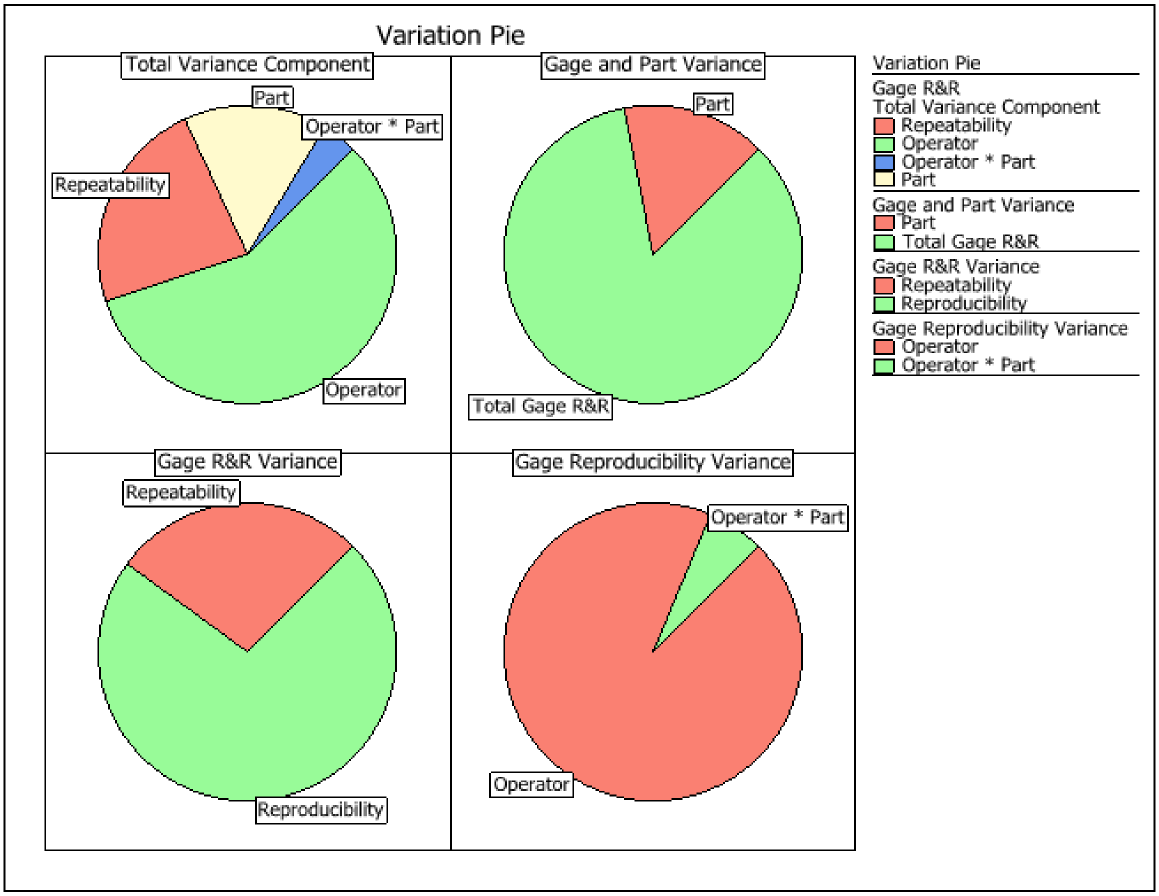
$$159.488/68.6006 = 232.49\%$$

The gage-to-total-variation-ratio is the standard deviation for Total Gage R&R divided by the standard deviation for Total Variation.

$$159.488/173.6159 = 91.86\%$$

Clearly, all these ratios are too large. As a result, the experiments concluded that the operators should be trained so they perform measurements more consistently, and a better gage may need to be used.

To further examine the breakdown of difference sources of measurement variation, click the **Plot** icon and view the variation pie charts to see the contribution of each variance component. The charts are shown next.

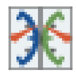


Gage Agreement Folio

The gage agreement folio is used to compare the accuracy and precision of two measuring devices (i.e., gages). For example, this folio could be useful if you wish to determine whether replacing an old gage with a new one is possible without sacrificing the accuracy and precision of the measurements. (To perform a detailed analysis of the accuracy of a gage, use the linearity & bias folio. For precision, use the gage R&R folio.)

Note: When comparing the precision of two systems, only the *repeatability* (i.e., variation due to properties of the gage) is compared. To eliminate variation due to properties of the operators (i.e., *reproducibility*), make sure all the measurements used in the study are performed by the same operator.

To add a gage agreement folio to your project, choose **Home > Insert > Gage Agreement**, or right-click the **Test and Planning** folder on the current project explorer and choose **Add Measurement Systems Analysis > Gage Agreement** on the shortcut menu.



Gage Agreement Folio Data Tab

The Data tab of the gage agreement folio contains a control panel and a sheet with the following four columns for entering data:

- **Part** is the identifier for the part that was measured.
- **Measuring Device** is the identifier for the gage that was used to obtain the measurement.
- **Reading** is the measurement obtained from the gage under investigation.
- **Trial** is used to help you keep track of multiple trials that are used in a particular study. For example, the first measurements of a part by a particular gage would be the first trial, the second measurement of the same part by the same gage the second trial, and so on.

As an example, suppose two scales are being compared for accuracy and precision. To compare them, three parts are weighed (in pounds) by both scales. Part 1 is measured four times: twice using the first scale and twice using a second scale. Part 2 and Part 3 are measured similarly. The same operator performs all the measurements. The resulting data is shown next. For example, the first row shows that Part 1 was measured by Gage 1 (the first scale), which produced a measurement of 66.32 pounds.

	Part	Measurement Device	Reading	Trial
1	1	Gage 1	66.32	1
2	2	Gage 1	95.51	1
3	3	Gage 1	61.93	1
4	1	Gage 1	65.8	2
5	2	Gage 1	95.94	2
6	3	Gage 1	60.27	2
7	1	Gage 2	74.3	1
8	2	Gage 2	94.74	1
9	3	Gage 2	70.81	1
10	1	Gage 2	74.39	2
11	2	Gage 2	94.93	2
12	3	Gage 2	70.75	2

Gage Agreement Folio Control Panel

The gage agreement folio control panel allows you to control the settings for analyzing the measurement system data. It also displays the results of the analysis. This topic focuses on the Main page and Analysis page of the gage agreement folio control panel, which contains most of the tools you will need to perform the analysis. For more information about the control panel in general, see Control Panels.

CONTROL PANEL MAIN PAGE

- The **Settings** area on the control panel is used to view the settings and status of the analysis.

Settings
Risk Level: 0.1
Analyzed
10 Observations

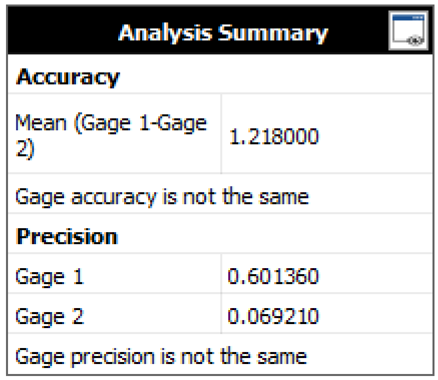
In this example:

- The specified risk level is 10%. This is also known as the alpha value, which is a measure of the risk that the analysis results are incorrect (i.e., alpha = 1 - confidence level). To configure the risk level used in the analysis, click the blue text, which will take you to the Settings page of the control panel.

Risk Level
0.1

- "Analyzed" is shown in green, which indicates that the current response data has been analyzed using the current settings. Otherwise, this would show "Modified" in red.
- The last row indicates that the analyzed data consists of 17 observations.

- The **Analysis Summary** area is shown when the current measurement data has been analyzed using the current settings. This area displays the following values.

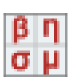


  - The **Accuracy** area displays the difference between the average observation of the first gage and that of the second gage. If the difference is significant according to the paired *T*-test that is performed on the data, then the accuracy of the gages is not the same.
  - The **Precision** area displays the variance due to repeatability for each gage. If the difference is significant, then the precision of the gages is not the same.

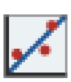
  Click the **View Analysis Summary** icon in this area to view the details of the analysis results.

Folio Tools

The folio tools are arranged on the left side of the control panel. Use these tools to configure your analysis and experiment with the analysis results.



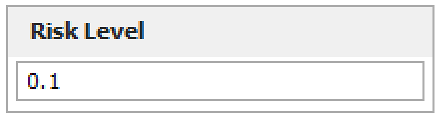
Calculate analyzes the measurement data and displays a summary of the results on the control panel.

 **Plot** creates a plot based on the analysis. If you click this icon before the current data set has been analyzed, an analysis will be performed automatically.

 **View Analysis Summary** opens a window that contains detailed information about current and past analysis results.

CONTROL PANEL ANALYSIS SETTINGS PAGE

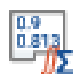
The Analysis page of the gage agreement folio is used to enter the **Risk Level**, or alpha value, which is a measure of the risk that the analysis results are incorrect (i.e., alpha = 1 - confidence level).



Gage Agreement Folio Analysis Results

When accessed from a gage agreement folio, the Analysis Summary window will contain a detailed comparison of the gages' accuracy and precision.

If the current response data has been analyzed, you can open the window by clicking the **View Analysis Summary** icon on the control panel.



If the current response data has not been analyzed, the icon will still be available so you can view the folio's analysis history.

Select an item in the Available Report Items panel to display it on the spreadsheet. Each item is described next.

ANALYSIS RESULTS

The **Part Summary table** compares the average readings from each gage.

Part Summary Table Columns

- **Subject** is the identifier of the measured part.
- The **Number of Readings** columns show the number of observations taken of a particular part using a particular gage. The name of the gage is shown at the top of the table.
- The **Average Reading** columns show the average of the measurements that were taken of a particular part by a particular gage.

- **Difference** is the Average Reading from the first gage minus the Average Reading from the second.
- **Grand Average** is the average of all measurements taken of a particular part.

The Difference and Grand Average values in this table are plotted in the difference vs. mean plot.

The **Average Reading Comparison table** compares the average readings of the two gages in the study using a paired *T*-test.

Average Reading Comparison Table Columns

- The **Mean** column shows the mean value of the difference between the averages of the two gages.
- **Standard Deviation of Mean** is the standard deviation of mean value of the difference.
- The **Lower Bound** and **Upper Bound** columns show the confidence bounds of the mean value of the difference. The confidence level is determined by the risk level entered on the Analysis Settings page of the control panel (e.g., if the risk level is 0.1, then the 95% one-sided upper and 95% one-sided lower bounds are calculated).
- **T Value** is the ratio of the absolute value of the Mean column and the Standard Deviation of Mean column. It is used to calculate the *p* value.
- **P Value** is calculated from the *T* value. If the *p* value is less than or equal to the risk level specified on the Analysis Settings page of the control panel, then the two gages have different average readings in this study. In this case, the *p* value will be shown in red.

Note: If you suspect that the gages may differ in linearity (e.g., you see a trend in the difference vs. mean plot), then you should check the Regression Study table and/or the average reading plot to make sure the gages have the same accuracy.

The **Regression Study** compares the accuracy (linearity and bias) of the two gages. It includes two tables.

Regression Study Tables

The **Individual Coefficient Test** table shows the calculated coefficients for the regression line fitted to the average readings of the first vs. second gages (also shown in the Average Reading plot). It includes the following columns.

- **Test** is the null hypothesis. Two hypotheses are tested individually with this table: that b0 of the fitted regression line is 0 (i.e., no difference in bias, assuming there is no difference in linearity) and that b1 is 1 (i.e., no difference in linearity). Both of these hypotheses are true if the gages have the same accuracy. Note that if there is a difference in linearity, then there is also a difference in bias, since linearity describes how the bias changes with the size of the measured parts.
- **Coefficient** is the coefficient of b0 and b1 in the regression equation.
- **Standard Error** is the standard deviation of the regression coefficient.
- The **Low Confidence** and **High Confidence** are the lower and upper confidence bounds on the regression coefficient. For example, if the bounds on b0 include 0, then the b0 = 0 null hypothesis is not false.
- **T Value** is the normalized regression coefficient, which is equal to Coefficient/Standard Error.
- **P Value** is the probability that the null hypothesis is true. If the *p* value is less than or equal to the risk level specified on the Analysis Settings page of the control panel, then the null hypothesis is considered false. In this case, the value is shown in red.

The **Simultaneous Coefficient Test** table shows the results of the *F*-test that is used to determine whether the gages have the same accuracy. It includes the following columns.

- **Test** is the null hypothesis that b0 = 0 and b1 = 1. If this is true, then the two gages have the accuracy.
- **F Value** is the critical value of the F-distribution. It is used to calculate the *p* value.
- **P Value** is the probability that the null hypothesis is true. If the *p* value is less than or equal to the risk level, then the two gages do not have the same accuracy. In this case, the value is shown in red. The folio's conclusion about whether the accuracy is the same is based on this value.

The **Precision Comparison table** compares the repeatability of the two gages.

Precision Comparison Table Columns

- **Gage** is the name of the measuring device specified on the Data tab.
- **Repeatability Variance** is the repeatability of each gage (i.e., the variation when multiple readings for the same part are performed by the same operator).

- **Degrees of Freedom** is the number of parts measurement by each individual gage.
- **F Ratio** is the ratio of the first Repeatability Variance value to the second value. An *F* ratio of 1 means the gages have the same repeatability.
- The **Lower Bound** and **Upper Bound** columns show the confidence bounds on the *F* ratio. If the bounds include 1, then the two gages have the same repeatability.
- **P Value** is the probability that the *F* ratio is 1. If the *p* value is less than the risk level/2 and 1-(risk level/2), then the two gages do not have the same repeatability. (The risk level is specified on the Analysis Settings page of the control panel.) If the gages do not have the same repeatability, then this value is shown in red.

Gage Agreement Folio Plots

The following plots are available in gage agreement folios. They allow you to visually display the data and compare the accuracy and precision of the gages. For general information on working with plots, see Plot Utilities.

- The **Difference vs. Mean** plot shows the difference between the gage's measurements of each part versus the mean measurement of the part. (These are the same values in the Difference and Grand Average columns in the Part Summary.) If points seem to follow a trend, then the linearity of the gages may be different. The control limits are marked with red dotted lines near the top and bottom of the chart. If a point is out of control, then the difference (i.e., y-axis value) associated with that point is significantly larger than the differences associated with the other points.
- The **Average Reading** plot is used to investigate whether the two gages have the same accuracy (linearity and bias). If they do, then the average readings from the first vs. second gages should follow the black 45-degree reference line that passes through the origin. The blue line is a regression line fitted to the data points. The closer the fitted line is to the reference line, the more likely it is that the gages have the same accuracy.
- The **Gage Repeatability** plot shows the calculated repeatability variance for each gage, as well as the upper and lower confidence bounds on the repeatability. The confidence level used is determined by the risk level specified on the Analysis Settings page of the Data tab control panel. If the confidence bounds for the two gages overlap, then the gages have the same repeatability.

Gage Agreement Folio Example

The data set used in this example is available in the example database installed with the software (called "Weibull21_DOE_Examples.rsgz21"). 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 example project is "Gage Agreement Study."

A company needs to investigate if the gages from two different vendors have the same accuracy and precision. Experiments were conducted and the resulting measurements were entered on the Data tab of the example folio. The first 13 rows of data are shown next.

	Part	Measurement Device	Reading	Trial
1	1	Gage 1	66.32	1
2	2	Gage 1	95.51	1
3	3	Gage 1	61.93	1
4	4	Gage 1	163.08	1
5	5	Gage 1	76.6	1
6	6	Gage 1	127.35	1
7	7	Gage 1	93.07	1
8	8	Gage 1	134.39	1
9	9	Gage 1	115.54	1
10	10	Gage 1	117.92	1
11	1	Gage 1	65.8	2
12	2			
13	3			

Note: Complete data set is not shown.

All of the measurements were taken by the same operator (in order to eliminate error due to differences between operators). For example, in the first ten rows, you can see that the operator measured 10 different parts using the same gage. Each measurement is shown in the Reading column.

The **Settings** area on the control panel shows that a risk level of 10% was used to determine whether a difference in accuracy/precision is considered significant.

Settings
Risk Level: 0.1
Analyzed
40 Observations

The following summary of the results is also shown on the control panel.

Analysis Summary	
Accuracy	
Mean (Gage 1-Gage 2)	1.218000
Gage accuracy is not the same	
Precision	
Gage 1	0.601360
Gage 2	0.069210
Gage precision is not the same	

According to these results, the gages differ in both accuracy and precision. To see more results, click the **View Analysis Summary** icon. The Average Reading Comparison table is used to determine whether the average readings of the gages are different.

Average Reading Comparison

Average Reading Comparison					
Mean (Gage 1 - Gage 2)	Standard Deviation of Mean	Lower Bound	Upper Bound	T Value	P Value
1.218	2.333891	-3.060286	5.496286	0.521875	0.614356

Since the P value is greater than the risk level, the average readings of the two gages are the same.

As you can see, Gage 1 produced measurements that were, on average, 1.218 units greater than those of Gage 2, but this difference was not found to be significant.

The Regression Study tables are used to determine if there are any differences in accuracy (i.e., linearity + bias) between the gages.

Regression Study

Individual Coefficient Test						
Term	Coefficient	Standard Error	Low Confidence	High Confidence	T Value	P Value
b0 = 0	22.387335	1.247147	20.068206	24.706464	17.950842	9.510205E-8
b1 = 1	0.77496	0.011408	0.753746	0.796174	19.726454	4.539458E-8

Simultaneous Coefficient Test		
Test	F Value	P Value
b0 = 0 and b1 = 1	200.575411	3.434355E-8

Since the P value is less than or equal to the risk level, the two gages do not have the same accuracy.

The **Simultaneous Coefficient Test** is used to evaluate the hypothesis that the gages have the same accuracy. The second row in the Individual Test Table (b1 = 1) specifically tests the hypothesis that the gages have the same linearity. Since the *p* value of that row is less than the risk level, and since linearity describes how the bias changes with the size of the measured parts, you can conclude that the gages are also different with respect to bias.

Finally, the Precision Comparison table shows how the precision (i.e., repeatability, or variation due to the gage) differs across the gages.

Precision Comparison

Gage Repeatability						
Gage	Repeatability Variance	Degrees of Freedom	F Ratio	Lower Bound	Upper Bound	P Value
Gage 1	0.60136	10	8.688918	2.917466	25.877615	0.998969
Gage 2	0.06921	10				

Since the P value is outside the range of (risk level/2) and 1-(risk level/2), the two gages do not have the same precision.

The Repeatability Variance column shows that the gages exhibited different variance in measurements. Since the *p* value is higher than 1-(risk level/2), or 0.95, it is concluded that the difference in repeatability is significant.

These results shows that Gage 2 is better in terms of repeatability, and that the gages are significantly different with respect to accuracy. Linearity and bias studies should be conducted if you wish to determine which gage is the more accurate one.

Maintenance Planning

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

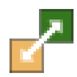
the inspections such that oncoming failures are detected and the component can be scheduled for replacement before any failures occur.

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.

Generating a Cost vs. Time Plot

Follow the steps outlined below to generate a cost vs. time plot.

1. Choose **Home> Insert > 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.

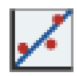
2. Click the **Plot Type** drop-down list on the control panel and select which plot to generate. You can choose to create a Planned Replacement plot, an Inspection Replacement plot, or place both on the same plot.

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. To calculate the results and generate the plot, choose **Maintenance Planning > Analysis > Plot** or click the icon on the control panel.

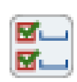


The **Parameters** area of the control panel shows the parameters of the distribution of the data set, while the **Results** area shows the estimated minimal cost and optimal time for the replacement. To see a summary report of the analysis, click anywhere within these two areas. To open the life data folio that contains the data set, click the blue link that displays the name of the folio and data sheet.

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).

1. 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.

2. Click **OK** to create the 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.

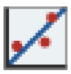
   You can also access the task via the Scheduled Tasks page of the Resource Manager (**Home > ReliaSoft > Resource Manager**).

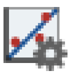
Maintenance Planning Control Panel

The Maintenance Planning tool's control panel consists of two pages: Analysis page and Settings page.

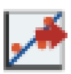
Analysis Page

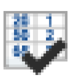
The Analysis page contains the options for generating a cost vs. time plot (see Maintenance Planning).

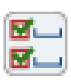
 **Plot** updates the plot to reflect any changes that have been made, and recalculates the estimates for the minimal cost and optimal time.

 **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.

 **RS Draw** launches ReliaSoft Draw, which allows you to annotate your plot and view your plot in greater detail.

 **Export Plot Graphic** opens the Save As window, which allows you to save the current plot graphic in *.jpg, *.gif, *.png or *.wmf format.

 **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.

The following settings affect how the plot is displayed:

- **Units** allows you to choose the unit for the plot's time scale.
- **Auto Refresh** automatically refreshes the plot if any of the inputs or settings are modified.
- **Keep Aspect Ratio** maintains the ratio of the horizontal size to the vertical size of the plot graphic when you resize the plot sheet.

- **Scaling** shows the minimum and maximum values for the X and Y axes. To change a value, clear the check box beside the value range. If selected, the application will automatically choose an appropriate value for the range.

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.
- **Convergence Criteria** 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.

Reliability Data Warehouse (RDW)

IN THIS CHAPTER

The Reliability Data Warehouse (formerly called the "Synthesis Data Warehouse (SDW)") enables you to access data from an XFRACAS failure reporting system for the purpose of life data analysis or repairable system/reliability growth analysis. You can also set up custom connections to obtain reliability-related data from external data sources (including Access, Oracle and SQL Server). You can also get data directly from custom reports created in XFRACAS.

You can then transfer the data to an analysis folio, or use the flexible Dashboard utility to explore and present the data in a variety of graphical charts and grids. (See Transferring from the RDW to an Analysis Folio or RDW Dashboards.) In addition, if an SEP web portal has been implemented for an enterprise database, users can also access RDW dashboards from any web-enabled device.

To open the RDW, choose **Home > ReliaSoft > Reliability Data Warehouse**.

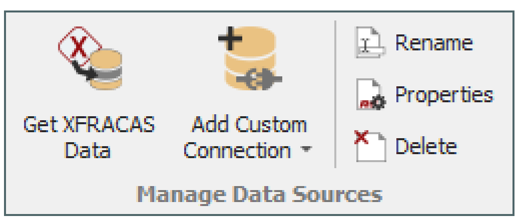
Managing Data Sources

The available data sources are listed in the data source manager on the left side of the window. They may include:

- Static data collections that were extracted from XFRACAS at a particular point in time (and will not change if incidents are later added or updated). (See Extracting Data from XFRACAS.)
- Custom connections that have a live link to a specified external database or to a predefined report created in XFRACAS. (See Custom Connections in the RDW.)
- Static data collections that were extracted from a custom connection at a particular point in time (and will not change if the original data source changes). (See Import to a Static Data Collection.)

If you are working with a long list of available data sources, this panel can utilize the same categories, identifiers and item filters that are available in many other locations throughout ReliaSoft desktop applications.

To create, delete, rename or edit the properties for these data sources, use the **Manage Data Sources** commands, or right-click inside the panel.



The Properties window allows you to enter identifiers for each data source, and to view its history.

Built-in Find/Filter, Configuration and Grouping Tools

Double-click any data source to display the data in a grid on the right side of the window. The RDW offers the same filter, column configuration and grouping tools that are built in to other utilities that use a similar grid (e.g., the Analysis Explorer, Actions Explorer, etc.). For details about how to use each feature, see:

- Finding and Filtering Records
- Configuring Columns
- Grouping Panel

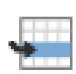
Building the Data Set

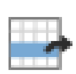
The "Include in Analysis?" column shows whether each row will be included in the data set when it is transferred to a folio.

- **Include**: Data points that will be included in the analysis are shown in green.
- **Invalid**: In XFRACAS data collections, incidents with a State Time of less than 0 are automatically set to invalid and will not be included in the analysis. These rows are shown in red.
- **Ignore**: In XFRACAS data collections, incidents with a State Time of 0 that are marked as suspensions in the State FS column are automatically set to ignore and will not be included in the analysis. These rows are shown in gray.
- **Exclude**: Data points that have been manually set to exclude from the analysis are shown in blue.

To view only those rows that will be transferred to a folio, choose **View > Show Only Included**. You can toggle this command off to show all records.

To change the status for a specific data point before transferring data to a folio, select the row and choose **Build Data Set > Include** or **Build Data Set > Exclude**.

 **Include**

 **Exclude**

Note: Changes to the "Include in Analysis?" indicator will not be saved with the data source. If you close the data source and open it again later, and if you view a dashboard in the SEP web portal, the data will not reflect changes that were applied manually.

In XFRACAS data collections, and only if you have permission to view incidents in the XFRACAS entity that the data were extracted from, you can select a row and choose **Build Data Set > View XFRACAS Incident** to open the incident in your web browser.

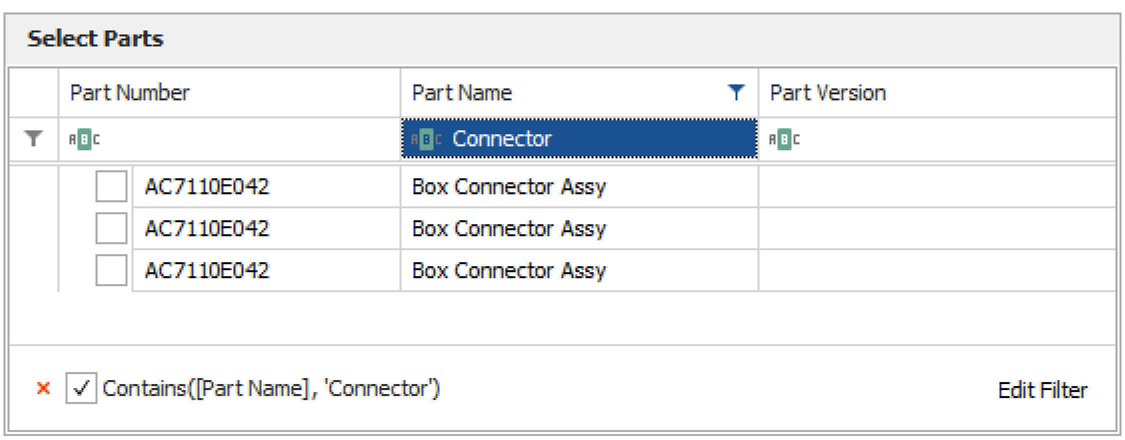
Extracting Data from XFRACAS

When you are working with an enterprise database that contains XFRACAS data, or a standard database that has a connection to external XFRACAS tables, you can use the Reliability Data Warehouse to extract static data collections. The collections will not change if incidents are later added or updated in XFRACAS. (Alternatively, if you need a live link to a predefined report that has already been created in XFRACAS, see Connect to XFRACAS Report.)

To extract a static data collection from XFRACAS, first open the RDW and then choose **Manage Data Sources > Get XFRACAS Data**.

(In a secure database, this is available only to users with the "Create/edit/delete RDW data collections" permission. Additionally, users must have the "ReliaSoft - Read Data" permissions for the associated XFRACAS tables.)

In the XFRACAS to RDW window, choose an **Entity** to import from, then select the check box for each part that you want to extract the incident (i.e., failure) data. If needed, you can use the Auto Filter Row to filter the grid by matching text in one or more columns. (For extremely large data sets, this row will be hidden and a Filter window will be available instead.)



Click **Next**, then specify other settings or fields to import:

- **Time Metric** allows you to choose which metric to use with the data. The options are based on the metrics that your system administrator has enabled for the XFRACAS entity.
- **Incident Detail Fields** allow you to map customized detail fields from the XFRACAS incident to one of the user-defined field columns in the RDW collection. The following field types can be mapped:
  - **StringUDF1-3**: Alphanumeric Input Box, Check Box, Company, Contacts, Currency, Date, Numeric Input Box, Users, Yes/No Button, or the Select List, Administrative Controlled field type
  - **NumberUDF1-3**: Currency or Numeric Input Box
  - **DateUDF1-3**: Date
- For reliability growth analysis (RGA) data, additional options are available:
  - **Include non-chargeable** extracts incidents that are marked as either chargeable or non-chargeable in XFRACAS. If you clear the check box, it will extract only chargeable incidents.
  - **Only system down events** extracts only those incidents that brought the system down.
  - **Filter by date commissioned** extracts only incidents for systems that were commissioned between the specified dates.

Click **Next**, then enter a name for the data collection and any Identifiers that you want to use.

When you click **OK**, the extraction process will begin. Depending on the amount of data that you are extracting, this may take some time.

You can then use the data collection to transfer data to an analysis folio or explore/present in the dashboard utility. For information about the fields that will be available either in the Build Data Set tab or in the Dashboard Designer, see Data Fields for XFRACAS Data Collections.

Data Fields for XFRACAS Data Collections

The following fields are available for use in dashboard layouts for XFRACAS data collections and for data sets created via the ReliaSoft API. These are the same columns available in the Build Data Set tab. The fields in custom connections will depend on the data source itself.

- **Address Location**: the location of the owner of the system. This corresponds to the **Location** field in the XFRACAS contact.

- **Category**: the incident category. This corresponds to the **Category** field in the XFRACAS incident and, in conjunction with the failure type assigned during part repair/replacement, determines whether the data row is chargeable (i.e., is considered a failure for reliability calculations) or non-chargeable (i.e., is considered a suspension). For more information, see the "Incident Disposition Area" topic in the XFRACAS documentation.
- **Chargeable**: the chargeable code value (where 0 = non-chargeable, 1 = chargeable): equivalent to the Category Chargeable and Category Non-Chargeable lists in the XFRACAS Admin List page. This is to be used to calculate the failure state (StateFS). XFRACAS categories can also be viewed within an XFRACAS incident.
- **CompanyOwner**: the owner of the system. This corresponds to the **Company** field in the XFRACAS contact.
- **DataID**: the database ID of the row.
- **DataSetID**: the ID of the data set that the row belongs to. This is useful if you are viewing multiple data sets at the same time.
- **DateUDF1-3**: user-defined fields for dates. You can choose up to 3 detail fields in XFRACAS incidents to import to these columns.
- **ExtractedBy**, **ExtractedDate** and **ExtractedName**: identify the user who extracted the data from XFRACAS, the date of extraction and the name of the RDW data collection.
- **FailureMode**: the failure mode associated with the incident. This is chosen or created in the **Failure Mode** field in the XFRACAS incident.
- **IncidentAction**: the action taken to address the incident (whether a part was removed or installed). This is not related to XFRACAS actions. In XFRACAS data sets, repaired and replaced parts will be represented using two rows of data—one will be for removing the part, and one for installing it.
- **IncidentEntityDisplayID**: the incident number assigned by XFRACAS, which includes the prefix for the entity that it is associated with (e.g., E1-201).
- **IncidentID**: the database ID of the incident.
- **IncidentOccurrenceDate**: the date when the incident happened. This corresponds to the **Occurrence Date** field in the XFRACAS incident.
- **IncidentRepairDate**: the date that part repair or replacement was completed. This corresponds to the **Completed Date** field that applies to the XFRACAS incident resolution.

- **IncidentResolution**: notes regarding the resolution of the incident. This corresponds to the **Incident Resolution** field in the XFRACAS incident.
- **IsLRU**: indicates whether the item is a line replaceable unit (LRU). This does not correspond to an XFRACAS field; it can be used when creating data sets via the ReliaSoft API.
- **LastInspectedTime**: the last inspected time. For exporting XFRACAS data sets to Weibull++, this is equal to the state time for the data row. For exporting XFRACAS data sets for Reliability Growth (RGA), it is the amount of time the system has accrued at the last reported incident.
- **NumberInState**: equivalent to the Number in State column in a Weibull++ life data folio for grouped data. In data sets from XFRACAS, this value will always be 1.
- **NumberUDF1**-3: user-defined fields for numbers. You can choose up to 3 detail fields in XFRACAS incidents to import to these columns.
- **ParentPartID**, **ParentPartName**, **ParentPartNumber**, **ParentPartVersion** and **ParentPartSerialNumber**: identifiers for the "parent" of the part that the incident pertains to. If the part doesn't have a parent (i.e., it is a top level part), this information will be identical to the information for the part itself.
- **PartHID** and **PartID**: the part's hierarchy identification number and database identification number. In XFRACAS, these are assigned by the system and shown in the HID and Part ID fields that are displayed for the generic part on the Template page.
- **PartName**, **PartNumber** and **PartVersion**: identifiers for the part that the incident pertains to. In XFRACAS, this information is specified for the generic part, and serialized parts based on the generic part will use the same information.
- **PartOrder**: the order in which the part was replaced. This corresponds to the order in the repair/replace table in the XFRACAS incident.
- **PartSerialHID**, **PartSerialMfgCode** and **PartSerialNumber**: relevant only if the incident was assigned to a part in a serialized system. They represent the part's serial hierarchy identification number, manufacturing code and serial number. In XFRACAS, the PartSerialHID is assigned by the system and shown in the Serial HID field that is displayed for the serialized part on the Serialized page.
- **ReportType**: how the incident was reported. This corresponds to the type chosen in the **Report Type** field in the XFRACAS incident.
- **RootCause**: the root cause of the failure mode. This is chosen or created in the **Root Cause** field in the XFRACAS incident.

- **StateFS**: This allows the user to understand if a data row is considered a failure (F) or a suspension (S) for the purposes of reliability calculations. In XFRACAS data sets, this is based on the incident category assigned in the XFRACAS incident, in conjunction with the failure type assigned during part repair/replacement. For more information, see the "Incident Disposition Area" topic in the XFRACAS documentation.
- **StateTime**: the amount of time accrued on the part. This is based on the run hours calculated by XFRACAS.
- **StateTimeRestore**: the amount of time required for the repair. This corresponds to the **Repair Duration** field in the XFRACAS incident.
- **StringUDF1-3**: user-defined fields for strings. You can choose up to 3 detail fields in XFRACAS incidents to import to these columns.
- **TimeMetric**: the time metric to use for the data. The options are based on the metrics that have been enabled for the XFRACAS entity.
- **TopLevelCommissionDate**: the commission date of the top level part in the system. This corresponds to the commission date of the associated CSI in XFRACAS.
- **TopLevelPartID**, **TopLevelPartName**, **TopLevelPartNumber** and **TopLevelPartVersion**: identifiers for the top level part in the system that the incident pertains to.
- **TopLevelSerialHID** and **TopLevelSerialNumber**: relevant only if the incident was assigned to a part in a serialized system. They represent the serial hierarchy identification number and serial number of the top level part in the serialized system. This information is used by Weibull++ to distinguish the system in which the failure occurred.

Transferring from the RDW to an Analysis Folio

Follow the steps below to transfer a data set from the Reliability Data Warehouse (RDW) to an analysis folio.

1. Choose **Home > ReliaSoft > Reliability Data Warehouse**.

2. In the RDW, double-click the data you want to transfer to an analysis folio.
3. In the data table, use the **Include in Analysis?** column to mark any data points that need to be excluded from the transfer. You can double-click the cell to toggle the options, or select the

row and choose **Build Data Set > Include** or **Exclude**.

In data collections imported from XFRACAS:

a. Incidents with StateTime = 0 and StateFS = S (suspension) are automatically marked **Ignore**.

b. Incidents with StateTime < 0 are automatically marked **Invalid**.

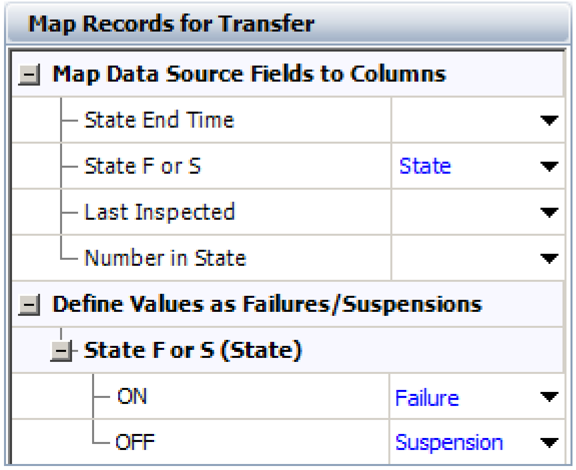
Include in Analysis?	StateTime	StateFS	LastInspectedTime
Invalid	-10	S	-10
Include	121	F	121
Ignore	0	S	0
Include	1000	S	1000
Include	131	S	131
Ignore	0	S	0
Exclude	1000	S	1000
Include	2160.02	S	2160.02
Ignore	0	S	0
Exclude	1000	S	1000
Include	2160.02	S	2160.02

4. Choose **Transfer > Transfer to New Folio**.

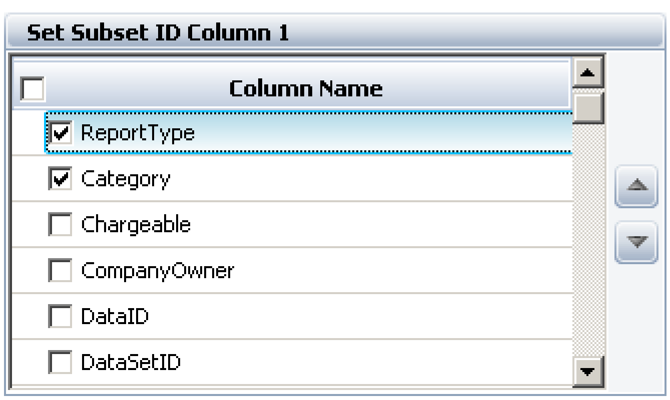
Follow the on-screen prompts to select the data type and map the RDW fields to columns in the new analysis folio.

Tips for Mapping the Columns

- **State F or S** (for life data and life-stress data analysis) indicates whether a data point represents a failure (F) or suspension (S). If the data source field does not use the F and S codes, use the **Define Values as Failures/Suspension** area to specify which value(s) represents each state.



- **Event** (for growth data analysis) indicates whether the data point represents a failure (F) that will be transferred for analysis, or some other event that will not be analyzed as a failure. When a data point is marked as a suspension (S) for transfer to a growth data analysis data set, it will be considered only if it affects the recorded system end time. For example, if the System End Time column indicates that the system was last inspected at 1,000 hours but there is a suspension for that system at 1,100 hours, the later time will be used in the folio.
- Time and quantity columns, such as **State End Time** and **Number in State**, can only be mapped to fields with a numeric data type (e.g., number, currency, etc.).
- **Classification** and **Mode** columns (for growth data analysis) can be mapped to any RDW user-defined field that contains the data. (See Extracting Data from XFRACAS.)
- **Subset ID** columns (for life data and life-stress data analysis) or **Comment** columns (for growth data analysis) allow you to choose multiple RDW fields to display in the same folio column. The data from the fields will be concatenated and separated by dashes (e.g., "Report Type – Category"). Use the check boxes to select the fields and then use the arrows to specify the order.



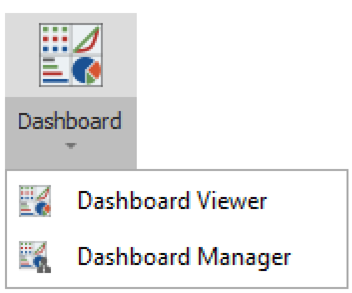
RDW Dashboards

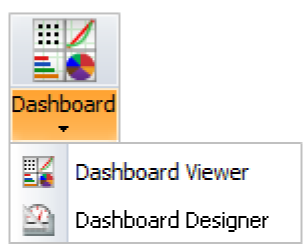
This topic describes how to use the Dashboard utility for exploring and presenting data from the Reliability Data Warehouse. If an SEP web portal has been implemented for an enterprise database, users can also access RDW dashboards from any web-enabled device.

For static data collections, a variety of dashboard layouts can be predefined. For custom connections, there can be only one layout for each connection.

1. Choose **Home > ReliaSoft > Reliability Data Warehouse**.

2. Double-click the static data collection or custom connection that contains, or links to, the data you want to view in the dashboard.
3. On the Build Data Set tab, select which rows to include or exclude. (See Building the Data Set.)
4. If an appropriate dashboard layout has already been predefined, switch to the Dashboard Viewer tab or choose **Dashboard > Dashboard Viewer**. If multiple layouts are available, use the drop-down list at the top of the viewer to choose what to display.
5. If you need to create a new layout, choose **Dashboard > Dashboard Manager** (for a static data collection) or **Dashboard Designer** (for a custom connection).





(In a secure database, the Dashboard Layout Manager and the Dashboard Layout Designer are available only for users with the "Manage dashboard layouts" permission.)

For information on the data fields available for use in dashboard layouts for RDW data collections, see Data Fields for XFRACAS Data Collections.

Tip: For dashboards that are intended to be viewed in both desktop applications and the SEP web portal, it is recommended to use the **Local Colors** option on the Design tab of the Dashboard Layout Designer to save your preferred colors with the layout. The global colors definition may not be the same in both web and desktop.

Custom Connections in the RDW

The Custom Connections feature in the Reliability Data Warehouse allows you to create a live link to an external data source (Access, Oracle or SQL Server). This enables you to get data from your own custom and third-party databases for transferring to an analysis folio, or for viewing in the RDW dashboards.

You can also get data directly from your own custom reports created in XFRACAS.

A custom connection data source always shows the latest information from the original database or report and you can only create one dashboard layout per connection. You can also use these connections to import data into a static data collection, if desired.

Connect to an External Database

To create a live link to an external database:

1. Choose **Manage Data Sources > Add Custom Connection > To External Database**.

2. In the Add Custom Connection window:
   a. Enter the **Display Name** that will identify this custom connection in the RDW data source list.
   b. Enter the connection settings for a **Microsoft SQL Server**, **Oracle** or **Access** database. (See Connection Issues below for more information about the **Use impersonation** option for SQL Server.)
   c. After you have specified the database, the **Table Name** field shows a list of the available tables. Select the main one that contains the data you want to use in the RDW.
   d. If you want to create aliases for column names, build a query that combines data from multiple tables or enter your own SQL, select the **Open Query Editor/Builder** check box.

   e. If you want to create the dashboard layout for this data source immediately after defining the connection/query, select the **Open Dashboard Designer** check box.

   f. Click **OK** to proceed.

3. If you selected to customize the query, you can use the Query Editor to type or paste your own query, or select a stored procedure. If you need further tools, click the **Run Query Builder** button. The Query Builder can serve three purposes:

   a. The bottom-center panel shows the fields that will be included in the data set. If desired, you can use the Alias column to change the names that will be displayed in the RDW grid and dashboard layouts.

   b. If the tables are linked by foreign keys, you can use the tool to build a query that combines fields from multiple tables.

   c. You can also enter your own SQL in this tool; select the **Allow SQL Editing** check box and type or paste your own query.

   When you are finished in the Query Builder, click **OK** to return to the Query Editor, where you can click **Finish** to save your changes.

4. If you selected to open the Dashboard Designer, you can use it to create a single dashboard layout for this data source.

If you want to change the query or create/modify the dashboard layout at a later time, select the custom connection in the data source manager and choose **Dashboard > Dashboard Designer**.

From within the Dashboard Designer window, choose **Home > Query > Edit** to customize the query.

Connection Issues

There will be a "connection failed" message if the database is not found at the specified name/-location or if you don’t have permission to access it.

If your organization has implemented an SEP web portal, the administrator may need to take additional steps to make the dashboards visible to all users via the portal. For details, consult the print-ready implementation guide (*.pdf).

- **Oracle** - the password is stored with the custom connection; therefore, both the desktop applications and SEP web portal will attempt to connect in the same way for all users.
- **SQL Server** - the **Use impersonation** option in the custom connection allows you to enter a login for a one-time extraction to an RDW data collection, but this login is not saved in RDW.

  For subsequent attempts, the desktop applications will connect with the current user’s Windows login, whereas SEP will use the login that it uses to connect with the ReliaSoft database (if the ReliaSoft database is SQL Server) or with the IIS "application pool identity" (if the ReliaSoft database is Oracle).
- **Access** - the RDW must have access to the folder where the database is stored. It is recommended to use the UNC pathname (e.g., \\servername\foldername rather than P:\foldername) when you create the custom connection.

  The desktop applications will attempt to access the file with the current user’s Windows login, whereas SEP can only access files stored directly on the web server or in a network folder that can be accessed by its IIS "application pool identity."

  Tip: For Access databases with the *.accdb file type, the dashboard can only be displayed if the database was created with the same version of Microsoft Office (32-bit vs. 64-bit) that is installed on each individual user's computer (for ReliaSoft desktop applications) or on the web server (for SEP).

  To ensure that the dashboard will display regardless of which version of Microsoft Office is installed, use the *.mdb file type instead of *.accdb.

Connect to XFRACAS Report

Creating a custom connection to a predefined XFRACAS report enables you to use the RDW to view any type of XFRACAS data (not just data collections extracted for life data or repairable systems analysis). If an SEP web portal has been implemented for an enterprise database, users can also access the dashboards created for these reports from any web-enabled device.

Tip: For information about creating RDW reports in XFRACAS, see the "Report Builder" topic in the XFRACAS documentation.

To create a live link to an RDW report that has already been created in XFRACAS:

1. Choose **Manage Data Sources > Add Custom Connection > To XFRACAS Report**.

2. In the Select Report window, select any of the XFRACAS RDW reports that have been pre-defined in the database.
3. Click **OK** to load the data.

Import to a Static Data Collection

In the Reliability Data Warehouse, a custom connection provides a live link to an external database or XFRACAS report, while a static data collection stores data that was extracted at a particular point in time.

A static data collection can contain data extracted directly from XFRACAS. If desired, you can also create a static data collection by importing data from a custom connection. To do this:

1. Right-click the custom connection in the data source manager and choose **Import**.
2. In the Import Data from Custom Connection window:
   a. The **Map Data Source Fields to Columns** section specifies which field from the custom connection will be mapped to a column that's available in a data collection.
   b. The **Define Values as Failures/Suspensions** section displays all of the unique values from the field that is currently mapped to the State FS column. Specify whether each value should be considered a failure or a suspension.
   c. The Preview tab allows you to see what the data collection will look like based on your selections.
3. Click **OK** to create the static data collection.

Regression Tools

IN THIS CHAPTER

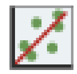
Multiple Linear Regression Folio

The multiple linear regression folio is a general statistical tool intended for performing a simple analysis of existing data in order to investigate the effects of predictors on normally distributed responses. Only quantitative data can be analyzed. After you designate columns for predictors and responses and enter the data, you can perform the analysis in much the same way that it is performed on the Data tab of a design folio. You can also use the analysis results to predict response values given user-specified levels for the predictors.

Note: If you are analyzing existing data specifically from a DOE perspective, then the free form folio is more appropriate. The free form folio can analyze both quantitative and qualitative factors, and it can analyze responses that are not normally distributed (e.g., life data that fits a Weibull distribution). In addition, the free form folio can be used to investigate interaction effects and optimize factor settings.

Also note that the multiple linear regression and free form folios are typically used for analyzing existing data. If you need to plan and implement an experiment to obtain data, use the design folio. (See Building a New Design.)

To add a multiple linear regression folio to your project, choose **Home > Insert > Multiple Linear Regression**.



To use the folio, click the arrows in the column headers to specify which columns will contain predictors and which will contain response data. After you enter the data and select the appropriate analysis settings on the control panel, click **Calculate** to analyze the data.

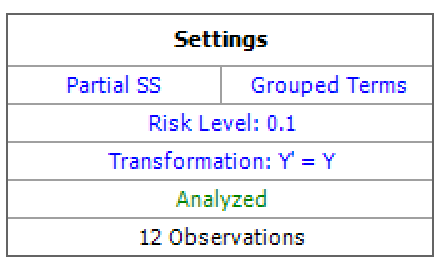
Multiple Linear Regression Folio Control Panel

The multiple linear regression folio control panel allows you to control the settings for analyzing response data and displays the results of the analysis. This topic focuses on the Main page and Analysis page of the multiple linear regression folio control panel, which contains most of the tools you will need to perform the analysis. For more information about the control panel in general, see Control Panels.

Control Panel Main Page

- The **Response** drop-down list is used to select the response that all the displayed settings and analysis results will apply to.

- The **Settings** area is used to view/edit the settings for analyzing the selected response. An example is shown next. To configure the settings, click the blue text. These settings are also accessible on the Analysis Settings page of the control panel.

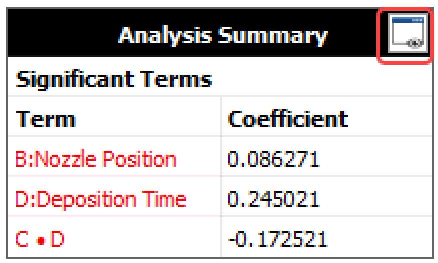


In this example:

- The partial sum of squares ("Partial SS") and grouped terms are used.
- The specified risk level is 10%.

    - No transformation is used (i.e., the "transformed" response, *Y'*, is equal to the non-transformed response, *Y*). To configure this setting, you can click the text or the Select Transformation icon in the folio tools.
    - "Analyzed" is shown in green, which indicates that the current response data has been analyzed using the current settings. Otherwise, this would show "Modified" in red.
    - The last row indicates that the analyzed data consists of 12 observations.

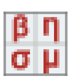
- The **Analysis Summary** area is shown when the current response data has been analyzed using the current settings. This area displays the terms that were found to be significant, as well as their associated regression coefficients. Click the **View Analysis Summary** icon in this area to view the details of the analysis results (including an ANOVA table).



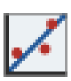
Analysis Summary	
Significant Terms	
Term	Coefficient
B:Nozzle Position	0.086271
D:Deposition Time	0.245021
C • D	-0.172521

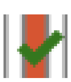
FOLIO TOOLS

The folio tools are arranged on the left side of the control panel. Use these tools to configure your analysis and experiment with the analysis results. Depending on the type of design you are working with, the control panel may contain some or all of the following tools:



Calculate analyzes the data for each response that is selected to be included in the analysis. To exclude a response from the analysis, clear the check box in its column heading.


Plot creates a plot based on the analysis. If you click this icon before the current data set has been analyzed, an analysis will be performed automatically.


Select Columns opens a window that allows you to specify which columns will contain predictors (i.e., factor level settings) and which will contain response data.

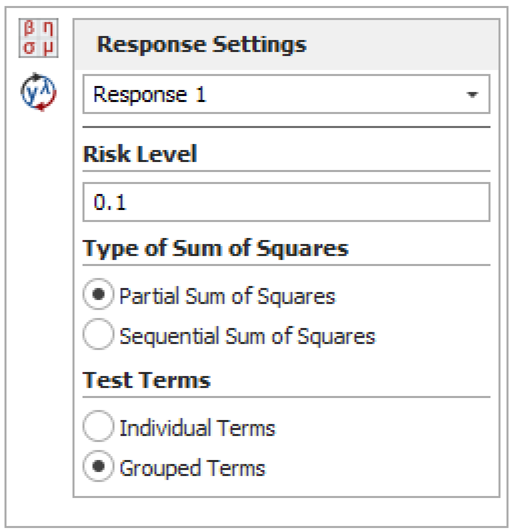
View Analysis Summary opens a window that contains detailed information about analysis results.

Select Transformation opens a window that allows you to select a transformation to apply to each response.

Prediction opens a window that allows you to enter values for each factor and see predicted results for the selected response.

Control Panel Analysis Settings Page

The Analysis Settings page of the control panel includes additional settings and appears as shown next.



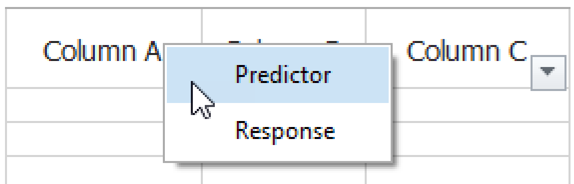
- The **Response Settings** drop-down list is used to choose the response that all the visible settings and analysis results will apply to. When you choose a response here, it will automatically be chosen when you return to the Main page.
- The **Risk Level**, or alpha value, is a measure of the risk that the analysis results are incorrect (i.e., alpha = 1 - confidence level).
- The next area allows you to specify the **Type of Sum of Squares** to use in the analysis. Select the **Partial Sum of Squares** option to test if a term is significant given that all other terms are considered in the model. Select the **Sequential Sum of Squares** option to test if an additional term is significant given that the terms before it are already in the model.
- The **Test Terms** area specifies what the information in the ANOVA table in the analysis results applies to. If you select **Individual Terms**, the software will separately examine the effects of each individual factor and/or factorial interaction. If you select **Grouped Terms**, it will examine groups of effect types, such as main effects, two-way interactions, etc.

You can also analyze the selected response or open the Select Transformation window by clicking the icons to the left of the **Response Settings** area.

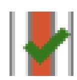
Multiple Linear Regression Folio Data Tab

The Data tab of the multiple linear regression folio includes a control panel for specifying analysis settings and a spreadsheet for entering data. You must enter numerical data in at least two columns (one for predictor data and one for response data) in order to perform the analysis. Note that all columns containing non-numerical data will be ignored.

For each column containing data, you must specify whether that column contains predictors (e.g., factor levels) or response data. To do this, click the arrow inside the column heading and choose the appropriate option.



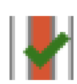
Alternatively, you can click the **Select Columns** icon on the control panel.



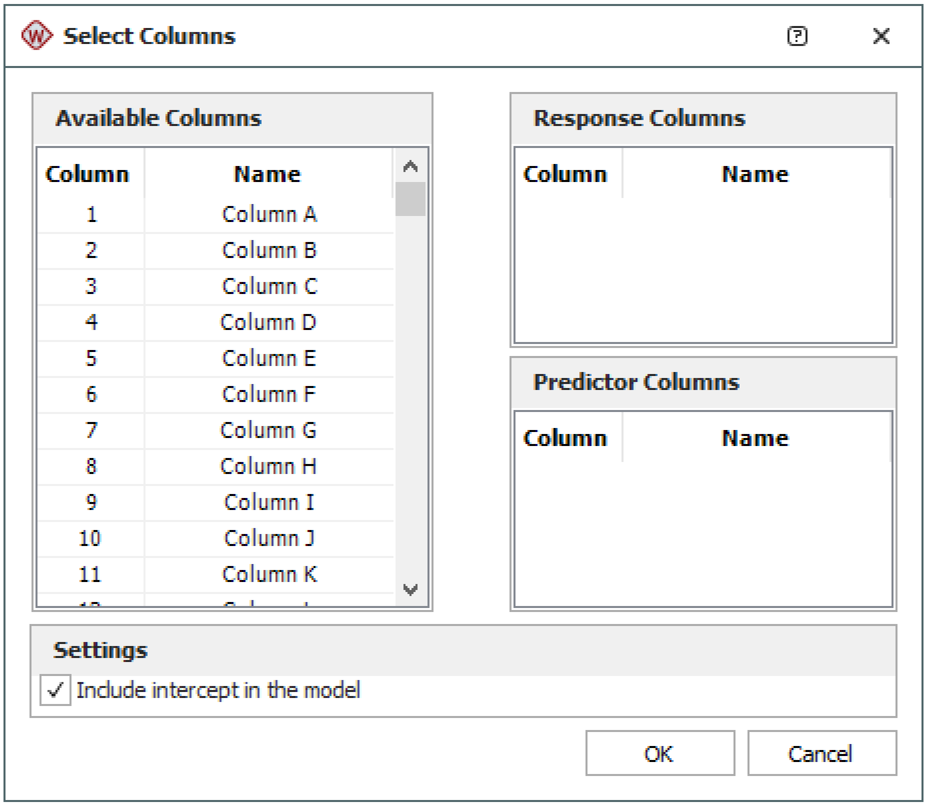
If you wish to rename any of the columns, right-click the column heading and choose **Rename Column** on the shortcut menu.

Select Columns Window

The Select Columns window allows you to specify, for each column on the Data tab of the multiple linear regression folio, whether that column contains response data or predictors (e.g., factor levels). It is accessed by clicking the **Select Columns** icon on the Main page of the Data tab control panel.



This window will also appear if you try to calculate the data or generate plots without specified response and predictor columns.



- The **Available Columns** area displays the columns on the Data tab that have not been specified as containing either response data or factor settings.
- The **Response Columns** area displays the columns that have been specified as containing response data.
- The **Predictor Columns** area displays the columns that have been specified as containing values for predictor levels.
- Select one or more columns in the **Available Columns** area and then drag them to either of the other two areas. To select multiple columns, click the first one and then press and hold **CTRL** while you individually select additional columns. You can also press **SHIFT** to automatically select all the columns in between the first and second column you click.
- The **Include intercept in the model** option includes the intercept in the analysis for all responses, thereby removing any effects that do not vary.

Once you have specified the columns you want to use in the analysis, click **OK** to apply the changes to the folio.

Multiple Linear Regression Folio Analysis Results

When accessed from a multiple linear regression folio, the Analysis Summary window will contain detailed information about analysis results, including information that describes how each predictor affects response that is currently selected on the control panel.

If the current response data has been analyzed, you can open the window by clicking the **View Analysis Summary** icon on the control panel.

If the current response data has not been analyzed, the icon will still be available so you can view the folio's analysis history.

Select an item in the Available Report Items panel to display it on the spreadsheet. Each item is described next.

Analysis Results

The **Analysis of Variance (ANOVA) table** provides general information about the effects of the predictors on the selected response. This information may be presented for individual predictors and or for all the predictors treated as a single group, depending on your analysis setting on the control panel.

ANOVA Table Columns

- **Source of Variation** is the source that caused the difference in the observed output values. The main effects of each predictor will be listed individually or grouped as "main effects," depending on whether you selected to use individual or grouped terms in the analysis (specified on the Analysis Settings page of the control panel). Sources displayed in red are considered to be significant.
- The number of **Degrees of Freedom** for the **Model** is the number of regression coefficients for the effects included in the analysis (e.g., two coefficients might be included in the regression table for a given main effect). The number of degrees of freedom for the **Residual** is the total number of observations minus the number of parameters being estimated.
- **Sum of Squares** is the amount of difference in observed output values caused by this source of variation.

- **Mean Squares** is the average amount of difference caused by this source of variation. This is equal to Sum of Squares/Degrees of Freedom.
- **F Ratio** is the ratio of Mean Squares of this source of variation and Mean Squares of pure error. A large value in this column indicates that the difference in the output caused by this source of variation is greater than the difference caused by noise (i.e., this source affects the output).
- **P Value** (alpha error or type I error) is the probability that an equal amount of variation in the output would be observed in the case that this source does not affect the output. This value is compared to the risk level (alpha) that you specify on the Analysis Settings page of the control panel. If the *p* value is less than alpha, this source of variation is considered to have a significant effect on the output. In this case, the term and its *p* value will be displayed in red.
- The following values are shown underneath the ANOVA table:
  - **S** is the standard error of the noise. It represents the magnitude of the response variation caused by noise.
  - **R-sq** is the percentage of total difference that is attributable to the factors under consideration. It is equal to Sum of Squares(factor)/Total Sum of Squares.
  - **R-sq(adj)** is an R-sq value that is adjusted for the number of parameters in the model.
  - **PRESS** is the prediction error sum of squares, which provides a measure of the model's validity. The lower the PRESS value, the better the model's predictive ability.
  - **R-sq(pred)** is a measure of how well the model predicts new observations. It is equal to 1-PRESS/Total Sum of Squares. The larger the value, the more accurate the model's predictions are likely to be.

The **Regression table** provides specific information on the contribution of predictor to the variation in the response and an analysis of the significance of this contribution.

Regression Table Columns

- **Term** is the factor under consideration. Terms displayed in red are considered to be significant. In cases where there is no error in the model, significant effects are determined according to Lenth's method and the term names are displayed in red and followed by an asterisk (*).

- **Coefficient** is the regression coefficient of the term, which represents the contribution of the term to the variation in the response.
- **Standard Error** is the standard deviation of the regression coefficient.
- **Low Confidence** and **High Confidence** are the lower and upper confidence bounds on the regression coefficient.
- **T Value** is the normalized regression coefficient, which is equal to Coefficient/Standard Error.
- **P Value** (alpha error or type I error) is the probability that an equal amount of variation in the output would be observed in the case that this term does not affect the output. This value is compared to the risk level (alpha) that you specify on the Anaysis Settings page of the control panel. If the *p* value is less than alpha, this source of variation is considered to have a significant effect on the output. In this case, the term and its *p* value will be displayed in red.
- **Variance Inflation Factor** is a measure of the correlation, if any, between the term (predictor) and the other predictors. The lower the value, the less likely it is that the predictors are correlated. If the correlation of a predictor with other predictors is extremely high, that predictor should be removed from the model. If predictors are 100% correlated, they are aliased and will automatically be removed from the model. This will be noted directly above the Regression Table.

The **Regression Equation** information is presented using multiple tables.

Regression Equation

- The **Response** table displays the response that the regression equation applies to and the units of measurement that were entered for the response (if any).
- The **Additional Settings** table shows the transformation and risk level you entered for the response.
- The **Significant Terms** table is applicable only when at least one term was found to be significant. It shows the significant terms in the Name column and the associated regression coefficients in the Coefficient column.
- The **Equation** tables show the regression coefficients for the model of the selected response. For example, consider this table:

Equation	
Yield =	
-29.8750	
-22.5625	A:Aperture Setting
+1.7250	B:Exposure Time
-0.1500	C:Develop Time

The corresponding model for this table is $y = -29.8750 - 22.5625x_1 + 1.7250x_2 - 0.1500x_3$.

Additional Results

All of the following tables provide information that was generated from the main calculations. The available tables will vary depending on the design type you are working with. The results that could be available include:

Alias Structure

This item is available for all designs with at least two factors. It describes the alias structure for the design, taking into account only the terms you've selected to include in the analysis. Together with your engineering knowledge, you can use this information to help determine whether any important interaction information was lost due to aliasing. When aliased terms exist, the following areas will be shown:

- **Terms selected to be in the model** lists all the terms that are considered for inclusion in the regression model (i.e., the selections in the Select Terms window).
- **Terms included in the model** lists all the selected terms that are included in the model. The alias structure determines which terms are excluded.
- **Alias Structure** lists the aliased effects based on the selected terms. For example, A • B = A • B + C • D means the interaction effect A • B is aliased because it is indistinguishable from effect C • D. Therefore, the model cannot include both interaction terms; it will include only one (e.g., A • B).

Alias Summary

The terms in the first column of this table are aliased with the terms shown in the second column. Only the terms in the first column are included in the model.

Var/Cov Matrix

This shows the variance/covariance matrix, which is available for one factor R-DOE designs and all other designs with two or more factors. The diagonal elements in this matrix are used to calculate the coefficients in the MLE or Regression Information table.

Diagnostic Information

This table is available for one factor R-DOE designs and all other designs with two or more factors. It displays various analysis results for each run and highlights significant values. The following columns are included:

- **Run Order** is the randomized order, generated by the software, in which it is recommended to perform the runs to avoid biased results. Note that any changes made to the Run Order column on the Data tab will be reflected here.
- **Standard Order** is the basic order of runs, as specified in the design type, without randomization. Note that any changes made to the Standard Order column on the Data tab will be reflected here.
- **Actual Value (Y)** is the observed response value for the run, as entered in the response column on the Data tab.
- **Predicted Value (YF)** is the response value predicted by the model given the factor settings used in the run.
- **Residual** (or "regular residual") is the difference between the actual value (Y) and the predicted value (YF) for the run.
- **Standardized Residual** is the regular residual for the run divided by the constant standard deviation across all runs.
- **Studentized Residual** is the regular residual for the run divided by an estimate of its standard deviation.
- **External Studentized Residual** is the regular residual for the run divided by an estimate of its standard deviation, where the run in question is omitted from the estimation.
- **Leverage** is a measure of how much the run influences the predicted values of the model, stated as a value between 0 and 1, where 1 indicates that the actual response value of the run is exactly equal to the predicted value (i.e. the predicted value is completely dependent upon the observed value).
- **Cook's Distance** is a measure of how much the output is predicted to change if the run is deleted from the analysis.

Values that are considered to be significant, or outliers, are displayed in red. For the residual columns, significant or critical values are those that fall outside the residual's upper or lower bounds, calculated based on the specified alpha (risk) value.

The ReliaWiki resource portal has more information on how significant values are determined for the Leverage and Cook's Distance columns at: http://www.reliawiki.org/index.php/Multiple_Linear_Regression_Analysis.

Least Squares Means

This table shows the predicted response values for the given factor levels. It includes the following columns:

- **Effect** is the main effect or interaction used to predict the response. The coefficients for effects not used in the prediction are set to zero.
- **Level** is the combination of factor levels used to predict the response.
- **Mean** is the predicted response value.

Multiple Linear Regression Folio: Plots

The following plots may be available on the Analysis Plot tab of the multiple linear regression folio. Note that the contour plot and residual vs. factor plot are available only when there are at least two factors included in the analysis. For general information on working with plots, see Plot Utilities.

Effect Plots

Effect plots allow you to visually evaluate the effects of factors and factorial interactions on the selected response.

- The **Pareto Chart - Regression** plot shows the standardized effects of the selected terms (i.e., factor or combination of factors). The vertical blue line is the threshold value. If a bar is beyond the blue line, it will be red, indicating that the effect is significant.
- The **Pareto Chart - ANOVA*** plot shows the inverse p value (1 - p) of each selected term. The vertical blue line is the threshold value. If a bar is beyond the blue line, it will be red, indicating that the term is significant.
- The **Scatter Plot** shows the observed values of the currently selected response plotted against the levels of the selected factor. A 3-dimensional version of this plot is available in the 3D plot

folio.

- The **Contour Plot** shows how varying two selected factors affects the predicted response values, which are represented as colors. See Contour Plots. A 3-dimensional version of this plot ("Surface Plot") is available in the 3D plot folio. For mixture designs, this plot applies only to process factors (i.e., process variables).

Residual Plots

Residuals are the differences between the observed response values and the response values predicted by the model at each combination of factor values. Residual plots help to determine the validity of the model for the currently selected response. When applicable, a residual plot allows the user to select the type of residual to be used:

- **Regular Residual** is the difference between the observed Y and the predicted Y.
- **Standardized Residual** is the regular residual divided by the constant standard deviation.
- **Studentized Residual** is the regular residual divided by an estimate of its standard deviation.
- **External Studentized Residual** is the regular residual divided by an estimate of its standard deviation, where the observation in question is omitted from the estimation.

The plots are described next.

- The **Residual Probability*** plot is the normal probability plot of the residuals. If all points fall on the line, the model fits the data well (i.e., the residuals follow a normal distribution). Some scatter is to be expected, but noticeable patterns may indicate that a transformation should be used for further analysis. Two additional measures of how well the normal distribution fits the data are provided by default in the lower title of this plot. Smaller values for the Anderson-Darling test indicate a better fit. Smaller *p* values indicate a worse fit.
- The **Residual vs. Fitted*** plot shows the residuals plotted against the fitted, or predicted, values of the selected response. If the points are randomly distributed around the "0" line in the plot, the model fits the data well. If a pattern or trend is apparent, it can mean either that the model does not provide a good fit or that Y is not normally distributed, in which case a transformation should be used for further analysis. Points outside the critical value lines, which are calculated based on the specified alpha (risk) value, may be outliers and should be examined to determine the cause of their variation.

- The **Residual vs. Order*** plot shows the residuals plotted against the order of runs used in the design. If the points are randomly distributed in the plot, it means that the test sequence of the experiment has no effect. If a pattern or trend is apparent, this indicates that a time-related variable may be affecting the experiment and should be addressed by randomization and/or blocking. Points outside the critical value lines, which are calculated based on the specified alpha (risk) value, may be outliers and should be examined to determine the cause of their variation.

- The **Residual vs. Factor*** plot shows the residuals plotted against values of the factor selected in the **Residual Factor** area. It is used to determine whether the residuals are equally distributed around the "0" value line and whether the spread and pattern of the points are the same at different levels. If the size of the residuals changes as a function of the factor's settings (i.e., the plot displays a noticeable curvature), the model does not appropriately account for the contribution of the selected factor. Points outside the critical value lines, which are calculated based on the specified alpha (risk) value, may be outliers and should be examined to determine the cause of their variation.

- The **Residual Histogram*** is used to demonstrate whether the residual is normally distributed by dividing the residuals into equally spaced groups and plotting the frequency of the groups. The **Residual Histogram Settings** area allows you to:

  - Select **Custom Bins** to specify the number of groups, or bins, into which the residuals will be divided. Otherwise, the software will automatically select a default number of bins based on the number of observations.

  - Select **Superimpose pdf** to display the probability density function line on top of the bins.

- The **Residual Autocorrelation*** plot shows a measure of the correlation between the residual values for the series of runs (sorted by run order) and one or more lagged versions of the series of runs. The default number of lags is the number of observations, *n*, divided by 4. If you select **Custom Lags** in the **Auto-Correlation Options** area, you can specify up to *n* -1 lags. The correlation is calculated as follows:

$$r_k = \frac{\sum_{i=1}^{N-k}\left(Y_i - \overline{Y}\right)\left(Y_{i+k} - \overline{Y}\right)}{\sum_{i=1}^{N}\left(Y_i - \overline{Y}\right)^2}$$

  where:

- k is the lag.
- $\overline{Y}$ is the mean value of the original series of runs.

For example, lag 1 shows the autocorrelation of the residuals when run 1 is compared with run 2, run 2 is compared with run 3 and so on. Lag 3 shows the autocorrelation of the residuals when run 1 is compared with run 4, run 2 is compared with run 5 and so on. Any lag that is displayed in red is considered to be significant; in other words, there is a correlation within the data set at that lag. This could be caused by a factor that is not included in the model or design, and may warrant further investigation.

- The **Fitted vs. Actual** plot shows the fitted, or predicted, values of the currently selected response plotted against the observed values of the response. If the model fits the data well, the points will cluster around the line.

Diagnostic Plots

- The **Leverage vs. Order** plot shows *leverage* plotted against the order of runs used in the design. Leverage is a measure (between 0 and 1) of how much a given run influences the predicted values of the model, where 1 indicates that the actual response value of the run is exactly equal to the predicted value (i.e. the predicted value is completely dependent upon the observed value). Points that differ greatly from the rest of the runs are considered outliers and may distort the analysis.
- The **Cook's Distance*** plot can show Cook's distance (i.e., a measure of how much the output is predicted to change if each run is deleted from the analysis) plotted against either the run order or the standard order for the currently selected response. Points that differ greatly from the rest of the runs are considered outliers and may distort the analysis.
- The **Box-Cox Transformation*** plot can help determine, for the currently selected response and model, what transformation, if any, should be applied. The plot shows the sum of squares of the residuals plotted against lambda. The value of lambda at the minimum point of this curve is considered the "best value" of lambda, and indicates the appropriate transformation, which is also noted by default in the lower title of the plot.

* These plots are available only when there is error in the design, indicated by a positive value for sum of squares for Residual in the ANOVA table of the analysis results.

Multiple Linear Regression Folio: Example

*The data set used in this example is available in the example database installed with the software (called "Weibull21_DOE_Examples.rsgz21"). To access this database file, choose **File > Help**,* *click **Open Examples Folder**, then browse for the file in the Weibull sub-folder.*

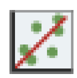
The name of the example project is "Multiple Linear Regression - Ice Cream Consumption."

The multiple linear regression folio provides an easy way to quickly analyze existing data. In this example, ice cream consumption per capita was measured in four-week periods from March 18, 1951 to July 11, 1953 in order to determine whether it depended on the price of ice cream, the weekly family income and/or the mean temperature.

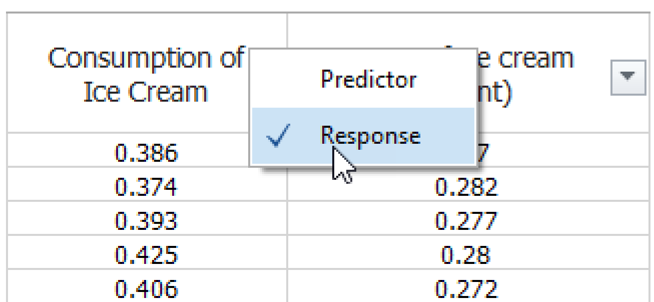
Creating the Folio

To investigate the predictors, the analysts create a multiple linear regression folio, and enter the predictor and response values on the Data tab. The design matrix and the response data are given in the "Ice Cream Consumption" folio. The following steps describe how to create this folio on your own.

1. On the Home tab, click the **Multiple Linear Regression** icon in the Insert gallery.

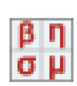


2. Right-click the column headings to rename each of the columns.
3. Click the arrow inside a heading to specify whether the column contains predictor data (e.g., mean temperature) or response data (i.e., ice cream consumption).



Analysis and Results

The data set for this example is given in the "Ice Cream Consumption" folio of the example project. After you enter the data from the example folio, you can perform the analysis by clicking the **Calculate** icon.



The ANOVA table and the Regression Information table for the model are available in the Analysis Summary window after you click the **View Analysis Summary** icon on the control panel. The R-sq and R-sq(adj) values in the ANOVA table indicate that the model fits the data relatively well.

ANOVA Table

Source of Variation	Degrees of Freedom	Sum of Squares [Partial]	Mean Squares [Partial]	F Ratio	P Value
Model	3	0.090251	0.030084	22.174889	2.450504E-7
Main Effects	3	0.090251	0.030084	22.174889	2.450504E-7
Residual	26	0.035273	0.001357		
Lack of Fit	26	0.035273	0.001357		
Total	29	0.125523			
S =	0.036833		PRESS =	0.047845	
R-sq =	71.90%		R-sq(pred) =	61.88%	
R-sq(adj) =	68.66%				

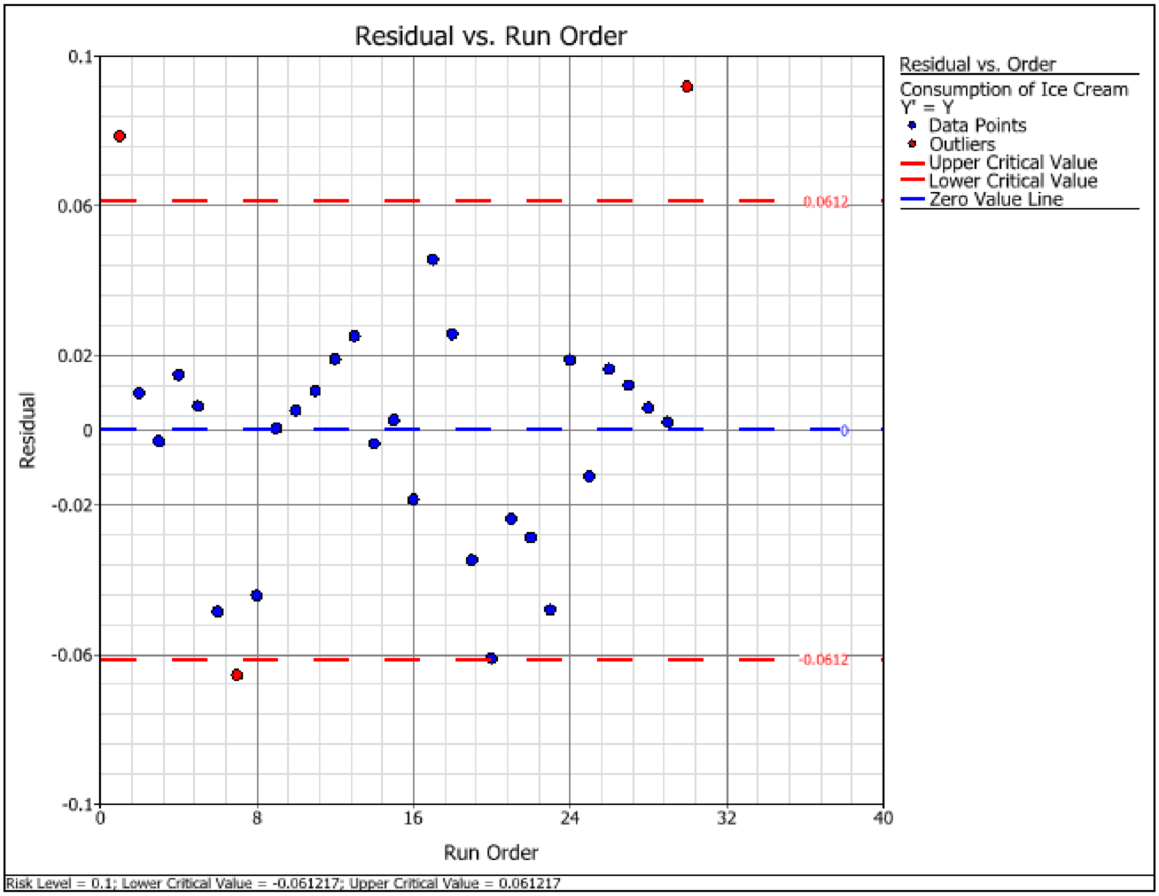
The Regression Information table indicates that weekly family income and mean temperature are good predictors for ice cream production.

Term	Effect	Coefficient	Standard Error	Low Confidence	High Confidence	T Value	P Value	Variance Inflation Factor
Intercept		0.197315	0.270216	-0.263571	0.658201	0.730212	0.471789	
Price of ice cream($/pint)	-2.088828	-1.044414	0.834357	-2.467509	0.378681	-1.251759	0.221803	1.035673
Weekly family income($)	0.006616	0.003308	0.001171	0.00131	0.005306	2.823722	0.008989	1.144186
Mean temperature(F)	0.006917	0.003458	0.000446	0.002698	0.004218	7.762213	3.100024E-8	1.144367

Since the price of ice cream does not seem to predict ice cream consumption, the folio was duplicated in the "Reduced Model" folio, where the price column was removed from the analysis by clicking the arrow in the heading and clearing all the check boxes. Click the **View Analysis Summary** icon in the new folio to view the final regression model.

Term	Effect	Coefficient	Standard Error	Low Confidence	High Confidence	T Value	P Value	Variance Inflation Factor
Intercept		-0.113195	0.10828	-0.297627	0.071237	-1.045395	0.30511	
Weekly family income($)	0.00706	0.00353	0.00117	0.001537	0.005523	3.017346	0.005506	1.117863
Mean temperature(F)	0.007087	0.003543	0.000445	0.002785	0.004301	7.963273	1.470071E-8	1.117863

Click the **Plot** icon to view the residual vs. order plot, which you can use to look for outliers in the data set.



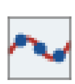
Conclusions

Based on the analysis, weekly family income and mean temperature are found to be the most significant factors in ice cream consumption. From the residual vs. order plot, we see that there are a few outliers but the majority of our data set fits the model. Thus, if desired, we could use the Prediction window to predict future ice cream consumption based on different combinations of family income and mean temperature.

Equation Fit Solver

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++.

To add an Equation Fit Solver folio to a project, choose **Home > Insert > Equation Fit Solver**, or right-click the **Regression 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 (Hours)
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:

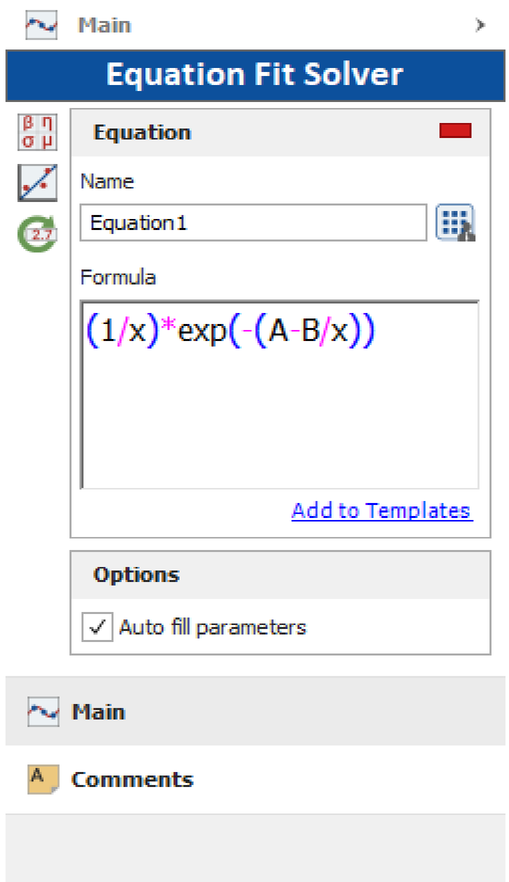
- m is the mean life.
- T is the operating temperature (i.e., use stress level).
- A and B are the parameters of the equation.

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. 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.

   a. 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:

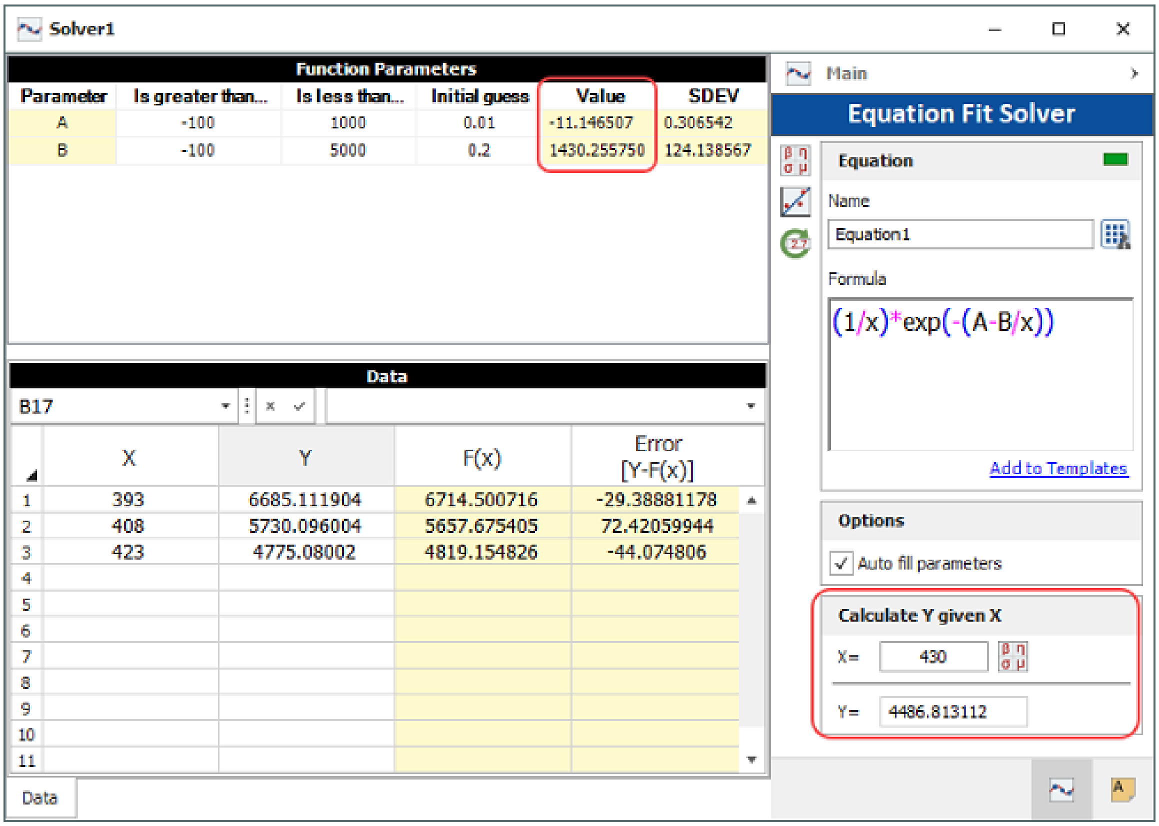
      - A is estimated to be greater than **-100** and less than **1000**.
      - B is estimated to be greater than **-100** and less than **5000**.

   b. 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.

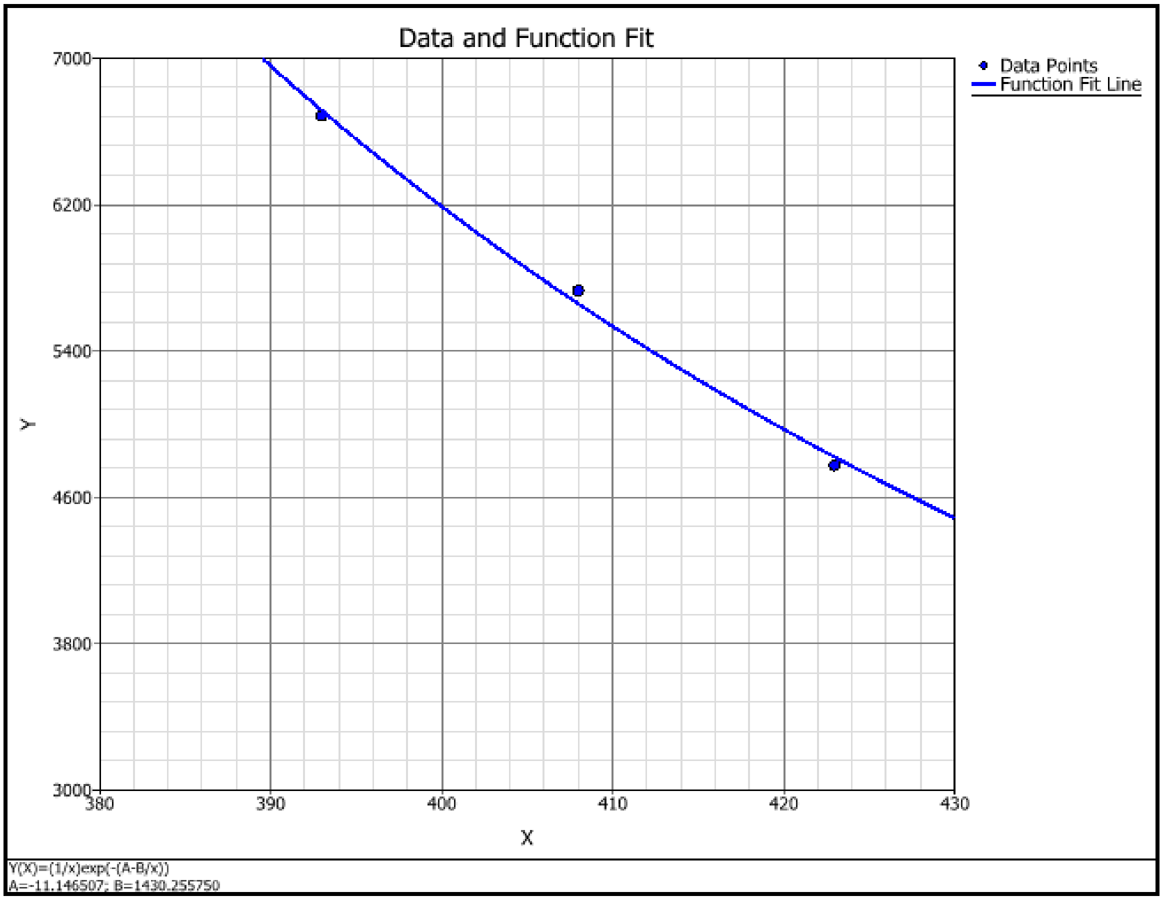


The following columns allow you to evaluate the results of the parameter estimation:

- **SDEV** shows the standard deviation of the values of the parameters, which helps you to evaluate the amount of variability in the data set.
- **F(x)** 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.
- **Error [Y-F(x)**] 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

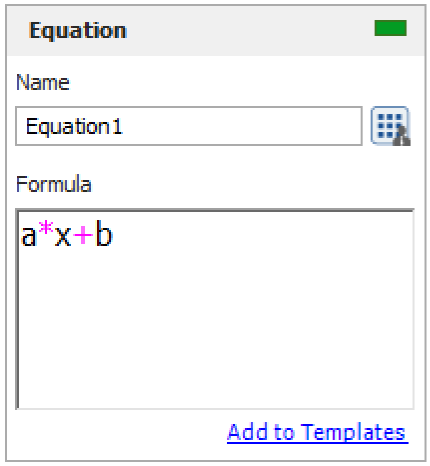
adjusted).



Equation Fit Solver Control Panel

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.

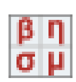


- To use a saved template, choose **Equation Fit Solver > Solver Templates > Select from Templates** or click the icon on the control panel.

  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 automatically copies 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 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 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. This tool is also available by choosing **Equation Fit Solver > Analysis > Initialize Parameters**.

Simulation

IN THIS CHAPTER

Monte Carlo and SimuMatic

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 or reliability growth tests

- Compare different parameter estimation methods
- Evaluate the impact of different censoring schemes

When using reliability growth analysis, the engineer will typically fit a reliability growth model to actual data obtained from developmental testing or fielded repairable systems operating in the field. However, in some situations, it may be useful to generate simulated data sets containing values that are distributed according to a specified set of parameters. For example, simulated data could be used to:

- Design reliability growth tests.
- Obtain simulation-based confidence bounds.
- Experiment with the influences of sample sizes and data types on analysis methods.
- Evaluate the impact of allocated test time.

You can use Monte Carlo simulation to produce data sets based on various user inputs, such as data type, the beta and lambda parameters of the Crow-AMSAA (NHPP) model and sample size. The software will randomly generate input variables that follow a specified probability distribution. In the case of reliability growth and repairable system data analysis, the goal is to generate failure times for systems that are assumed to have specific characteristics. Therefore, the inter-arrival times of the failures will follow a non-homogeneous Poisson process with a Weibull failure intensity (as specified in the Crow-AMSAA model).

Weibull++ offers the following utilities for generating and analyzing simulated data:

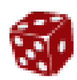
- The Monte Carlo utility (which comes in a life data version, a life-stress version and a reliability growth version) uses Monte Carlo simulation to generate a single data set based on various user inputs, such as distribution type, distribution parameters and sample size. The data set is then automatically placed in an analysis folio, where it can be analyzed like any other data set. For growth analysis, the Repairable Systems Monte Carlo utility uses simulation to generate a single data set containing values that are distributed according to the Crow-AMSAA model with specified beta and lambda parameters. The data set is then automatically placed in a growth data folio, where it can be analyzed like any other data set.
- SimuMatic (which also comes in a life data version, a life-stress version and a reliability growth version) generates a large number of data sets using Monte Carlo simulation. It then analyzes the group of data sets as a whole. You can use SimuMatic, for example, to find the average reliability at a given time for a thousand simulated data sets. For growth analysis, the Repairable Systems SimuMatic folio generates a large number of data sets using Monte Carlo simulation. It then automatically analyzes the group of data sets as a whole in order to explore

a variety of questions. For example, you can use SimuMatic to calculate the simulation-based confidence bounds on the demonstrated MTBF.

Life Data and Life-Stress Data Monte Carlo Utility

The 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 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 life data folio or life-stress data folio, where it can be analyzed like any other data set.

To access the Monte Carlo utility, choose **Home > Insert > Monte Carlo**. If you have Accelerated Life Testing activated on your computer, you will be offered a choice between the **Monte Carlo** or **Stress-Dependent Monte Carlo** utility.



In general, you can produce a data set in just a few steps:

- Use the Main tab of the setup window to specify the distribution/model and its parameter values.
- Use the Censoring tab of the setup window to specify the censoring type.
- Use the Settings tab of the setup window to specify the number of data points to generate.
- In Accelerated Life Testing, additional steps are required:
  - Use the Stress tab of the setup window to specify an appropriate use level and transformation (if applicable) for each stress.
  - Use the data sheet on the right side of the setup window to specify the stress levels and the number of data points to generate for each stress level.

To learn more, the following sections provide complete descriptions of all available options in the utility.

Main Tab

The Main tab of the setup window allows you to select a distribution/model, enter its required parameter values and select the time or usage units (e.g., hours) for the failures/suspensions in the data set.

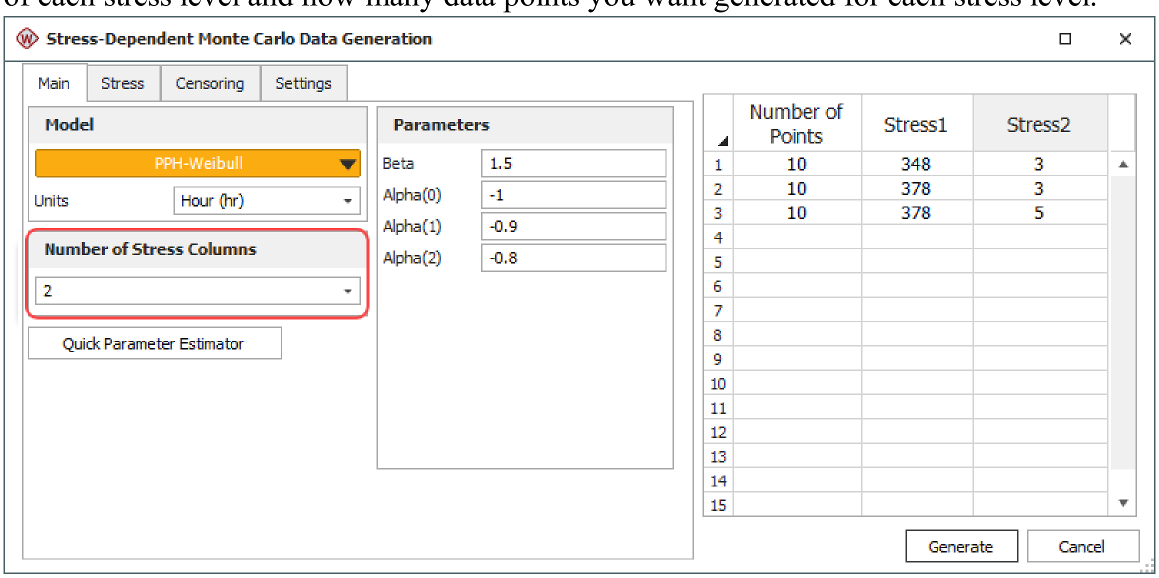
If the estimated values of the distribution parameters are not available, you can use the **Quick Parameter Estimator** button to solve for them.

For the life data Monte Carlo utility:

- If you select the normal, logistic or generalized gamma distributions, the **Allow Negative Values** check box will be 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.
- If you select the mixed Weibull distribution, you will need to specify parameters for each subpopulation. The Portion field is for the 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.
- If you select the User-Defined distribution, you can manually enter an equation that relates different random variables. (See User-Defined Equations.)
  - In the **Equation** area, use *R* to insert a uniform random variable.
  - Click the **Insert Data Source** button if you wish to generate simulated values based on distributions calculated from a folio in the project.

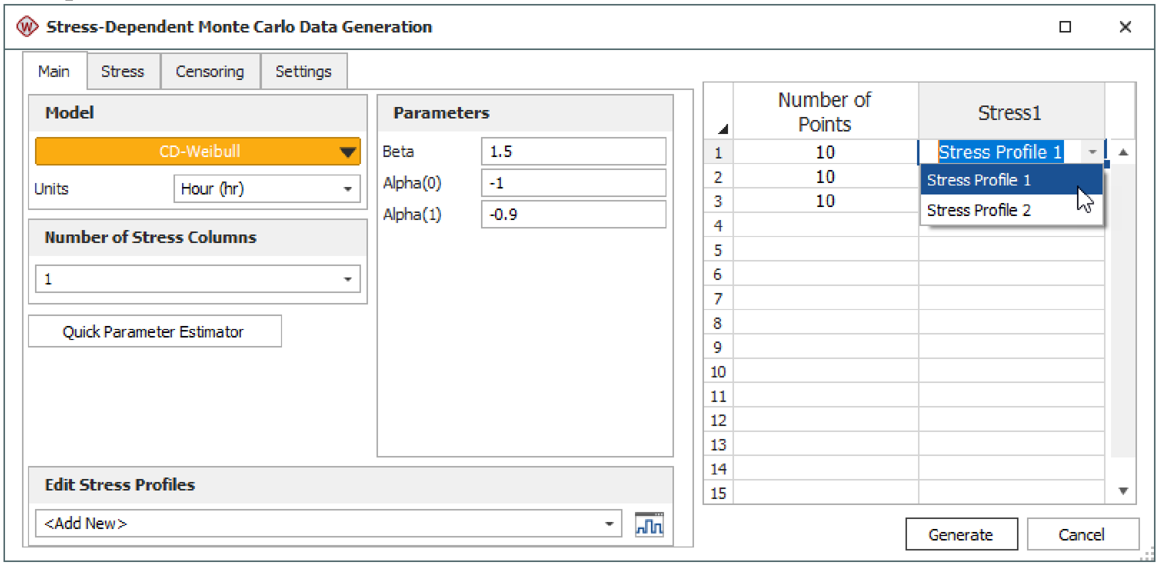
For the Stress-Dependent Monte Carlo utility:

- If you select the proportional hazards (PPH), general log-linear (GLL) or cumulative damage (CD) model, the **Number of Stress Columns** area will be enabled. It allows you to add stress columns to the data sheet on the right side of the window. In the data sheet, specify the value of each stress level and how many data points you want generated for each stress level.



- If you select the cumulative damage (CD) model, you must assign time-dependent stress profiles to the stress columns in the data sheet.

  To create a new profile, choose **<Add New>** from the **Edit Stress Profiles** drop-down list and then click the **Create/View Profile** icon. To edit an existing profile, choose it from the drop-down list and then click the icon.



Stress Tab (Accelerated Life Testing Only)

The Stress tab of the setup window applies to the Stress-Dependent Monte Carlo utility only. It allows you to specify a use level for each stress type. There will be a separate input box for each stress column in the data sheet.

If you have selected the general log-linear (GLL) or cumulative damage (CD) model, you must also select an appropriate **Transformation** for each stress:

- **None X=V** is 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).
- **Reciprocal X=1/V** is for stress types associated with the Arrhenius LSR. This transformation is commonly used for thermal stresses.
- **Logarithmic X=ln[V]** is for stress types associated with the inverse power law LSR. This transformation is commonly used for non-thermal stresses.

Censoring Tab

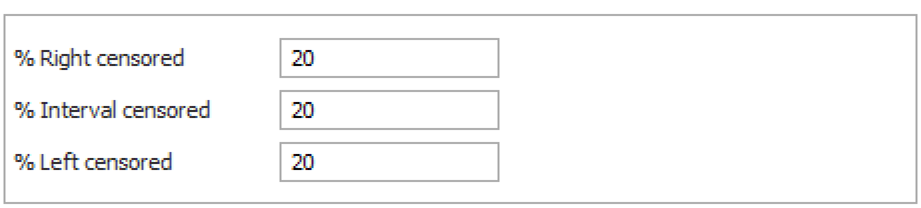
- **No censoring**: The generated data set will contain only exact failure times (i.e., it will contain no right censored, interval or left censored data).

- **Right censoring after specific number of failures**: You specify the number of failures, *n*, that the generated data set will contain. After the data are generated, they are 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 in the Stress-Dependent Monte Carlo utility, 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).

- **Random censoring** (available in for life data only): 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, the settings in following picture shows that 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.

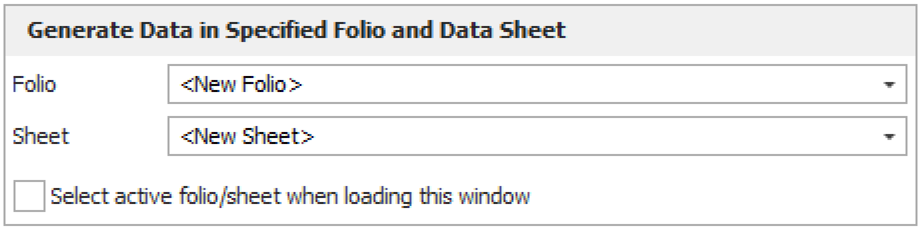
Settings Tab

The Settings tab of the setup window allows you to specify how you want to generate the data set and where you want the data to be stored:

- **Use Seed** allows you 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.
- **Math Precision** allows you to specify the number of decimal places to use for each simulated data point.
- **Data Points** sets the number of data points you wish to generate.
  - For the life data Monte Carlo utility, use the **Number of points** field.
  - For the Stress-Dependent Monte Carlo utility, use the data sheet on the right side of the window. For example, if you have selected a multi-stress model, the following settings would generate 10 data points for Stress 1 = 348 and Stress 2 = 3, 10 data points for Stress 1 = 378 and Stress 2 = 3, and 10 data points for Stress 1 = 378 and Stress 2 = 5.

	Number of Points	Stress1	Stress2
1	10	348	3
2	10	378	3
3	10	378	5

- **Generate Data in Specified Folio and Data Sheet** allows you to choose where to put the simulated data. Click the **Active** button to select the folio and data sheet that were active when the utility was opened. If you want the utility to always select the active folio and data sheet, select the **Select active folio/sheet when loading this window** check box.

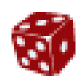


Repairable Systems Monte Carlo Utility

For reliability growth data analysis only.

Weibull++'s Repairable Systems Monte Carlo utility uses simulation to generate a single data set containing values that are distributed according to the Crow-AMSAA (NHPP) model with specified beta and lambda parameters.

To access the Monte Carlo utility, choose **Home > Insert > Monte Carlo** and choose the **Repairable Systems Monte Carlo** option.



The setup window will appear. Follow the steps outlined below to set up the simulation:

1. On the Main tab of the window, select a data type and time units in the **Data Type** area. In the **Parameters** area, enter the beta and lambda values for the Crow-AMSAA model that will be used to generate the data set.
2. In the **Data Sets / Points** area, enter the following information:
   - Specify the **Number of systems** for which data will be generated. This value is fixed at 1 unless you selected a multiple systems data type on the Main tab (i.e., Multiple Systems - Concurrent or Repairable Systems).
   - Use the **Test termination** field to specify what will determine the end of the observation period for the simulated data.
     - If the period will end after a specific number of failures have been observed, choose **Failure Terminated** from the drop-down list and enter the number of **Failures** in the next field.
     - If the observation period will end at a specific time, choose **Time Terminated** from the drop-down list and enter a **Time** in the next field.

- If you selected to generate Grouped Failure Times data, click the **Set Intervals** link to open a window that allows you to specify the intervals that will contain the failures. You need at least three intervals to simulate this type of data.

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** area, enter the number of decimal places that will be used for each simulated data point. (This does not affect the precision used in the calculated parameters and other results.)
    - Use the **Generate Data in Specified Folio** drop-down list to specify where to put the simulated data. If you select a folio that already exists in the project, the data will be placed into a new data sheet at the end of that folio, and any existing data sheets and plots will remain unchanged. If you choose **<New Folio>** the software will create a new folio to place the data into.

4. After the simulation is set up, click **Generate** to create a new growth data folio data sheet populated with a data set that meets your specifications.

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 (see User-Defined Equation Example).

Entering a User-Defined Equation

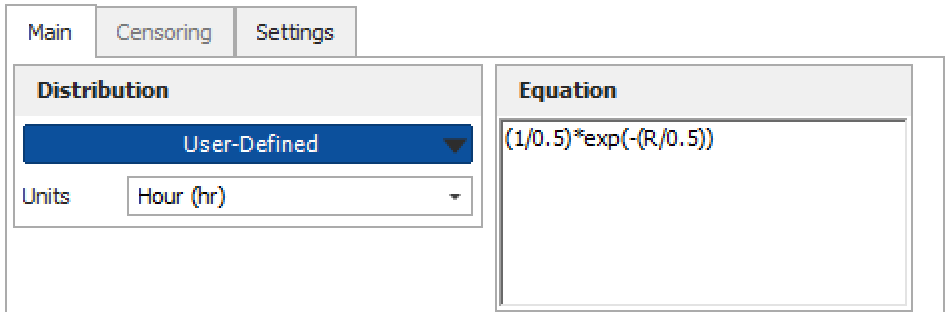
When you select **User-Defined** in the **Distribution** area on the Main tab of the Monte Carlo utility, the **Parameters** area will be replaced with an **Equation** field.

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 $\beta = 0.5$.

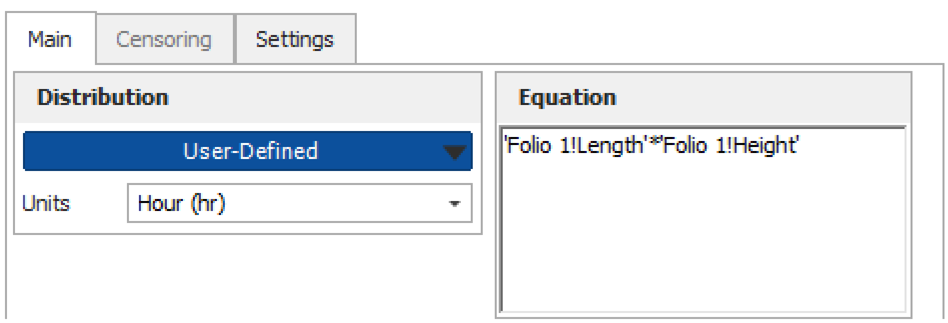
$$f(x, \beta) = \frac{1}{\beta} \times e^{-(x/\beta)}$$

Substituting the value of β = 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 enter **R*R**, the software will compute the product of two different uniform random numbers.

- **Draw a random number from a distribution calculated in a life data folio** If you want to obtain a random number from the distribution in an existing life data 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:

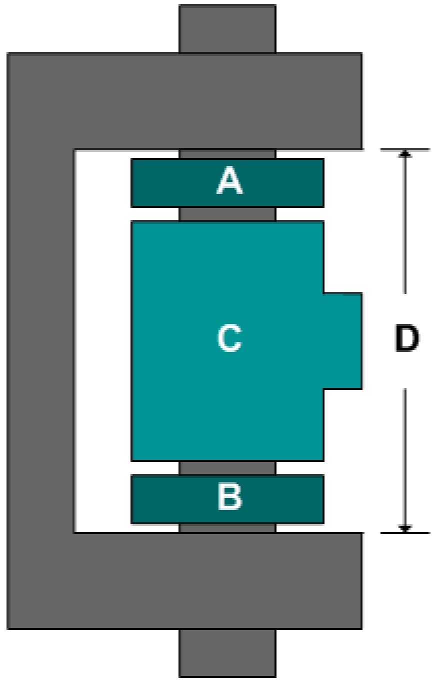


User-Defined Equation Example

*The data set used in this example is available in the example database installed with the Weibull++ software (called "Weibull21_Examples.rsgz21"). 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 "Monte Carlo Simulation - User-Defined Equation - Hinge."

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.

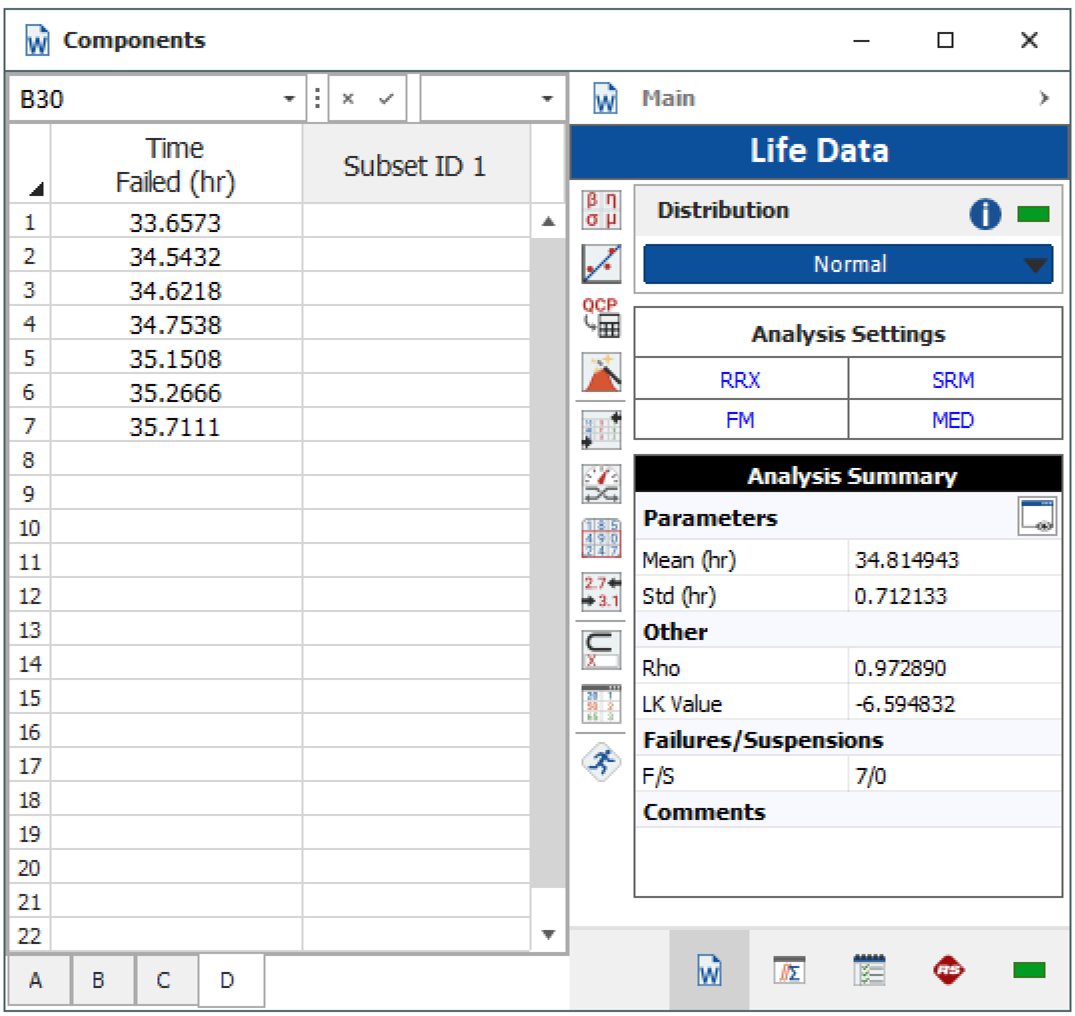
Dimensions for A (cm)	Dimensions for B (cm)	Dimensions for C (cm)	Dimensions for D (cm)
2.0187	1.9795	30.4216	33.6573
1.9996	2.0288	29.9818	34.5432
2.0167	1.9883	29.9724	34.6218
2.0329	2.0327	30.192	34.7538
2.0233	2.0119	29.9421	35.1508

2.0273	2.0354	30.1343	35.2666
1.984	1.9908	30.0423	35.7111

Obtain Distributions for the Part Dimensions

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++ life data 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.



Use Monte Carlo Simulation to Generate a Data Set

On the Home tab, click the **Monte Carlo** icon in the Insert gallery. (If you have Accelerated Life Testing activated on your computer, you will be given a choice between "Monte Carlo" or "Stress-Dependent Monte Carlo." Choose **Monte Carlo**.)

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.

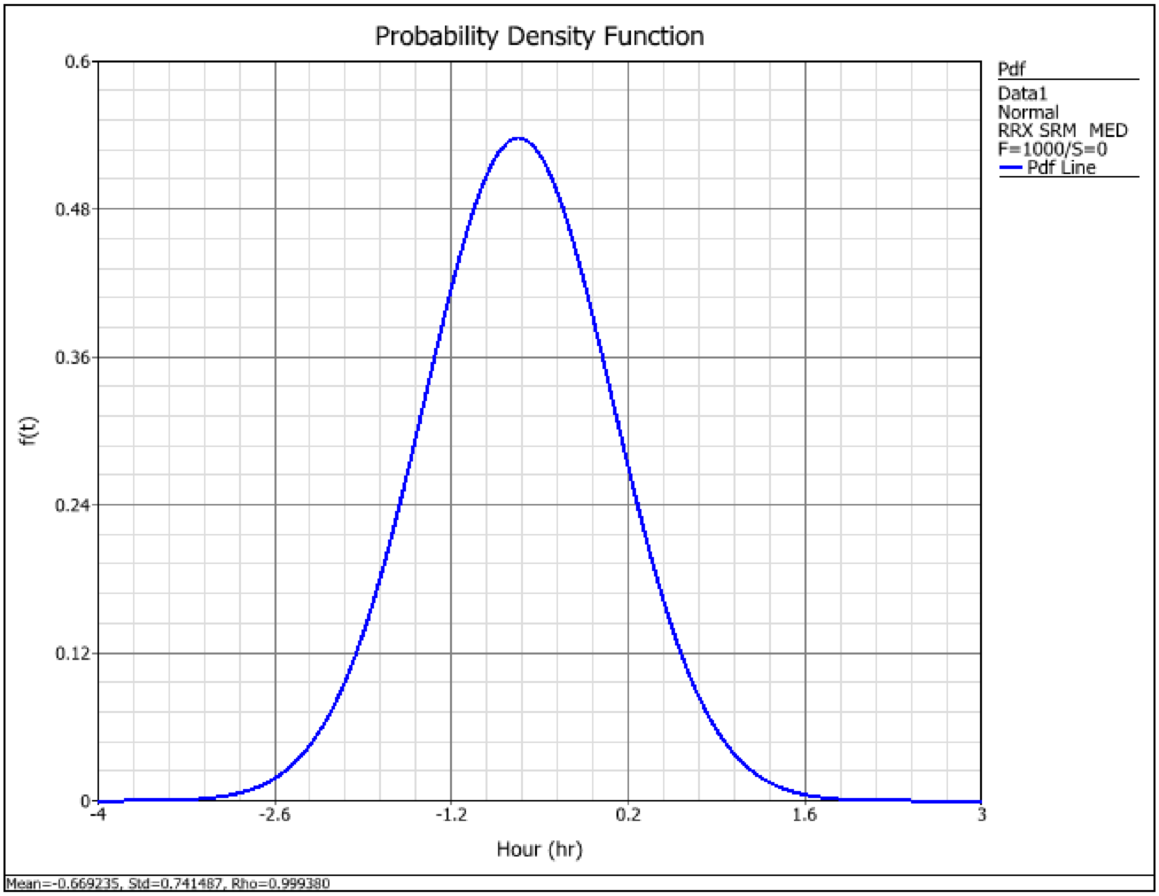
```
'Components!A' + 'Components!B' + 'Components!C' - 'Components!D'
```

In the **Data Points** area on the Settings tab, enter **1000** for the number of data points. Choose to generate the data set in a new folio and new sheet, then click **Generate** to create the simulated data.

Analyze the Data Set

In the new data sheet that contains the simulated data, calculate the parameters, assuming a normal distribution and RRX as the analysis method.

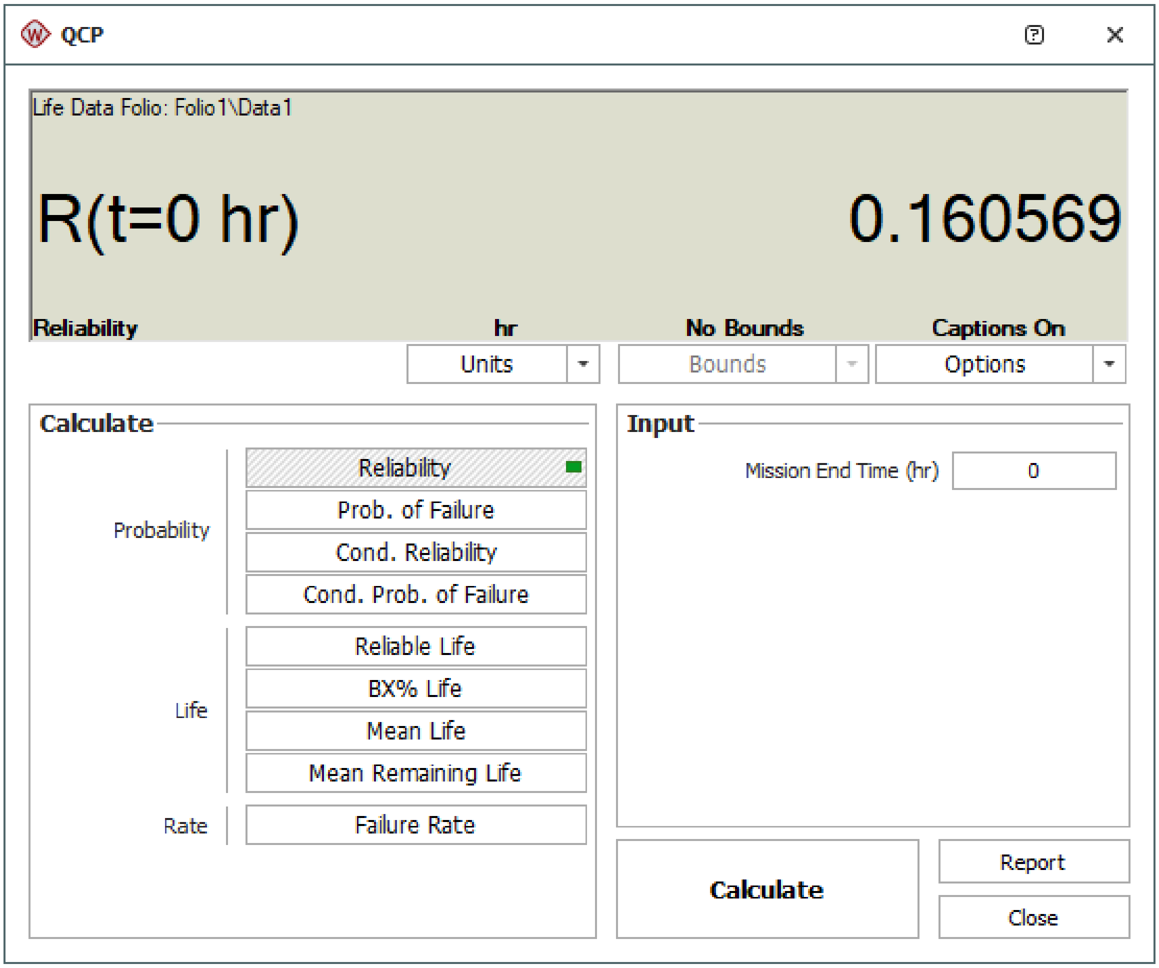
The following is a *pdf* plot of the simulated data for (A+B+C-D).



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 **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 16.05%. Note that your results may vary because the data were obtained through simulation.

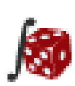


SimuMatic

Life Data and Life-Stress Data SimuMatic Setup

SimuMatic generates multiple data sets using Monte Carlo simulation. (See Monte Carlo Utility for information about how each data set is generated.) 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 create a SimuMatic folio for life data or life-stress data, choose **Home > Insert > SimuMatic** and choose either the **SimuMatic** or **Stress-Dependent SimuMatic** option.



MAIN, CENSORING AND SETTINGS TABS

The Main, Censoring and Settings tabs of the SimuMatic setup window are the same as in the Monte Carlo Utility. The settings in these tabs determine how each data set is generated.

On the Settings tab, you must enter the **Number of data sets** to generate. In most cases, you will want this number to be reasonably large (e.g., 1000).

ANALYSIS TAB

- **Analysis Method** and **Rank Method** (available in for life data only) specifies the methods to use for calculating the parameters of the simulated data sets. Select the **Use RS Regression Method** 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).
- **Confidence Bounds on Plots** displays the specified confidence bounds in plots that are based on the generated data points. For example, if you enter 90, the plot sheet of the SimuMatic folio will display the 90% two-sided confidence bounds on the plots.

TEST DESIGN TAB (OPTIONAL)

The Test Design tab of the setup window allows you to use SimuMatic to design or evaluate a reliability test. The software will 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 the specified target reliability would be reached. 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.

RELIABILITIES AND TIMES TAB (OPTIONAL)

The Reliabilities and Times tab of the setup window allows you to calculate the time for every given reliability, and/or calculate the reliability for every given time. You can enter up to 10 reliability values (as decimals less than 1) and up to 10 time values.

For example, if you enter 0.5 in the **Reliabilities** area, 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, the reliability at time 100 will be displayed.

Repairable Systems SimuMatic Setup

For reliability growth data analysis only.

Weibull++'s Repairable Systems SimuMatic tool generates multiple data sets using Monte Carlo simulation. It then analyzes each data set individually (e.g., to find the instantaneous MTBF at a specified time) and also analyzes the group of data sets as a whole (i.e., it shows the average and median parameter values for all data sets).

To create a SimuMatic folio for growth data, choose **Home > Insert > SimuMatic** and choose the **Repairable Systems SimuMatic** option.

The setup window will appear. Follow the steps outlined below to generate and analyze the simulated data sets.

1. On the Main tab of the window, select a data type and time units in the **Data Type** area. In the **Parameters** area, enter the beta and lambda values for the Crow-AMSAA model that will be used to generate the data set.

   In the **Data Sets / Points** area, enter the following information:

   - Specify the **Number of Data Sets** that will be generated.
   - Specify the **Number of systems** for which data will be generated. This value is fixed at 1 unless you selected a multiple systems data type on the Main tab (i.e., Multiple Systems - Concurrent or Repairable Systems).
   - Use the **Test termination** field to specify what will determine the end of the observation period for the simulated data.
     - If the period will end after a specific number of failures have been observed, choose **Failure Terminated** from the drop-down list and enter the number of **Failures** in the next field.
     - If the observation period will end at a specific time, choose **Time Terminated** from the drop-down list and enter a **Time** in the next field.

- If you selected to generate Grouped Failure Times data, click the **Set Intervals** link to open a window that allows you to specify the intervals that will contain the failures. You need at least three intervals to simulate this type of data.

2. 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** area, enter the number of decimal places that will be used for each simulated data point. (This does not affect the precision used in the calculated parameters and other results.)

3. On the Analysis tab, enter the bounds that will be shown in the SimuMatic plots. For example, if you enter **90**, then the plots will show 90% one-sided upper and 90% one-sided lower confidence bounds.

4. On the Test Design tab, select the **Calculate Target Time** check box if you wish to a) calculate the time at which the demonstrated MTBF reaches a certain value and b) mark the lower confidence bound on this value on the instantaneous MTBF vs. time plot. For example, if you wanted to know the time for an instantaneous MTBF of 1,000 hours, then you would enter **1,000** in the **Instantaneous MTBF** field. To mark the 90% lower confidence bound on this value in the IMTBF vs. time plot, you'd enter **90** in the **Lower 1-sided Confidence Level** field.

5. On the Results tab, specify the metrics that will be calculated.

   - The first four areas are for instantaneous MTBF calculations (i.e., the MTBF over a small interval *dt* that begins at a specified time) and cumulative MTBF calculations (i.e., the MTBF from time = 0 up to a specified end time). Click the arrow to open a drop-down table where you can enter the user-specified values.

     - The two areas on the left (e.g., **Instantaneous MTBF Given Time**) allow you to specify up to ten different times and SimuMatic will calculate the MTBF(s).

     - The two areas on the right (e.g., **Time Given Instantaneous MTBF**) allow you to specify up to ten different MTBFs and SimuMatic will calculate the time(s).

   - In the **Metrics to Calculate** area, select the metrics you wish to calculate for each data set. You can then use the results on the Sorted sheet to estimate, for example, the one-sided 90%

lower confidence bound on any of these values at the end of the test. (See SimuMatic Example.)

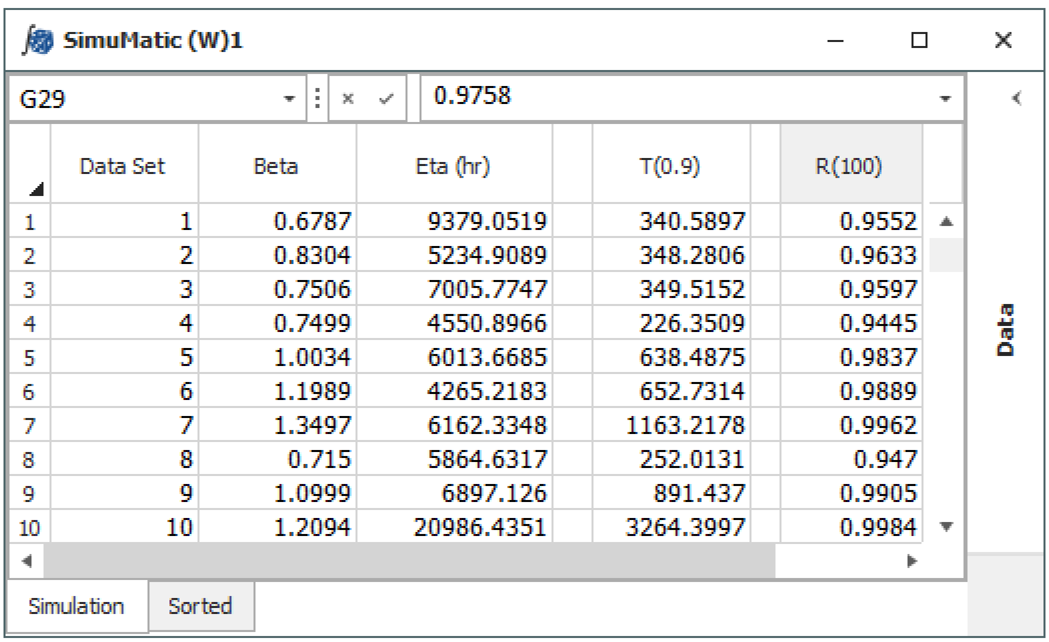
6. Click the **Generate** button to create and analyze the data. The results will be displayed in the Repairable Systems SimuMatic folio.

Life Data and Life-Stress Data 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.

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.



	Data Set	Beta	Eta (hr)	T(0.9)	R(100)
1	1	0.6787	9379.0519	340.5897	0.9552
2	2	0.8304	5234.9089	348.2806	0.9633
3	3	0.7506	7005.7747	349.5152	0.9597
4	4	0.7499	4550.8966	226.3509	0.9445
5	5	1.0034	6013.6685	638.4875	0.9837
6	6	1.1989	4265.2183	652.7314	0.9889
7	7	1.3497	6162.3348	1163.2178	0.9962
8	8	0.715	5864.6317	252.0131	0.947
9	9	1.0999	6897.126	891.437	0.9905
10	10	1.2094	20986.4351	3264.3997	0.9984

- **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 heading.
- **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

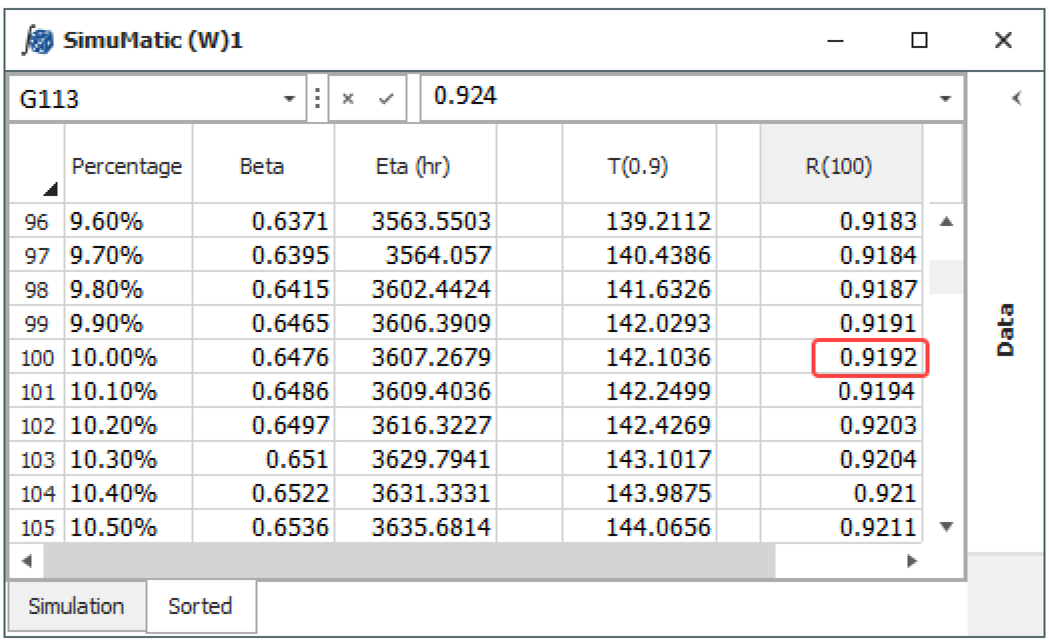
window. In this example, it shows the time at which the product's reliability is expected to be 0.9, or 90%.

- **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.

SORTED SHEET

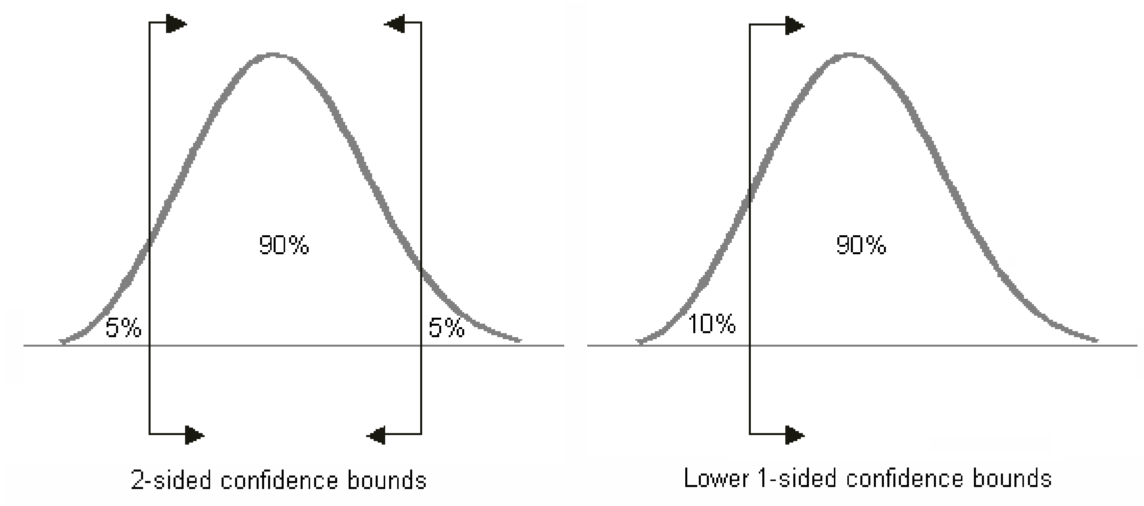
In the Sorted sheet, the calculated values from the Simulation sheet are sorted from least to greatest in order to show the confidence bounds. Assuming all the simulated data sets have calculated parameters, the Percentage column will display the percentage of data sets that are equal to or less than 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%	0.6371	3563.5503		139.2112	0.9183
97	9.70%	0.6395	3564.057		140.4386	0.9184
98	9.80%	0.6415	3602.4424		141.6326	0.9187
99	9.90%	0.6465	3606.3909		142.0293	0.9191
100	10.00%	0.6476	3607.2679		142.1036	0.9192
101	10.10%	0.6486	3609.4036		142.2499	0.9194
102	10.20%	0.6497	3616.3227		142.4269	0.9203
103	10.30%	0.651	3629.7941		143.1017	0.9204
104	10.40%	0.6522	3631.3331		143.9875	0.921
105	10.50%	0.6536	3635.6814		144.0656	0.9211

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.



SIMUMATIC CONTROL PANEL

The SimuMatic control panel and its components are described next.

- **Distribution** area displays the type of distribution that was used to generate each data set.
- **Analysis Settings** area displays what analysis settings were used to analyze the simulated data.
- **Analysis Summary** area displays the parameters that were used to generate each data set.
- **Additional Results** area:
  - **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.

  - **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 Life Data to Selected Folio is available only for life data. It transfers the simulated data set to an existing Weibull++ life data folio, with one data sheet for each parameter of the model that was used to generate the data.

Transfer Life Data to New Folio is available only for life data. It transfers the simulated data set to a new Weibull++ life data folio, with one data sheet for each parameter of the model that was used to generate the data.

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. See Plot Utilities for more general information on plot sheets.

- **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.
- **Bounds on Reliability** and **Bounds on Time** show confidence bounds on the plot. The percentile of the confidence bounds is determined by what you entered in the Analysis tab of the setup window. For example, if you entered 90% in the setup window and selected **Bounds on Time**, then the displayed confidence bounds would mark the simulated time values at the 10th (90% lower bound) and 90th (90% upper bound) 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.

Repairable Systems SimuMatic Folio

For reliability growth data analysis only.

The Repairable Systems SimuMatic folio gives you access to the generated data sets and to the results of analyses performed automatically for those data sets.

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 Analysis and Results tabs of the setup window. The following columns may be included in the Simulation sheet, depending on the metrics you chose to calculate in the Repairable Systems SimuMatic Setup window.

- **Beta** and **Lambda** of the Crow-AMSAA model are the calculated parameters for each data set.
- The following values are calculated according to the inputs you provided on the Analysis tab of the setup window.
  - **Target DMTBF** is the time required to demonstrate the specified MTBF.
  - **DMTBF** is the demonstrated MTBF at the end of the test.
  - **DFI** is the demonstrated failure intensity at the end of the test.
  - **Growth Rate** is equal to 1 - beta. A larger growth rate means faster MTBF growth.
- The following values are calculated according to the inputs you provided on the Results tab of the setup window.
  - **IMTBF(*t*)** and **CMTBF(*t*)** are the instantaneous/cumulative MTBF at a specified time, *t*.
  - **T_IMTBF(*m*)** and **T_CMTBF(*m*)** are the time given a specified instantaneous/cumulative MTBF, *m*.

As an example, the following picture shows the results from the first ten data sets for a particular simulation. In this case, the analysts chose to display only the Crow-AMSAA model parameters (Beta and Lambda), the growth rate, the instantaneous MTBF at 15,000 hours (IMTBF(15000) and the lower confidence bound on the time at which the demonstrated MTBF reached a specified target value (Target DMTBF).

C20 | 0.7528

	Data Set	Beta	Lambda (hr)	Growth Rate	IMTBF(15000 hr)	T(IMTBF=15000 hr)
1	1	1.1	0.3177	-0.1	1.0934	6.6325E-38
2	2	0.7625	0.1728	0.2375	74.5181	7.4897E+13
3	3	0.2577	2.7621	0.7423	1767.7001	267436.5486
4	4	1.0521	0.0887	-0.0521	6.4911	4.6128E-61
5	5	0.4058	1.1785	0.5942	633.5864	3082600
6	6	0.2847	0.5192	0.7153	6569.1817	47575.7161
7	7	0.6726	0.2125	0.3274	162.9433	14973000000
8	8	1.5009	0.0316	-0.5009	0.1709	2.0278E-06
9	9	0.5989	1.0074	0.4011	78.4589	7304900000
10	10	0.2994	2.0076	0.7006	1402.6553	441560.6163

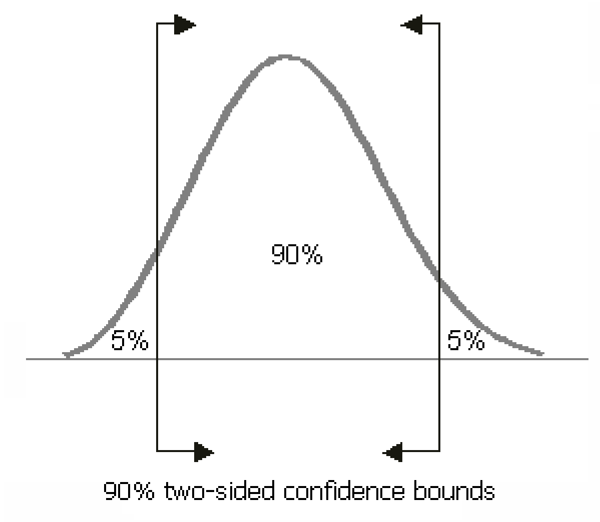
SORTED SHEET

In the Sorted sheet, the calculated values from the Simulation sheet are sorted from least to greatest in order to show the confidence bounds. The Percentage column displays 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 on the instantaneous MTBF at 15,000 hours, you would look up the value of **IMTBF(15000) (hr)** that corresponds to 10% (i.e., 100% - 90%), as shown next.

C7 | 0.0119

	Percentage	Beta	Lambda (hr)	Growth Rate	IMTBF(15000 hr)
7	0.07	0.2366	0.0119	-1.2248	0.0003
8	0.08	0.2577	0.0316	-0.9713	0.0014
9	0.09	0.2709	0.0321	-0.923	0.002
10	0.1	0.2782	0.033	-0.8325	0.0031
11	0.11	0.2847	0.0358	-0.5009	0.0872
12	0.12	0.285	0.0404	-0.4983	0.1709
13	0.13	0.2973	0.0453	-0.482	0.1987
14	0.14	0.2994	0.0588	-0.3931	0.2282
15	0.15	0.3117	0.0635	-0.3437	0.4051
16	0.16	0.316	0.0744	-0.3279	0.6028

Alternatively, if you wished to obtain the 90% two-sided confidence bounds of IMTBF(15,000), you would look up the values that correspond to 5% (for the lower bound) and 95% (for the upper bound), as illustrated in the next figure.



CONTROL PANEL

The SimuMatic control panel and its components are described next,

- The **Model** area always displays "Crow-AMSAA (NHPP)" because this is the only model Reliability Growth SimuMatic uses to generate simulated data.
- The **Analysis Settings** table shows what settings were used in the analysis of each data set. These settings will vary depending on the data type selected.

  As an example, the following picture shows the settings that are used for the Failure Times data type. The settings show the type of data (Developmental – Failure Times), parameter estimation method (MLE), confidence bounds method (Crow), whether a gap interval has been defined in the analysis (No Gap) and whether the failure times are entered as cumulative or non-cumulative (Cumulative).

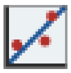
Developmental	
Failure Times	
MLE	Crow
No Gap	Cumulative

- The **Parameters** area displays the Crow-AMSAA model parameters that were used to generate the data.
- In the **Additional Results** area:
  - The **T(i) (...)** button opens a report that displays each data set in its own column. (Note that this report will contain multiple sheets when there are too many data points to fit onto one sheet.)

- The **Summary (...)** button opens a report that displays the settings used to generate the data. It also displays the average and median parameter values of the data sets.

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 the Plot sheet, which allows you to view the five different plots based on the simulated data.

PLOT SHEET

In the plot sheet, five different kinds of plots are available in the **Plot Type** drop-down list on the control panel.

- **Cumulative Number of Failures** gives an indication of how the number of failures is increasing over time. It plots the times on the x-axis and the cumulative number of failures on the y-axis. The points represent the actual failure times in the data set, and the solution line represents the expected cumulative number of failures based on the calculated model parameters. The vertical line represents the test termination time.
- The **Cumulative MTBF vs. Time** and **Instantaneous MTBF vs. Time** plots show how the time between consecutive failures increases, decreases or remains constant over time.
  - The cumulative MTBF is the MTBF from time = 0 up to a given end time. 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.
  - The instantaneous MTBF is the MTBF over a small interval *dt* that begins at a given time. For example, an instantaneous MTBF of 5 hours at 100 hours duration means that, over the next small interval *dt* that begins at 100 hours, the average MTBF will be 5 hours.
- The **Cumulative Failure Intensity** and **Instantaneous Failure Intensity** plots show how the failure intensity changes over time.
  - The cumulative failure intensity is the average failure intensity from the beginning of the test (i.e., t=0) up to a given time.
  - The instantaneous failure intensity is the failure intensity over a small interval *dt* that begins at a given time.

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. See Plot Utilities for more general information on plot sheets.

- **Simulation Lines** plots the line (e.g., cumulative MTBF vs. time) for every simulated data set.
- **True Parameter Line** plots the line defined by the parameters of the Crow-AMSAA model. (These parameters are visible on the control panel of the Simulation sheet.)
- **Median Line** and **Average Line** plots the lines defined by the median and average parameters of all generated data sets.
- **CB on Function** and **CB on Time** plot two-sided confidence bounds on the plot. The "Function" is determined by the current plot type (e.g., cumulative MTBF is the function for the cumulative MTBF vs. time 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.
- **Target** is available when you have selected the **Calculate Target Time** option on the Analysis tab of the setup window. This option marks the target time on the instantaneous MTBF/FI vs. time plots.

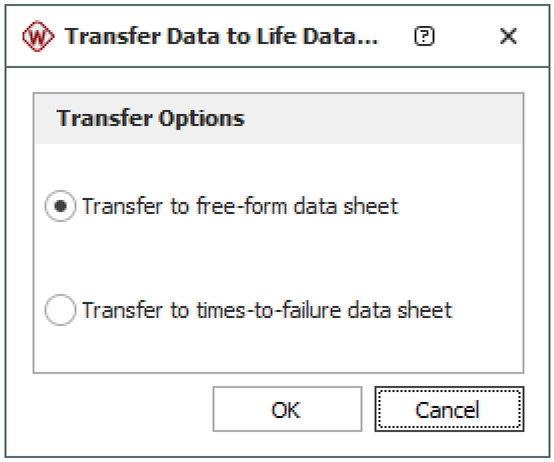
Transferring Data to a Life Data Folio

The Life Data SimuMatic folio allows you to transfer the calculated parameter values for each simulated data set to a new or existing life data 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.

TRANSFERRING 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.

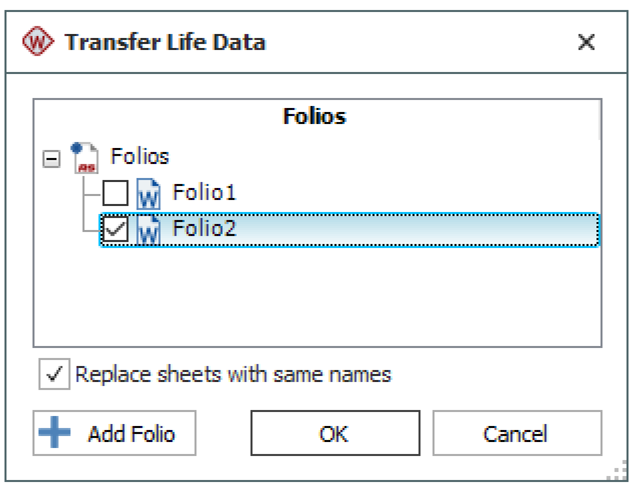
Select whether the data will be transferred to a free-form or times-to-failure data sheet, as shown next. After you click **OK**, a new life data folio will be created containing the simulated data in the selected format.



TRANSFERRING 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, 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 life data folio and display it in the window.

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.

RECOMMENDED SETTINGS

Below are some recommended settings for using SimuMatic to design a reliability life test. The options 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.

On the Settings tab:

If you are solving for the sample size, start with a large number of data points. You can later repeat the simulation with smaller values until you arrive at an acceptable test plan. (In the Stress-Dependent Monte Carlo utility, use the data sheet on the right side of the window to specify the number of data points for each stress level.)

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.

On the Censoring tab:

- 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: In the Life Data SimuMatic utility, 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 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.

EVALUATING THE RESULTS

After generating the simulated data set, click the **Summary (...)** icon inside the **Additional Results** area of the control panel to view a report of the results. 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 or suspension 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, note that:

- 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.

See Life-Stress SimuMatic Example for an example of designing a reliability test with SimuMatic.

Life Data SimuMatic Example

This example uses the Life Data 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.

GENERATE DATA SETS

1. Choose **Home > Insert > SimuMatic** and choose the **SimuMatic** option.
2. On the Main tab of the setup window, select the **2P-Weibull** distribution and select **Hour (hr)** for the time unit. Enter **2** in the **Beta** field and **100** in the **Eta** field.
3. On the Censoring tab, select **No censoring**.
4. On the Settings tab, enter **1000** in the **Number of data sets** field and enter **10** in the **Number of points** field.
5. 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.

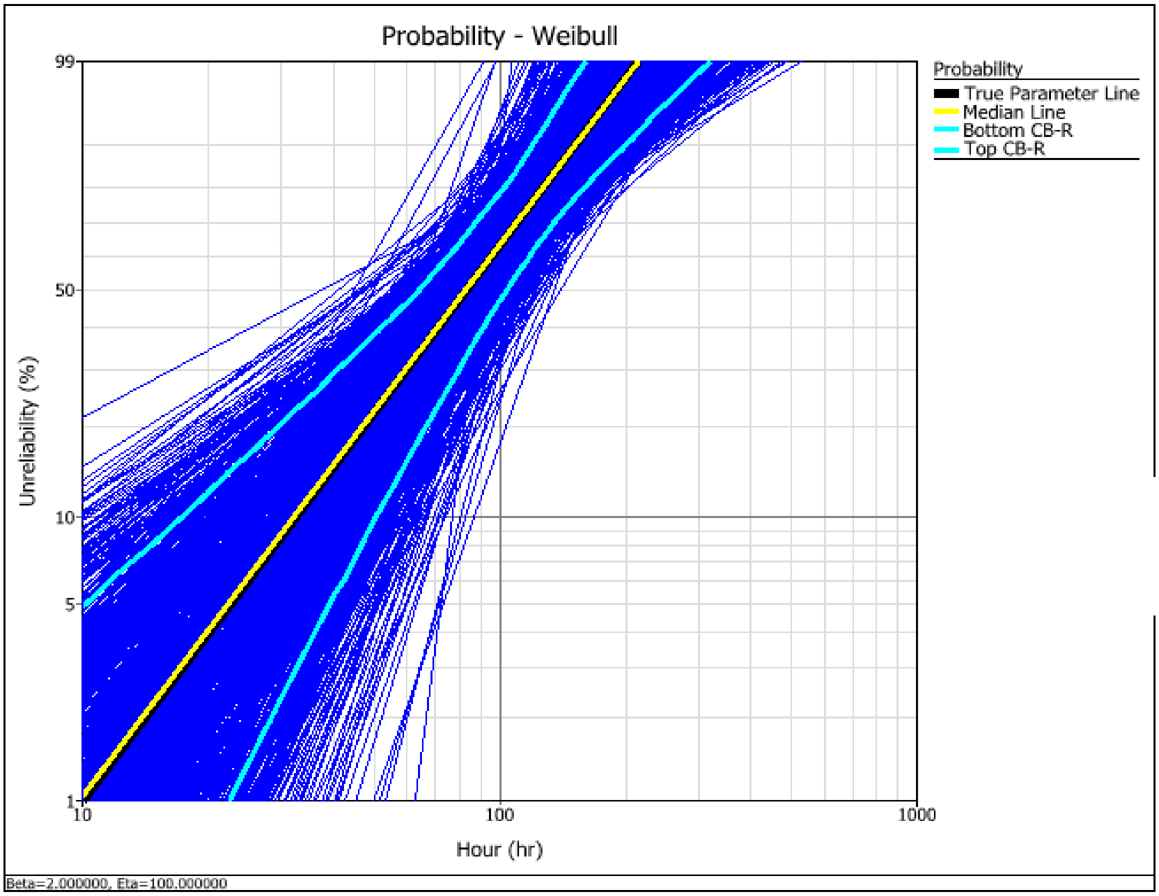
- Choose **Median Ranks** from the **Rank Method** drop-down list and clear the **Use RS Regression Method** check box. Enter **90** in the **Confidence Level** field.

6. Click **Generate** to create and analyze the data. A SimuMatic folio will appear.
7. Click **Plot** on the control panel. You will be taken to the plot sheet 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
   - Bounds on Reliability
8. Click **Plot** again to refresh the plot display all the selected lines.

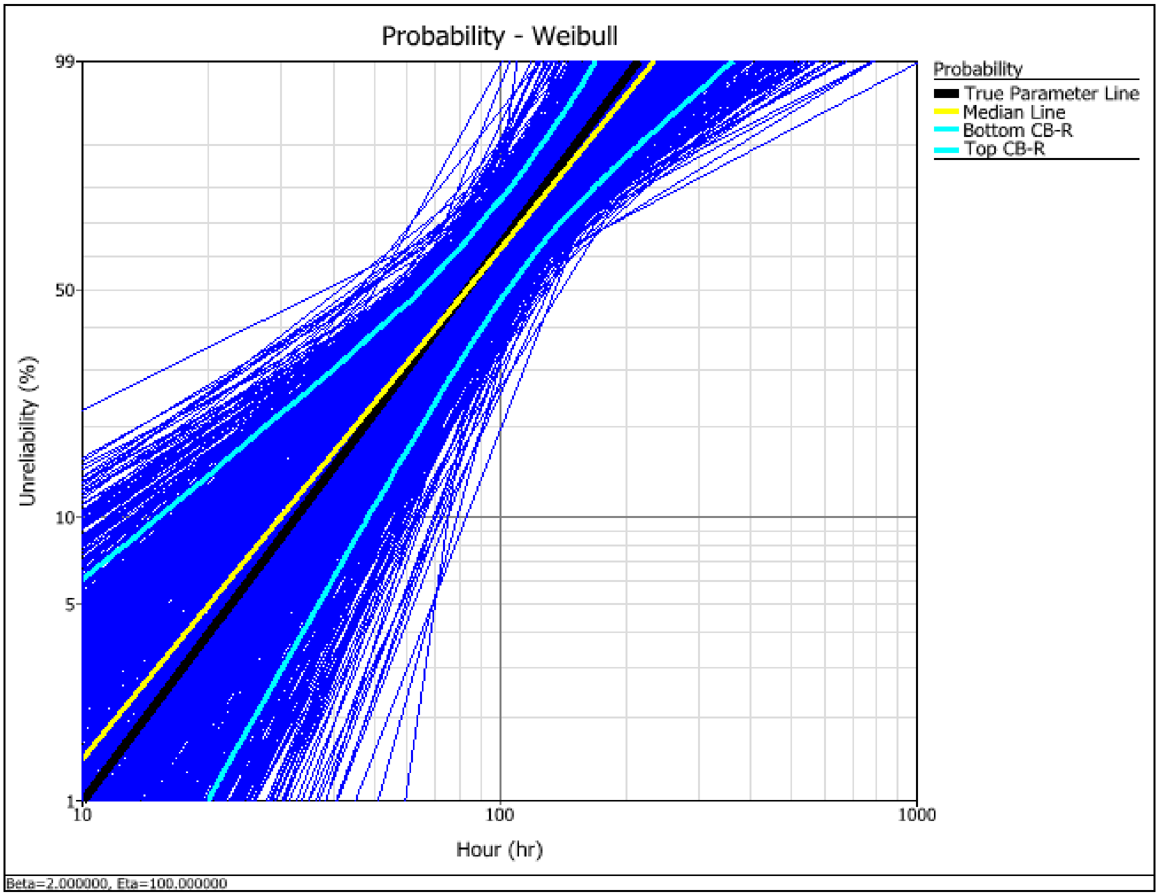
To compare RRX to the rank regression on Y (RRY) and maximum likelihood estimation (MLE) methods, repeat the steps above, but choose a different option in the **Analysis Method** drop-down list on the Analysis tab. Then compare the three plots.

COMPARE THE PLOTS

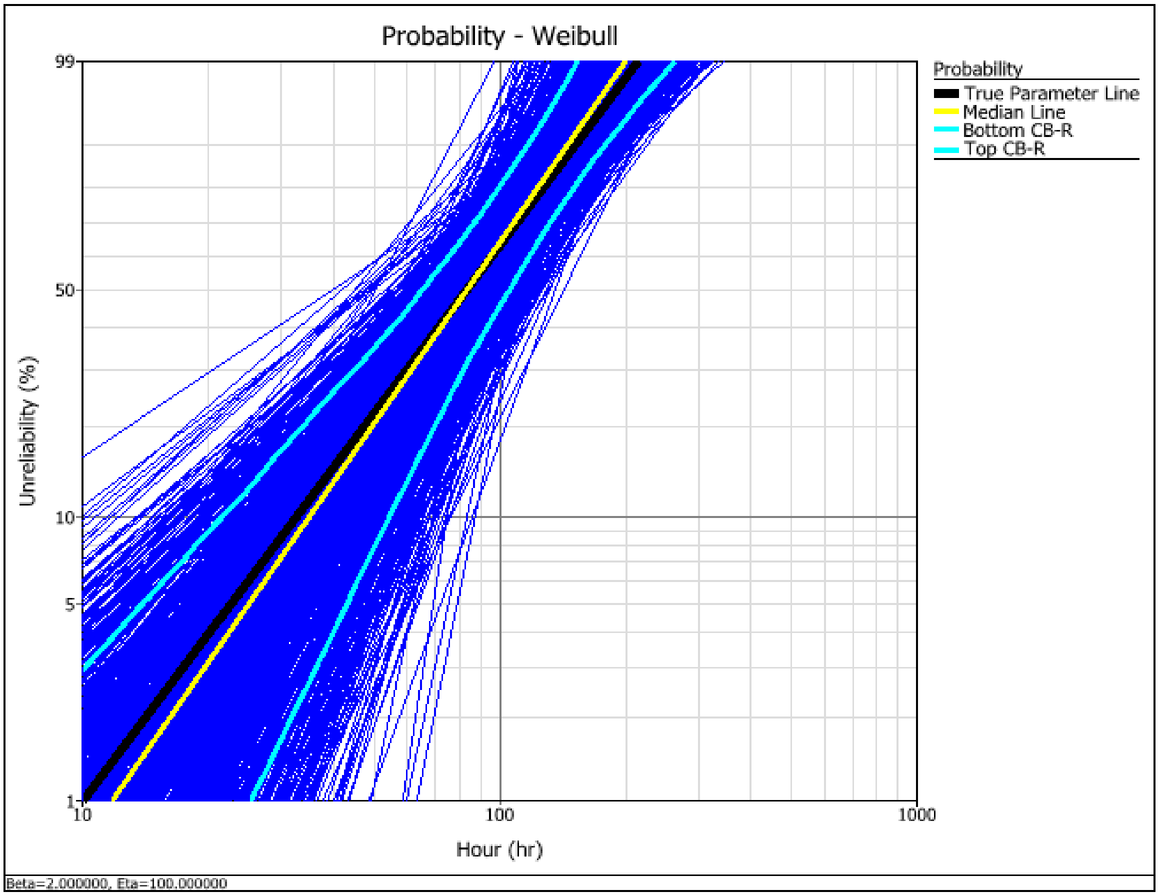
The plots for the three methods are shown below:


Probability - Weibull
Unreliability (%)
Hour (hr)
99
50
10
5
1
10
100
1000
Probability
True Parameter Line
Median Line
Bottom CB-R
Top CB-R
Beta=2.000000, Eta=100.000000

RRX Method


Probability - Weibull
Unreliability (%)
Hour (hr)
99
50
10
5
1
10
100
1000
Probability
True Parameter Line
Median Line
Bottom CB-R
Top CB-R
Beta=2.000000, Eta=100.000000

RRY Method



MLE Method

The true parameter line (black) shows the time vs. unreliability defined by the parameters of the distribution that was used to generated the simulated data. The median line (yellow) 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 (pink) 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).

Life-Stress Data SimuMatic Example

For life-stress data only.

This example uses the Stress-Dependent SimuMatic utility 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. The objective is to determine whether this test plan will meet the reliability goal:

GENERATE DATA SETS

1. Choose **Home > Insert > SimuMatic** and choose the **Stress-Dependent SimuMatic** option.
2. On the Main tab of the setup window, select the **Arrhenius-Weibull** model and select **Hour (hr)** for the time unit. Enter **1.5** for the beta parameter.
3. 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**.
    - 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 go the second page. Then enter **340** for the accelerated stress level and **500** for the characteristic life (i.e., eta).
    - Click **Next>** to see the calculated values for the B and C parameters, which are shown to be 7871.5321 and 4.4093E-08, respectively. Click the **Update** button to transfer the values to the SimuMatic setup window.
4. On the Stress tab of the SimuMatic window, enter the use stress level of **300** K.

5. On the Censoring tab, select **No censoring**.
6. On the Settings tab, set the **Number of data sets** to **1000**. Use the default values in the other fields.
7. 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

8. On the Test Design tab, enter **90** in the **Target Reliability** field and **95** in the **Lower 1-Sided Confidence Level** field.
9. Click **Generate** to create and analyze the simulated data sets. A SimuMatic folio will appear.

VIEW THE RESULTS

In the SimuMatic folio, open the Results window by clicking **Summary** in the **Additional Results** area of the control panel.

In the results, T1 confirms that the product's intrinsic reliability is high enough to meet the target reliability. Also, because T2 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.

Test Planning Results	
	Required test time
Expected Test Duration (T3)	3374.22311
	Demonstrated Reliability
Demonstrated Life at Given Confidence (T2)	907.0385402
Target Reliability (R)	90.00%
1-Sided Lower Confidence Level (CL)	95.00%
	Variation from True Value
Expected Life (T1)	2443.626628
Ratio of Expected Life/Demonstrated Life (DELTA)	2.6940714

Repariable Systems SimuMatic Example

For reliability growth data analysis only.

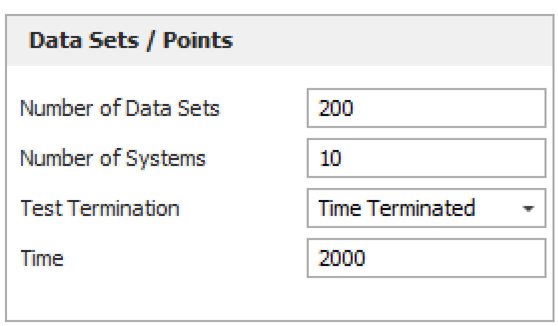
*The data set used in this example is available in the example database installed with the software (called "Weibull21_ReliabilityGrowth_RGA_Examples.rsgz21"). To access this database file, choose **File > Help**, click **Open Examples Folder**, then browse for the file in the Weibull sub-folder.*

The name of the project is "RGA SimuMatic."

A reliability engineer is developing a reliability growth test plan for a new system. The target MTBF that she needs to demonstrate at the end of the test is 800 hours with 90% confidence. There will be 2,000 hours of test time available. Before she can perform the test, she needs to determine the number of prototypes that must be tested in order to demonstrate the target. Based on historical data, the expected beta and lambda parameters for the Crow-AMSAA model are 0.55 and 0.21, respectively.

The SimuMatic folio called "Test Design - 10 Systems" contains all the simulated data and analysis results based on the assumption that 10 systems will be tested. To recreate this folio on your own, follow the steps described next.

1. Choose **Home > Insert > SimuMatic** and choose the **Repairable Systems SimuMatic** option to open the SimuMatic Setup window.
2. On the Main tab of the setup window, select the **Multiple Systems - Concurrent** data type and choose **Hour (hr)** for the time units. Then, in the **Parameters** area, enter **0.55** for beta and **0.21** for lambda.
3. In the **Data Sets / Points** area on the Main tab, specify that you will create 200 data sets in order to simulate a time terminated test that ends at 2,000 hours. As a conservative guess, enter **10** for the **Number of systems** that will be tested. The area will appear as shown next.



4. On the Settings tab, select the **Use Seed** check box and enter a seed value of 1. Then enter **4** in the **Math Precision** area.

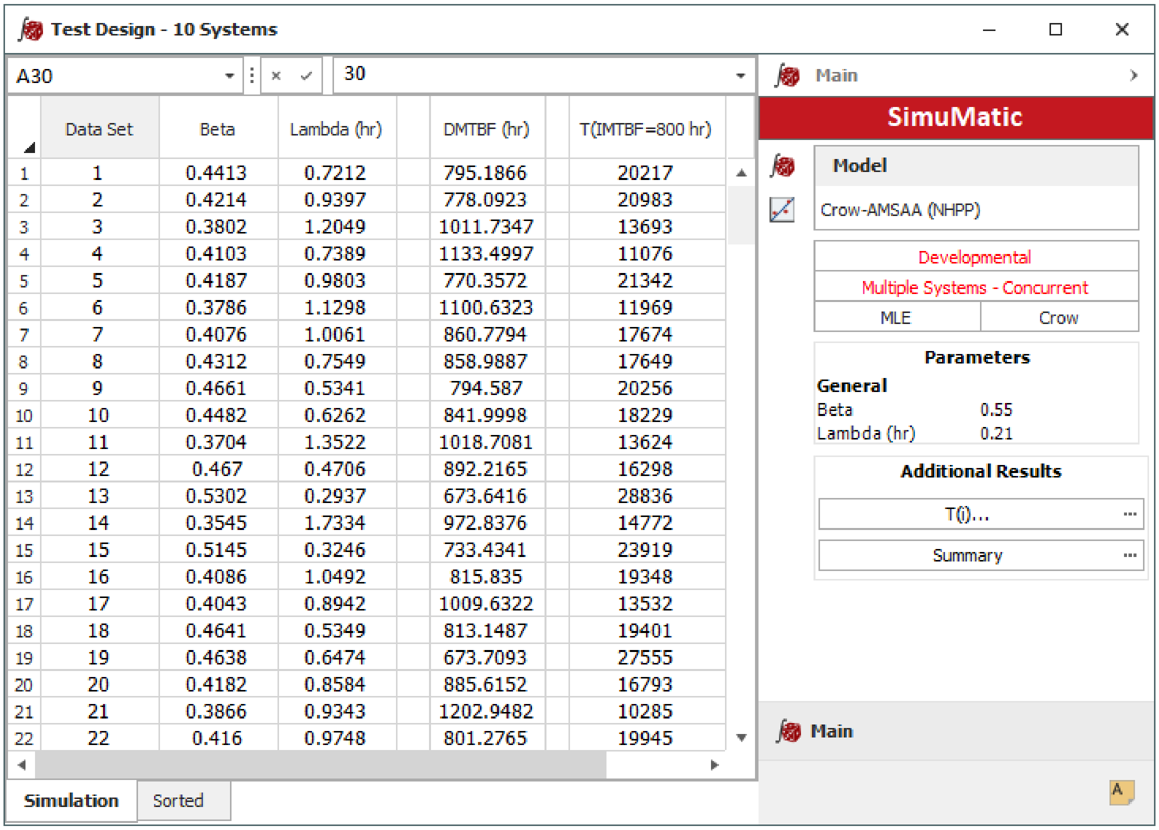
5. On the Analysis tab, enter **90** in the **Confidence Bounds on Plot** area to specify the bounds that will be available to show in all the plots. In this case, lines will be available that show the 90% one-sided lower bound and 90% one-sided upper bound.

6. On the Test Design tab, select the **Calculate Target Time** option, then specify that you will estimate the test time required to demonstrate an instantaneous MTBF of **800** hours. To place a target point on the instantaneous MTBF vs. time plot that marks the 90% lower one-sided confidence bound on the required test time, enter **90** in the **Lower 1-sided Confidence Level** field.

7. On the Results tab, select the **Demonstrated MTBF** option and clear all other options and inputs. You can use the **Clear All** icon  to delete any values entered in the first four areas.

8. Click the **Generate** button to create and analyze the data.

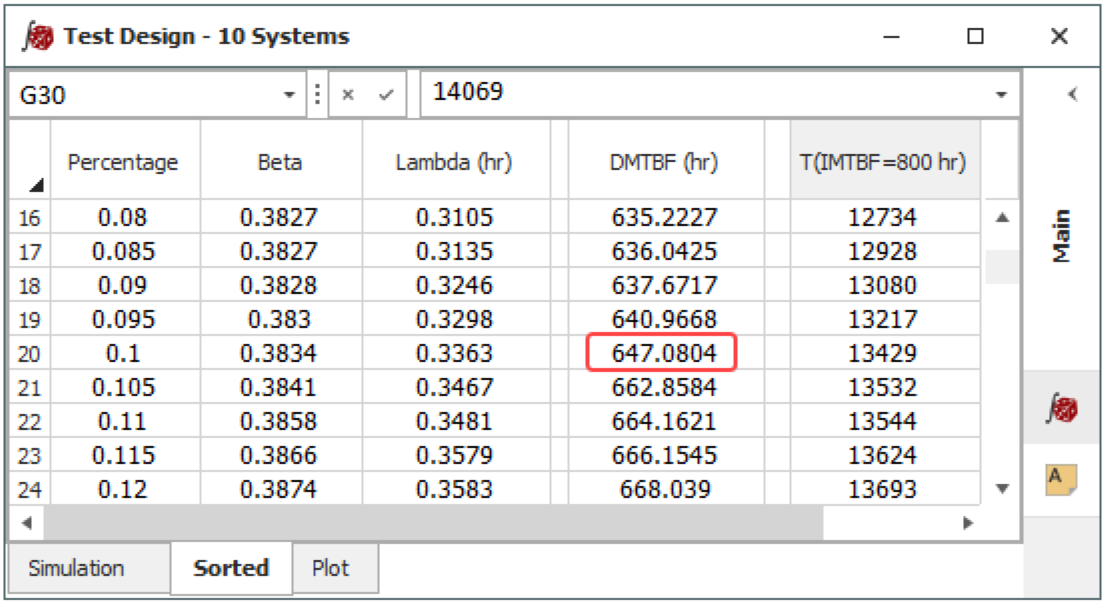
The Simulation sheet of the new SimuMatic folio displays the results for each data set in separate rows on the Simulation sheet, as shown next (where the parameters and calculated results for the first 22 data sets are visible).



	Data Set	Beta	Lambda (hr)	DMTBF (hr)	T(IMTBF=800 hr)
1	1	0.4413	0.7212	795.1866	20217
2	2	0.4214	0.9397	778.0923	20983
3	3	0.3802	1.2049	1011.7347	13693
4	4	0.4103	0.7389	1133.4997	11076
5	5	0.4187	0.9803	770.3572	21342
6	6	0.3786	1.1298	1100.6323	11969
7	7	0.4076	1.0061	860.7794	17674
8	8	0.4312	0.7549	858.9887	17649
9	9	0.4661	0.5341	794.587	20256
10	10	0.4482	0.6262	841.9998	18229
11	11	0.3704	1.3522	1018.7081	13624
12	12	0.467	0.4706	892.2165	16298
13	13	0.5302	0.2937	673.6416	28836
14	14	0.3545	1.7334	972.8376	14772
15	15	0.5145	0.3246	733.4341	23919
16	16	0.4086	1.0492	815.835	19348
17	17	0.4043	0.8942	1009.6322	13532
18	18	0.4641	0.5349	813.1487	19401
19	19	0.4638	0.6474	673.7093	27555
20	20	0.4182	0.8584	885.6152	16793
21	21	0.3866	0.9343	1202.9482	10285
22	22	0.416	0.9748	801.2765	19945

The Beta and Lambda columns display the parameters that were calculated for each data set. The DMTBF (hr) column shows the demonstrated MTBF at the end of the test, and the last column shows the test time required to demonstrate the target MTBF (an instantaneous MTBF of 800 hours, as specified on the Analysis tab of the setup window).

The Sorted sheet shows the results for each column sorted from least to greatest. On this sheet, you can see the demonstrated MTBF at the 10th percentile of the results (i.e., 90% one-sided lower bound), shown next.

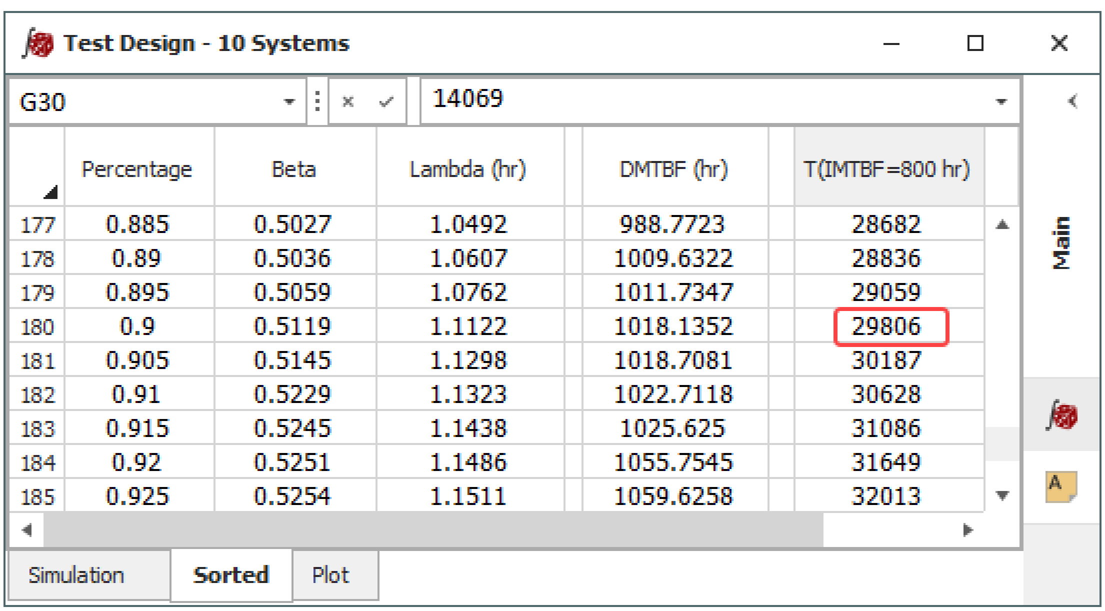


	Percentage	Beta	Lambda (hr)	DMTBF (hr)	T(IMTBF=800 hr)
16	0.08	0.3827	0.3105	635.2227	12734
17	0.085	0.3827	0.3135	636.0425	12928
18	0.09	0.3828	0.3246	637.6717	13080
19	0.095	0.383	0.3298	640.9668	13217
20	0.1	0.3834	0.3363	647.0804	13429
21	0.105	0.3841	0.3467	662.8584	13532
22	0.11	0.3858	0.3481	664.1621	13544
23	0.115	0.3866	0.3579	666.1545	13624
24	0.12	0.3874	0.3583	668.039	13693

This shows, with 90% confidence, that the demonstrated MTBF at the end of a 2,000-hour test will be at least 647.0804 hours.

The T(IMTBF=800 hr) column displays the total accumulated test time required to demonstrate the MTBF goal of 800 hours. You will use the required test time to calculate the number of systems that need to be tested.

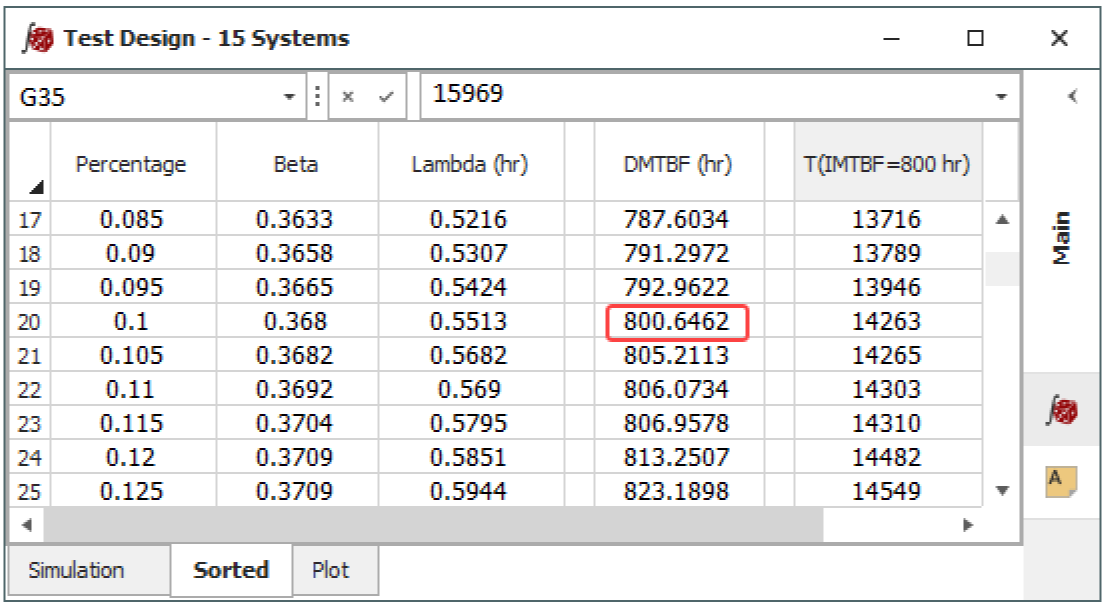
On the Sorted sheet, you can see that the 90% upper one-sided confidence bound on the required test time is 29,806 hours, as shown next.



	Percentage	Beta	Lambda (hr)	DMTBF (hr)	T(IMTBF=800 hr)
177	0.885	0.5027	1.0492	988.7723	28682
178	0.89	0.5036	1.0607	1009.6322	28836
179	0.895	0.5059	1.0762	1011.7347	29059
180	0.9	0.5119	1.1122	1018.1352	29806
181	0.905	0.5145	1.1298	1018.7081	30187
182	0.91	0.5229	1.1323	1022.7118	30628
183	0.915	0.5245	1.1438	1025.625	31086
184	0.92	0.5251	1.1486	1055.7545	31649
185	0.925	0.5254	1.1511	1059.6258	32013

Given that each system in the test accumulates 2,000 hours of testing, the current plan of 10 systems will lead to a total accumulated time of 20,000 hours. Therefore, approximately 10,000 hours of additional test time are required in order to accumulate 29,535 test hours and reach the target MTBF. In other words, an additional 5 systems (each tested for 2,000 hours) will be needed to accumulate enough test time.

To confirm that 5 additional systems will be enough to demonstrate the goal MTBF, return to the RGA SimuMatic Setup window and change the **Number of systems** on the Main tab to **15**. Then perform the simulation again and check the 90% one-sided lower bound on the demonstrated MTBF. The "Test Design – 15 Systems" folio contains all the simulated data and analysis results based on the assumption that 15 systems will be tested. The lower bound on the demonstrated MTBF with 15 systems is shown next.



	Percentage	Beta	Lambda (hr)	DMTBF (hr)	T(IMTBF=800 hr)
17	0.085	0.3633	0.5216	787.6034	13716
18	0.09	0.3658	0.5307	791.2972	13789
19	0.095	0.3665	0.5424	792.9622	13946
20	0.1	0.368	0.5513	800.6462	14263
21	0.105	0.3682	0.5682	805.2113	14265
22	0.11	0.3692	0.569	806.0734	14303
23	0.115	0.3704	0.5795	806.9578	14310
24	0.12	0.3709	0.5851	813.2507	14482
25	0.125	0.3709	0.5944	823.1898	14549

According to this result, you know with 90% confidence that testing 15 systems for 2,000 hours will demonstrate an MTBF of at least 800.6462 hours at the end of the test. You conclude that 15 systems will be sufficient to demonstrate the goal MTBF.

Simulation Worksheets

Simulation worksheets allow you to vary values that are used when simulating an RBD or an event analysis flowchart in BlockSim. This allows you to investigate the effect of one or more settings on the simulation results.

Simulation worksheets consist of columns that can be defined as inputs or outputs. The input values are used during simulation, and they can be entered manually or transferred from a DOE experiment design. Values in output columns are determined through simulation. For example, you might design an experiment in a Weibull++ design folio and transfer all the factor level combinations to the simulation worksheet. Then you could simulate the experiment in BlockSim and transfer the output values back to the Weibull++ design folio to analyze the results. If, on the other hand, you do not want to test all combinations of all factor settings, you can enter the desired combinations of settings by hand.

In general, using a simulation worksheet will include the following steps:

- Create the Variables and the Diagram in BlockSim
- Define the Factor Settings
- Define the Simulation Worksheet Columns in BlockSim
- Simulate the Worksheet in BlockSim
- Analyze the Results in Weibull++
- Verify the Results in BlockSim

Depending on what you are trying to achieve, some of the steps may not be necessary.

In this topic, we will consider an example in which we explore options for maintaining a spare part pool, with a goal of maximizing system availability while minimizing the cost of the pool. The factors that we will vary are the number of spare parts initially in the pool, the frequency with which the pool is restocked and the number of parts added to the pool at each restock.

Create the Variables and the Diagram in BlockSim

Build the diagram for the system in BlockSim. Use variables to represent the factors whose settings you want to investigate. Depending on the factors you are studying, you might use the variables directly (as field inputs for blocks and resources, or in equations and functions in event analysis flowcharts) or via a dynamic model. In RBDs, the following fields can be defined using a variable:

- In standard blocks:
  - **Current age**
  - **Duty cycle**
  - **Throughput**
- In contained standby blocks and contained load sharing blocks:
  - **Duty cycle**

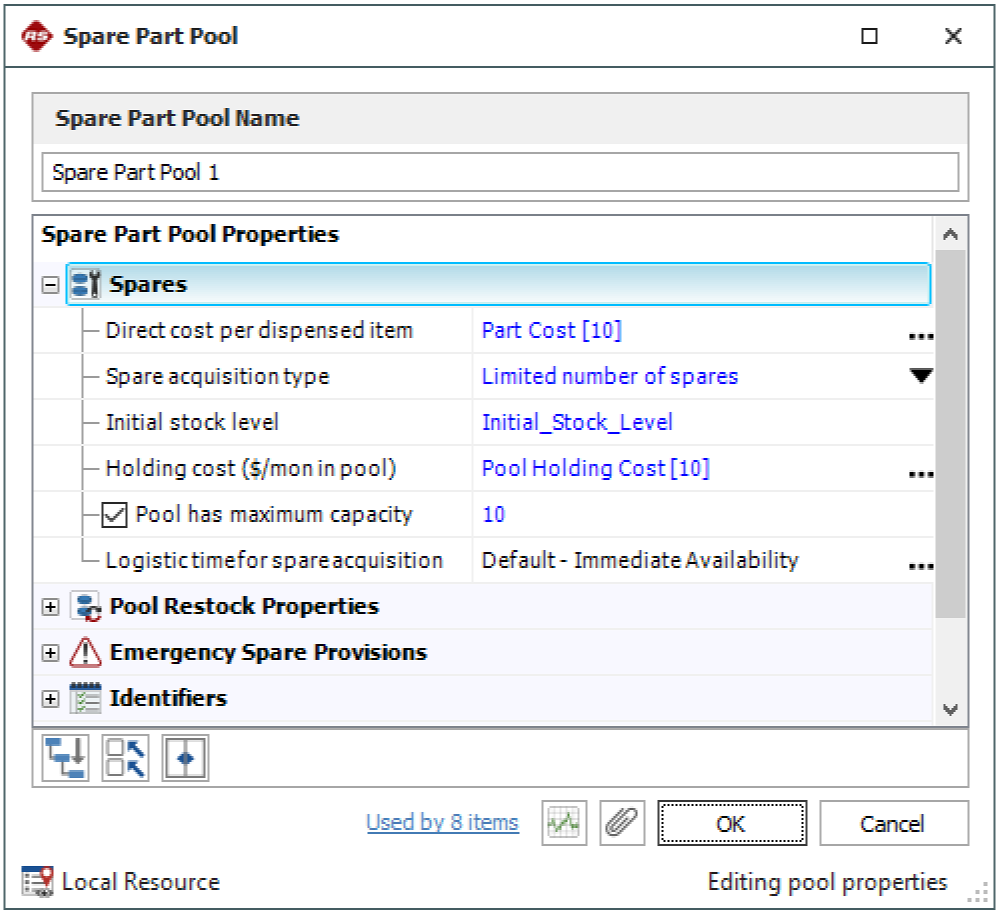
- In subdiagram blocks:
  - **Duty cycle**
- In crews:
  - **Number of tasks** (visible when there is a limit to the number of tasks the crew can perform at the same time)
- In spare part pools:
  - **Initial stock level**
  - Capacity if **Pool has maximum capacity** is selected
  - For scheduled restocks:
    - **Restock every**
    - **Number added per restock**
  - For restocking as needed:
    - **Restock when stock drops to**
    - **Number added per restock**
  - For emergency spare provisions:
    - **Number added per emergency**
- In scheduled tasks performed at fixed or dynamic intervals:
  - Interval value

Any field that takes a model can use a dynamic model; it is up to you to decide if exploring various values for that field will yield useful results.

IMPORTANT: In event analysis flowcharts, you will need to store the results of interest in variables. The **When to Reset** field for output variables must be set to **Never Reset** in order for the results to be available to the simulation worksheet. This ensures that the results are available to the simulation worksheet, rather than the variables being reset to their initial values.

Example

The spare part pool that is used by the tasks in our example is configured as shown next. The cost per item is specified as a constant model equal to $10 and the holding cost is specified as a constant model equal to $10 per month. Since the factors of interest (i.e., the initial stock level, the frequency of restocking and the number added per restock) will not be held constant, they are assigned to variables. These variables represent factor values that will be specified in the simulation worksheet.



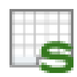
Define the Factor Settings

For the factors of interest, you will decide on the settings to be used in simulation and the combinations of factor settings that should be tried (i.e., that will be used as inputs to the simulation).

- One way to do this is to build an experiment design in a Weibull++ design folio and then transfer the factor settings to a new simulation worksheet. After creating the DOE in Weibull++, choose **Data > Simulation Worksheets > Transfer to Simulation Worksheet**.

This will ensure that the factor settings are tested in combinations appropriate to determine the effects of each factor and of factorial interactions.

- In some situations, not all factor setting combinations may be possible. For instance, let's say that you are considering two versions of a part, one that costs $100 and has a reliability of 0.97 at 1,000 hours and another that costs $85 and has a reliability of 0.92 at 1,000 hours. You can vary the cost of replacement and the block reliability to account for these scenarios. You would not, however, want to run a simulation using a cost of $85 and a reliability of 0.97, since that combination does not reflect an available part. In cases like this, you can add a new simulation worksheet to the project by choosing **Home > Insert > Simulation Worksheet**.



After adding the new worksheet, you can enter the desired factor values manually, as shown next. (Note that the column names were changed by right-clicking the column heading and choosing **Rename Column** on the shortcut menu.)

	Replacement Cost	Reliability at 1,000 Hours
1	100	0.97
2	96	0.92

While you have the simulation worksheet open, it will be read-only in other applications. For instance, if you create the simulation worksheet in Weibull++, it will be read-only in BlockSim until you do one of the following:

- Close the simulation worksheet, or
- Click **Commit and Lock** on the control panel. This will commit the data in the simulation worksheet to the database and lock the simulation worksheet within the current application so that it will be available within other applications.

In either case, if you already have the simulation worksheet open in a read-only state, you will need to click **Unlock and Edit** on the control panel once you have released the lock in the original application.

Example

The experiment used in our example is a general full factorial design using the following settings:

General Full Factorial

Response Properties	
Name	**Units**
System Availability	
Pool Cost	

Factor Properties									
Abbreviation	**Name**	**Units**	**Type**	**Number of Levels**	**Level 1 (Low)**	**Level 2**	**Level 3**	**Level 4**	**Level 5 (High)**
A	Initial Stock Level		Quantitative	5	1	2	3	4	5
B	Restock Time		Quantitative	3	500	1000	2000	-	-
C	Number Added per Restock		Quantitative	5	1	2	3	4	5

Additional Settings	
# of Unique Runs	75
# of Blocks	1
Center Points per Block	0
# of Replicates	1
Block on Replicates	No
Total # of Runs	75

The first fifteen rows of the simulation worksheet that is created in the transfer process are shown next.

	Initial Stock Level	Restock Time	Number Added per Restock	System Availability	Pool Cost
1	1	500	5		
2	5	1000	1		
3	4	2000	5		
4	2	1000	4		
5	1	2000	1		
6	3	1000	2		
7	4	1000	4		
8	5	1000	5		
9	2	2000	1		
10	2	500	5		
11	1	2000	5		
12	4	500	1		
13	2	2000	5		
14	4	500	5		
15	3	1000			

Note: Complete data set is not shown.

Define the Simulation Worksheet Columns in BlockSim

Once the factor settings have been entered in the simulation worksheet, you will simulate the worksheet in BlockSim. The software will perform a set of simulations for each combination of factor settings in order to obtain response values. In BlockSim, open the simulation worksheet and click the **Simulate** icon on the control panel.

The Select Diagram window will appear, allowing you to select the diagram that will be used to simulate the response values (i.e., the data source) for the simulation worksheet.

Note: You can also click the **Associated Data Sources** icon to specify or change the source diagram.

The Worksheet Simulation window then appears, allowing you to specify the information contained in each column of the simulation worksheet.

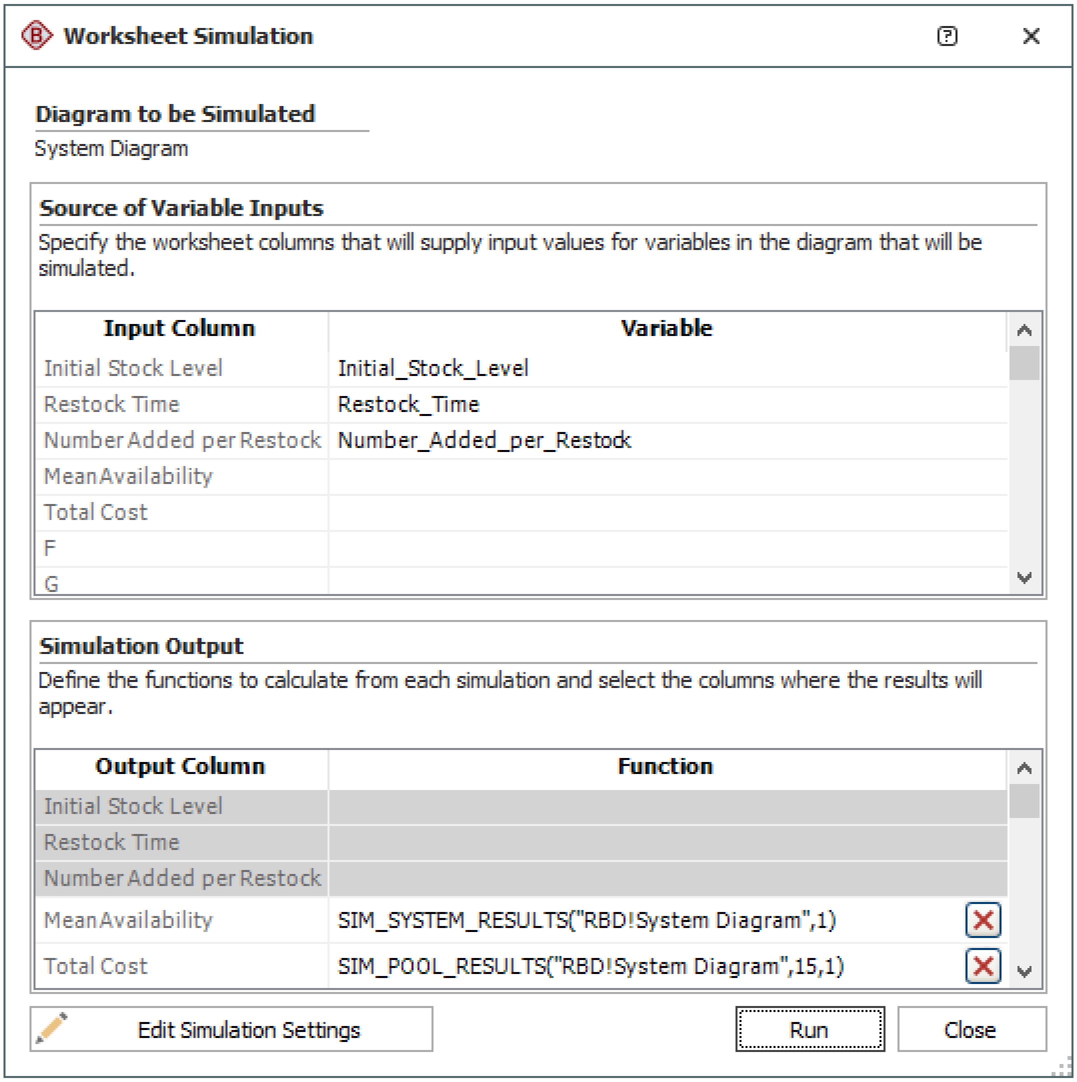
- In the **Source of Variable Inputs** area, assign the variables representing the factors of interest to the appropriate input columns in the worksheet. Available variables will be shown in the drop-down list for each column. If a factor of interest is represented by a dynamic model, choose the variable upon which the dynamic model is based.

  Starting in Version 11, you can choose to assign the simulation end time to one of the input columns. This allows you to obtain results as a function of time.

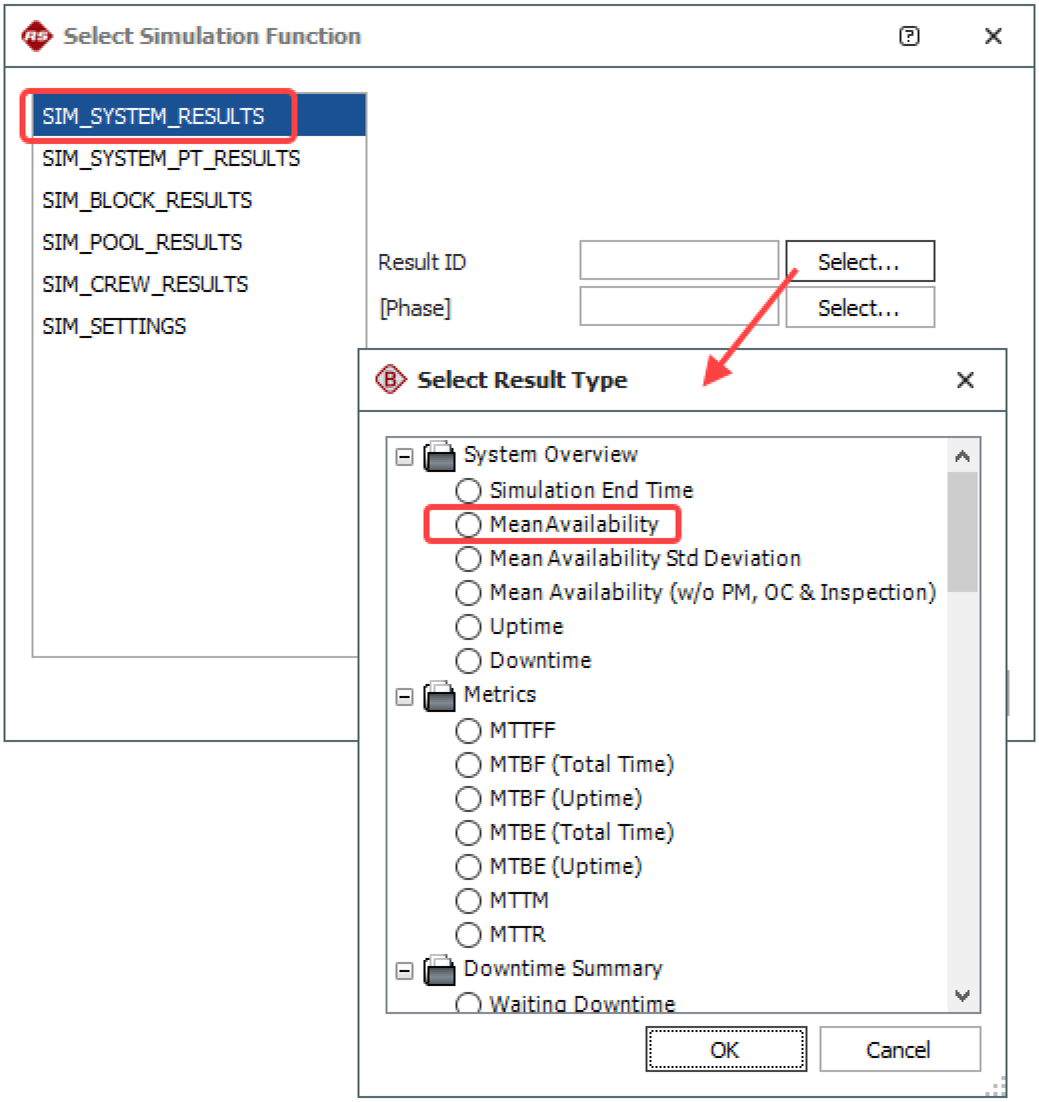
- In the **Simulation Output** area, for each result column, specify the result to be entered:
  - If the associated diagram is a simulation RBD, define the function to calculate from each simulation. This is done by clicking the cell associated with the result column to open a limited version of the Function Wizard, where only the simulation functions are available.
  - If the associated diagram is an event analysis flowchart, select the variable to read for the result.

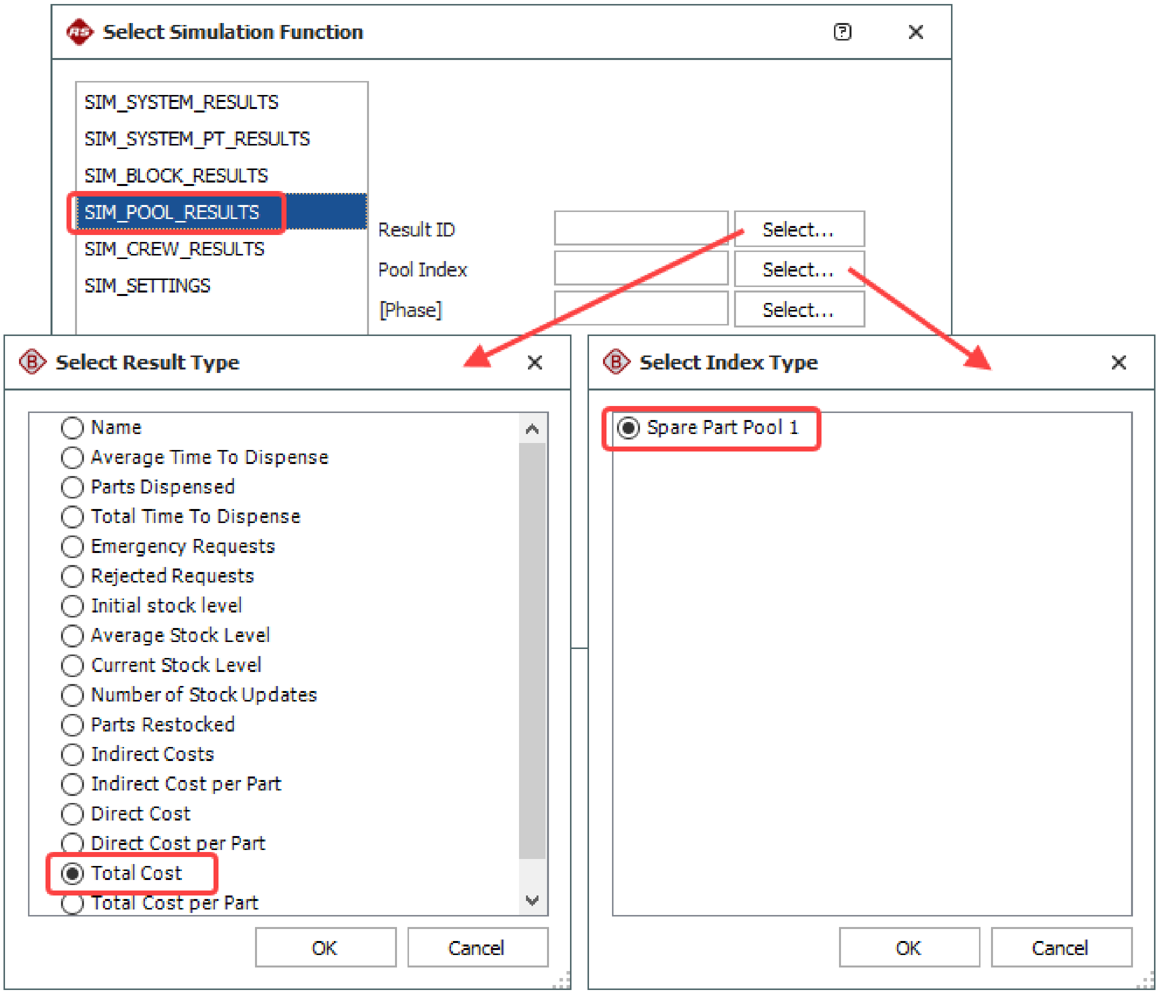
Example

The Worksheet Simulation window used in our example is shown next.



The output columns were specified as follows:


Select Simulation Function
SIM_SYSTEM_RESULTS
SIM_SYSTEM_PT_RESULTS
SIM_BLOCK_RESULTS
SIM_POOL_RESULTS
SIM_CREW_RESULTS
SIM_SETTINGS
Result ID
[Phase]
Select...
Select...
Select Result Type
System Overview
Simulation End Time
Mean Availability
Mean Availability Std Deviation
Mean Availability (w/o PM, OC & Inspection)
Uptime
Downtime
Metrics
MTTFF
MTBF (Total Time)
MTBF (Uptime)
MTBE (Total Time)
MTBE (Uptime)
MTTM
MTTR
Downtime Summary
Waiting Downtime
OK
Cancel



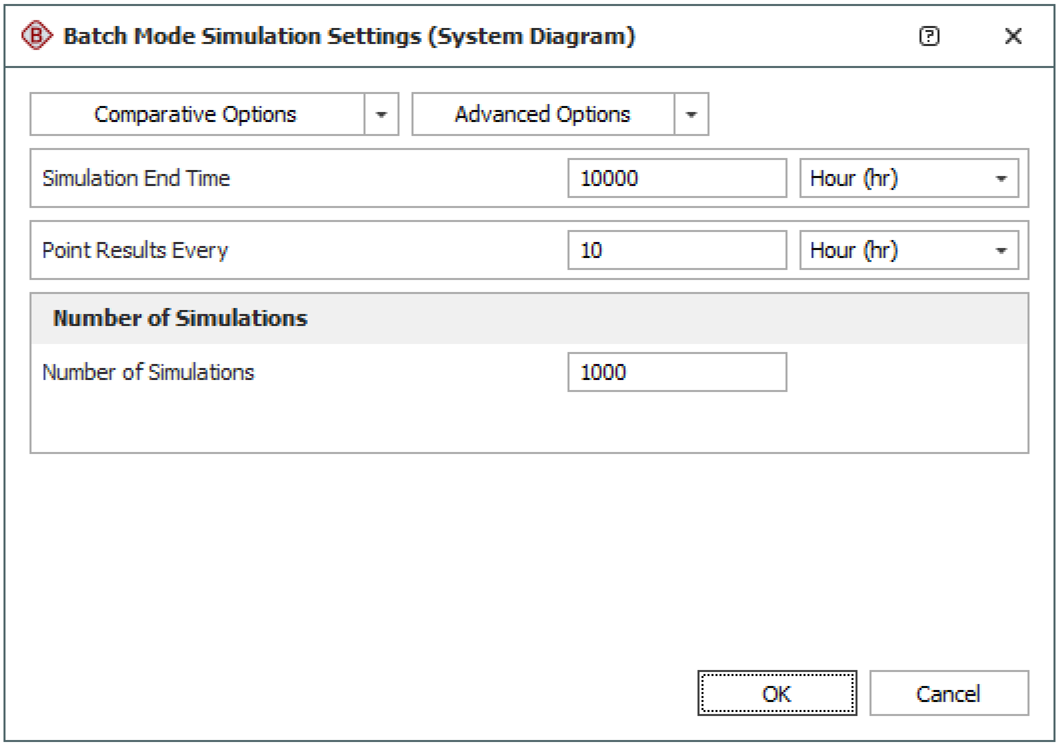
Simulate the Worksheet in BlockSim

With all columns defined, you can perform the simulation in BlockSim.

- Click the **Edit Simulation Settings** button to open the Batch Mode Simulation Settings window. This is an abbreviated version of the Maintainability/Availability Simulation window or of the Flowchart Simulation window, depending on whether the associated diagram is a simulation RBD or an event analysis flowchart. The settings that you specify will apply to the simulation performed for each row (i.e., each factor setting combination). Specify the desired simulation settings and click **OK** to return to the Worksheet Simulation window.
- Click **Run** to perform the simulations. The results will be entered in the simulation worksheet.
- Commit and lock the simulation worksheet to make it available for analysis in Weibull++.

Example

The simulation settings used in our example are shown next.



The first fifteen rows of the simulation worksheet, with results, are shown next.

	Initial Stock Level	Restock Time	Number Added per Restock	Mean Availability	Total Cost
1	1	500	5	0.902354227	1411.382
2	5	1000	1	0.458572644	302.46664
3	4	2000	5	0.702400236	454.30698
4	2	1000	4	0.898430723	925.96507
5	1	2000	1	0.210193871	76.596271
6	3	1000	2	0.627604674	293.08958
7	4	1000	4	0.902806462	1118.2979
8	5	1000	5	0.903143039	1309.8705
9	2	2000	1	0.239007925	114.28311
10	2	500	5	0.903072022	1425.6487
11	1	2000	5	0.628109835	386.94246
12	4	500	1	0.674737851	365.58001
13	2	2000	5	0.655075933	400.84377
14	4	500	[illegible]	[illegible]	[illegible]
15	3	1000			

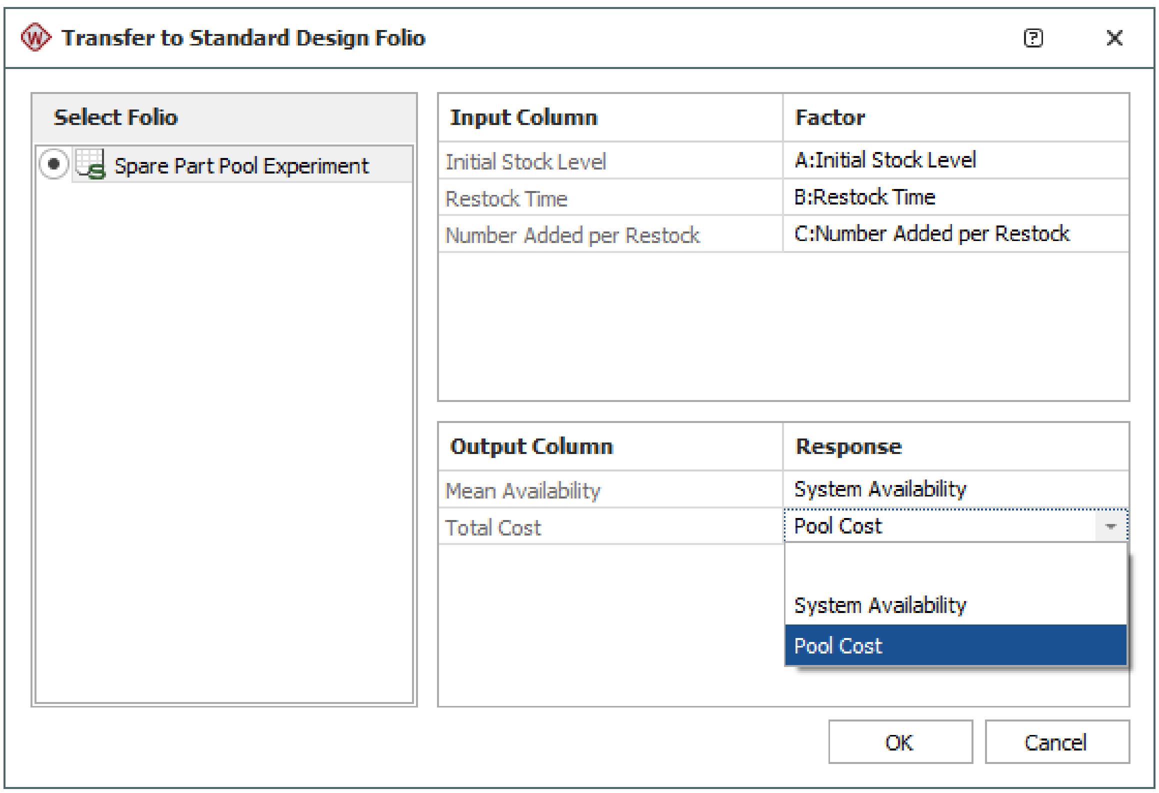
Note: Complete data set is not shown.

Analyze the Results in Weibull++

The results that were derived from the simulation can now be analyzed. To do this:

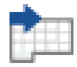
- Open the simulation worksheet in Weibull++. (If it was already open, click the **Refresh** icon on the control panel to load the results and unlock the simulation worksheet for use in Weibull++.)
- Transfer the results to a design folio for analysis. You can do either of the following:
  - **Transfer to Standard Design Folio** transfers the defined input and output columns to the factor and response columns in an existing standard design folio.

In the window that appears, select the design folio to transfer to and match up the columns in the design folio (factor/response columns) with the columns in the simulation worksheet (input/output columns), as shown next.



Note that all data will be transferred from the simulation worksheet, so if you have made changes to the input/factor columns in the simulation worksheet, those changes will be transferred to the design folio.

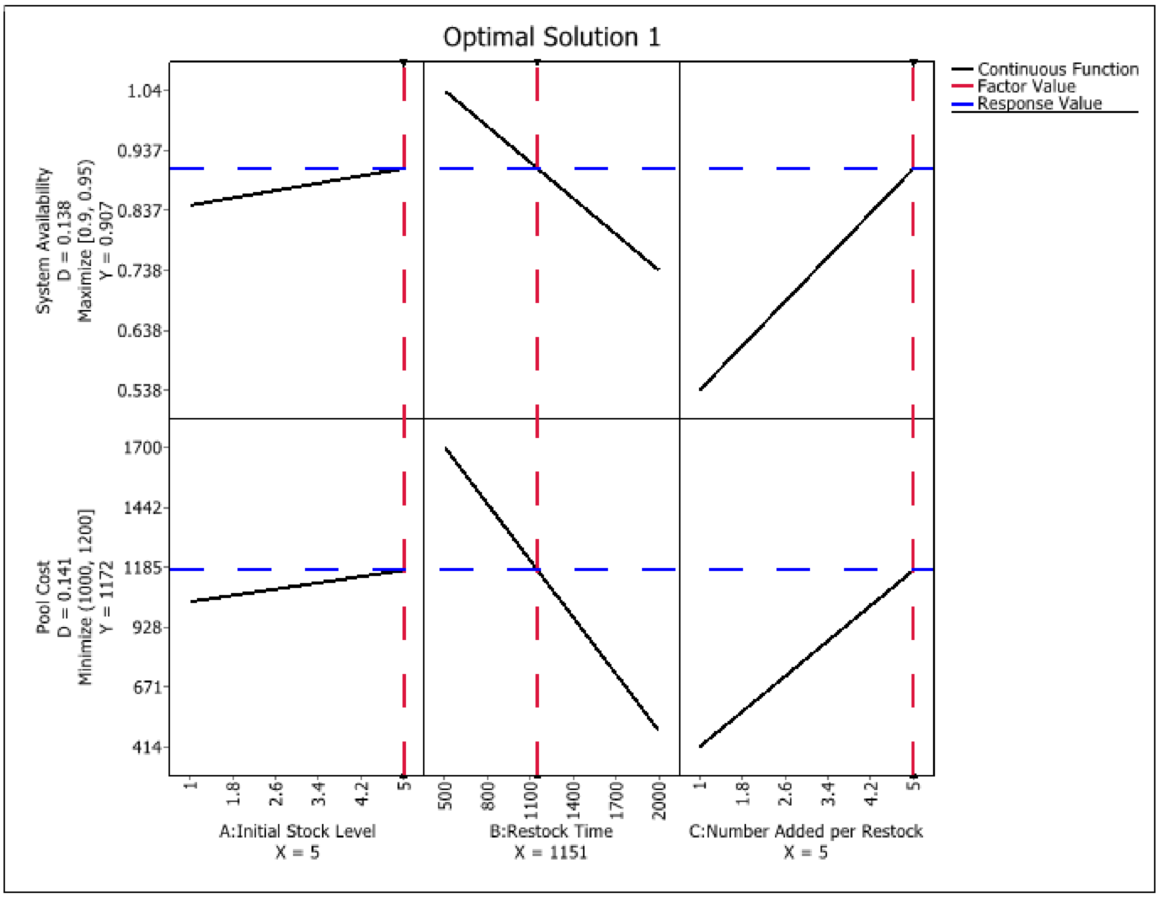
- **Transfer to Free Form Folio** transfers the defined input and results columns to a new free form folio.



You can then perform any analysis you choose in Weibull++, including optimization.

Example

The results of optimization for our example are shown next. A single optimal solution is found with Initial Stock Level = 5, Restock Time = 1151 and Number Added per Restock = 5. This gives a system availability of .907 and pool cost of 1172.

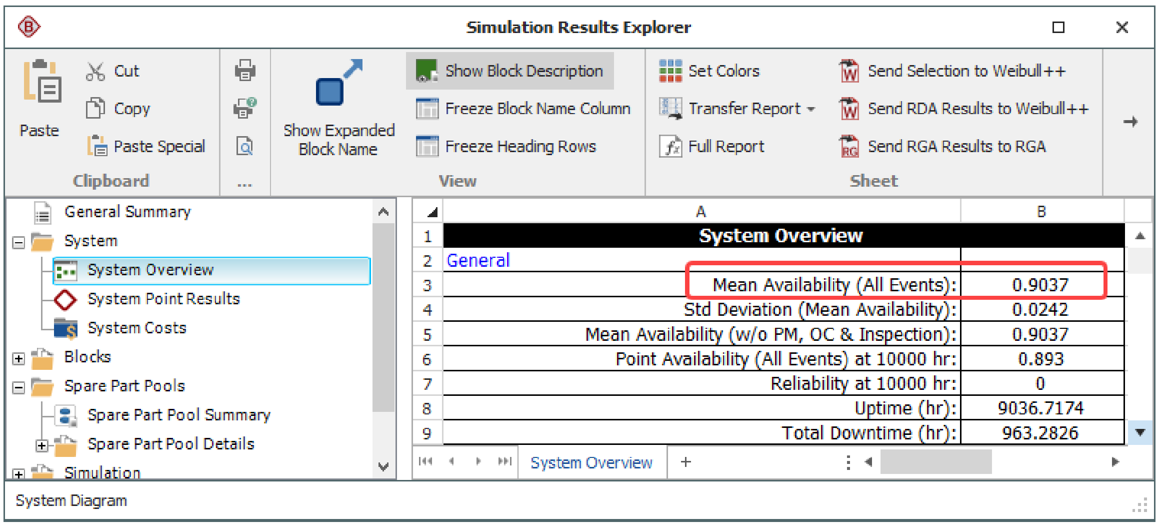


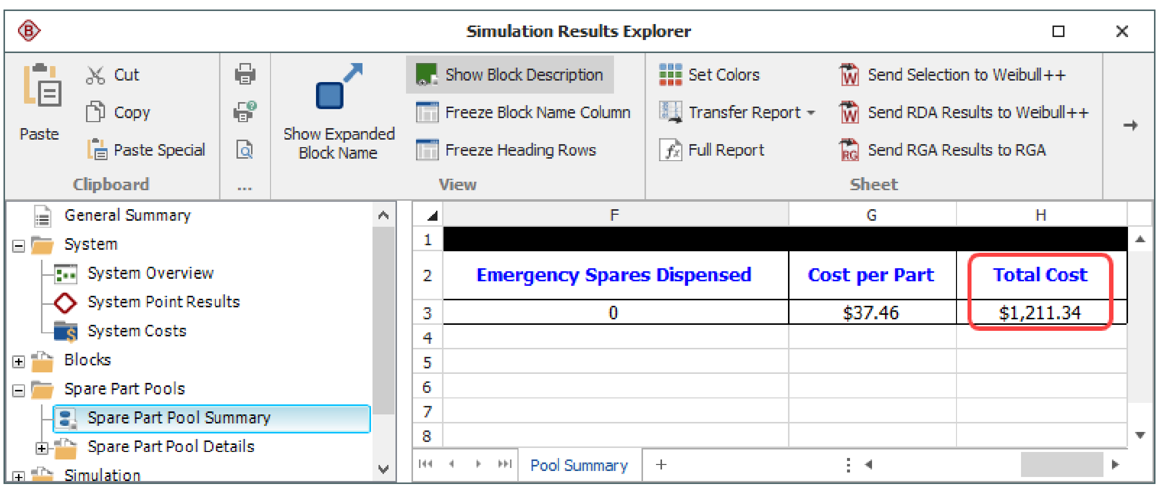
Verify the Results in BlockSim

Because BlockSim and Weibull++ rely on different methodologies to arrive at results, predictions made by Weibull++ will vary from the simulated results generated by BlockSim. Therefore, the optimal settings selected by Weibull++ must be simulated in BlockSim to ensure that the predicted results are reliable. To do this, return to BlockSim and, for each of the variables, set its initial value to the value given in the optimal solution and then simulate the diagram directly (i.e., not via the simulation worksheet) to make sure that the results are close to those estimated in Weibull++.

Example

The simulation results from BlockSim are shown next. The results show a mean system availability of 0.904 and a total cost for the pool of $1212.55. These values are very close to the estimated values in Weibull++.





Quick Statistical Reference

IN THIS CHAPTER

The Quick Statistical Reference (QSR) tool allows you to 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 > ReliaSoft > Quick 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.

Calculating Statistical Values

To calculate a statistical value, select the appropriate function option on left side of the Function Option page. Then enter the required inputs and click **Calculate**. The following options are available:

- 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.

- 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.

- Standard Nominal 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.

- 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.

- 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.

- 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).

- F-Distribution Values

The F-Distribution option returns $Q(F|n1,n2)$, 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 $Q(F|n1,n2)$ 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:

- $n1$ is the degrees of freedom for the first sample.
- $n2$ 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.

- 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:

- $n1$ is the degrees of freedom for the first sample.
- $n2$ is the degrees of freedom for the second sample.

- $Q(F|n1,n2)$ 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.

- 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.

- Incomplete Beta Function

The Incomplete Beta Function option returns the value of $Ix(a,b)$. The required inputs are the values of x, a and b.

The output is obtained by solving for $Ix(a,b)$ 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 $0 < x < 1$.

- 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 $n > 0$.

- 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.

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.

Minimum Data Requirements

IN THIS CHAPTER

Requirements for Weibull++

The following sections describe the minimum data requirements for using different folios in Weibull++.

Life Data 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 life data 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:

- Case 1: No Data
- Case 2: No Failures and 1+ Suspensions
- Case 3: 1 Failure and No Suspensions
- Case 4: 1 Failure and 1+ Suspensions
- Case 5: 2+ Failures

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 life data folios.

Non-Parametric Life Data Analysis (LDA) Folios

For non-parametric LDA folios, there must be at least 1 data point entered.

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 life data 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 life data 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 life data 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 life data 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 life data 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 life data 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 life data 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.

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.

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.

Non-Parametric Recurrent Event Data Analysis (RDA) Folios

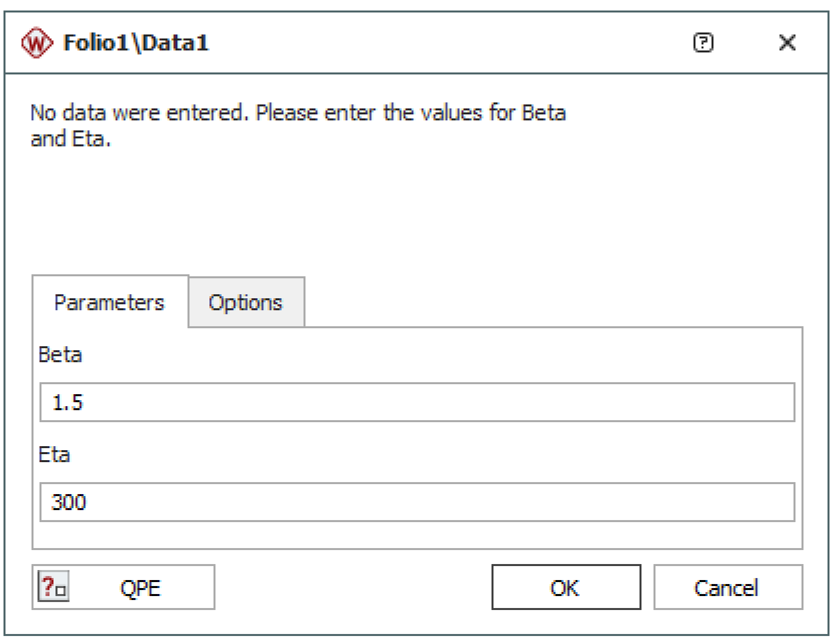
The minimum data requirements for non-parametric RDA folios are as follows:

- 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.

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.

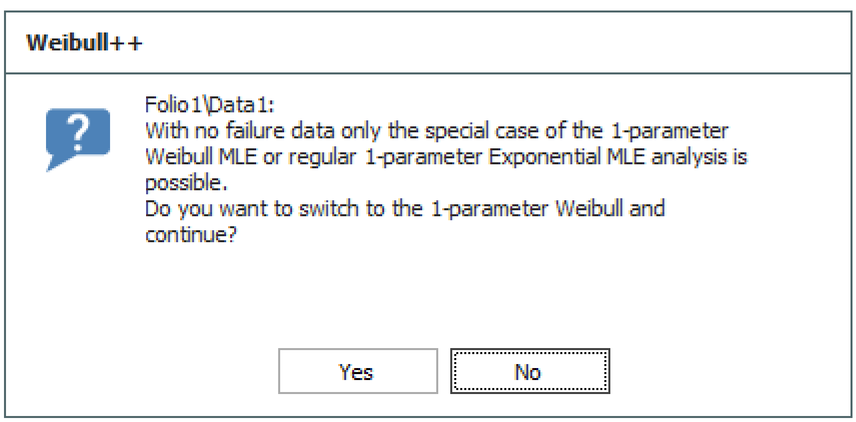


Specify values for the parameters based on the distribution selected and click **OK**. All options and calculations supported by Weibull++ will be available except calculations involving confidence bounds.

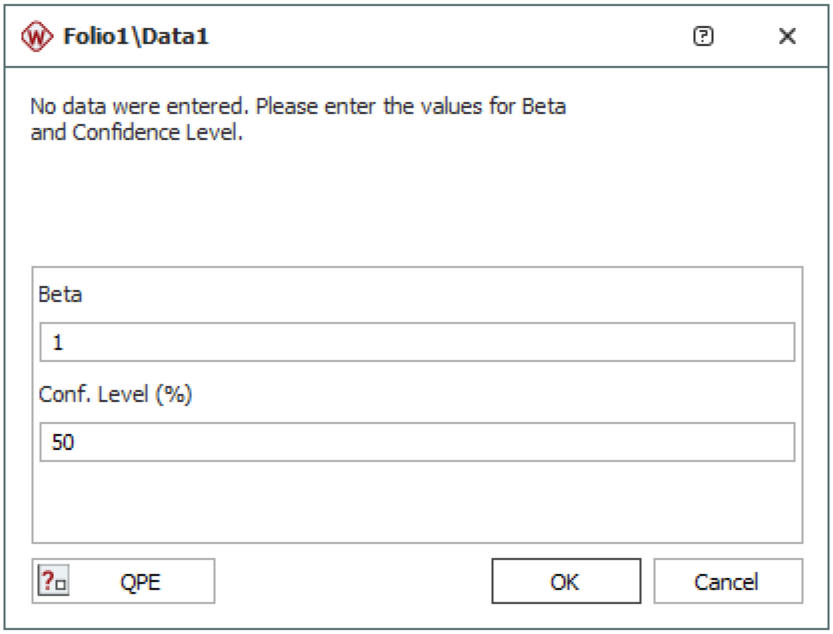
Case 2: No Failures and 1+ Suspensions

Data Types:	All except free-form
Methods 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.

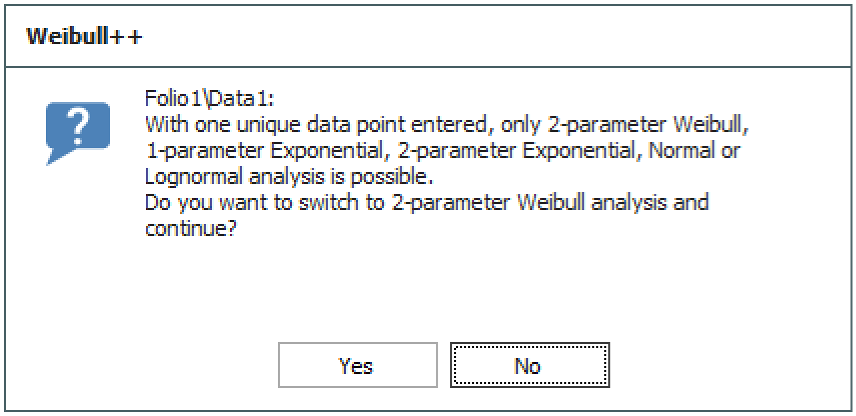
Case 3: 1 Failure and No Suspensions

Free-Form Data

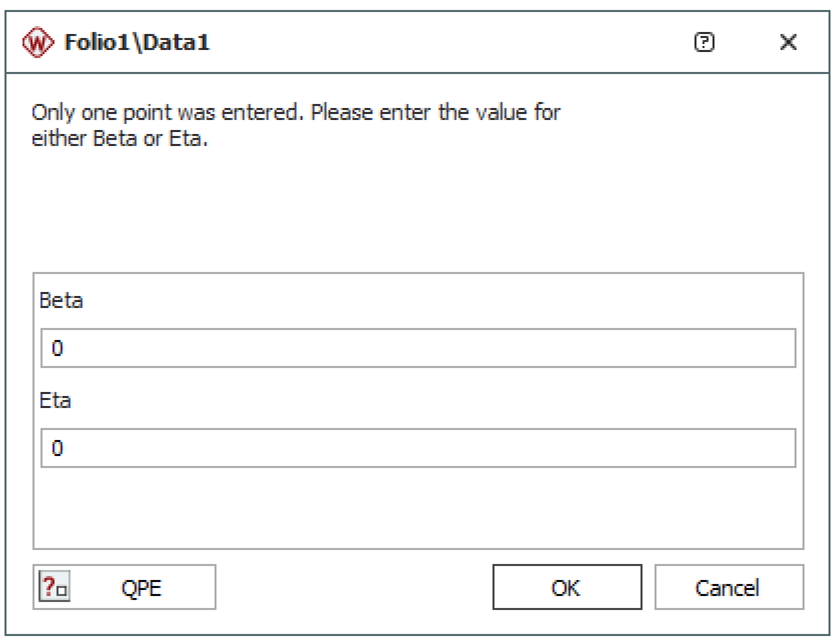
Data Types:	Free-form only
Methods of Analysis	MLE
Confidence Bounds Calculations?	No
Distributions	2-parameter Weibull, 1-parameter exponential, 2-parameter exponential, normal and lognormal
Competing Failure Modes?	No

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.



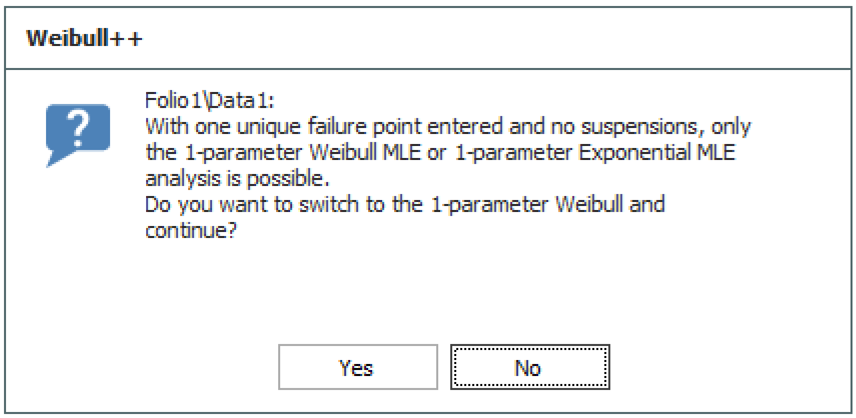
If you select to use a 2-parameter distribution, you will then be prompted to enter a value for one of the parameters. Remember that when using free-form data, you are supplying both the X and Y position for the entered data point. Given this point and a value for one of the parameters, the application will then solve for the other parameter. The parameter that you would like to solve for should be left as zero as shown below.



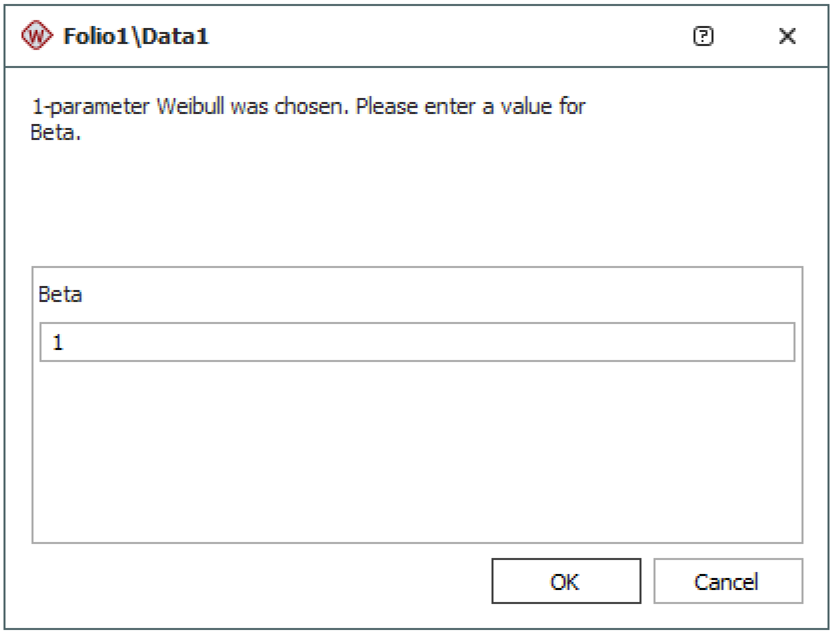
Other Data Types

Data Types:	All except free-form data
Methods 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.



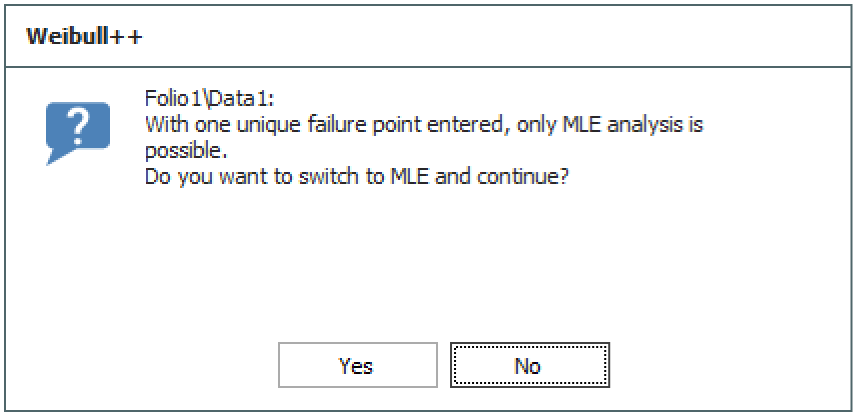
If you select the 1-parameter Weibull, you will then be prompted to enter the value of the known parameter.



Case 4: 1 Failure and 1+ Suspensions

Data Types:	All data types with suspensions
Methods 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.



Case 5: 2+ Failures

Data Types:	All
Methods 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.

Requirements for Accelerated Life Testing

For life-stress data only.

The following sections describe the minimum data requirements for using different folios in Accelerated Life Testing.

Life-Stress Data Folios

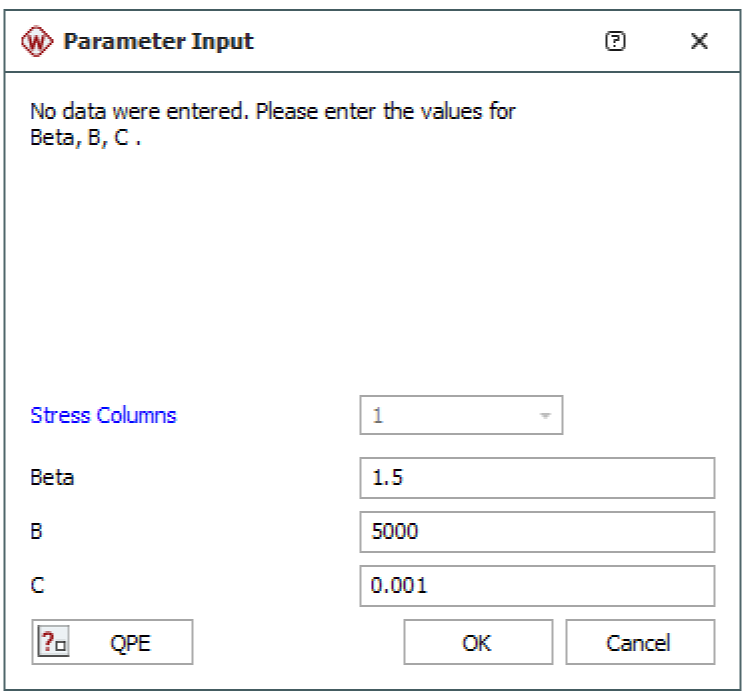
The minimum data requirements for life-stress data folios are broken down by model in the table below. Note the following:

- "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-Non-thermal	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, Weibull++ 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 Accelerated Life Testing will be available except calculations involving confidence bounds and plots that require individual stress levels.

Degradation Analysis Folios

The minimum data requirements for Accelerated Life Testing 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.
- Within each unique Unit ID, the inspection times must be increasing.
- The data must produce enough extrapolated failure/suspension times to perform accelerated life testing data analysis with the selected Accelerated Life Testing model. See the requirements for life-stress data folios.

Requirements for Reliability Design

The minimum data requirements for DOE reliability designs are as follows:

- For the Weibull and lognormal distributions, the number of unique failures must be greater than the number of parameters in the model.
- For the exponential distribution, the number of unique failures must be greater than or equal to the number of parameters in the model.

Depending on the type of design you are working with, these requirements affect calculations in different ways.

- For two level factorial and Plackett-Burman, if the data set does not meet the requirement, the software will automatically reduce the number of parameters in the model until it can be calculated. Therefore, the following requirements apply.
  - To use the Weibull or lognormal distribution, you need at least two failures.
  - To use the exponential distribution, you need at least one failure.
- In one factor reliability designs, you must have more than one factor level regardless of the distribution chosen. Therefore, the following requirements apply.

- To use the Weibull or lognormal distribution, you need at least one failure at each level and at least two different failures for at least one level.
- To use the exponential distribution, you need to have at least one failure at each level.

Requirements for Reliability Growth

Minimum Data Requirements - Times-to-Failure Data

For reliability growth data analysis only.

The minimum data requirements for analysis of developmental times-to-failure data are presented next:

Data Type	Minimum Requirements	Models
Failure Times	At least 3 unique failures.	Crow-AMSAA, Duane
		Crow Extended: BD modes - at least 3 unique BD modes - at least 3 unique
Grouped Failure Times	At least 3 unique failures.	Crow-AMSAA, Duane
		Crow Extended: BD modes - at least 3 unique BD modes - at least 3 unique
Multiple Systems - Known Operating Times	At least 3 unique failures for overall analysis.	Crow-AMSAA, Duane
		Crow Extended: BD modes - at least 3 unique BD modes - at least 3 unique
Multiple Systems - Concurrent Operating Times	At least 3 total failures for overall analysis.	Crow-AMSAA, Duane
	At least 3 unique failures on an individual system to return individual system results.	Crow Extended: BD modes - at least 3 unique BD modes - at least 3 unique

Multiple Systems with Dates	At least 3 total failures for overall analysis.	Crow-AMSAA, Duane
	At least 3 unique failures on an individual system to return individual system results.	Crow Extended: BD modes - at least 3 unique BD modes - at least 3 unique
Multiple Systems with Event Codes	At least 3 total failures for overall analysis. At least 3 unique failures on an individual system to return individual system results.	Crow Extended: BD modes - at least 3 unique BC modes - at least 3 unique I events apply only to BC modes and there must be an I event for each BC mode

Minimum Data Requirements - Discrete Data

For reliability growth data analysis only.

The minimum data requirements for analysis of developmental discrete data are presented next:

Data Type	Minimum Requirements	Models
Sequential	At least 3 reliability points (this is not equal to the number of data points or failures).	Crow-AMSAA, Standard Gompertz, Lloyd-Lipow, Duane, Logistic
	At least 4 reliability points	Modified Gompertz
Sequential with Mode	At least 3 reliability points (this is not equal to the number of data points or failures).	Standard Gompertz, Lloyd-Lipow, Logistic
	At least 4 reliability points	Modified Gompertz
Grouped per Configuration	At least 3 data entries (configurations).	Crow-AMSAA, Standard Gompertz, Lloyd-Lipow, Duane, Logistic
	At least 4 reliability points.	Modified Gompertz

Mixed Data	At least 3 intervals with at least one failure in each.	Crow-AMSAA
	Number of failures in the first interval must be less than the number of units in the interval.	Crow Extended: BD modes - at least 3 unique

Minimum Data Requirements - Multi-Phase Data

For reliability growth data analysis only.

The minimum data requirements for analysis of developmental multi-phase data are presented next:

Data Type	Minimum Requirements	Models
Failure Times	At least 3 unique failures.	Crow Extended - Continuous Evaluation: maximum of 10 phases
Grouped Failure Times	At least 3 unique failures.	Crow Extended - Continuous Evaluation: maximum of 10 phases
Mixed Data	At least 3 intervals with at least 1 failure in each BD modes - at least 3 unique	Crow Extended - Continuous Evaluation: maximum of 10 phases

Minimum Data Requirements - Reliability Data

For reliability growth data analysis only.

The minimum data requirements for analysis of developmental reliability data are presented next:

Data Type	Minimum Requirements	Models
Reliability	At least 3 unique failures.	Lloyd-Lipow: Initial Time/Stage > 0 Logistic: Initial Time/State > = 0
	At least 4 unique failures.	Standard Gompertz and Modified Gompertz: Initial Time/Stage >= 0

Minimum Data Requirements - Change of Slope Calculations

For reliability growth data analysis only.

The minimum requirements for Change of Slope calculations require at least 3 failures up to the break point and at least 2 failures after the break point. Therefore, there must be at least 5 total failures or intervals (in grouped data) to conduct the analysis.

Change of Slope calculations are available only for developmental testing data using Crow-AMSAA, for the following data types:

- Failure Times
- Grouped Failure Times
- Known Operating Times
- Concurrent Operating Times
- Multiple Systems with Dates
- Mixed

Minimum Data Requirements - Fielded Data

For reliability growth data analysis only.

The minimum data requirements for analysis of fielded data are presented next:

Data Type	Minimum Requirements	Models
Repairable Systems	At least 3 total failures for overall analysis.	Power Law
	At least 3 unique failures on an individual system to return individual system results.	Crow Extended: BD modes - at least 3 unique
Fleet	At least 3 total failures for overall analysis.	Crow-AMSAA
	At least 3 unique failures on an individual system to return individual system results	Crow Extended: BD modes - at least 3 unique BC modes - at least 3 unique

Application Setup

IN THIS CHAPTER

The Application Setup contains personal preferences that are stored per computer/username. These include default settings for new analyses/plots that you create from this computer, as well as personal preferences that don't affect shared analysis data stored in the database.

In addition, note that the region and language settings will determine how dates, times, decimals, currency, etc. are displayed on your computer.

To open the Application Setup window, choose **File > Application Setup**.

The settings on the pages listed under **Common Settings** are the same for all ReliaSoft desktop applications; the pages listed under the application name contain settings that apply only to the current applications.

ReliaSoft Settings

Common Settings

The Common Settings page of the Application Setup can be changed from within any ReliaSoft desktop application.

- **Interface** sets the language and the skin (color scheme) for the software interface. Selecting a skin shows you a preview of the changes in the Application Setup window. The change takes effect in the rest of the interface the next time you launch each application.

  Note that a language change will not affect applications that are currently available only in English.

- **Open With** sets the ReliaSoft application to launch by default when you double-click an *.rsr21 or *.rserp file from this computer. If you want to be prompted to choose an application each time, select **Show the ReliaSoft Launcher**.

- **Recent Databases List** sets the maximum number of recently saved database files to be displayed in the "Recent Databases" list in the Backstage view.

- **Auto Save** specifies when folios and diagrams will be saved automatically. The application always saves folios/diagrams upon calculate or close. If you want to also save periodically while you're editing, specify an interval. This can be useful if you're working across a network in an enterprise database.

- **Other**

  - **Allow multiple projects (project explorer)** applies to ReliaSoft desktop applications that use the current project explorer (i.e., Weibull++, BlockSim and Lambda Predict). Select the check box if you want to have multiple projects open at the same time. There is a separate setting for project windows in XFMEA/RCM++. (See Working with Multiple Projects.)

- **Show project name in opened folios** applies to ReliaSoft desktop applications that use the current project explorer (i.e., Weibull++, BlockSim and Lambda Predict). It shows the project name in [brackets] on the tab or window caption. This may be useful when you have multiple projects open at the same time.
- **Use web-based help**. Select the 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.
- **Highlight 'Active' category in ribbon** applies a highlight color to the label that shows which tabs are currently active.
- **Use non-modal Resource Manager** locks the Resource Manager in a top window position so it can remain open while you have access to the project. If not selected, you must close the Resource Manager to return to the project.
- **Warn when changing metric associations** shows a warning when you select an existing metric in a situation that will change the metric's associated model. (See Showing Metrics in Folios/Diagrams.)
- **Hide unused project folders** applies to ReliaSoft desktop applications that use the current project explorer (i.e., Weibull++, BlockSim and Lambda Predict). It hides from view all folders that do not contain any project items.

- **Highlight colors** sets the colors used to color-code values in the following analyses:
  - Results in the FMRA in BlockSim and XFMEA/RCM++
  - Certain BlockSim plots
  - BlockSim FRED reports
  - BlockSim allocation analysis
  - Spread of failure rate contributions in a prediction folio in Lambda Predict.

Backup/Check Out Options

The Backup/Check Out Options page of the Application Setup can be changed from within any ReliaSoft desktop application.

- **Standard Database Maintenance** applies to standard database files (*.rsr21) only. It allows you to automatically save a backup of a database in the location you specify when you close the application. The backup file will be named after the database file with the backup date appended (e.g., RSDatabase1_2021-01-15.rsr21).

You can specify how many days of backups can be automatically saved (up to 10), but 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).

Note that you can also create backups manually at any time by choosing **File > Save As**. For more tips on keeping databases running smoothly, see Backups and Database Maintenance: Protecting Your Data.

- **Check In/Out** specifies the default save location for projects that you check out from a database.

Other Common Settings

The Common Settings > Other page of the Application Setup can be changed from within any ReliaSoft desktop application.

- **Model, Variable and Workbook Names** disallows spaces and special characters in the names of models (including those published from any ReliaSoft desktop application) and variables, and in the names of workbooks created in BlockSim. This will ensure that all models, variables and workbooks can be used in equations in event analysis flowcharts.
- **Display Object IDs**. Select the check box to always display the unique object identifier (object ID) of each project and resource in the database. The project IDs will be displayed in the Manage Projects window and in the Edit Project Properties window, while the resource IDs will be displayed in the Resource Manager. In BlockSim, diagram IDs and block IDs will be shown in the control panel and in the Block Details simulation results worksheet.
- **Copy Plot Graphic** sets the default file type to use when copying a plot to the Clipboard as an image. If you will be pasting copied plots into ReliaSoft Workbooks, choose **Metafile for ReliaSoft 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**. If you want to choose the graphic on the spot, you can do so by right-clicking the plot and choosing the **Copy Plot Graphic** command on the shortcut menu.
- **Hierarchical Trees** sets the maximum number of lines for displaying names and descriptions in any of the hierarchical tree interfaces used in XFMEA/RCM++, Lambda Predict and MPC.
- **Alternative Credentials** applies to secure databases. It allows you to either save your alternative credentials (so your account is automatically logged in whenever you open a secure database from a different domain) or clear the credentials from this computer.

- **Diagram Skins Utility** converts diagram skins created in Version 9 to the latest version. Click the **Convert** link, and in the window that opens, select the name of the skin(s) that you wish to convert. You will need to close and re-launch the application to use the converted skin(s). (See "Converting Version 9 Skins" in the BlockSim, Weibull++ or XFMEA/RCM++ documentation.)

Application-Specific Settings

Weibull++ Life Data Folios Settings

The Weibull++ Life Data Folios page of Weibull++'s Application Setup contains default settings that will be applied when you create a new life data folio. Note that some of the configurable settings that are saved with the folio can still be modified from the control panel and others can now be modified via the Item Properties window.

- **Analysis Method** sets the default analysis method for time-to-failure data sheets and for free-form data sheets.
- **Ranking Method** sets the default ranking method for rank regression analyses.
- **Confidence Bounds Method** sets the default method used to calculate the confidence bounds.
- **Grouped Data Settings** affect how individual median rank values are assigned to each data point (observation) within each group.
  - **Ungroup on regression**. Select the check box 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 the line.
  - **Ungroup on MLE**. Select the check box 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**. Select which metrics will be shown in the **Analysis Summary** area of the control panel:
  - **Show Theta** displays the theta parameter when using the Weibull distribution. Theta is defined as the sum of the scale and location parameters (eta and gamma):

$$\theta = \eta + \gamma$$

- **Show correlation coefficient (Rho)** applies only when using rank regression as the analysis method. It displays the correlation coefficient, which may help determine how well the parameters of a distribution fit the data.
- **Show likelihood value (LK)** displays the likelihood function value, which may be used to compare how well different models fit the same data set.

ALTA Life-Stress Folios Settings

The ALTA Life-Stress Folios page of Weibull++'s Application Setup contains default settings that will be applied when you create a new life-stress folio. Note that some of the configurable settings that are saved with the folio can still be modified from the control panel and others can now be modified via the Item Properties window.

- **Data Sheet Options for ALTA**
  - **Show likelihood value (LK)** displays the likelihood function value in the **Analysis Summary** area of the control panel. The LK value may be used to compare how well different models fit the same data set.
  - **Show activation energy** applies only to temperature-related models such as Arrhenius, Eyring, generalized Eyring, temperature-humidity, and temperature-nonthermal. It displays the activation energy (Ea) in the **Analysis Summary** area of the control panel. A large activation energy means temperature has a large effect on the life of the product. Click the **Units** drop-down list to specify which unit to use (Kelvin or Rankine).
  - **Indicate stress columns used** highlights the borders of the selected stress columns in the data sheet. Click the **Color** drop-down list to select the border color.
- **Grouped Data Settings** affect how individual median rank values are assigned to each data point (observation) within each group. Select the check box 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.
- **Other Options.** Select the check box to allow the Use Stress Level window to remain open, while you have access to all life-stress folios. If not selected, you must close the Use Stress Level window to return to the folios.
- **Default Life Distribution** sets the default life distribution for life-stress folios. Note that 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 Accelerated Life Testing PRO), the life distribution will automatically reset to Weibull.

- **Cumulative Damage Model (ALTA PRO)**. These options apply to the cumulative damage life-stress model in Accelerated Life Testing PRO.
  - **Time-dependent stress profiles** specifies the time at which the stress will be applied 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, or **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.
  - **Allow large Beta values** displays beta values up to 250. If not selected, the default maximum beta value is 100.

RGA Growth Data Folios Settings

The RGA Growth Data Folios page of Weibull++'s Application Setup contains default settings that will be applied when you create a new growth data folio. Note that some of the configurable settings that are saved with the folio can still be modified from the control panel and others can now be modified via the Item Properties window.

- **Data Sheet Options**
  - **Show Alter Data Type** enables the Alter Parameters command in all growth data folios. The command enables you to experiment with possible alternative scenarios by allowing you to alter the values of the calculated parameters.
  - **Use advanced systems view** uses the Advanced Systems View by default when you create a new data sheet that offers both the Normal View and Advanced Systems View. (See Normal and Advanced Systems View.)
  - **Warn when transferring data with excluded events** displays a message when you transfer a data set that has some coded events excluded from the calculations. This applies only when you transfer data from a Multiple Systems Event Code data sheet to another type of data sheet.
- **Default Number of Systems**
  - **For 'Multiple Systems - Known Operating Times' data sheets** specifies the number of "Time System" columns that will appear by default in the data sheet.

  - **For all other fielded/developmental multiple systems data types** specifies the number of systems that appear by default in the navigation panel of the Advanced Systems View. (See Normal and Advanced Systems View.)

- **Statistical Tests.** When you use the Crow-AMSAA (NHPP) or Crow Extended models, Weibull++ automatically performs statistical tests on the calculated data set. Use the check boxes to select which test results to display in the Result area of the growth data folio control panel.

  - **Show Chi-Squared** applies to grouped failure times only. The test evaluates the hypothesis that the failure times follow a non-homogeneous Poisson process (NHPP).

  - **Show Cramér-von Mises (CVM)** applies to non-grouped failure times where there are no gaps in the data. The test evaluates the hypothesis that the failure times follow a non-homogeneous Poisson process (NHPP).

  - **Show Laplace Trend** applies to multiple systems analysis only. The test evaluates the hypothesis that a trend does not exist in the data, and displays in the analysis results whether the system reliability is improving, deteriorating or staying the same.

  - **Show Common Beta Hypothesis (CBH)** applies to multiple systems analysis only. The test evaluates whether all the systems in the data set have similar beta values (i.e., operated in a similar manner).

  - **Default Significance Level** specifies the significance level used in the statistical tests.

- **Failure Discounting** applies only to the Discrete Sequential with Mode data type. Specify the confidence level that will be used to define the fractional decrease in failure value. (See Failure Discounting.)

- **User Warnings**

  - **Beta = 1 hypothesis is invalid** displays a message when the beta = 1 hypothesis fails. This warning applies only to the Crow Extended or Crow Extended - Continuous Evaluation models.

  - **Model parameters out of range** displays a message when the value of the k parameter (logistic model) or Alpha parameter (Lloyd-Lipow model) is less than or equal to zero. Negative values indicate that the analysis results may not be valid.

  - **Effectiveness factors of zero** displays a message when one or more effectiveness factors equal to zero. This warning applies only to the Crow Extended or Crow Extended - Continuous Evaluation models.

- **Other Options**
  - **Use percents (not decimals) for reliability** displays reliability values as percentages (e.g., 90) rather than decimals (e.g. 0.90). This applies to the Reliability data type only.
  - **Data input is cumulative** configures the data sheet for cumulative failure times, by default, instead of non-cumulative failure times. (See Cumulative vs. Non-Cumulative Data.)
  - **Calculate unbiased beta** removes the bias of the MLE estimate of the beta parameter, when applicable. An unbiased estimate will be labeled in the results as "Beta (UnB)" instead of "Beta." For more information, the "Unbiasing Beta for the Crow-AMSAA (NHPP) Model" article at: http://www.weibull.com/hotwire/issue141/hottopics141.htm demonstrates how the bias is corrected for Crow-AMSAA (NHPP). This is also applicable to the Crow Extended, Crow Extended - Continuous Evaluation and power law models.

Results	
Parameters	
Beta (UnB)	0.689808
Lambda (hr)	0.527989
Growth Rate	0.310192
DMTBF (hr)	16.107538
DFI	0.062083

IMPORTANT: Starting in Version 2020, Weibull++ calculates the ratio to convert the MLE beta to the unbiased beta as "(N-2)/(N-1)" instead of the old ratio of "(N-2)/N" that was used in RGA Version 2019 and earlier. *This change only affects failure terminated data.*

Design Folios Settings

The Design Folios page of Weibull++'s Application Setup contains settings that will be applied when you create a new DOE design folio.

- **Formatting Options**. Select which colors to use for the applicable columns in the data sheet. Click the **Color** drop-down list to select the border color.
  - **Indicate selected response** highlights the borders of the cells in every response column.
  - **Indicate response columns** and **Indicate factor columns** sets the background color of every response and factor column.

- **Indicate generated response columns** sets the background color of columns that are automatically populated based on entered observations (e.g., the Y Mean column in robust design folios).

- **Default Analysis Settings** set the default options for the settings on the Analysis Settings page of the control panel.
- **Analysis History** sets the maximum number of historical analysis results to keep in the Analysis Summary window. Once the maximum is reached, older results are deleted.

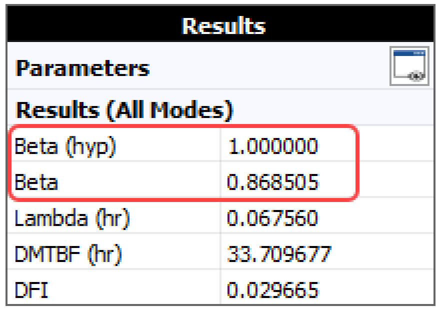
Calculations Settings

The Calculations page of Weibull++'s Application Setup contains default settings that will be applied to calculated results when you create a new folio. Note that some of the configurable settings that are saved with the folio can still be modified from the control panel and others can now be modified via the Item Properties window.

- **Precision to be Displayed on Calculations** sets the math precision (number of decimal places) and 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** applies to analyses in Weibull++ that use the location parameter (gamma).
    - **Warn if location parameter is negative** generates a warning when the location parameter contains a negative value.
    - **Discard if location parameter is negative** causes the software to behave 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.
  - **Reset if location parameter > T1 on Exponential** applies to the exponential distribution for both rank regression on X (RRX) and rank regression on Y (RRY). If the location parameter is greater than the first failure time, it resets the value of the location parameter equal to the first time-to-failure. The parameters will then be recalculated based on the new value of the location parameter.
  - **Use true 3-P MLE on Weibull** applies a true MLE solution to the 3-parameter Weibull distribution. 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 $T' = (T - \gamma)$.
  - **Use Theta parameter scheme on Weibull** replaces the Weibull scale parameter eta in all results with theta, where theta = eta + gamma.
  - **Allow negative location parameter for G-Gamma** allows the lambda parameter of the generalized gamma distribution to take negative values.
- **Precision for Convergence Criteria** sets 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).
- **Crow/Power Law Models** applies only to calculations using the Crow-AMSAA (NHPP), Power Law, Crow Extended or Crow Extended - Continuous Evaluation models. Select whether to display the value of the power law parameter, **Lambda**, or the Weibull parameter, **Eta**, in the analysis results.
- **Duane Model** applies only to calculations using the Duane model. Select whether to display the value of the **A** parameter (cumulative failure intensity) or the **b** parameter (cumulative MTBF) in the analysis results.
- **Crow Extended Model** applies only to calculations using the Crow Extended or Crow Extended - Continuous Evaluation models.

- **Show Crow-AMSAA beta for beta = 1 hypothesis** displays the value of the beta parameter for the Crow-AMSAA (NHPP) model in addition to the hypothetical value of beta calculated for the Crow Extended or Crow Extended - Continuous Evaluation models, as shown in the following example.



Results	
Parameters	
Results (All Modes)	
Beta (hyp)	1.000000
Beta	0.868505
Lambda (hr)	0.067560
DMTBF (hr)	33.709677
DFI	0.029665

- **Allow EF values for BD modes with implemented fixes** adds a second tab in the Effective Factors window when you are using the Crow Extended - Continuous Evaluation model. This allows you to record the effectiveness factors for BD modes that were fixed during testing. Note that this functionality is for your information only; it is not used in calculations.

- **Instantaneous MTBF/FI at the End of Development Testing** sets whether the instantaneous MTBF and failure intensity at the end of the test will be shown in the analysis results with the term "demonstrated" (i.e., DMTBF/DFI) or "achieved" (i.e., AMTBF/AFI).

- **Other Options**

  - **Use mean time for the exponential distribution.** By default, lambda is used as the scale parameter of the exponential distribution. Select the check box if you prefer to use MTTF (1/lambda) instead.

  - **Sort before calculation** sorts the times in ascending order before calculating the data sheet. This does not affect the results of the calculation.

  - **Use unbiased Std on Normal data** applies to analyses in Weibull++ that use the normal distribution with complete data. Select the check box to correct the MLE standard deviation (sigma) value for biasedness.

  - **Use special sort (place F before S)** applies to analyses in Weibull++. The software always sorts entries based on failure times, but in special cases where two or more failure times are identical to the 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 this check box 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.

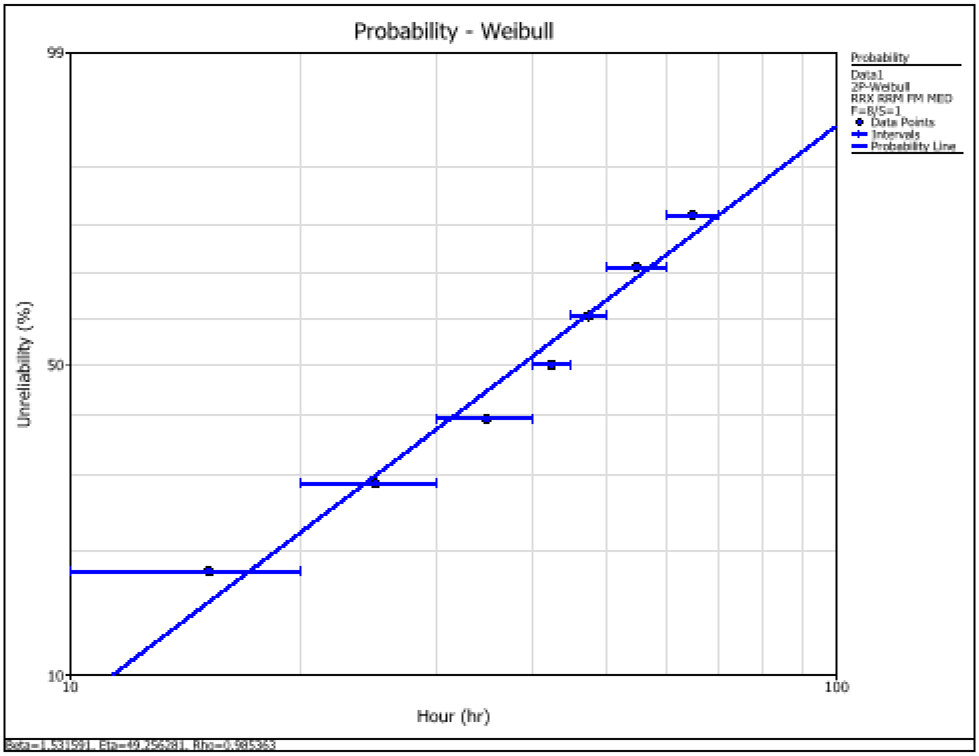
- **Use plotted Y points for K-S test** applies to the Kolmogorov-Smirnov test, which is used by the Goodness of Fit Results and the Distribution Wizard in Weibull++. Select the check box to use the plotted points (using median ranks or Kaplan-Meier) to calculate the difference between the observed and model-estimated probability. If not selected, the non-parametric (empirical) values of the probability of failure are used as the observed values for the test.

Plot Settings

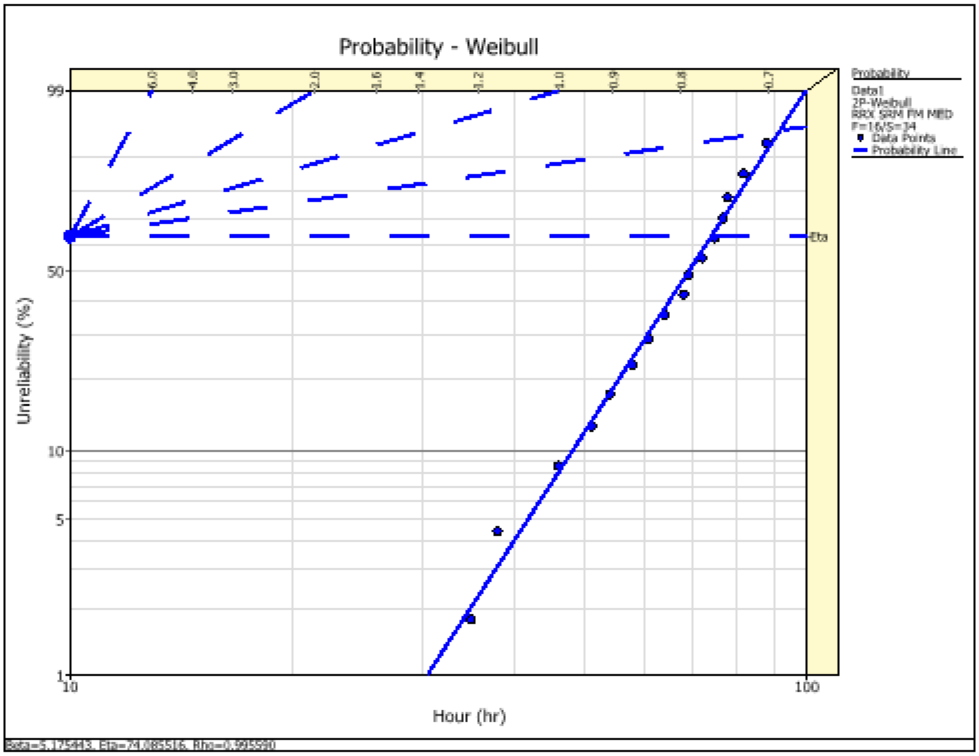
The Plot page of Weibull++'s Application Setup contains default settings that will be applied when you create a new plot sheet. Note that some of the configurable settings that are saved with the plot can still be modified from the control panel and others can be modified via the Show/Hide Plot Items window (**Plot > Actions > Show/Hide Plot Items**).

What's Changed? Starting in Version 2019, several options, including those for setting the numbers in axis labels and showing the user, date and time on plots, are now specified only with the Plot Setup utility.

- **Settings for Plot Sheets**
  - **Enable auto refresh** activates the auto refresh option.
  - **Show lines around interval points** shows lines around the points for interval censored data. The lines represent the starting and ending time for each interval. For example:



- **Show parameter scales** shows the parameter scales on a probability plot. The probability scales allow you to read the parameters directly from the plot.
- **Show parameter bars** places a box around the parameter scales on a probability plot. For example:



- **Show left censored points** displays the left censored points as triangles on the bottom of the plot when plotting interval or left censored data.
- **Show suspensions** displays the suspension (right-censored) data points as triangles on the bottom of the plot when plotting suspended data.

- **Other Options**
  - **Enable interactive plot** highlights and shows the coordinates/values of a point, line, slice or bar on the plot when you move the mouse pointer over it. Note that if you change this setting, you must redraw the plot to see the change.
  - **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).
  - **Straighten Gamma line** straightens 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 redraw an affected plot to see the change.
  - **Display interval lines (when applicable)** applies only to Reliability Growth times-to-failure data sheets. Select the check box to display the instantaneous interval lines on the MTBF vs. time plot or failure intensity vs. time plot.

Other Settings

The Other Settings page of Weibull++'s Application Setup contains default settings that will be applied when you create a new folio.

- **Preferences**
  - **Show the Project Item Wizard when creating new projects** displays a wizard that helps you select the first new folio to create in a project. If not selected, the new project will be blank.
  - **While QCP is open, have access to all folios** locks the QCP in a top window position (i.e., non-modal) so it can remain open while you have access to all folios. 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.
  - **While Analysis Summary is open, have access to all folios** lets you view other folios while the DOE Analysis Summary window is still open. If not selected, you must close the Analysis Summary window to return to the folios.
  - **Show warning when deleting columns with data** displays a confirmation message when using the Add or Remove Columns window to delete a column that has data.
  - **Show warning when deleting data sheets** displays a confirmation message when deleting a data sheet.
- **Diagram**. The following settings apply to diagrams created in Weibull++:
  - **Truncate block names when the number of characters exceeds** sets the maximum number of characters displayed for block names.
  - **Ask for confirmation when connecting multiple blocks**. 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. Select the check box to be prompted to add connections to all of the selected blocks.
  - **Ask to apply changes when using the Format Painter** displays a window with the Format Painter that allows you to specify which settings will be applied to another block.

- **Other Options**
  - **Default folio font** sets the default font for data sheets in new folios.
  - **Show list of distributions on hovering** displays the distribution/model drop-down list in control panels when you move the pointer over the bar. If not selected, you must click the bar to display the list.

Customize Folio Tools Settings

The Customize Folio Tools page of Weibull++'s Application Setup contains default control panel settings that will be applied when you create a new life data folio and life-stress data folio.

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.

Reset Settings

The Reset Settings page of the Application Setup allows you to restore some or all of the settings to their original configuration. Note that it's recommended to close all windows before resetting settings. If the change does not take effect immediately, restart the application.

- **Reset Common Settings** resets the settings under the **Common Settings** heading in the Application Setup.
- **Reset Application Settings** resets the settings under the current application's heading in the Application Setup. It also clears any saved default column headings in life data, life-stress data and growth data folios.
- **Reset Application Form Settings** resets the form settings for the current application (e.g., size and location of windows).
- **Reset Plot Settings** resets the Plot Setup settings that are applied by default when you create a new plot in the current application.
- **Reset FIDES Settings** (Lambda Predict only) resets the predefined FIDES settings (e.g., saved pi factors, process audits, etc.) that are applied by default when you create a new database. (See FIDES Settings Manager in the Lambda Predict documentation.)
- **Reset Component Defaults** (Lambda Predict only) resets the component property values that are applied by default when you add a new component to a prediction. (See Default Component Properties in the Lambda Predict documentation.)

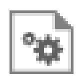
- **Reset All Settings** resets all the saved settings for the current application. This is the same as clicking all of the individual "Reset" buttons above.

Project Item Settings

ReliaSoft desktop applications offer a variety of configurable settings that are stored per computer/username and managed via the Application Setup.

Your personalized application setup determines the default settings for new folios, diagrams and flowcharts that you create in Weibull++ and BlockSim. The relevant settings are also saved with each individual folio/diagram and are accessible from the Item Properties window. This makes it possible to have different settings for different analyses, and also ensures that any given analysis will be the same for all database users.

To view and edit the configurable settings in the Item Properties window, select the folio, diagram or flowchart in the current project explorer and choose **Project > Current Item > Item Properties**.



The settings are displayed on the second tab of the properties window. (In a secure database, the settings can be edited only if the user is the project owner or has the "Create/edit project items" permission.)

If you modify the settings for a particular project item, the new preferences will be saved with the folio/diagram. The default preferences for new folios/diagrams in the application setup will remain unchanged.

Region and Language Settings

The Region and Language settings for your particular computer will have some impact on the way some information is displayed in ReliaSoft applications.

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, 2019 is stored in the database, User A might see it as "9/25/2019" while User B might see "25-Sep-19."

This topic first explains how to view or change the Region and Language settings on your computer and then discusses some specific considerations for ReliaSoft applications, including:

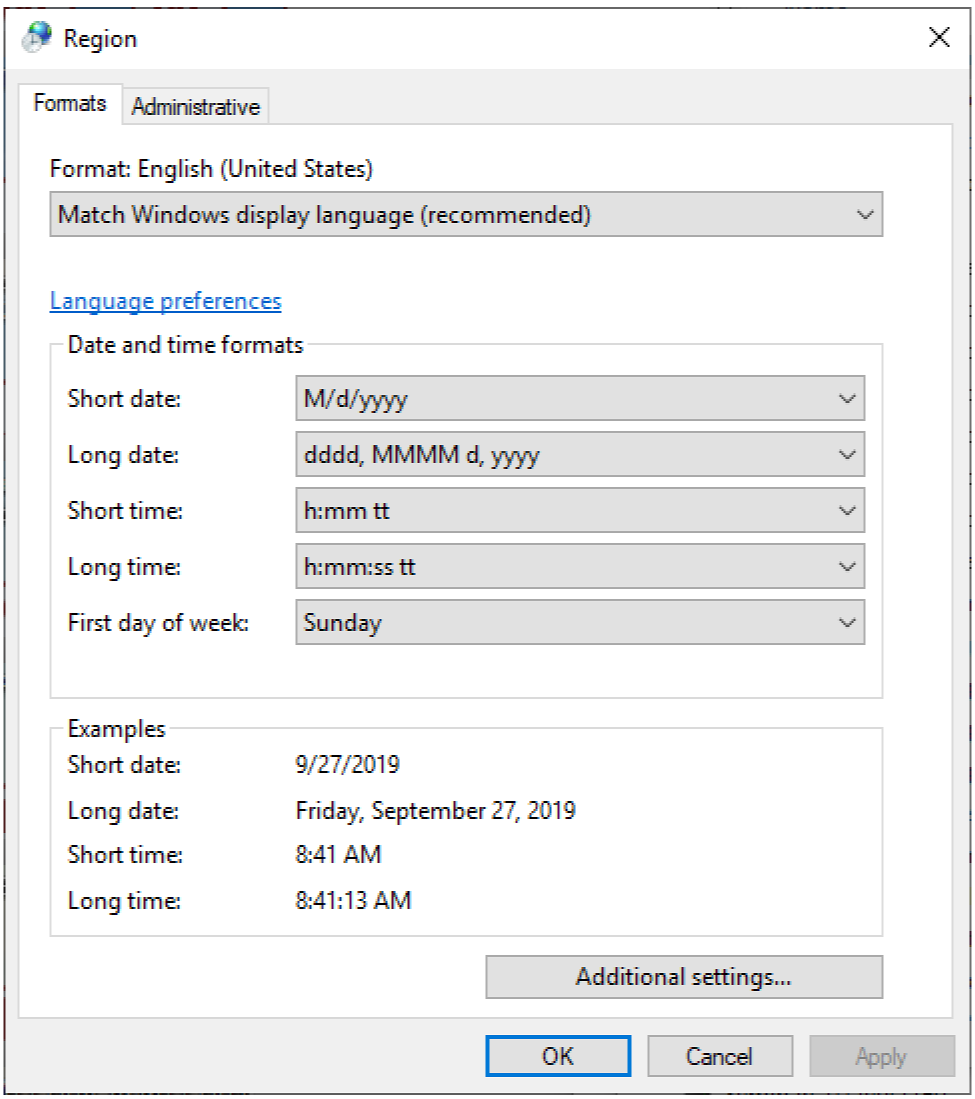
- Which language is selected by default when you install the software
- How dates and times will be entered and displayed
- How decimal values and currency will be entered and displayed

Viewing or Changing the Region and Language Settings for Your Computer

To view or change the region and language settings:

- In Windows 10, move the pointer to the lower left corner of the desktop, then right-click and choose **Control Panel**. Click the **Clock and Region** option then click the **Region** link.

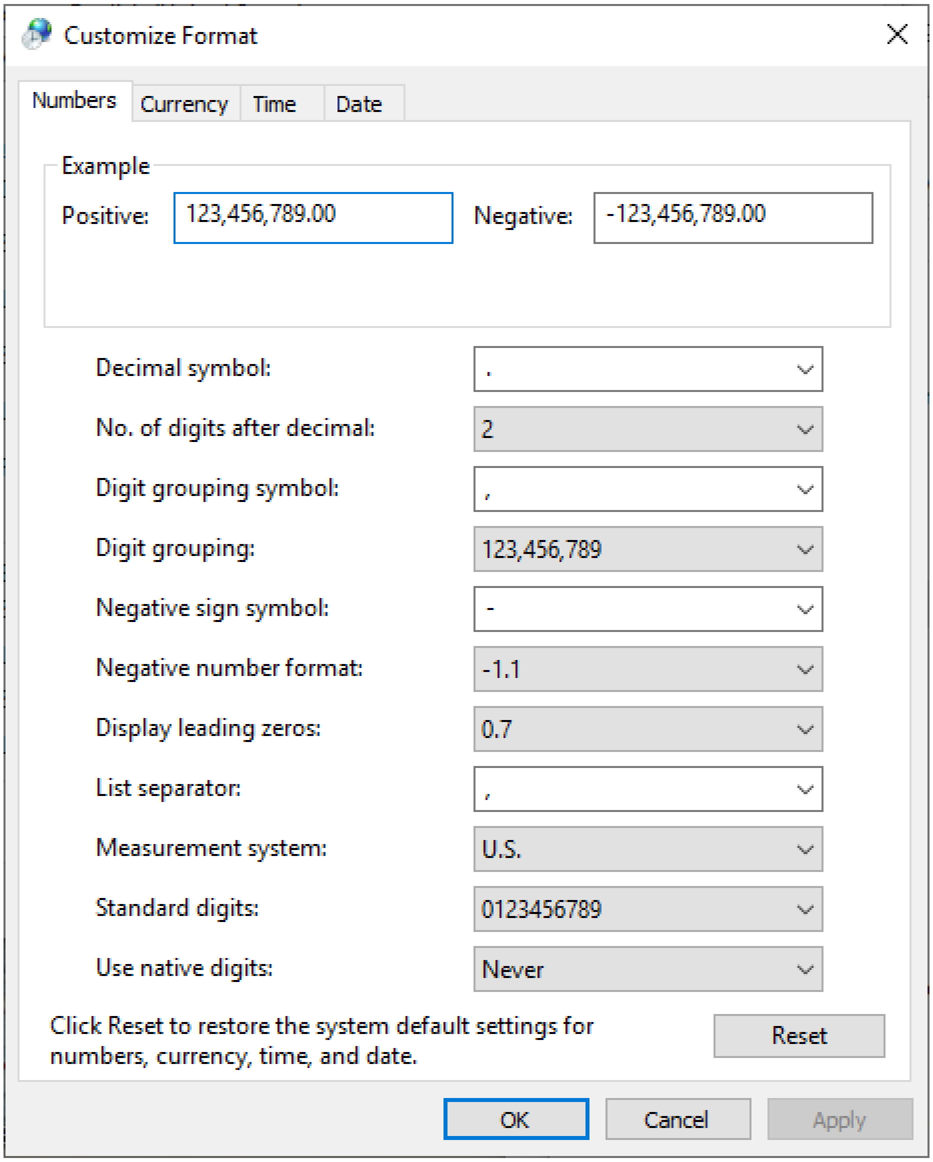
As an example, the Windows 10 interface is shown here.



Other relevant settings are managed via the Customize Format window:

- In Windows 10, click the **Additional settings** button.

As an example, the Windows 10 interface is shown next.



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

Which Language is Selected by Default When You Install the Software

The user interface for ReliaSoft desktop applications is available in several languages. You can change this language at any time by choosing an option from the **Language** drop-down list on the

Common Settings page of the Application Setup.

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 the ReliaSoft applications, they will use those settings; however, if that language is not supported, they will use the default setting of English.

Defining Date and Time Formats

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

- The dates in the plot legend area of a plot in all ReliaSoft applications.
- The history provided throughout ReliaSoft applications (e.g., for resources, FMEA records, diagrams, in history logs, etc.).
- The dates in the worksheet view and in the record properties windows in XFMEA/RCM++ and MPC.
- The dates in the "dates of failure" format and "usage" format of the Weibull++ warranty folios.

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

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

The Windows settings do not apply to the following items:

- Dates and times in spreadsheet utilities (e.g., a spreadsheet module in ReliaSoft 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.

Changing the Decimal Symbol, the List Separator and the Currency Symbol

How decimal values are displayed depends on the value of the **Decimal symbol** field on the Numbers page of 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 a spreadsheet module in ReliaSoft Workbooks or 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("Weibull!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("Weibull!Folio1!Data 1",500,100,0.9)).`

The currency symbol that is displayed in the software depends on the selection in the **Currency symbol** field on the Currency page of the Windows Customized Format window.

ReliaSoft Common Tools

IN THIS CHAPTER

Categories, Identifiers and Filters

Defining Categories

In all ReliaSoft desktop applications, you can use flexible *categories* and *identifiers* to filter and group analysis data in a way that fits your specific needs. There are two types:

- **Project Categories** can be applied to projects in all desktop applications.
- **Item Categories** can be applied to project items (e.g., folios, diagrams, system hierarchy items) and resources (e.g., models, URDs) in all desktop applications except MPC.

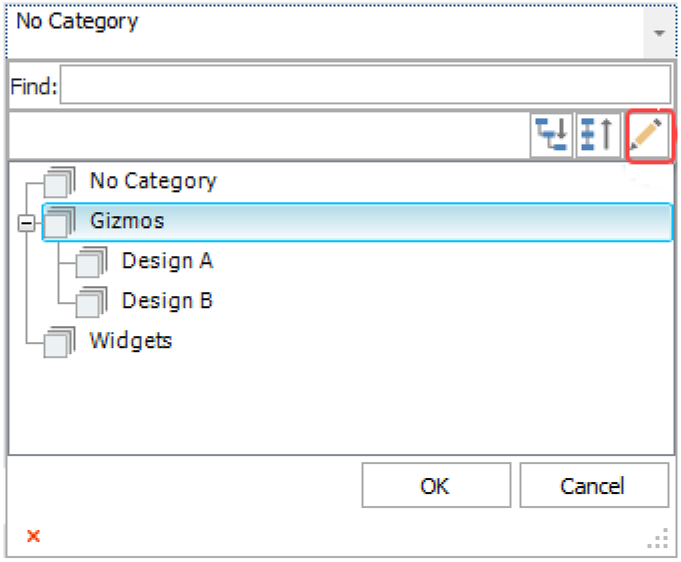
This topic describes how to define the categories that will be available throughout the current database. To learn how to use them to filter and group data, see Project and Item Filters. (For additional options related to categories for system hierarchy items, see Item Categories in XFMEA/RCM++ in the XFMEA/RCM++ documentation.)

Accessing the Project/Item Categories Window

There are two ways to access the Project/Item Categories window and specify which categories will be available throughout the current database. (In a secure database, these are available only for users with the "Manage project/item categories" permission.)

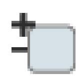
To open the window from the backstage view, choose **File > Manage Database > Project/Item Categories**.

Alternatively, you can click inside any **Category** drop-down list (e.g., in a properties window, filter, etc.) and click the **Edit Categories** icon.

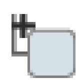


Adding, Renaming and Deleting Categories

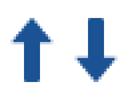
- To rename an existing category, double-click inside the cell and edit the text.
- To add a new category in the same level as the one that is currently selected, click **Add**.



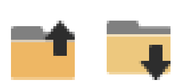
- To add a new category in the level below the one that is currently selected, click **Add Below**.



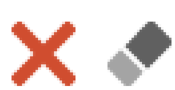
- To move a selected category up or down within the same level, select the row and click **Up** or **Down**.



- 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.***



Identifiers

In all ReliaSoft desktop applications, you can use flexible *identifiers* and *categories* to filter and group analysis data in a way that fits your specific needs. The *identifiers* are available for project explorer items (e.g., folios, diagrams, plots, etc.) and there is one standard set of identifier fields for all relevant locations.

Identifiers	
Category	Bulb A
Part Number	PN12345
Version	Rev 1
Supplier	Acme Chandelier
Application	Chandelier
Description	Light Bulb
Comments	Based on in-house test data.
Keywords	Bulb, Life, Test Data

This topic describes how to view and edit identifiers for each type of resource or analysis, and how existing identifiers will be converted from Version 8 or 9. To learn how to use identifiers to filter and group data, see Project and Item Filters.

For Resources

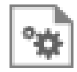
To view and edit the identifiers for most types of resources, you can simply open the properties window and select the Identifiers node. They can also be edited via the Batch Properties Editor. Note that:

- For published models, the identifiers are obtained from the analysis (e.g., Weibull++ folio, BlockSim diagram, etc.). To make a change, you must first edit the identifiers for the analysis and then republish the model.
- Identifiers are not applicable for the following resource types: Variables, Maintenance Groups, Mirror Groups, Actions, Controls and Profiles.

For Project Explorer Items

To view and edit the identifiers for folios, diagrams, plots and other analyses in Weibull++, BlockSim or Lambda Predict, select the item in the current project explorer and choose **Project >**

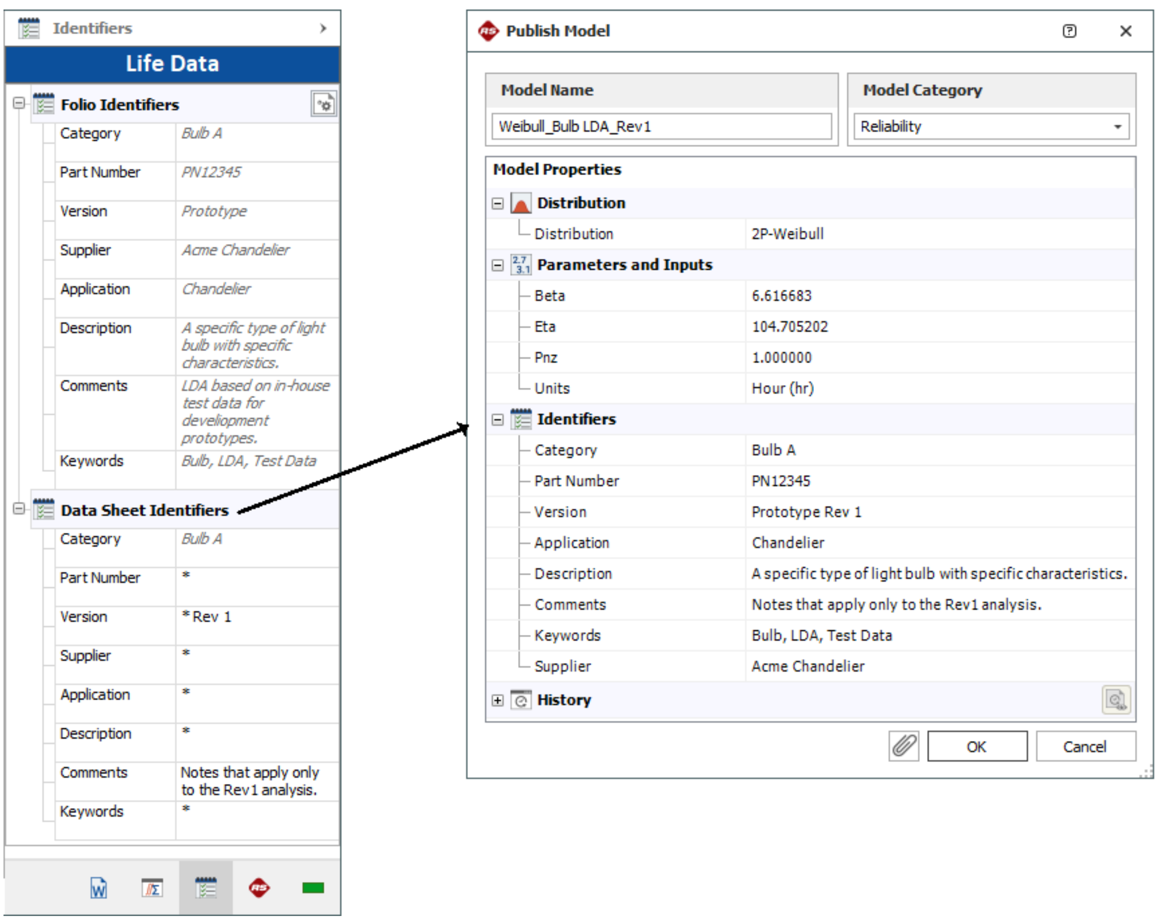
Current Item > Item Properties.



The identifiers are always displayed on the first tab of the properties window.

When applicable, the fields are also visible on the Identifiers page of the control panel. If a folio has multiple data sheets, you can use this page to define separate identifiers for each sheet. Note that the **Folio Identifiers** will be used in item filters and in the Analysis Explorer; while the Data Sheet Identifiers will be used when publishing a model.

You can use an asterisk (*) in any or all of the data sheet fields to apply the same text from the corresponding folio field. As an example, the following picture shows the identifiers for a Weibull++ life data folio that contains multiple data sheets for each design prototype of a new chandelier bulb in development. Although most of the data sheet identifiers will be the same as the folio (indicated with *), the Version and Comments fields have been modified for each data sheet. The data sheet identifiers will be used when you publish a model.

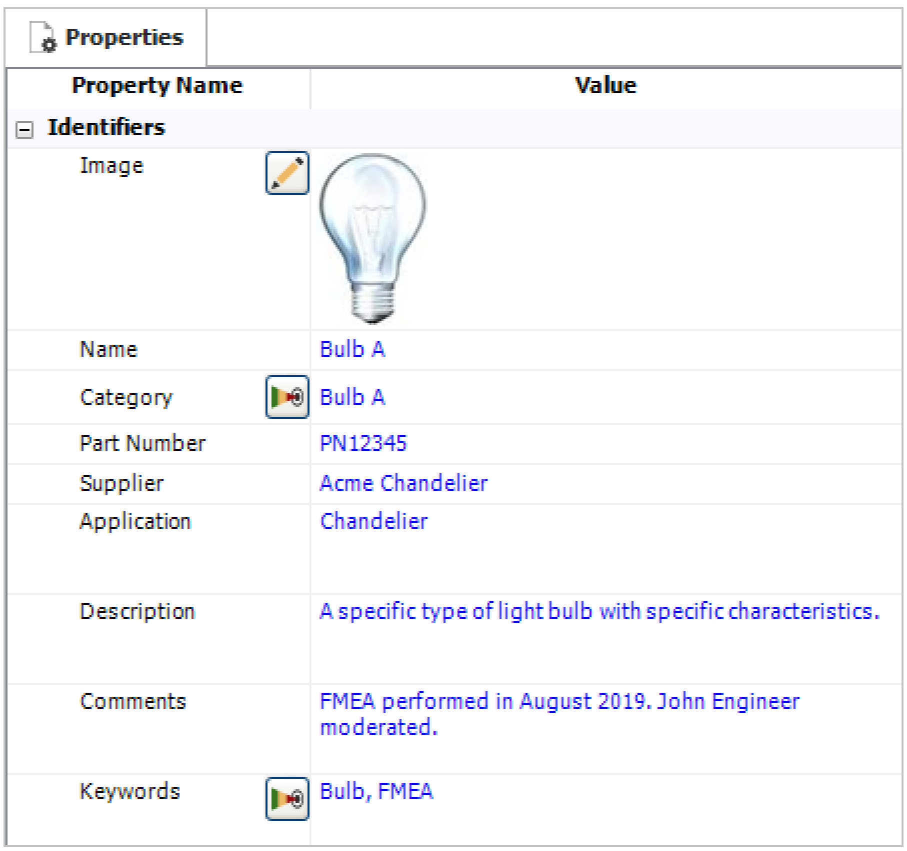


For System Hierarchy Items

To view and edit the identifiers in Lambda Predict and XFMEA/RCM++, select an item in the system hierarchy and go to the Properties tab in the Analysis panel.

The fields are grouped together under the Identifiers heading.

- In Lambda Predict, Name will always be displayed. The remaining identifiers may be hidden or displayed via the Application Setup. (See Properties Settings in the Lambda Predict documentation.)
- In XFMEA/RCM++, Name, Category and Keywords will always be displayed. The remaining identifiers may be hidden or renamed based on the configurable settings for the current project. (See About Configurable Settings in the XFMEA/RCM++ documentation.)

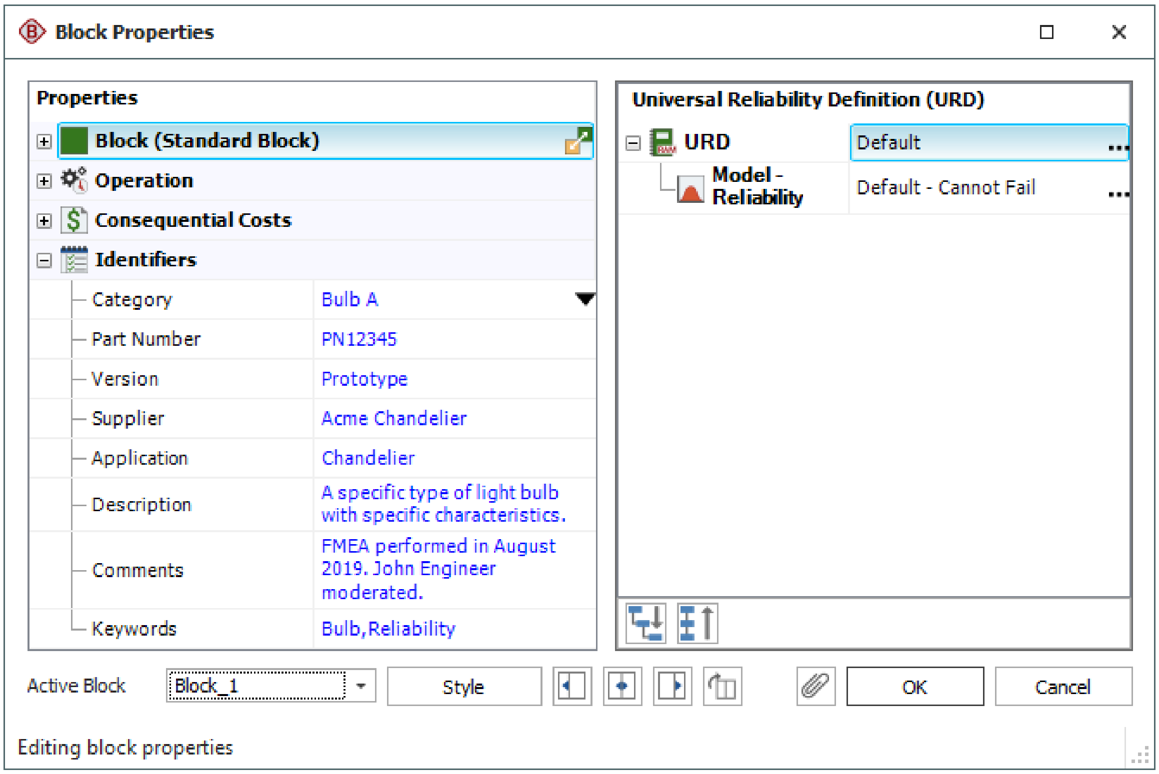


In XFMEA/RCM++, the same set of identifiers will be used for all analyses associated with a given system hierarchy item (e.g., FMEA, control plan, etc.).

For Diagram Blocks

To view and edit identifiers for diagram blocks in BlockSim, open the block properties and select the Identifiers node. The fields are also visible on the Identifiers page of the control panel when the block is selected.

You can use these identifiers in BlockSim's Find utility and in the Batch Properties Editor.

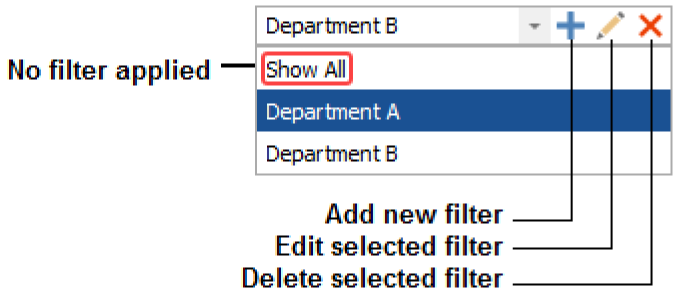


Conversion from Version 8/9

The following table shows how the "global identifiers" from previous versions will be mapped when you convert existing data in a Version 8/9 database (where " - " indicates no change).

Project and Item Filters

In many locations throughout all ReliaSoft desktop applications, you can use flexible filters to limit a list of projects, project items (e.g., folios, diagrams, system hierarchy items, etc.) or resources (e.g., models, URDs, etc.).



The drop-down list contains the filters you have saved in this database, plus any filters that another user has chosen to share with other database users.

Use the drop-down list to apply an existing filter. To stop filtering the list, select **Show All** (formerly called "Default filter").

Sharing Filters with Other Users

To share a saved filter, open the filter's properties window and select the **Show to all database users** check box.

In a secure database, a shared filter can be edited or deleted only by a) the user who created the filter or b) a user with the "Manage other database settings" permission.

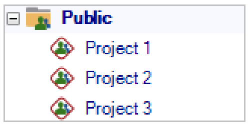
Project Filters

Project filters can be applied to lists that contain project names such as the Project Manager and Analysis Explorer. The filter can either be a query based on specified criteria, or a selected list of projects.

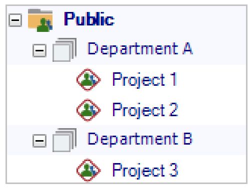
- **Based on Criteria**. You can match **All Criteria** (AND query) or **Any Criteria** (OR query):
  - Owner (project owner)
  - Category (project category)
  - Analysis/Feature (whether the project contains analyses associated with the specified application or feature)
  - Last Updated By
  - Last Updated Date
- **Based on Selected Projects**. Click the **Add** button to select the projects you want to include, or the **Remove** button to remove a selected project from view.

Clear the **Show locked projects** check box if you want to exclude locked projects from the results, even if they meet the criteria.

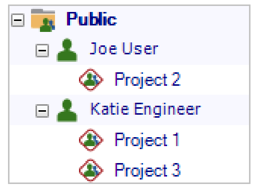
Projects are always grouped based on whether they are public, private or reference. For additional grouping, select the **Group by category** and/or **Group by owner** check boxes. If you select both, the results will be grouped first by project category and then by project owner. For example:



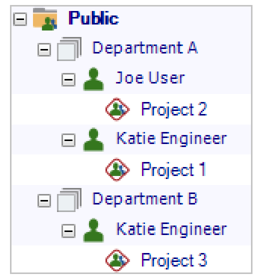
Not Grouped



Grouped by Category



Grouped by Owner



Grouped by Category and Owner

Item Filters

Item filters can be applied to lists that contain project items and resources such as the Current Project Explorer and Resource Manager.

Select whether the item must match **All Criteria** (AND query) or **Any Criteria** (OR query):

- Category (item category)
- Created By
- Last Updated By
- Last Updated Date
- Identifiers (Name, Part Number, Version, Application, Supplier, Description, Comments and Keywords)

Analysis Explorer

The Analysis Explorer is available in all ReliaSoft desktop applications except MPC. You can use this flexible tool to explore all of the different analyses that are stored in the current database.

What's Changed? In versions prior to Version 2020, the Analysis Explorer was called the Synthesis Explorer.

To access the utility, choose **Home > ReliaSoft > Analysis Explorer**.

To reduce the amount of time required to populate the grid, first use the project and item filters (discussed below) to limit your search, then click **Load Data**.

The analyses shown in the explorer may include:

- Project items in Weibull++, BlockSim and Lambda Predict (e.g., folios, diagrams, plots, ReliaSoft Workbooks, etc.).
- Analyses that are associated with system hierarchy items in XFMEA/RCM++ (e.g., FMEAs, control plans, risk based inspection analyses, etc.).
- Project Plans that can be created for any project.

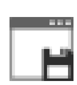
You can filter/sort/group the analyses based on ReliaSoft application, analysis type, analysis creator and many other properties. You can also present the information in a wide variety of dashboard charts.

This topic summarizes the tools you can use to find and organize data in the Analysis Explorer's flexible grid. For information about presenting the data graphically, see Analysis Explorer Dashboards. For a list of the properties that can be used in either the grid or the dashboard, see Analysis Explorer Properties.

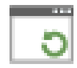
Save and Apply Views

Once you have customized the Analysis Explorer's grid to suit a particular purpose (using the built-in find/filter, column configuration and grouping features described below), you can save the preferences as a view that can be used again whenever you need it. A view is saved in the database and available only to the user who created it.

To create a view, first configure the grid to suit your particular needs and then click **Save View**.



To quickly apply these same preferences again at any time, click **Apply View** and select one of the saved views from the list.

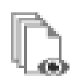


Note: The view does not affect filtering that has been applied from the Project Filter, Item Filter or Category Panel.

Project and Item Filters

The Analysis Explorer can utilize the same project and item filters that are available in many other locations throughout ReliaSoft desktop applications. For example, with the custom filters shown below, the Analysis Explorer will show only analyses performed by Department A (project filter) that were modified within the last month (item filter).

To remove either filter, select **Show All** from the drop-down list. To remove both filters at the same time, select **Clear Filters**.

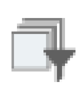


After you clear or change these filters, you must click **Load Data** again to update the grid.

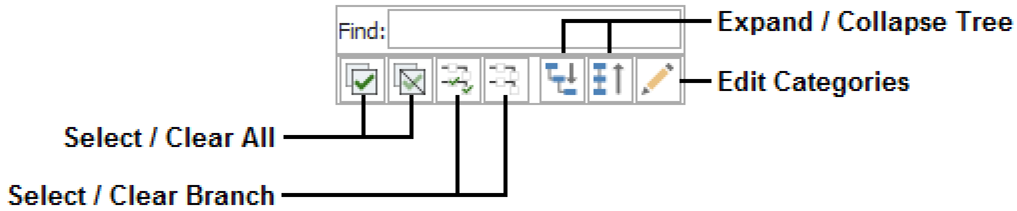
Category Panel

The category panel provides a quick way to filter the data based only on item category. For example, you might use the panel to first view the analyses for Category A and then quickly switch to see the analyses for Category B.

To show or hide the panel, click **Show Category Panel**.



When there are many categories, the tools at the top of the panel can help you find and select the one(s) you need.



Built-in Find/Filter, Configuration and Grouping Tools

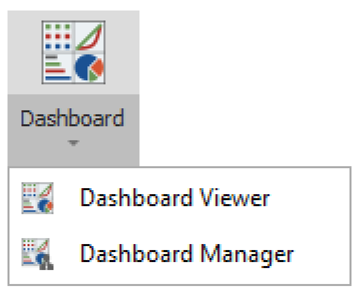
The Analysis Explorer offers the same filter, column configuration and grouping tools that are built in to other utilities that use a similar grid (e.g., the Resource Manager). For details about how to use each feature, see:

- Finding and Filtering Records
- Configuring Columns
- Grouping Panel

Analysis Explorer Dashboards

You can use the flexible Dashboard utility for presenting data from the Analysis Explorer.

As with any other dashboard, you can use the Dashboard Viewer to select any of the layouts that have been predefined for this type of data, and you can use the Dashboard Layout Manager to create or edit layouts.



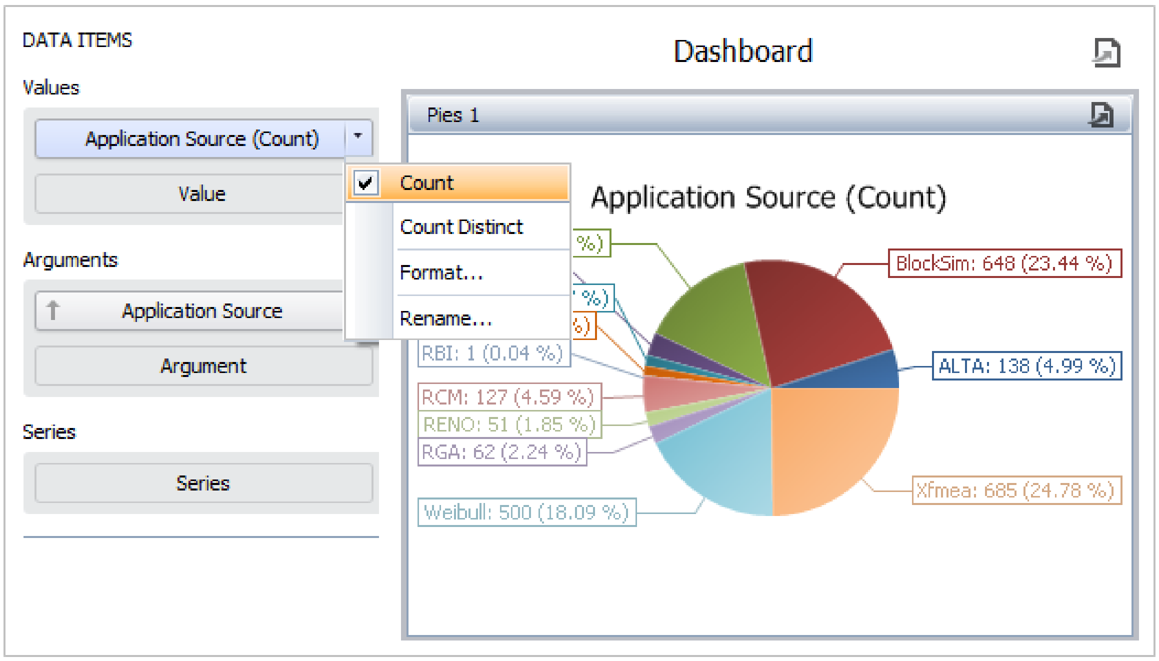
(In a secure database, the Dashboard Layout Manager is available only for users with the "Manage dashboard layouts" permission.)

Values and Arguments

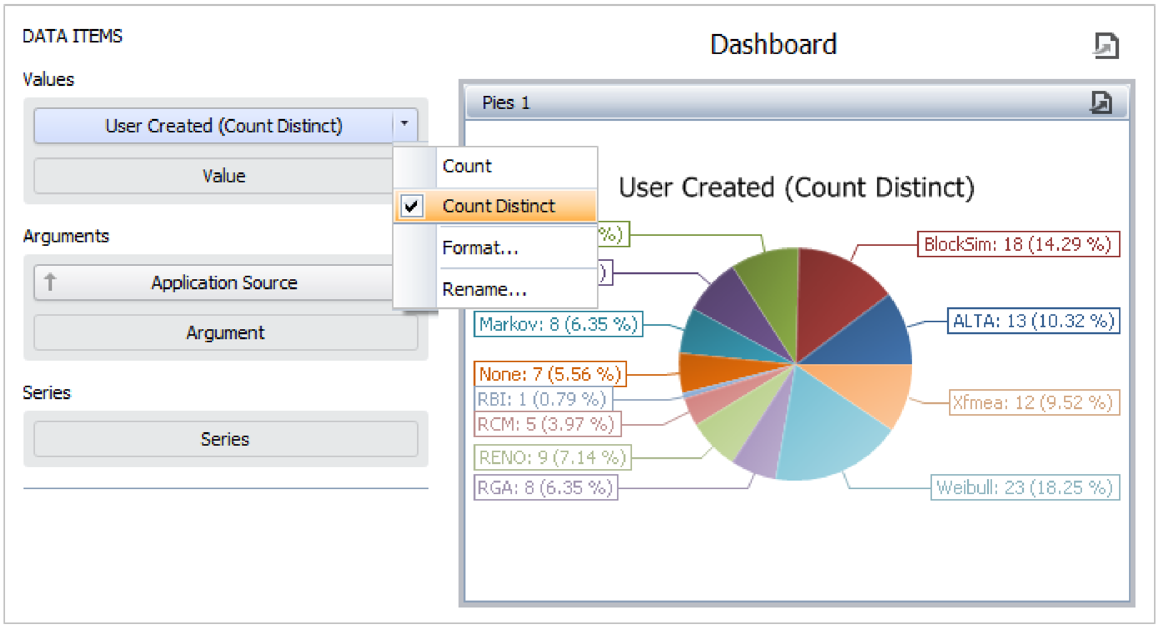
When you're creating bar charts or pie charts based on Analysis Explorer data, note that the *argument(s)* determine the bars or slices shown in the chart and the value will always be the quantity of analyses that fit the specified criteria.

If you use the default **Count** option when specifying the value, then any of the Analysis Explorer properties will return the same quantities. You may prefer to choose the one that gives the most appropriate chart label(s). For example, the following pie chart uses the same property ("Application Source") for both the **Value** and **Argument** in order to display the total quantity/percentage of analyses for each ReliaSoft application.

Tip: Use the **Design** tab of the ribbon to configure the chart labels and other settings to fit your particular preferences. For example, in this chart, the **Data Labels** are configured to display the Argument, Value and Percent (e.g., BlockSim: 648 (23.44%).



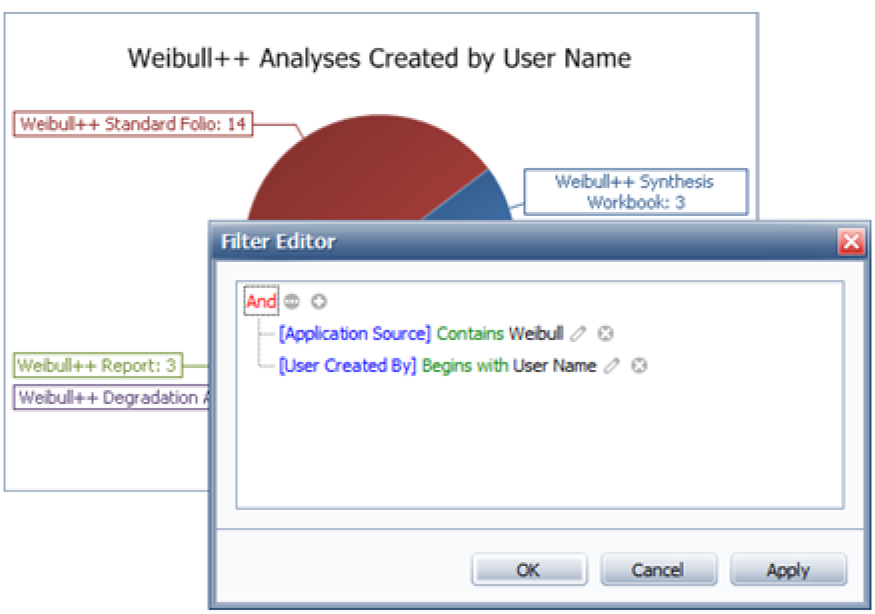
If there is ever a situation in which you want to show only the number of distinct values for a particular property, you can use the **Count Distinct** option instead. For example, the following pie chart uses **User Created By (Count Distinct)** for the value in order to show the number of users who created analyses with each ReliaSoft application.



As you can see from looking at both charts, the "Count" shows that there are 648 total BlockSim analyses; while "Count Distinct" shows that only 18 distinct users created BlockSim analyses.

Filtering the Data

Any filters that are currently applied in the Analysis Explorer grid will not be reflected in the dashboard charts. However, you can incorporate filters directly into the dashboard layout. For example, if you want to show all of the Weibull++ analyses that were created by a particular user, you can create a filter like the one shown next. (See Using the Filter Editor.)



In addition (or instead), you can configure the layout with one or more *master filters* that allow individual users to change the filters on-the-fly in the Dashboard Viewer. (See Configuring a Master Filter and Using Master Filters.)

Analysis Explorer Properties

The following properties will be available when you are using either the grid or dashboard in the Analysis Explorer.

Application

- **Application Source** is the primary ReliaSoft application (or product family) for viewing or editing the analysis. (Note that Project Planners and Simulation Worksheets are listed under ReliaSoft because they can be created/edited from more than one product family.)
- **Application Activated?** indicates whether the primary application is activated on your computer.

Project

- **Project Name** is the project in which the analysis is stored.
- **Project Type** indicates whether it is a public, private or reference project.

Analysis

- **Analysis Name** is the name of the folio, diagram, plot, system hierarchy item, etc.
- **Analysis Type** and **Variant** indicate the type of analysis. This can include project items (e.g., folios, diagrams, plots, etc.) and specific analyses that are associated with a system hierarchy item in XFMEA/RCM++ (e.g., FMEAs, risk based inspection analyses, etc.). The variant is used to distinguish:
  - Analytical vs. simulation RBDs and fault trees in BlockSim.
  - Discrete vs. continuous Markov diagrams in BlockSim.
  - The specific prediction standard in Lambda Predict (e.g., MIL-217, Telcordia, etc.). If the prediction folio uses more than one standard, this will show as "Multiple." If a standard has not yet been added to the prediction folio, the variant field will be blank.

Categories and Identifiers

Note that these properties are preceded with "SI" in the Dashboard Layout Designer, so that all identifiers can be grouped together in the data source panel.

- **Category** is the category assigned to the analysis.
- **Name**, **Part Number**, **Version**, **Supplier**, **Application**, **Description**, **Comments** and **Keywords** are the standard identifiers that can be defined for the analysis. Note that XFMEA/RCM++ analyses use the identifiers defined for the system hierarchy item they are associated with.

Creation and Last Update

- **User Created** and **Date Created** indicate the database user who originally created the analysis, and the associated date/time.
- **User Updated** and **Date Updated** indicate the database user who last modified the analysis, and the associated date/time.

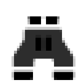
Record Lists

Finding and Filtering Records

This topic describes the find and filter tools that are built in to the Resource Manager and other utilities that use a similar grid. Each of these tools can be used separately, or in conjunction with the others.

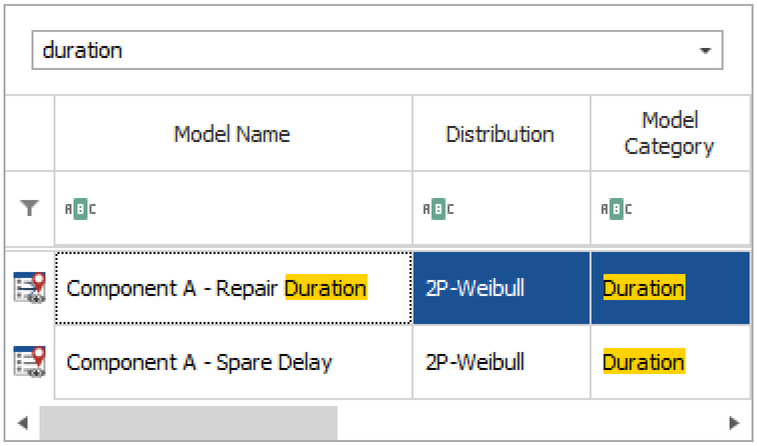
Find Panel

The Find panel allows you to search for a word or phrase across all of the available properties. To show or hide this panel, toggle the **Show Find Panel** command on the ribbon or in the shortcut menu that appears when you right-click a column heading.



When you enter text into the Find panel, it will return only the fields that contain the text you specified, as well as highlight all of the locations where the matching text occurs.

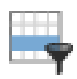
To stop filtering the data, clear the text box.



Auto Filter Row

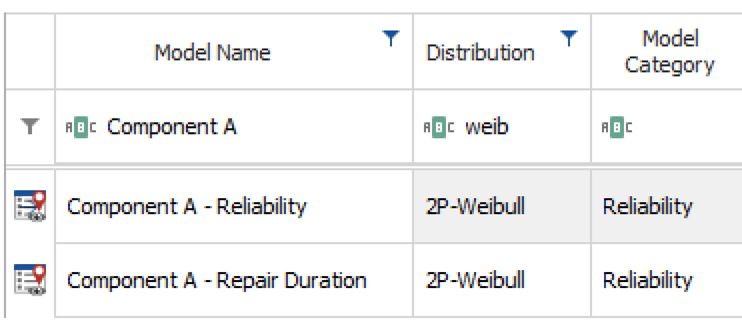
The Auto Filter Row allows you to filter the grid based on a selected filter operator.

To display or hide this row, toggle the **Show Auto Filter Row** command on the ribbon or in the shortcut menu that appears when you right-click a column heading.



Click the selector icon in the auto filer row and then enter the search text or value against which column will be filtered, as shown below. The set of filter operators available for a column depends on the type of data the column displays.

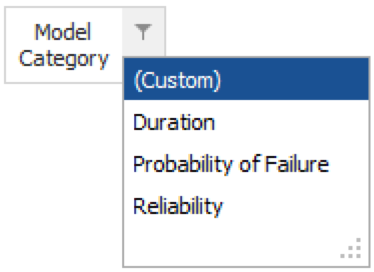
For example, the following filter shows only the rows in which the model name contains "Component A" and the distribution contains "Weib."



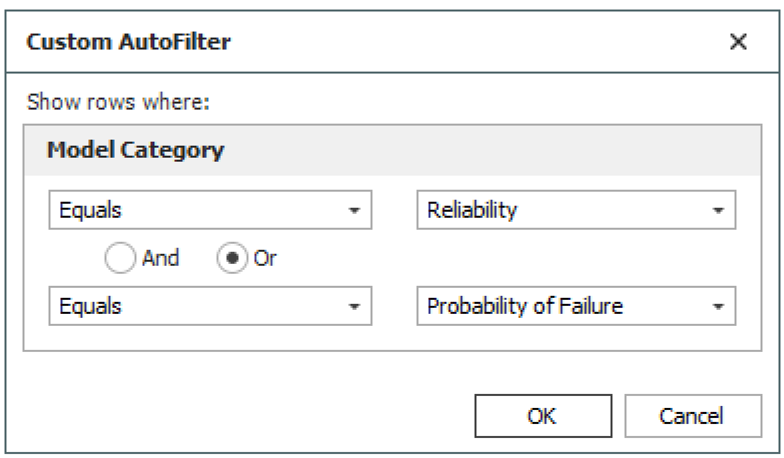
Advanced Filters

The Advanced Filters allow you to choose an automatically generated filter or create a quick custom filter. The automatically generated filters are based on the unique values that currently exist in the column.

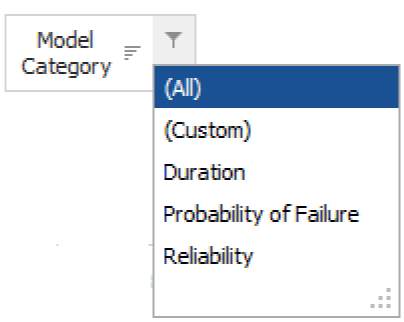
For example, the following picture is based on a data set that contains models from three different categories. If you choose one of the automatically generated filters (e.g., "Reliability"), the grid will display only models of that one selected category.



If you want to see models from two of those categories (e.g., either "Reliability" or "Probability of Failure"), you can choose **[Custom]** and create a quick custom filter like this:

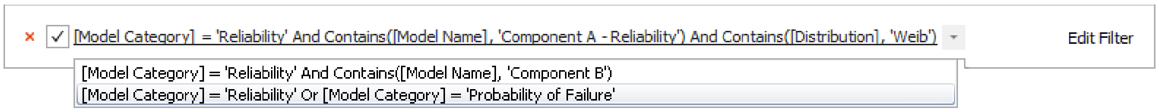


If a filter has been applied for a particular column, select **[All]** to remove it..



Current Query Criteria

When you are filtering the grid using the Auto Filter Row, Advanced Filters and/or Edit Filter window (described below), the current query criteria will be displayed at the bottom of the window. For example:

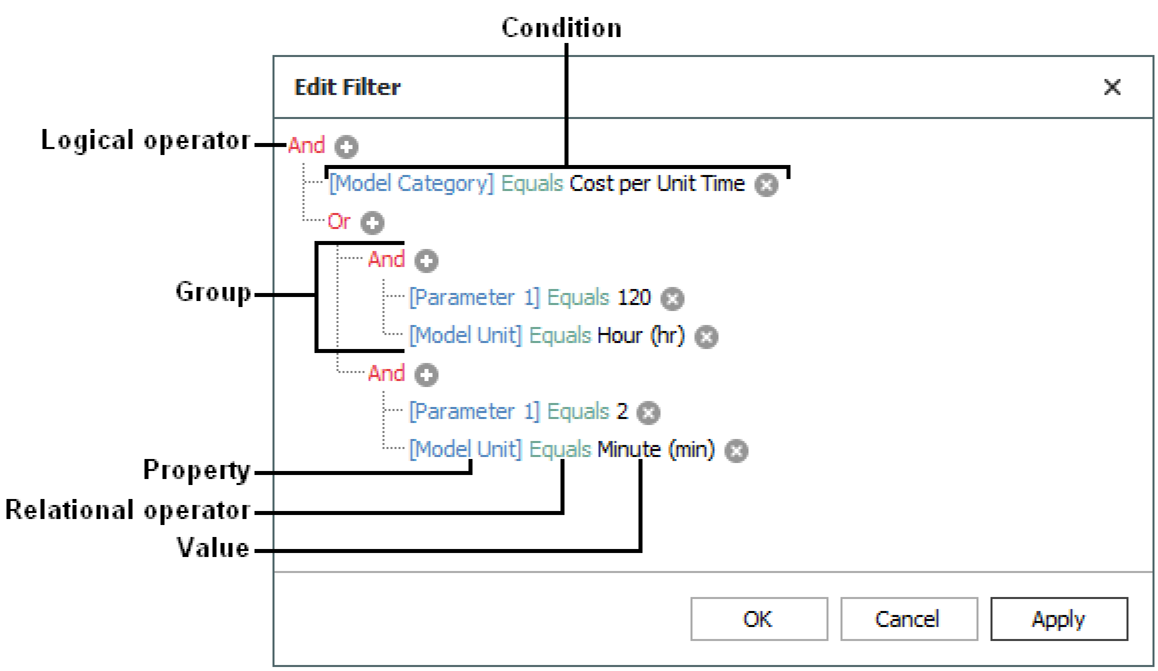


- Click the **X** to delete the current set of filter criteria.
- Clear the check box to remove the current set of criteria without deleting it. Select the check box if you want to re-apply it.
- Click to choose from a list of filter criteria that were recently applied.

Edit Filter

The Edit Filter window provides the most flexibility to refine and adjust a filter in either a grid or a Dashboard layout. To access the tool, click the **Edit Filter** button to the right of the current query criteria (if applicable), or choose **Edit Filter** from the shortcut menu.

As an example, the following filter from the Resource Manager will display only cost per unit time models in which the rate equals either $120 per hour or $2 per minute.



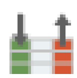
- Click a red label if you want to:
  - change the *logical operator* (And, Or, Not And, Not Or)
  - add or remove a *group*
  - add a *condition*
  - clear the entire filter
- Use the  and  icons to add or remove *conditions* within an existing group.
- Click a blue label if you want to change the *property* (e.g., Model Name, Created By, etc.).
- Click a green label if you want to change the *relational operator* (e.g., Equals, Contains, Begins with, etc.).
- Click a gray or black label if you want to enter or change the *value*.

Configuring Columns

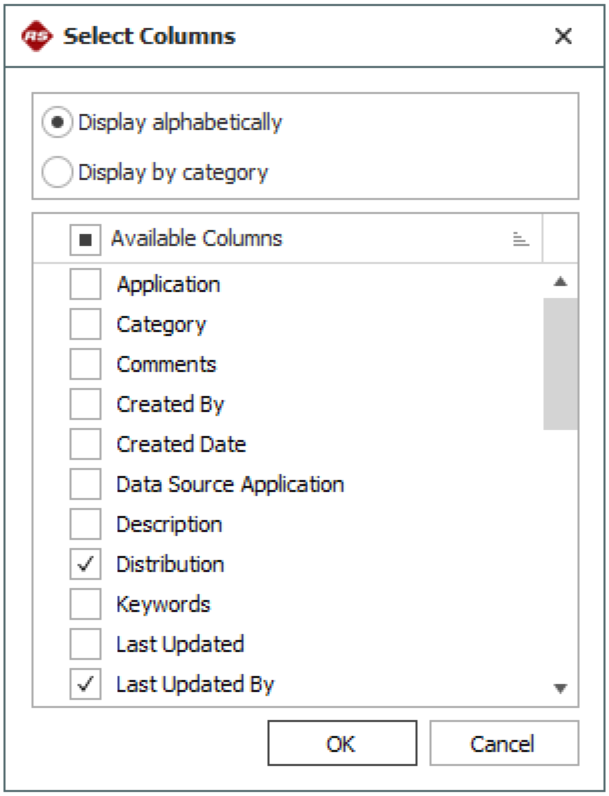
This topic describes the capabilities for selecting, reordering, resizing and sorting columns that are built in to the Resource Manager and other utilities that use a similar grid.

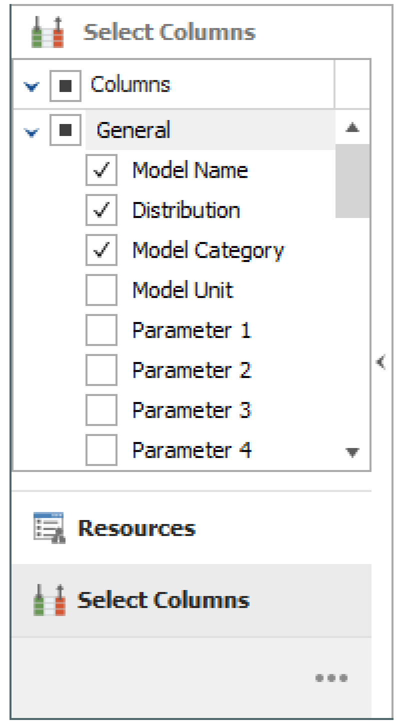
Selecting Columns

To specify which columns will be displayed in the grid, click **Select Columns**.



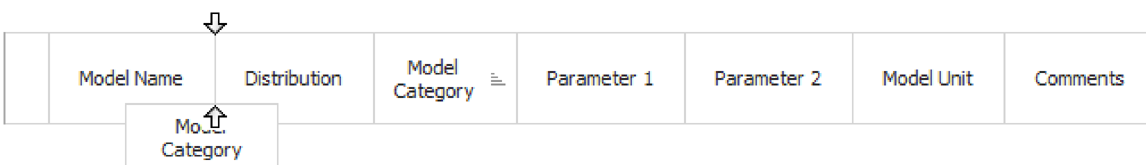
In the utility's window or panel for selecting columns, use the check boxes to choose which columns will be displayed. As an example, the following pictures show the window and panel for model resources in the Resource Manager.





Reordering Columns

To change the column order, drag the column heading into the desired position.



Resizing Columns

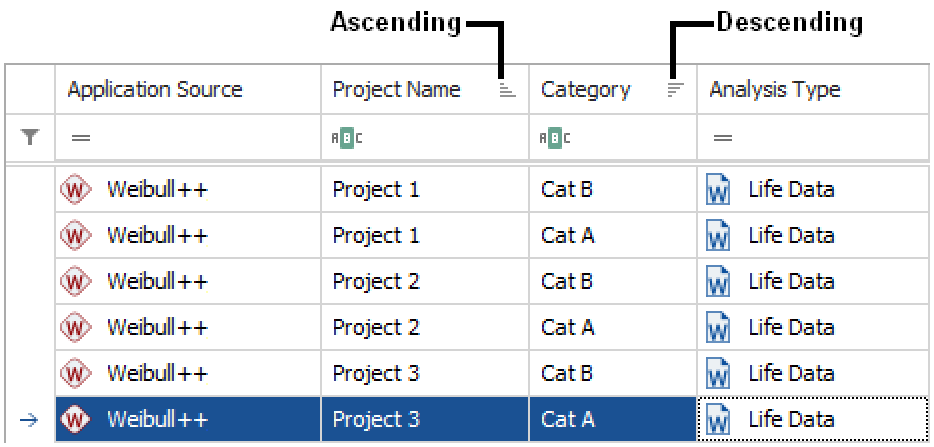
To resize columns, you can do any of the following:

- Drag the edge of the column heading to the desired position.
- Double-click the edge of the column heading or right-click it and choose **Fit This Column**. This will resize the column to fit its data.
- Right-click any column heading and choose **Fit All Columns**. This will resize all columns to fit their own data.

Sorting Columns

To sort the grid by one column at a time, you can simply click the column heading to sort in ascending order ( ), or click it again to sort in descending order ( ).

To sort the grid by multiple columns, right-click each column in the desired order and choose **Sort Ascending** or **Sort Descending** each time. For example, the following picture shows Analysis Explorer data sorted first by project (ascending) and then by category (descending).

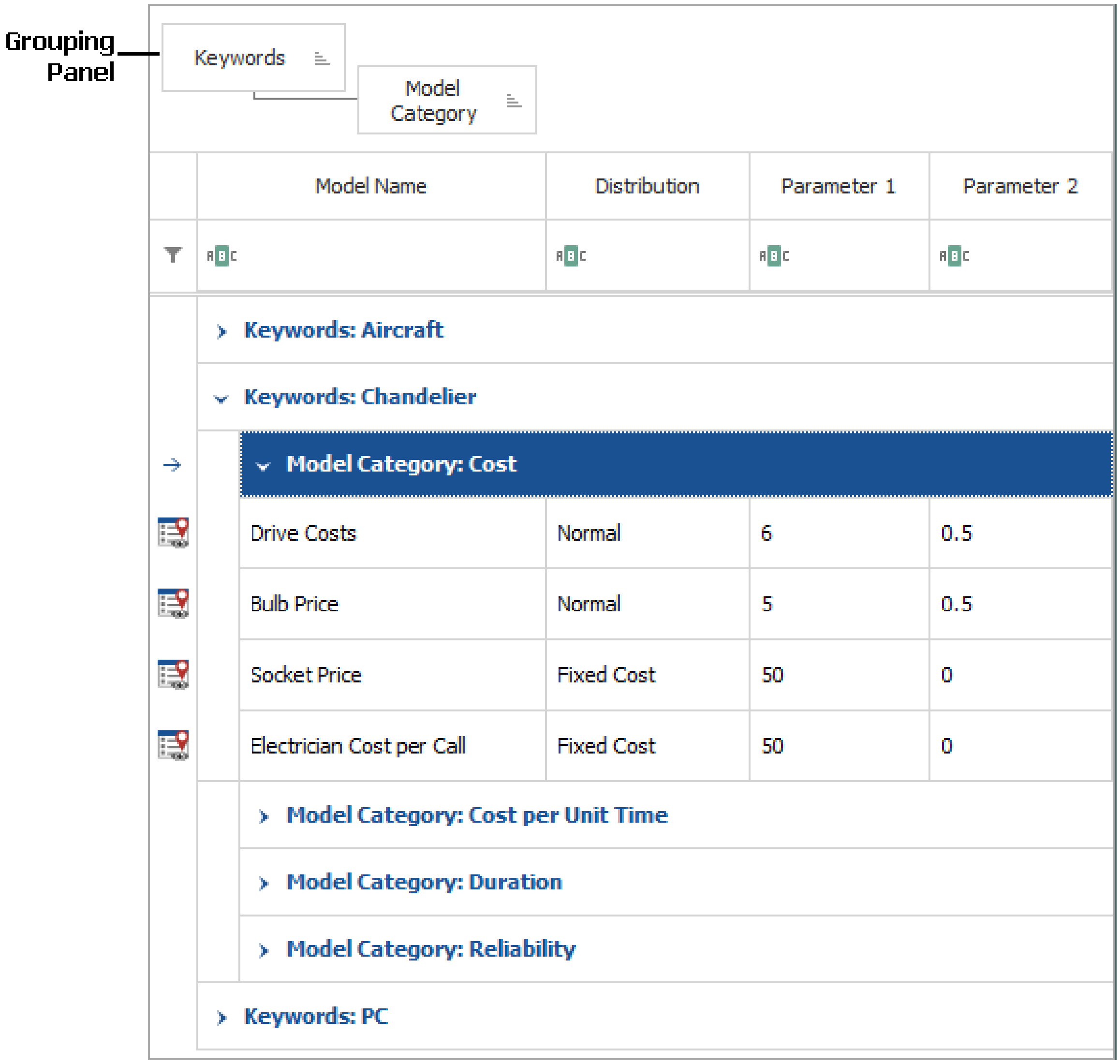


To stop sorting, right-click any column heading and choose **Clear All Sorting**.

Grouping Panel

This topic describes the grouping panel capability that is built in to the Resource Manager and other utilities that use a similar grid.

When you are working with a large amount of data, it may be convenient to group it based on one or more properties. For example, the following picture shows resources in the Resource Manager grouped first by keyword and then by model category.

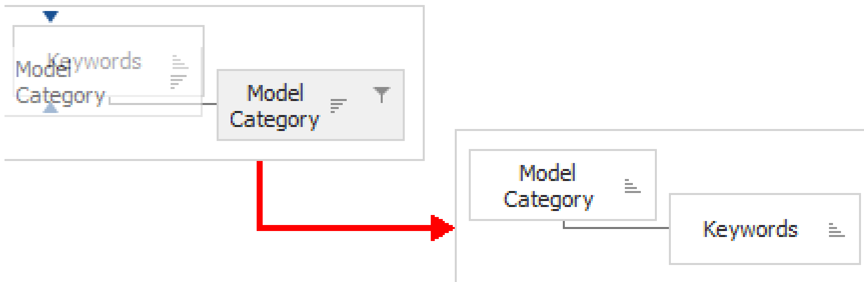


To group the data without opening the grouping panel, right-click any column heading and choose **Group By This Column**. Alternatively, you can choose **Show Grouping Panel** to display the panel and then drag column headings into or out of the panel to configure the groups.

When the grouping panel is displayed, you can also:

- Click the column heading to change the sort order between ascending ( ) or descending( ).

- Drag column headings within the panel to change the grouping order. For example, if you prefer to group first by model category and then by keyword, drag the column headings as shown below.



To stop grouping the data, drag all column headings out of the panel or choose **Clear All Grouping**.

SEP Web Portal

If your organization implements an SEP web portal for an enterprise database, the entire team — including managers and colleagues who don't have ReliaSoft desktop applications installed — can access key analysis and project details from any web-enabled device.

SEP Dashboards

The personalized home page provides an intuitive, at-a-glance overview of the information you're tracking via SEP, such as FMEA stats, actions, recent messages, reports, metrics and more. You can choose which tiles appear on your dashboard and change their order and settings.

Actions and Portal Messages

You can use SEP to create, view and edit the same actions and portal messages that are visible to you from the desktop applications. SEP also allows colleagues who don't use the desktop applications to stay up-to-date on assignments and team communication.)

Project Summary, Analysis Summaries and Project Plan

For each analysis project that you have permission to view, SEP provides a summary that includes metrics, assigned actions, attachments and other project details. It also displays summaries, plots and reports for selected analyses published to SEP from Weibull++, BlockSim and Lambda Predict.

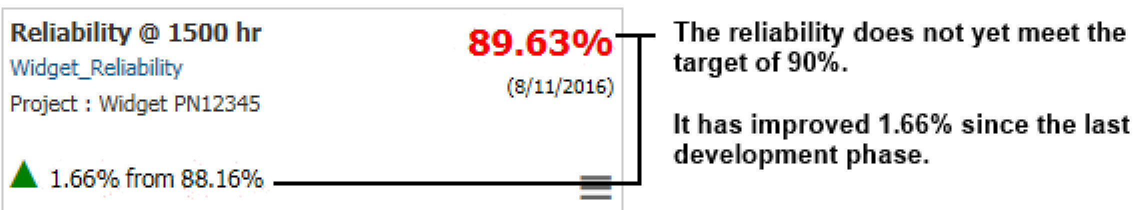
If your team is using the Project Planner in desktop applications, SEP shows a streamlined view of the project plan. Users can access this plan in SEP and update their progress for assigned actions.

System Hierarchies and FMEAs

SEP enables users throughout your organization to view FMEAs, queries and reports that were created in XFMEA/RCM++. This provides managers and others throughout your organization with convenient web-based access to the wealth of lessons learned and troubleshooting recommendations from your investment in these analyses.

Metrics (Key Performance Indicators)

SEP shows all of the KPIs (metric resources) that have been created in desktop applications for the projects you have permission to view; no publishing is required. Quick visual indicators help you monitor performance and support decision making.



Monitored Reports

SEP can display a variety of custom reports and dashboards created in desktop applications. The "Watch" feature makes it easy to manage the specific reports you want to access quickly.

SEP also shows all of the Analysis Explorer and Reliability Data Warehouse (RDW) dashboards that have been predefined in the desktop applications.

Publishing to SEP

SEP does not display all of the analyses that were performed in ReliaSoft desktop applications. Instead, your team can decide which analyses to make available and publish selected analysis summaries and reports.

In the desktop applications, these features are only available when:

- You are connected to an enterprise database that has an SEP web portal implemented and the "Enable publish to SEP web portal" option is set to "Yes" for the database.
- You have the "Publish to SEP web portal" permission.

The types of analysis summaries and reports that can be published vary for each ReliaSoft desktop application. In addition, some types of Weibull++ and BlockSim analyses/reports may not be available, depending on your organization's licensing.

Weibull++

Life Data

Non-Parametric LDA

Life-Stress Data

Weibull Degradation

ALTA Degradation

Parametric RDA

Standard / Robust / Mixture / Free Form Design

One Way ANOVA

Linearity & Bias, Gage R&R, Gage Agreement

Multiple Linear Regression

Growth Data

ReliaSoft Workbooks

XFMEA/RCM++

Reports (saved with the project)

System hierarchy and FMEAs are also visible in SEP — no publishing is required.

BlockSim

Analytical RBD, Simulation RBD

Analytical Fault Tree, Simulation RBD

Phase Diagram

Event Analysis Flowchart

Continuous Markov, Discrete Markov

ReliaSoft Workbooks

Lambda Predict

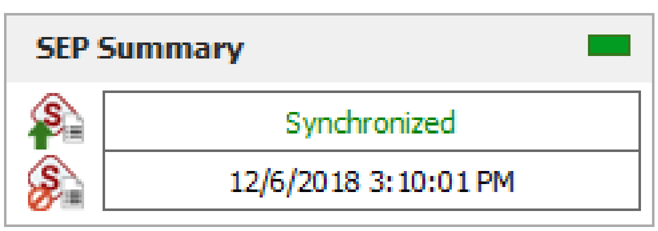
Reports

Stored Plots

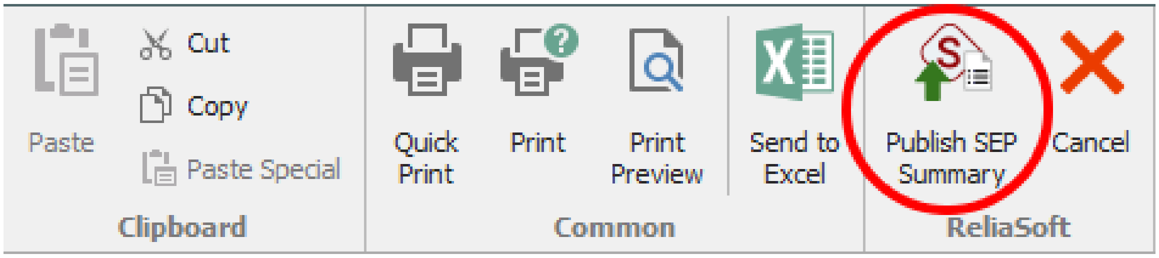
Note: If you have published analysis summaries and you see a "Link is not properly formatted" message when you attempt to open the original analysis from SEP, the link may be corrupted or using old encryption. Republish the analysis to refresh the link.

Data Sheets or Diagrams

To publish an analysis summary from Weibull++ or BlockSim, open the data sheet or diagram and click **Publish SEP Summary** on the Publishing page of the control panel. (For DOE design folios, the **Data** tab with analyzed response data must be selected.)



To publish a data sheet, you must also click **Publish SEP Summary** again in the preview of the spreadsheet summary.



After publishing, the panel displays "Synchronized" if the web version reflects the latest results from the analysis. If the analysis has been modified (e.g., if more data has been added, an analysis setting has changed, etc.), it displays "Out of Sync" and you will need to recalculate and republish to update the web portal.

To remove the summary from the web portal, click **Remove SEP Summary**.

ReliaSoft Workbooks

To publish a ReliaSoft Workbook from Weibull++ or BlockSim, open the module in the workbook that you want to share (spreadsheet or word processing document) and then click **Home > SEP > Publish Report**.

The status bar shows that the spreadsheet or document is visible via SEP and indicates when the web version was last updated.

If you subsequently make a change in the desktop application, you will need to republish to update the web portal.

If you want to remove the report from the web portal, open the module and click **Home > SEP > Remove Report**.

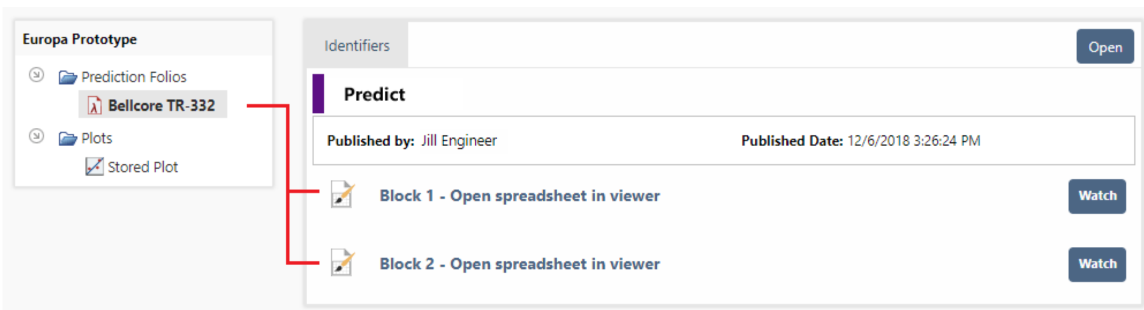
Prediction Reports and Plots

To publish a report for a standard item or block in Lambda Predict, select the item/block and choose Prediction Tools > Share > Publish SEP Summary.

In the Select Report window, select what to include in the report (e.g., template-based reports, default reports or plots) and click **OK**. (See Lambda Predict Reports in the Lambda Predict documentation.) There can be only one published report for each item/block. If you publish again for the same item, it will replace the report that was previously published.

To publish a stored plot from Lambda Predict, open the plot, then click the **Publish SEP Summary** icon on the **Publishing** page of the plot control panel.

The following example shows two published reports and a published plot.

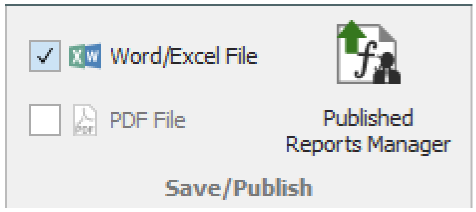


To remove a report from the web portal, select the standard item or block, then open the Publishing tab of the Properties panel and click **Remove SEP Summary**.

To remove a plot from the web portal, open the plot, then click the **Remove SEP Summary** icon on the Publishing page of the plot control panel.

XFMEA/RCM++ Reports

To publish a report generated in XFMEA/RCM++, use the Reports window to build the document, select at least one of the **Save/Publish** file formats on the ribbon (Word, Excel, or PDF) and click **Generate Report**.



To see all of the generated report documents that have been saved with the project (and therefore will be visible via SEP), click Published Reports Manager. You can also use the manager to edit the name and description that will show in the web portal.

For more information, see Save/Publish Reports in the XFMEA/RCM++ documentation.

ReliaSoft Locator Links

ReliaSoft Locator Links provide quick access to specific analyses in a database (similar to Windows shortcuts). You can save these link files anywhere on your computer or network, e-mail to a colleague, post on the Internet or an intranet, etc.

When you double-click (open) a ReliaSoft Locator Link file (*.rsllx), it will launch the appropriate ReliaSoft application and go directly to a specific project item (e.g., folio, diagram, plot, system hierarchy item, etc.) or FMEA record (e.g., function, failure, etc.), as long as the following conditions are met:

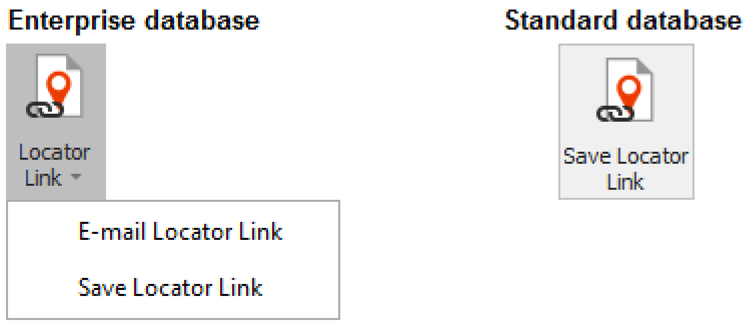
- The database still resides in the same location and you have access to the server.
- You have an active user account in the database.
- The required ReliaSoft application is activated on your computer.

Tip: In the case of a standard database, the locator link stores the pathname/filename that was open when the *.rsllx file was created. If that was a mapped network drive (e.g., P:\ReliaSoft\Database1.rsr21), the link will only work if all other users have that drive mapped to the same letter. To make sure the link works for all users, you may need to open the database from the UNC pathname (e.g., \\SharedDrive\ReliaSoft\Database1.rsr) before creating a locator link file.

ReliaSoft Locator Links created in Version 2019 or later do not include the software version information as part of the link. When you open a link, the software will attempt to connect to the database, regardless of the database's version. If you attempt to open a standard database that was created in an earlier version, the software will automatically convert it to the latest version. If you attempt to open an enterprise database that was created in an earlier version, the software will display a message; you must then upgrade the enterprise database to the later version.

Creating a Locator Link

In an enterprise database, you can choose to either send the file by e-mail or save it to a specified location. In a standard database, the only option is to save the file because it is likely that the path-name may only work on your own computer (as discussed above).



- For an item selected in a current project explorer (Weibull++, BlockSim or Lambda Predict), choose **Project > Current Item > Save Locator Link**.
- For an item selected in an XFMEA/RCM++ system hierarchy, choose **System Hierarchy > Current Item > Save Locator Link**.
- For an item, structure or zone selected in an MPC hierarchy, choose **Tools > Save Locator Link** on the relevant ribbon tab.

Note: If you get a "Locator Link is not properly formatted" message, the locator link may be corrupted or is using old encryption. Create a new locator link using the methods described above.

Posting Locator Links on a Web Page

Remember that ReliaSoft Locator Links are files that must be opened from a computer where ReliaSoft desktop applications are installed. If you post the *.rsllx as a standard link on an HTML web page, you will need to instruct users to download the file (instead of opening directly from the web browser).

What's Changed? In prior versions, ReliaSoft Locator Links were version-specific and were not compatible with other versions.

Watches and Alerts

Automated alerts can be an effective tool to facilitate communication and track the status of assigned responsibilities.

Alerts can be sent via e-mail, SMS text message and/or ReliaSoft portal messages, depending on the **Receive automated alerts** preference specified for your personal user account.

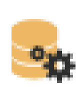
- Alerts via portal message are always available in any database.
- Alerts via e-mail or SMS are only available if they are enabled for the database, and a valid SMTP server has been defined. (See Enable Alerts via E-mail or SMS.)

Alerts can be sent for:

- Any resource that you have personally subscribed to "watch." (See Subscribing to a Watch.)
- Action resources, under any of the following conditions:
  - You have personally subscribed to watch the action.
  - You are the action's creator, person responsible or assigned reviewer/approver and the database has been configured to auto-subscribe a watch for that role. (See Action Alert Preferences.)
  - You are assigned via the Action Monitors window. (See Action and Gate Monitors.)
- Project Planner gates, under either of the following conditions:
  - You have personally subscribed to watch the gate.
  - You are assigned via the Gate Monitors window.
- Change logs you have been assigned to approve in XFMEA/RCM++. (See Electronic Approval Tracking in the XFMEA/RCM++ documentation.)

Enable Alerts via E-mail or SMS

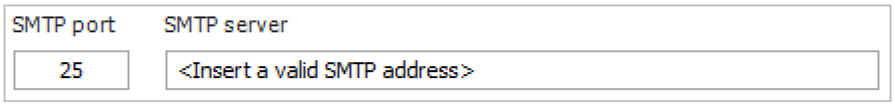
To configure a database to enable alerts via e-mail and/or SMS text message, choose **File > Manage Database > Database Settings**.



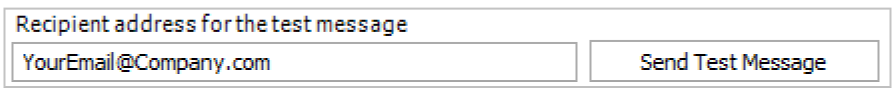
In a secure database, this is available only for users with the "Manage other database settings" permission.

What's Changed? In prior versions, similar preferences were defined in the E-mail and Other Settings window.

1. Select the **Enable Alerts via E-mail** or **SMS** 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 (or a valid SMS text messaging address if you prefer) in the Recipient address for test message field and then click **Send Test Message**.



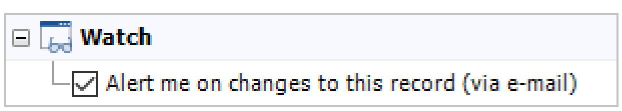
If the test message cannot be sent, an error will be displayed. If that happens, you can update the settings and try again until the message is delivered successfully.

Subscribing to a Watch

Users can personally subscribe to "watch" specific resources, Project Planner gates and MPC tasks that are of particular interest to them. This generates an alert (via e-mail, SMS text message and/or ReliaSoft portal message) each time the record is changed.

Subscribing and Unsubscribing

To personally subscribe/unsubscribe to a watch, open the resource, Project Planner gate or MPC task and select or clear the **Alert me on changes to this record** check box. The alert types shown for this option will depend on your alert preferences, as described in the section below.



Note that the "watch" feature is not applicable for the following types of resources: Variables, Profiles, Maintenance Groups, Mirror Groups, Controls, Maintenance Templates (in BlockSim) or resources specific to BlockSim's event analysis flowcharts (Functions, Static Functions, Simulation Definitions and Tables).

Receiving Alerts

Alerts can be sent via e-mail, SMS text message and/or ReliaSoft portal message, depending on the **Receive automated alerts** preference specified in your personal user account.

- Alerts via portal message are always available in any database.
- Alerts via e-mail or SMS are only available if they are enabled for the database, and a valid SMTP server has been defined. (See Enable Alerts via E-mail or SMS.)

No Cascading Alerts

Alerts are generated only when the record you're watching is edited directly. For example, if you are watching a universal reliability definition (URD):

- An alert will be sent if someone replaces Model A with Model B in the URD window.
- An alert will not be sent if there is a change to an existing model, crew, etc. that is already assigned to the URD, unless you have also subscribed to watch the dependent resource that was changed.

Special Considerations for Actions and Gates

For action resources only, there are some additional features designed to maintain continuity with prior versions and enhance the actions management functionality. Specifically:

- Action changes that are applied via the FMEA worksheet in XFMEA/RCM++ will not initiate an alert. The change must be saved via the action properties window.
- The database can be configured to auto-subscribe watches for users with the following roles: **Action Creator**, **Person Responsible** or **Reviewer** (aka "approver"). Each user can choose to personally unsubscribe later if desired. (See Action Alert Preferences.)
- The Action Monitors window allows you to specify individual users and/or groups of users who will always receive alerts for the action, regardless of whether they have personally subscribed to a watch. For example, this allows you to set up an alert for a user who doesn't have access to view or modify the project (and therefore can't personally subscribe to a watch).

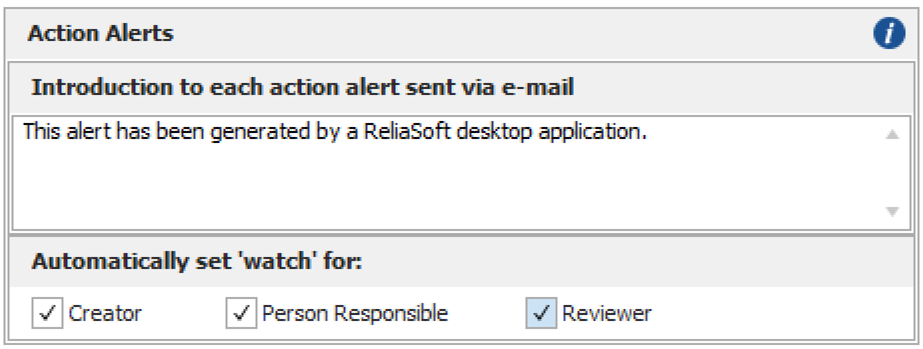
  A similar feature, the Gate Monitors window, is available for Project Planner gates. (See Action and Gate Monitors.)

Action Alert Preferences

To configure the action alert preferences for the database, choose **File > Manage Database > Database Settings**.

In a secure database, this is available only for users with the "Manage other database settings" permission.

What's Changed? In prior versions, similar preferences were defined in the E-mail and Other Settings window.



- **Introduction to each action alert sent via e-mail**. This field allows you to specify the default text that will be used at the beginning of each action alert e-mail sent from this database.
- **Automatically set 'watch' for**. Users can personally subscribe to "watch" specific resources and Project Planner gates that are of particular interest to them. For action resources only, the database can also be configured to auto-subscribe watches for users that have particular roles in a given action, as shown in the picture above. Note that each user can choose to personally unsubscribe later if desired (by opening the action record and clearing the check box under the **Watch** heading).

Action and Gate Monitors

The Action Monitors window (for action resources) and the Gate Monitors heading (for Project Planner gates) allow you to specify individual users and/or groups of users who will always receive alerts for the record, regardless of whether they have personally subscribed to a "watch."

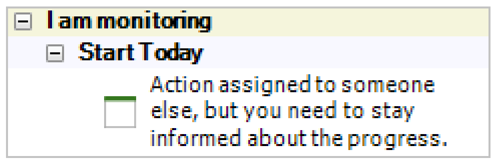
For example, this allows you to set up an alert for a user who doesn't have access to view or modify the project (and therefore can't personally subscribe to a watch).

Action Monitors

For actions, click the **Action Monitors** icon in the action's ribbon.

This replaces the "E-mail Notifications" feature from prior versions, and you can choose any of the user groups or individual user accounts defined in the current database.

The action will also appear under the **I am monitoring** heading in My Portal, Actions Explorer and SEP web portal (along with the actions that the user has personally subscribed to watch).



Gate Monitors

For Project Planner gates, the assigned users are listed under the **Gate Monitors** heading. Click the **Edit** icon to select the users you wish to assign.

This replaces the "Team" feature from XFMEA/RCM++/RBI Version 9, and you can choose to assign any of the individual user accounts in the database.

What is Your SMS Address?

If you choose to receive alerts via SMS text messaging, the address will usually be your cell phone number @ the provider's e-mail domain. As a courtesy, the table below provides a list of domains for popular providers in the United States (as of November 2020). Of course, you may need to check with your provider to obtain the most up-to-date information.

For example, if you use T-Mobile in the US and your cell number is 555-123-4567, your SMS address will be 5551234567@tmomail.net.

Provider	Domain
Alltel	@message.alltel.com
Boost	@myboostmobile.com
AT&T	@txt.att.net
Metrocall	@page.metrocall.com
Nextel	@mes-saging.nextel.com
Sprint	@mes-saging.sprintpcs.com
T-Mobile	@tmomail.net
US Cellular	@email.uscc.net
Verizon	@vtext.com
Virgin (USA)	@vmobl.com

History Logs

By default, all projects and project items automatically show information about when an item was created and last updated, and by whom. For example, system hierarchy items in XFMEA/RCM++ show this information under the History heading on the Properties tab, while folios and diagrams in Weibull++ and BlockSim show the information in the Item Properties window (**Project > Current Item > Item Properties**).

In addition to the basic information provided, you have the option to activate a full history log that shows more detailed information such as the type of change that was made (e.g., add, edit or delete), the record that was changed and the specific property, and the value before and after the change.

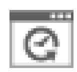
History logs will increase the size of a database and will cause a slight degradation in performance; hence, they must be activated on a per project basis. If desired, you can configure the database to automatically activate history logs for each new project by choosing **File > Manage Database > Database Settings**, and then selecting the option under the Other Settings tab in the window. (In secure databases, this is available only if the user has the "Manage other database settings" permission.)

Once a history log is activated, the *project history log* shows the changes for all items in the project, and individual *record history logs* show the changes for selected items only.

Tip: If you want to monitor and control revisions made to FMEAs, DVP&Rs, control plans, P-Diagrams or test plans in XFMEA/RCM++, the use of change logs may be more appropriate. See Change Logs in the XFMEA/RCM++ documentation.

Activating a Project History Log

To activate the history log for a project, choose **Project > Management > History Log**, or right click the project in the project list and choose the command on the shortcut menu. (In secure databases, this is available only if the user is the project owner, or has the applicable "manage all projects" permissions.)



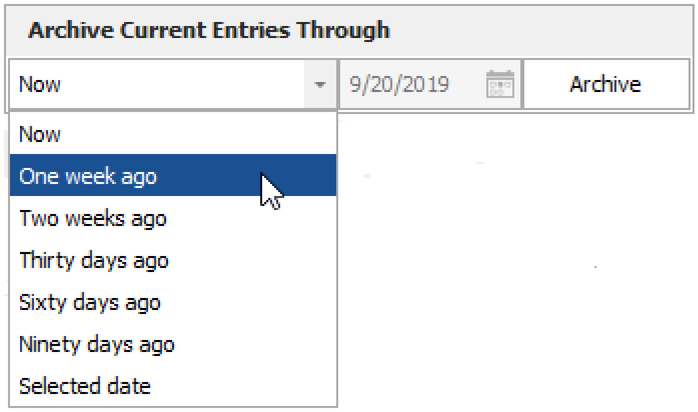
In the Project History Log window that appears, click **Activate**.

Once the log has been activated, the database will start keeping records of all changes that are performed throughout the entire project for any of the ReliaSoft applications. You can filter the records by the last update date, application, user or type of change. 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.

You can deactivate the history log at any time from this window. When you do this, the recorded data is retained unless you select to clear it in the confirmation message that appears.

Creating and Managing Archives

You can limit the amount of information that is visible in the log by archiving older entries that are no longer of interest. Entries that have been archived can still be viewed in the Project History Log window if you specifically select to view them, but they will no longer be visible in the Record History Log windows.



Archives are named based on the date they were created. For example, if you 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 after the current date.

To view the contents of an archive, select the archive of interest from the **Archive** drop-down list in the **Filter Based On** area. To delete an archive, click the **Delete** icon that appears in the field.

Note: When a restore point is created for a project or when a project is checked out, all current entries in the history log are automatically archived.

Viewing Record History Logs

In addition to the project history log, you can view a log that shows only the changes made to a selected item (archived entries will not be shown). The History Log icon  will be displayed whenever the log is available:

- For folios, diagrams, multiplots (such as overlay plots), reports and worksheets, right-click the item in the current project explorer and choose **Item Properties**, then in the window that appears, click the **History Log** icon in the Identifiers tab.
- For system hierarchy items in XFMEA/RCM++ and MPC, select the item then click the **History Log** icon in the Properties panel.
- For FMEAs, DVP&Rs and other analyses in XFMEA/RCM++, right-click the analysis's tab in the Analysis panel and choose the **History Log** command on the shortcut menu.
- For resources (e.g., models, actions, etc.), the **History Log** icon will be available when you edit the item or view its properties.

Import, Export and Data Conversion

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

In desktop applications, you can:

- Import data from an existing database when you are creating a new one.
- Convert and import data from prior version files.
- Import/export selected projects.

When applicable, you can also:

- Import/export selected project items or resources.
- Import from an Excel spreadsheet or delimited text file into a data folio.
- Use XML files to import/export system configuration information in BlockSim or Lambda Predict.

These features can be accessed either from the Import/Export Wizard (**Project > Management > Import/Export**) or from the Backstage View.

In addition to the common functionality described in this chapter, some ReliaSoft 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++, the ability to share analysis details between specific ReliaSoft applications.). For more information about these specialized tools, please consult the documentation for the particular application(s) involved.

Importing from an Existing Database

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

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

First, use the drop-down list or browse icon to select the database that you want to import from. This can be a standard database (*.rsr21) or an enterprise database connection file (*.rserp).

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

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

Note: If you are creating a new enterprise database on SQL Server, the **Create SQL Server** login check box will be displayed at the bottom of the window. (See SQL Server Logins or Using Windows Impersonation.)

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 Database)
- Importing to an existing database and project (Import > Other file)

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

- For Weibull++ and BlockSim, you can use either method.
- For Lambda Predict, XFMEA/RCM++ and MPC, you must use the File > Open Database 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 Converting XFMEA/RCM++ 5 Databases.)

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

For all ReliaSoft desktop applications, you can use the File > Open Database 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.

1. Choose **File > Open Database**.
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.
    - See Converting from BlockSim 6 or 7 Files.
    - See Converting MPC 3 Databases.

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. (See Converting XFMEA/RCM++ 5 Databases.)

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.

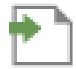
Importing to an Existing Database and Project (Import > Other file)

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

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 Converting from BlockSim 6 or 7 Files.)

   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. (See Converting XFMEA/RCM++ 5 Databases.)

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

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 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 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 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. Starting in Version 8, 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; subsequent versions calculates load using the weight proportionality factor as a multiplier.
  - Normal/lognormal distributions have increased precision starting in Version 8.
  - 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 subsequent versions, 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.
  - 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 subsequent versions, 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; subsequent versions treat them as the same group. This may produce results that are statistically different.
- Throughput is not available in fault trees starting in Version 8.
- The throughput property Send units to failed blocks works differently for subdiagrams in versions after Version 7, 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.
- Starting in Version 8, 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; subsequent versions calculate 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; subsequent versions treat 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 starting in Version 8. 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; subsequent versions calculate 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 subsequent versions. 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 subsequent versions.
  - The threshold (i.e., detection) information from Version 7 will be transferred to the failure detection properties of the on condition task in subsequent versions.
  - The preventive maintenance properties from Version 7 will be transferred to the on condition task (upon detection) properties of the on condition task in subsequent versions.
  - 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 subsequent versions 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.

Converting XFMEA/RCM++ 5 Databases

This topic describes how to convert/import data from an XFMEA/RCM++ 5 database into a ReliaSoft database.

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

STANDARD DATABASES (*.RX5)

For XFMEA/RCM++ 5 standard databases, you must convert the entire *.rx5 file to a new standard database. (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 current version of XFMEA/RCM++, choose **File > Open Database**.
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 *.rsr21; 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 an enterprise database that has already been created. You must be a member of the Admin group in the ReliaSoft enterprise database to perform this task.

Do the following:

1. Connect with the enterprise database you want to import into.
2. Choose **File > Manage Database > Import from Version 5**.

3. In the area at the top of the Import from Version 5 Enterprise Database window, enter the connection information for the Version 5 enterprise database and click **Connect**.
4. The table shows all of the data and settings in the original database. Use the check boxes to select which data you wish to import and then click **OK** to start the transfer.

Note: If you are importing into a SQL Server database, the **Create SQL Server login** check box will be displayed at the bottom of the window. This is applicable only if you are importing user accounts. (See SQL Server Logins or Using Windows Impersonation.)

CONVERSION ASSUMPTIONS AND TIPS

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

Conversion to Resources

Certain types of information in the Version 5 projects will be converted to *resources* in the current 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 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 the current version. Specifically:

- Database-level security: User permissions assigned by access level in Version 5 will be assigned by the corresponding security group in the current 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 current version. If there is no corresponding security group in the current 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 ReliaSoft database. 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 current version. Therefore, these groups will initially be assigned the default permissions; you can edit the permissions via the Security Groups tab of the Users and 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 current 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 current 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++ documentation.

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 ReliaSoft database.

Choose **File > Open Database**, 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 Security Options 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 *.rsr21 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 than they were in Version 3. Specifically:

- Some of the **Task Types** that were combined in Version 3 are now categorized separately (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. Now, 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 unit (e.g., 2000 flight hours can now be stored as value=2000 and 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++.
- The **Zonal** field from Version 3 is called **Zonal Candidate** in the current version. 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 the current version, the **Zone** field will be enabled only if Zonal Candidate is set to **Yes**.

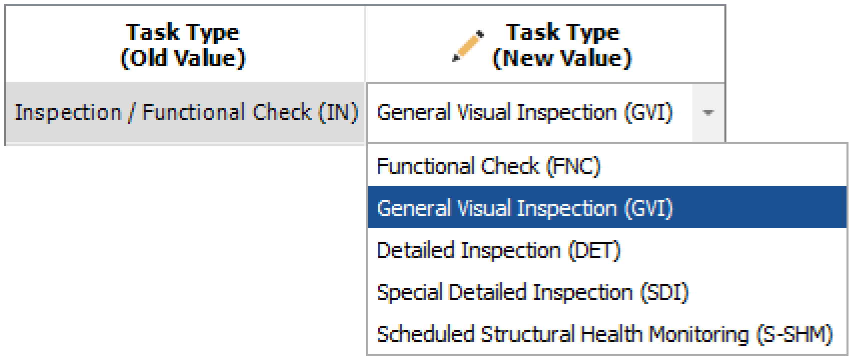
The following subsections describe the default conversion logic for Task Type, Interval and Zone.

Task Type

For the **Discard** and **Restoration** task types, there is no difference between Version 3 and subsequent versions. 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 Detailed Inspection
Health Monitor	Scheduled Structural Health Monitoring
Function	Functional Check
none of the words/phrases listed above	Detailed Inspection (DET)

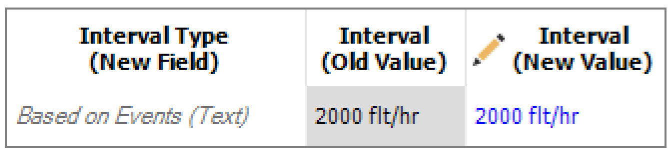
- When the task type was set to **Lubrication/Servicing (LU)** in Version 3, the default option is to import the task as Servicing (SVC) if the task description contains "Servic" and as Lubric-ation (LUB) if it does not.
- When the task type was set to **Operational/Visual Check (OP)** in Version 3, 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 Version 9 and later of MPC 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 Database > Task Types**.

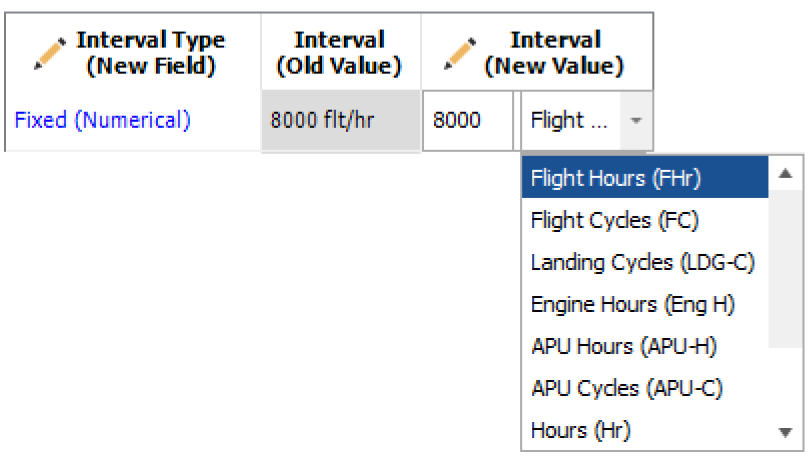
Interval Type and Interval

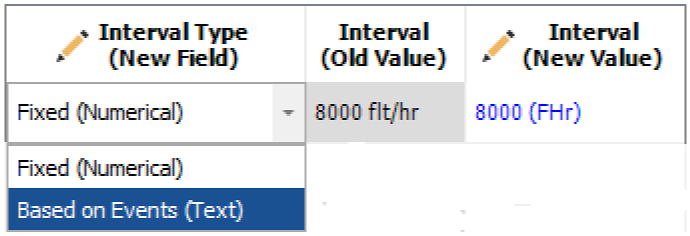
Two options are displayed at the top of the Tasks Conversion window to determine how the task intervals will be imported:

- 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 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 units displayed in the drop-down list, click the **Units** button. The changes that you make in the Unit Settings 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 Database > Unit Settings**.

The Tasks Conversion window uses a variety of techniques to try to match the text field from Version 3 with one of the specified units in subsequent versions. There will be a match if the name or abbreviation of a current version unit either starts with or contains the text in the Version 3 field. For some predefined units, the software also recognizes other commonly used abbreviations for the 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 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 **Units** button and define a new 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 "Unit Settings" window and create a new 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.

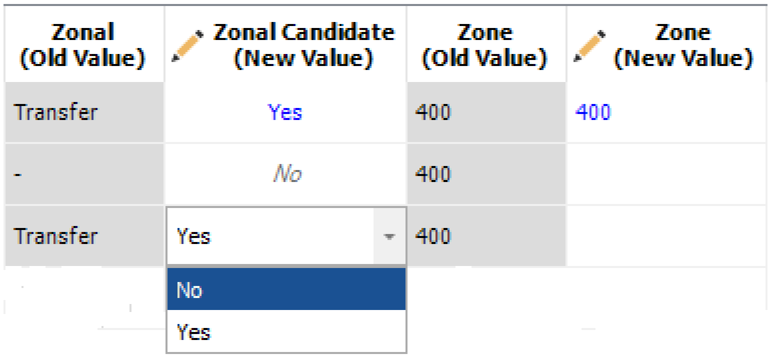
Zonal Candidate and Zone

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.



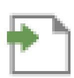
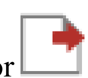
Zonal (Old Value)	Zonal Candidate (New Value)	Zone (Old Value)	Zone (New Value)
Transfer	Yes	400	400
-	*No*	400	
Transfer	Yes	400	

Importing/Exporting Projects

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

Note that when importing/exporting a project between databases, any resources and FMEAs used by the project will be automatically imported/exported along with the project as local resources, even if they were originally reference resources or global resources. If you do not want the reference resources to be converted to local resources, you must import the both the project and the reference project at the same time. (See Local, Global and Reference Resources.)

To import a project, first choose **Project > Management > Import/Export**.

 or 

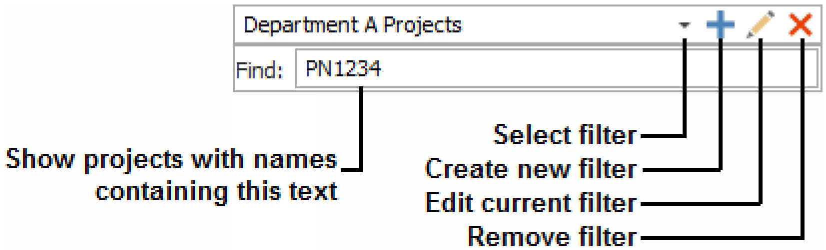
In MPC, the Import/Export window opens directly. In all other applications, a wizard displays the options that are relevant for the current application. Select **Projects** and click **OK**.

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

1. Use the drop-down list or browse icon  to select the database that you want to import from or export to.
    - This can be any existing standard database (*.rsr21) or enterprise database connection file (*.rserp).

- If you are exporting, you can also use the add icon 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. (See Project Manager.)



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

The following considerations apply:

- The names of projects must be unique within each ReliaSoft 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."
- If the project has a category, the application will first attempt to match it to an existing category in the destination database. If a matching category does not already exist, and you have the permission to create project categories in the destination database, it will be created automatically.
- If you select a project of a type that you don't have permission to create in the destination database (private, public or reference), it will be converted to a type that you do have permission to create. For example, if you select to import a reference project but you don't have the "Create and own reference projects" permission in the destination database, the project will be imported as a public or private project, and any links from other projects will be broken.

Importing/Exporting Project Items or Resources

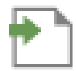
When applicable, ReliaSoft applications make it easy to import/export selected project items (e.g., folios, diagrams, plot sheets, ReliaSoft Workbooks, etc.) or resources (e.g., models, maintenance tasks, etc.) between existing projects. The projects can be in the same database or in different databases.

First, open the project you want to import to or export from. Then open the import/export window, there are two ways:

- If you have selected the **Allow multiple projects** option in the Common Settings page of the Application Setup, you can open both projects simultaneously and then drag/drop items from one project to another.

OR

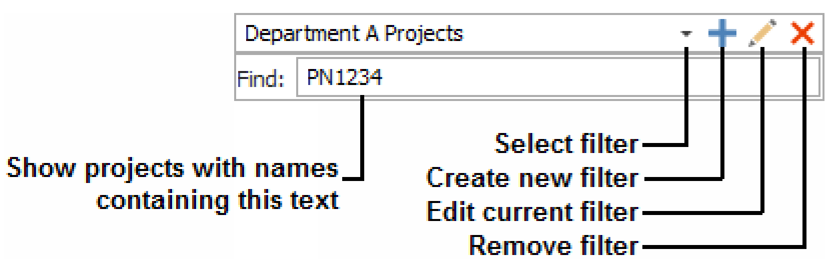
- Choose **Project > Management > Import/Export**.

 or 

In the wizard, choose **Items** (not available for XFMEA/RCM++ and MPC) or **Resources** (not available for MPC) and click **OK**.

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

1. Use the drop-down list or browse icon to select the database you want to import from or export to.
   - This can be any existing standard database (*.rsr21) or enterprise database connection file (*.rserp).
   - If you are exporting, you can create a new standard database and project by clicking the icon.
2. Use the tree in the **Source Project** or **Destination Project** area to select the project 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. (See Project Manager.)



3. Use the check boxes in the **Items to Import** or **Items to Export** 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 resource that exists in the selected project.

4. Click **Import** or **Export** to copy the data.

Note: The names of project items and resources must be unique within each 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

The application will automatically copy any item/resource that is linked to the imported or exported item, even if you did not specifically select to import/export them. Some examples:

- A **multiplot**, such as an overlay plot, side-by-side plot, or 3D plot, will automatically import/export a copy of the folios or diagrams that were used to generate the plot.
- A **BlockSim diagram** will automatically import/export a copy of any utilized subdiagrams and resources. This includes event analysis flowcharts, which will automatically import/export copies of any utilized subcharts, resources and ReliaSoft Workbooks.
- A **ReliaSoft Workbook** or **special analysis folio/tool** (e.g., allocation analysis, stress-strength analysis, etc.) will automatically import/export a copy of its data sources.
- A **URD** will automatically import/export a copy of the models, maintenance tasks, crews, etc. that were used to build the URD. The same applies to any resource that is built upon another resource.

Note that when importing/exporting project items between databases, the resources are imported/exported as local resources in the destination database, even if they were originally reference resources or global resources. For more information on how importing/exporting affects a resource, see Local, Global and Reference Resources.

Tip: If you do not want an event analysis flowchart to import/export an associated resource or workbook, you can enclose the resource/workbook name in single quotes to reference it by name only. For example, if you export a diagram that contains a block using the expression **'Model1'(1000)**, Model1 will not be transferred along with that diagram. If the destination project already contains a model called Model1, that model will be used in simulating the transferred diagram.

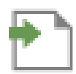
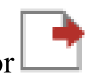
Importing from Excel or Delimited Text Files into a Folio

In Weibull++, 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 ReliaSoft 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++ 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 design folios, 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++, 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 rest of this topic describes how to map the data for import into a Weibull++ data folio.

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.

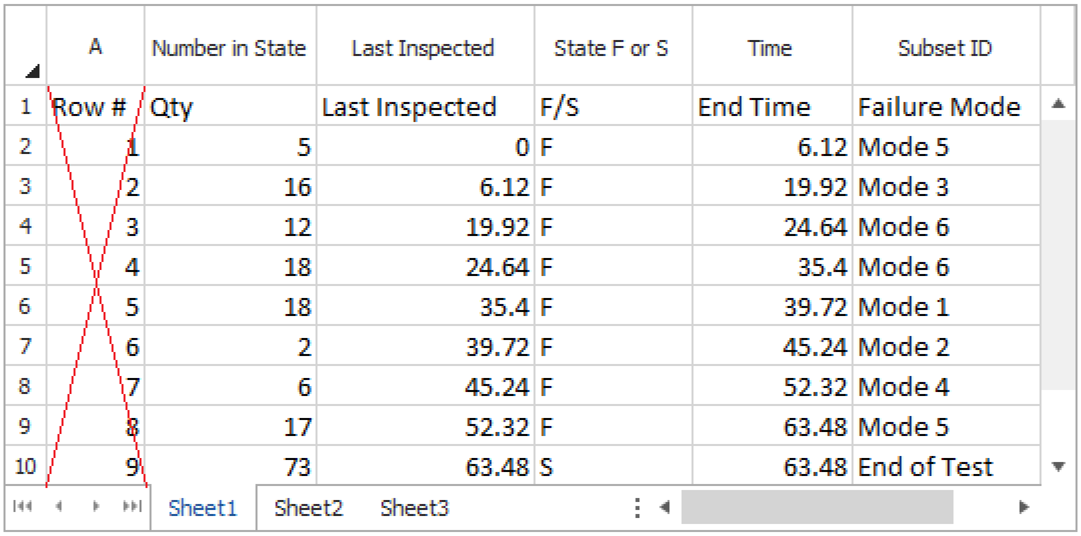
- **Do not import sheet**: The control panel will not contain any other options and the data will not be imported.
- **Import sheet**: The control panel displays the options you need to either manually map the columns or use a template to automatically apply the same mapping that was used for another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

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 heading and maps the data to a column in the data folio.
   - If the heading displays a letter, the data in that column will not be imported.
   - If the heading 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++ life data folio. Column A will not be imported. The rest of the data has been assigned to columns that are used in the 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	24.64	Mode 6
5	4	18	24.64	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

3. If you are importing to an life-stress data 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.

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 or *.rgaim in Weibull++) 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.

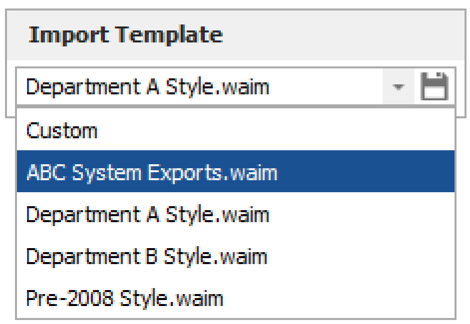
Creating a New Import Template

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.

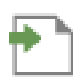
3. Specify a name and click **Save**.

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.



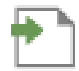
4. The utility will automatically assign the column mappings defined in the template. If desired, you can use the control panel to make further adjustments before import.
5. When you are satisfied with the mapping, click **Import** to create the new folio.



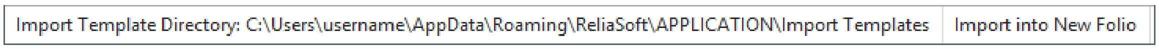
Tip: If you make adjustments after applying the template, you have the option to click **Save** in the **Import Template** area to either replace the existing template, or create a new template with a different name.

Importing the Data

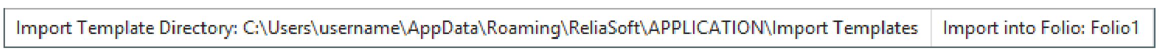
When you are ready to import the data, click the **Import** button.



By default, the application will import the data into a new 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.



Using XML in BlockSim and Lambda Predict

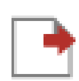
In BlockSim and Lambda Predict, you have the option to use XML (extensible markup language) files to import or export system configuration data and item properties. XML files can be used to transfer data to/from other applications or database systems.

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 event analysis flowcharts, plots, attachments, etc.) are not included.

Importing from XML

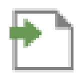
There are two ways to import from an XML file.

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

1. Choose **File > Open Database**.
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**.

Attachments

All ReliaSoft desktop 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 application you are using.

IMPORTANT: Starting in Version 20.0.3, users with the appropriate permission can prevent a **URL** or a **File Link** from being added to the database and/or limit the types of files that can be linked to or embedded to a list of allowable file extension types (see Database Settings). Embedded links will always be available but are also limited to the list of allowable file extension types.

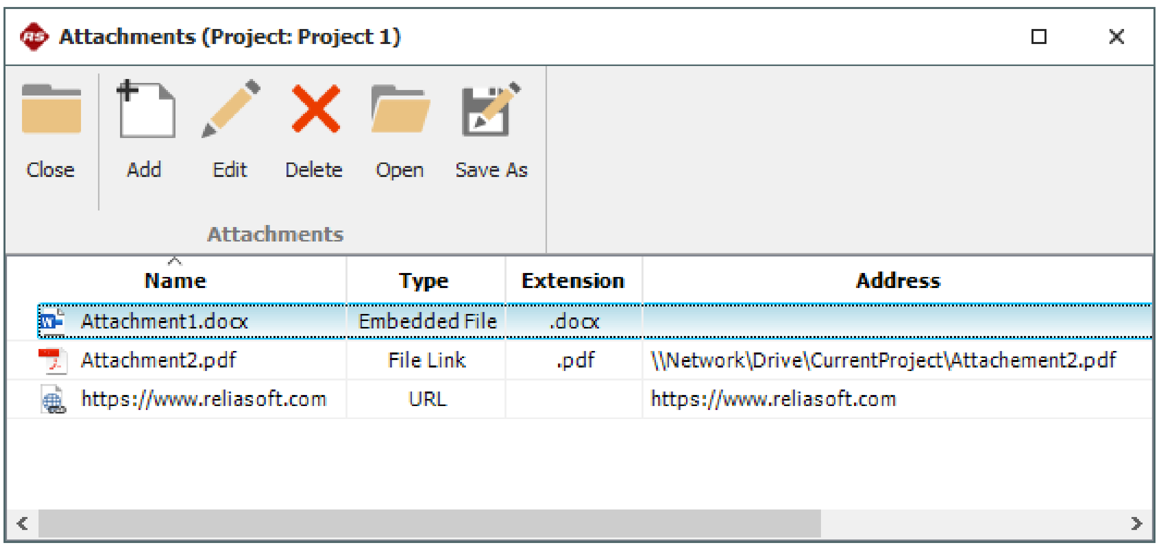
In all desktop applications, attachments can be either linked or embedded.

- **URL** and **File Link** attachments store the path to a web page 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. When you delete a file link or URL, this simply removes the link, leaving the original file untouched.

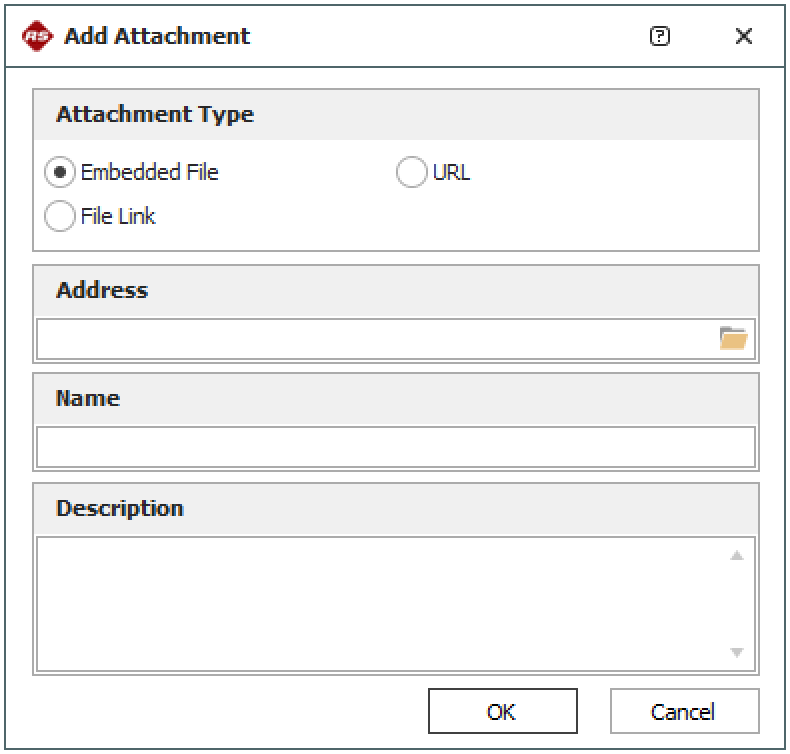
Note: If your organization has implemented an SEP web portal for an enterprise database, note that file link attachments (i.e., stored paths to local files on your computer or network) cannot be created, opened or saved within the SEP due to browser security.

- **Embedded** attachments are always files. For these attachments, the software stores a complete copy of the file inside the database. 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 or restore point that you could roll back to).

The Attachments window manages all of the URLs/files attached to a particular location (e.g., project, resource, hierarchy record or block).



The Add/Edit Attachment window specifies the details of an individual file or URL, and will be displayed whenever you choose to create or edit an attachment.



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.

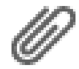
Attachment Locations

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

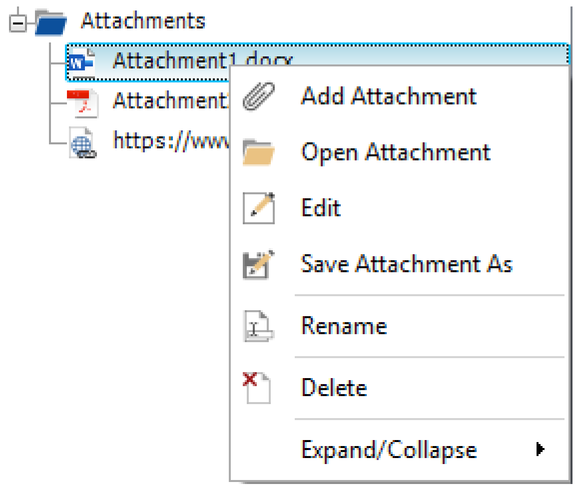
Tip: In most cases, the caption bar in the Attachments window will provide an indication of which type of attachment you are working with. For example, when you are working with project-level attachments, the caption bar will indicate "Project:" followed by the specific name of the project.

Project Attachments

All ReliaSoft desktop 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.

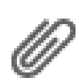


In XFMEA/RCM++ and MPC (which do not have a current project explorer), you can also access the project attachments by selecting **Project > Management > Attachments**.

Resource Attachments

All of the applications that utilize resources support attachments at the resource level. For example, you can attach a file to a URD, a model, etc.

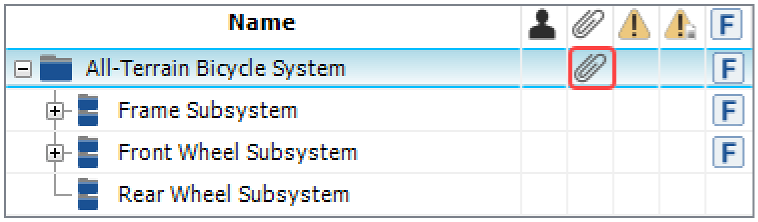
To access attachments for an existing resource, you can click the **Attachments** icon in the resource's properties window.



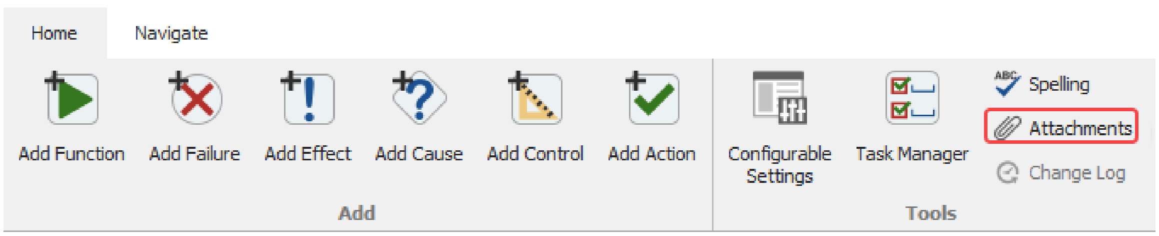
Hierarchy Attachments

Some applications, including XFMEA/RCM++, 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 headings and choose **Customize Columns**.) You can also choose **Tools > Attachments** on the ribbon tab for the hierarchy that you are currently working with.

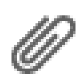


For records in an analysis hierarchy (such as FMEA or functional failure analysis), the attachment icon can also be accessed from within the record properties window.



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.



Select Existing Text Window

The Select Existing Text window provides a list of existing descriptions that might apply to the current text field or analysis. This can help to save time on data entry, ensure consistency and facilitate brainstorming. The utility is used extensively in applications such as XFMEA/RCM++ and MPC, and it is also available for certain text fields in other ReliaSoft desktop applications.

There are two ways this utility might be used:

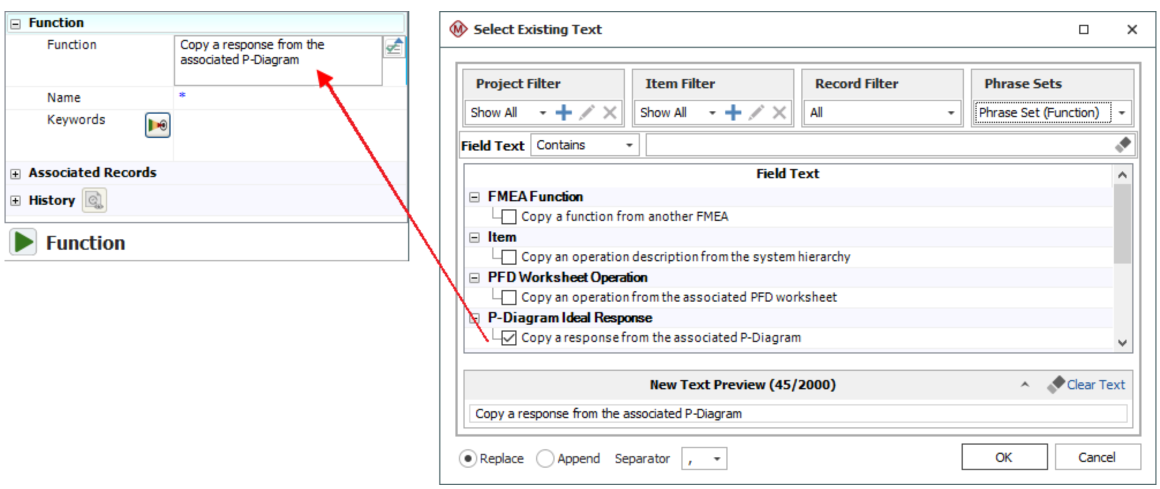
- **Replace or append text in the current field** - If you click the  icon inside a text field, you can select text and either replace the current text or append the new text to the end.

Starting in Version 2019, many text fields in XFMEA/RCM++ allow selection of multiple descriptions, which will be concatenated in the order in which they were selected using the separator specified in the **Separator** field. Available separators include commas, periods, semicolons, dashes and, for multi-line text fields, new lines.

When you are using the utility in this way, the **New Text Preview** area shows how the selected text will appear, and provides information on how many characters out of the character limit are being used (e.g., 45 characters out of a possible 2000 in the example below). If you exceed the text field's character limit, your text will be truncated.

- **Create one or multiple new records** - If you use one of the ribbon commands in an FMEA or P-Diagram (e.g., **Functions > Select Existing Text**), you can select multiple descriptions and the utility will use your selection(s) to create new record(s).

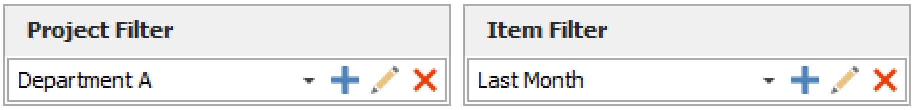
The options will vary depending on the record/field type, and whether you are updating a single description or adding new records. The following example shows how the tool may be used to populate the function description in an FMEA.



Tip: The descriptions are sorted alphabetically under each heading. When the table has focus, you can type a letter to move to the next description that begins with the letter.

Project and Item Filters

This window utilizes the same project and item filters that are available in many other locations throughout ReliaSoft desktop applications. For example, when searching for text for an item in the system hierarchy, you could use a Project Filter to search for analyses performed by Department A and then use an Item Filter to show only those analyses that were modified in the last month.

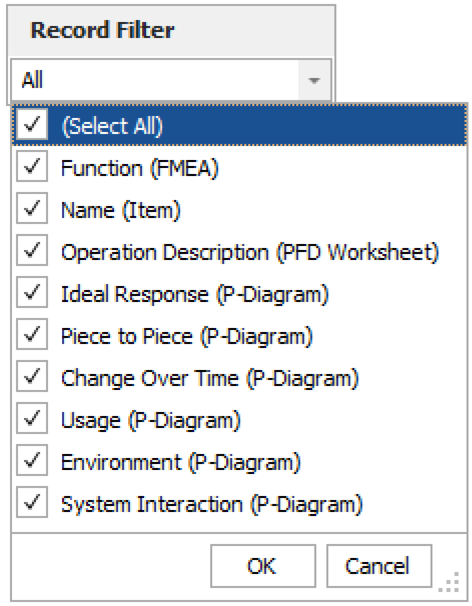


In addition, when applicable:

- **Project Filters**:
  - **Phrase Sets Only** limits the results to data in the selected phrase set(s), with no data from any projects.
  - **Current Project** limits the results to data in the current project.
- **Item Filters**:
  - **Current Item** limits the results to those found in the currently selected system hierarchy item.
  - **Parent Item** limits the results to those found in the immediate parent of the currently selected system hierarchy item.
  - **Next Lower Level Items** limits the results to those found in the immediate dependents of the currently selected system hierarchy item.
  - **Current Item Branch** limits the results to those found in the current branch of the system hierarchy.
  - If the currently selected system hierarchy item has a category assigned, **Match Category** limits the results to those found in system hierarchy items that have the same category assigned. The scope of this search is limited by your project filter, if any.
  - If the currently selected system hierarchy item has a part number assigned, **Match Part Number** limits the results to those found in system hierarchy items that have the same part number assigned. The scope of this search is limited by your project filter, if any.

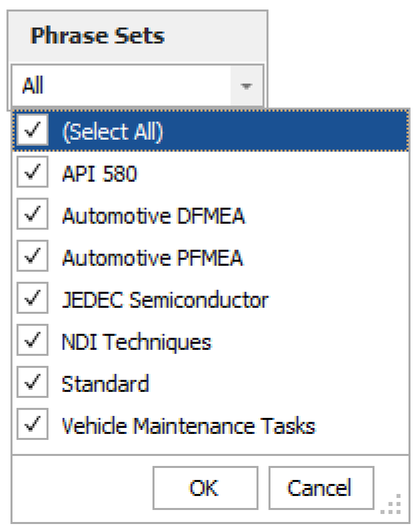
Record Filter

When applicable, the **Record Filter** shows the possible sources for the current record type and field that you can choose to include or exclude. The specific record types vary depending on the field.



Phrase Sets

FMEA records in XFMEA/RCM++ also have the option to include text from selected phrase sets.



Field Text

Use the drop-down list in the **Field Text** area to specify how to match the specified keywords, then type the keywords into the input box.

- **Contains** returns descriptions that contain the exact string entered.
- **Contains Any** and **Contains All** allow you to enter multiple keywords (separated by commas) and returns descriptions that contain at least one (any) or all of the keywords entered.
- **Begins With** returns descriptions where the exact string entered appears at the start of the field.
- **is** returns descriptions where the exact string entered matches the whole field.

Record Filter Options

When the descriptions can be obtained from more than one type of analysis (e.g., when PFMEA failures can be copied from FMEA causes, or when control plan operations can be copied from PFD Worksheet operations), the headings in the table identify the source. You can use the Record Filter to specify which source(s) to include.

FMEAs

FMEA Functions	FMEA Functions Next Higher Level Functions Next Lower Level Functions FMEA Function Requirements System Hierarchy Items PFD Worksheet Operations P-Diagram Functions P-Diagram Functional Requirements P-Diagram Ideal Responses P-Diagram Piece-to-Piece P-Diagram Change Over Time P-Diagram Usage P-Diagram Environment P-Diagram System Interaction DRBFM Functions
FMEA Functions: Requirements	FMEA Function Requirements FMEA Functions Next Higher Level Functions Next Lower Level Functions P-Diagram Functions P-Diagram Functional Requirements P-Diagram Non-Functional Requirements P-Diagram Input Signals DRBFM Functions

FMEA Functions: Next Higher Level Functions and Next Lower Level Functions	FMEA Functions Next Higher Level Functions Next Lower Level Functions FMEA Function Requirements P-Diagram Functions P-Diagram Functional Requirements P-Diagram Ideal Responses P-Diagram Piece-to-Piece P-Diagram Change Over Time P-Diagram Usage P-Diagram Environment P-Diagram System Interaction DRBFM Functions
FMEA Functions: Next Lower Level Item	FMEA Function: Next Lower Level Items System Hierarchy Items
FMEA Failures	FMEA Failures P-Diagram Error States FMEA Causes FMEA Function Requirements PFD Worksheet Product Characteristics DRBFM Concerns Related to Change DRBFM Other Concerns
FMEA Effects	FMEA Effects FMEA Failures P-Diagram Error States DRBFM Potential Effects of Concerns
FMEA Causes	FMEA Causes P-Diagram Piece-to-Piece P-Diagram Change Over Time P-Diagram Usage P-Diagram Environment P-Diagram System Interaction P-Diagram Control Factors PFD Worksheet Process Characteristics DRBFM Potential Causes of Concerns DRBFM Other Causes

FMEA Controls	FMEA Controls P-Diagram Control Factors Control Plan Methods DVP&R Test/Specification Methods DRBFM Current Design Controls - Detection DRBFM Current Design Controls - Prevention
FMEA Actions	FMEA Actions DVP&R Test/Specification Methods DRBFM Recommended Actions - Design DRBFM Recommended Actions - Validation/Test DRBFM Recommended Actions - Manufacturing/Supplier DRBFM Actions Taken

P-Diagrams

P-Diagram Input Signal	P-Diagram Input Signal System Hierarchy Items
P-Diagram Control Factors	P-Diagram Control Factors FMEA Causes
P-Diagram Error States	P-Diagram Error States FMEA Failures
P-Diagram Ideal Response	P-Diagram Ideal Response FMEA Functions
P-Diagram Piece-to-Piece	P-Diagram Piece-to-Piece FMEA Causes PFD Worksheet Product Characteristics Analysis Plan Assumptions
P-Diagram Usage	P-Diagram Usage Analysis Plan Conditions of Use
P-Diagram Environment	P-Diagram Environment Analysis Plan Conditions of Use

P-Diagram Change Over Time	P-Diagram Change Over Time FMEA Causes Analysis Plan Conditions of Use
P-Diagram Functions	P-Diagram Functions FMEA Functions FMEA Function Requirements
P-Diagram Functional Require-ments	P-Diagram Functional Requirements FMEA Functions FMEA Function Requirements
P-Diagram Non-Functional Requirements	P-Diagram Non-Functional Requirements FMEA Function Requirements

Control Plans

Control Plan Part/Process Num-bers	Control Plan Part/Process Numbers FMEA Function Op Seq Numbers PFD Worksheet Op Seq Numbers
Control Plan Process Name/Operation Descriptions	Control Plan Process Name/Operation Description FMEA Functions PFD Worksheet Operations
Control Plan Product Char-acteristics	Control Plan Product Characteristics PFD Worksheet Product Characteristics FMEA Causes FMEA Failures
Control Plan Process Char-acteristics	Control Plan Process Characteristics PFD Worksheet Process Characteristics FMEA Causes
Control Plan Methods	Control Plan Methods FMEA Controls

PFD Worksheets

PFD Worksheet Operation Descriptions	PFD Worksheet Operation Description System Hierarchy Items
PFD Worksheet Op Seq Numbers	PFD Worksheet Op Seq Numbers System Hierarchy Item Reference Numbers
PFD Worksheet Product Characteristics	PFD Worksheet Product Characteristics P-Diagram Piece-to-Piece

DVP&Rs

DVP&R Test/Specification Methods	DVP&R Test/Specification Methods FMEA Controls FMEA Actions

DRBFMs

DRBFM Functions	DRBFM Functions FMEA Functions
DRBFM Concerns Related to Change	DRBFM Concerns Related to Change FMEA Failures
DRBFM Other Concerns	DRBFM Other Concerns FMEA Failures
DRBFM Potential Effects of Concerns	DRBFM Potential Effects of Concerns FMEA Effects
DRBFM Potential Causes of Concerns	DRBFM Potential Causes of Concerns FMEA Causes
DRBFM Other Causes	DRBFM Other Causes FMEA Causes
DRBFM Current Design Controls - Detection	DRBFM Current Design Controls - Detection FMEA Controls

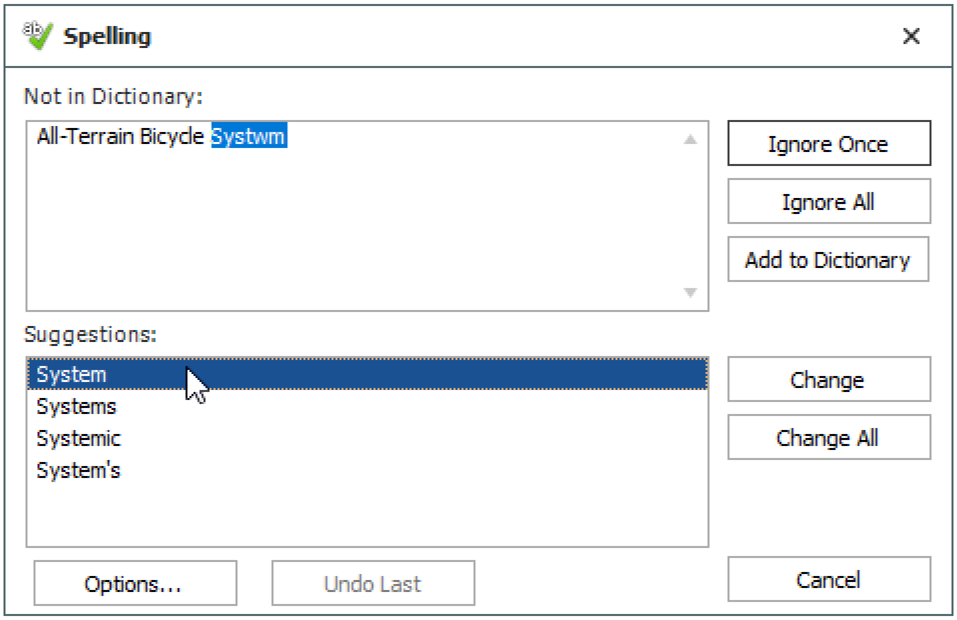
DRBFM Current Design Controls - Prevention	DRBFM Current Design Controls - Prevention FMEA Controls
DRBFM Recommended Actions - Design	DRBFM Recommended Actions - Design FMEA Actions
DRBFM Recommended Actions - Validation/Test	DRBFM Recommended Actions - Validation/Test FMEA Actions
DRBFM Recommended Actions - Manufacturing/Supplier	DRBFM Recommended Actions - Manufacturing/Supplier FMEA Actions
DRBFM Actions Taken	DRBFM Actions Taken FMEA Actions

Check Spelling

ReliaSoft desktop 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++, 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.

Results Window

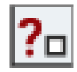
The Results window is used in many ReliaSoft applications to show detailed calculation results. From this window, you can edit the results, copy the results to the Clipboard or print the results.

- **Paste** pastes the contents of the Clipboard into the current control.
- **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 Special** opens the Paste Special window, which allows you to paste specific cell contents or attributes (such as formulas, formats, or comments) from the Clipboard.
- **Quick Print** sends the current document directly to the default printer without making changes.
- **Print** opens the Print window, which allows you to specify the printer and the printing options.
- **Print Preview** allows you to preview the current page before it is sent to the printer.
- **Send to Excel** allows you to save the current selection as a Microsoft Excel (*.xls or *.xlsx) file and then opens that file in Excel, if it is installed on your computer.
- **Select Columns** allows you to specify the columns shown in the spreadsheet. This option is available starting in Version 2020, for BlockSim fault trees only.

Note that the Results window may contain multiple sheets, accessible via the page index tabs at the bottom of the window.

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 ReliaSoft desktop applications, you can open the QPE from the Model properties window or the Model wizard by clicking the **QPE** icon.



In Weibull++, it's also accessible from the Home tab of the ribbon and several analysis interfaces (including the Weibull++ life data folio, the 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.

Quick Parameter Estimator 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 units.
- In Weibull++, 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.

Quick Parameter Estimator Expert View

Follow the steps below to use the Expert view of the Quick Parameter Estimator (QPE):

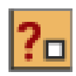
- Choose a distribution from the **Distribution** drop-down list. This is the distribution that you will solve the parameter(s) for. Then choose the appropriate time units from the Units drop-down list.
  - If you are not sure which distribution to select, consider using the Wizard view of the QPE instead.
  - 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 **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).
- Click the **Update** button to update the model using the selected distribution, calculated parameters and selected time units.

Quick Parameter Estimator (ALTA)

For life-stress data 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. You can open the QPE from the ribbon by choosing **Home > ReliaSoft > Quick Tools > Quick Parameter Estimator (ALTA)**.



You can also open the ALTA QPE from the life-stress data folio (by leaving the folio's data sheet empty and clicking Calculate), as well as the stress-dependent Monte Carlo and stress-dependent 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 **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.

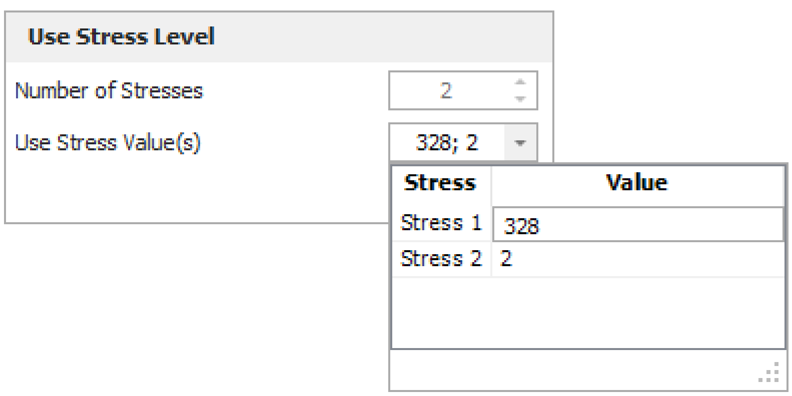
        - In the **Solve for Parameter** area, select the parameter that you wish to solve for. The remaining parameters in this area will have input fields enabled. Enter the known values of these parameters.

    - 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.

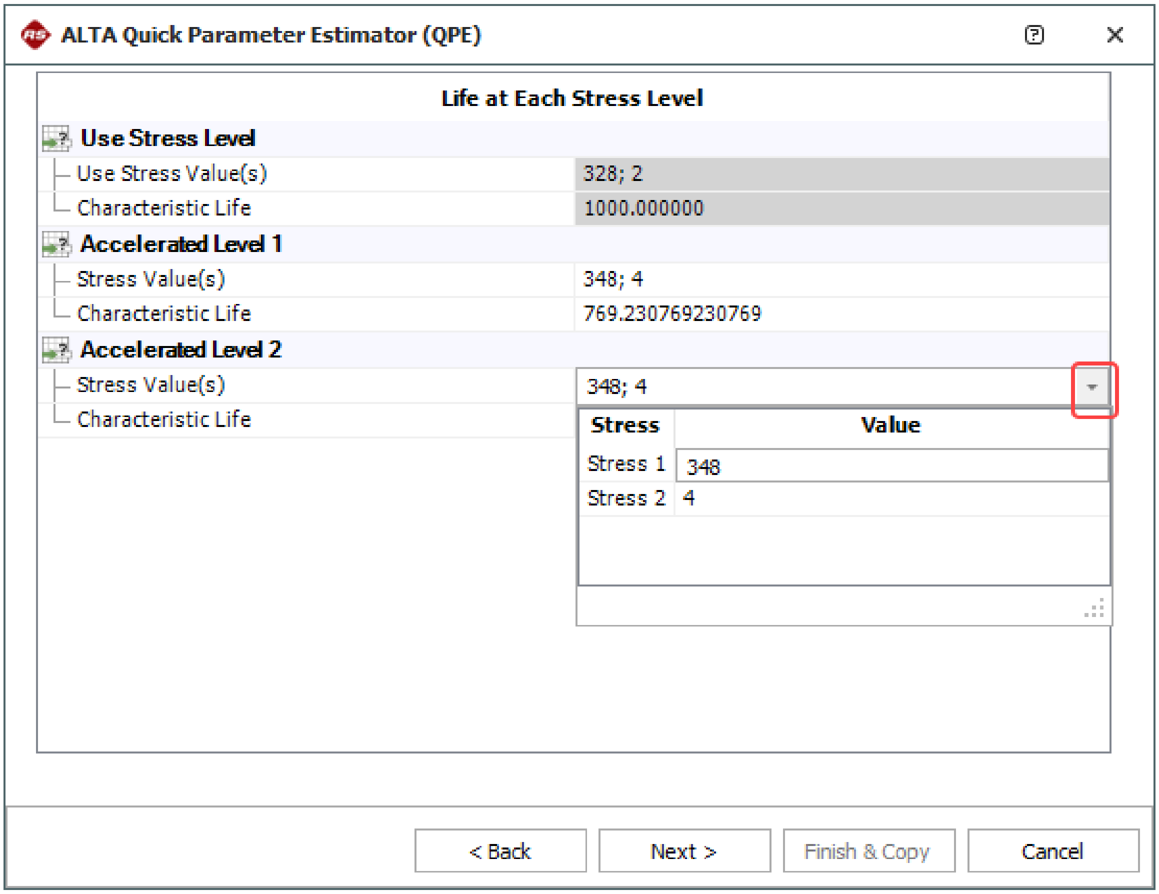
    - If the **Number of Stresses** field is enabled, enter the number of stresses that will be used in your model. (Note that different models have different requirements for the number of stresses that can be used.)

    - 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 another utility, such as the Monte Carlo tool, you may click the **Update** button if you wish to update the Monte Carlo tool with the model, parameters and time units defined in the QPE.
    - 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.

Resources

IN THIS CHAPTER

In ReliaSoft applications except MPC, *resources* contain various types of information that can be shared between analyses. Resources may be created in one application and then shared with other ReliaSoft applications where they may be relevant. When the resource is updated with new information, the change is reflected in all analyses that rely upon it.

An example of a resource is a *model* that represents a product's probability of operating successfully over time. The model may be defined manually or published from an application such as Weibull++ or Lambda Predict. Then it can be used in applications such as BlockSim or XFMEA/RCM++ as part of the *universal reliability definition (URD)* of a particular component/assembly.

Note: In a secure database, the ability to create, edit and delete resources is available only if the user a) is the project owner, b) has the applicable "resources" permissions, or c) has the applicable "manage all projects" permissions.

Types of Resources

The following tables describe the types of resources that you can use in different ReliaSoft applications. These resources can be created while you're performing an analysis and also from the Resource Manager. (In a secure database, the ability to create, edit and delete resources is available only if the user a) is the project owner, b) has the applicable "resources" permissions, or c) has the applicable "manage all projects" permissions.)

Used in Multiple Applications

Resource	Used In
Universal reliability definitions (URDs) describe a set of reliability and maintenance characteristics for a particular component/assembly.	BlockSim, RCM++, XFMEA
Variables store numerical values that can be programmatically varied during simulation.	BlockSim
Models can represent probabilities, durations or costs, either fixed or time-dependent. They are used by other resources to represent the reliability of a component, the duration of a task, the expected cost of a repair, etc.	Can be published from analyses in Weibull++, BlockSim and Lambda Predict, or created manually. Used in BlockSim and XFMEA/RCM++.
Tasks represent maintenance activities: • **Corrective** tasks are unplanned maintenance performed when a failure occurs. • **Scheduled** tasks can include preventive maintenance, inspections and on condition maintenance.	BlockSim, RCM++
Task packages represent groups of tasks that are performed together at scheduled intervals.	BlockSim, RCM++

Crews represent the personnel who will perform a maintenance task.	BlockSim, RCM++
Spare part pools determine whether a spare part will be available, how long it will take to obtain and how much it will cost.	BlockSim, RCM++
Metrics display any numerical value of interest, and track how it changes over time.	All applications except MPC via the Project Planner; analyses in Weibull++, BlockSim and XFMEA/RCM++
Maintenance groups are used to model situations in which some event within the group can trigger maintenance or state changes for other components/assemblies.	BlockSim, RCM++
Mirror groups are used to represent the exact same component/event in more than one location within your analysis.	BlockSim, RCM++, XFMEA
Actions describe specific assignments that need to be performed.	All applications via My Portal; Project Planner and FMEAs in RCM++ and XFMEA
Controls may be used in FMEAs and/or control plans to represent methods to reduce or eliminate the risk associated with potential failures.	RCM++, XFMEA
Profiles represent values that vary with time. They can be used for life-stress folios to describe stress levels under test conditions, or in process flow simulation diagrams to describe patterns of throughput. They can also be used to describe cost and duration in spare part pools.	BlockSim, RCM++

Used Only in BlockSim

Resource	Used In

Switches describe how the activity transfers between active and standby blocks in BlockSim's standby containers/gates.	BlockSim
Maintenance templates define the activities that will be performed during a maintenance phase in a phase diagram.	BlockSim
Flow groups are used to model situations in which meeting a particular flow threshold can trigger maintenance tasks. These groups are used only in process flow simulation diagrams (if supported by your license).	BlockSim
Event analysis functions store equations that are evaluated based on input values passed to the function during simulation. These resources are used only in event analysis flowcharts (if supported by your license).	BlockSim
Event analysis static functions store equations that are evaluated before simulation begins. These resources are used only in event analysis flowcharts (if supported by your license).	BlockSim
Simulation definitions are used to trigger simulation of a simulation RBD or fault tree from within an event analysis flowchart, allowing you to use one or more results from the simulation in the flowchart. These resources are used only in event analysis flowcharts (if supported by your license).	BlockSim
Event analysis tables store arrays of values in rows and columns. These resources are used only in event analysis flowcharts (if supported by your license).	BlockSim

Local, Global and Reference Resources

There are three types of resources:

Local resources can be used only within the project in which they were created, and are therefore available only to users with permissions within that project.

Reference resources (which are resources created in a reference project) can be used in any project throughout the database. However, in a secure database, they can be created and edited only by users who have the "Create/edit/delete local resources" permission within the reference project. These resources can be selected by any user who has at least the "Read" permission in the reference project. Users without permissions in the reference project can see the reference resources wherever they are used, but they will not be able to select or modify them.

 **Global resources** can be used in any project throughout the database. In a secure database, they can only be created and edited by users who have the “Create/edit/delete global resources” permission. However, they can be selected by any database user.

To make a resource global, select it in the Resource Manager and choose **Home > Actions > Make Global**.

Making a resource global cannot be undone and this option is not available for all resource types. 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, regardless of whether it was originally local or reference).

Parent/Child Resource Relationships

Certain resources can have other resources assigned to them (e.g., URDs can have models and tasks assigned to them, tasks can have models, crews and spare part pools assigned to them, etc.). This can be considered a "parent/child" relationship.

When you create a resource from within its parent, the child resource will be of the same type as its parent. For example, if you are working with a global URD and you add a model to it, the model will be global. If you are working with a reference task and you add a crew to it, the crew will be added to the reference project that contains the task.

When you assign existing resources to a parent resource, the following rules apply:

- Any resource assigned to a reference resource must be either a reference resource within the same project or a global resource (e.g., a local model cannot be assigned to a reference URD, nor can a model in "Reference Project 2" be assigned to a URD in "Reference Project 1").
- Any resource assigned to a global resource must be either a reference resource or a global resource (e.g., a local model cannot be assigned to a global URD).

Keeping Resources Together

The application will automatically keep copies of a project's resources and linked FMEAs when the project is restored, checked out, imported or exported.

- When you create a restore point, any reference resources, global resources or linked FMEAs used in the project are converted to local resources/FMEAs and stored with the backup.
- When you check out a project for local editing, any reference resources, global resources or linked FMEAs used in the project are converted to local resources/FMEAs and stored with the project.
- When importing/exporting a project item that uses reference resources, global resources or linked FMEAs:
  - If you import/export within the same database, the references will remain unchanged.
  - If you import/export between databases, the destination project will contain local copies of the original resources/FMEAs.
- When importing/exporting a project:
  - If you import/export a project that uses global resources, the destination project will contain local copies of the original resources.
  - If you import/export a project that uses reference resources or linked FMEAs:
    - The references/FMEAs will be maintained if you also import/export the reference project at the same time.
    - If you don't import/export the reference project at the same time, the new project will instead contain copies of the original resources/FMEAs.

Creating and Selecting Resources

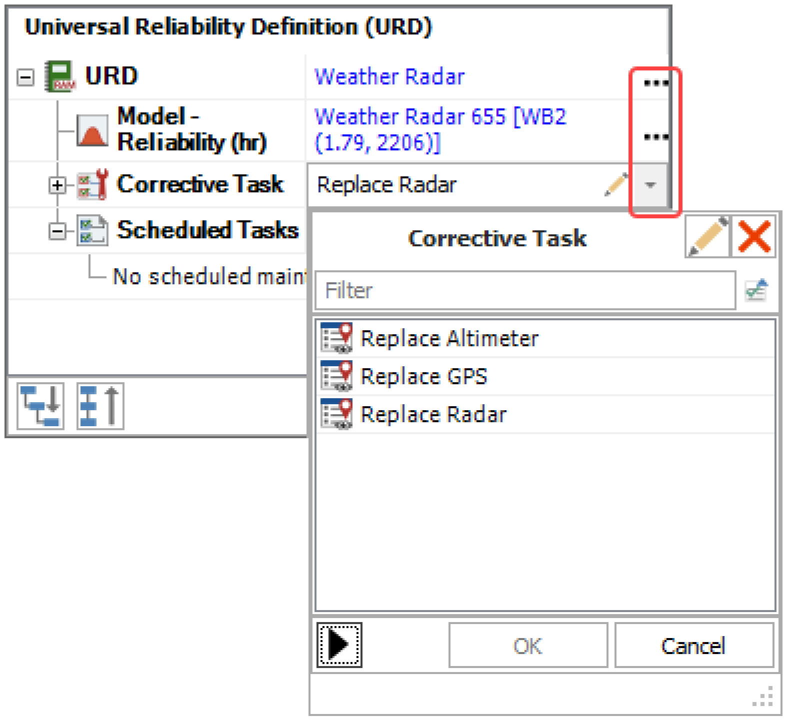
This topic describes how to use *resource wizards* to create and edit resources while you're performing an analysis or creating a URD. In addition, you may also need to:

- Use the Resource Manager to see all of the resources that are available to use in the current project. In addition to creating and editing resources, the manager also allows you to:
  - Delete resources that are no longer needed.
  - Create global resources that are available to all projects in the database.
- Publish a model that is linked to an existing analysis.

(In a secure database, the ability to create, edit and delete resources is available only if the user a) is the project owner, b) has the applicable "resources" permissions, or c) has the applicable "manage all projects" permissions.)

Opening a Resource Wizard

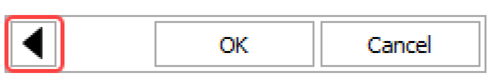
Throughout all ReliaSoft desktop applications, the availability of a resource wizard is usually indicated by an ellipsis (...) or an arrow in the resource field. Simply click twice in the field to open the wizard.



Tip: If a resource is already assigned in the field, you can edit it directly without opening the wizard by clicking the **Edit** icon that appears in the field.

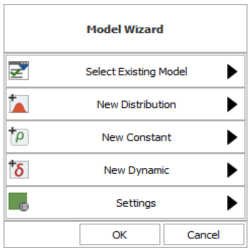
Back Arrow and Main Page

Each wizard contains multiple pages with different options on each page. If the option you need is not immediately visible, you can click the **Back** arrow to see the Main page, which shows all the options that are applicable for the current situation.

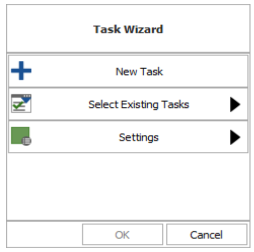


As an example, the following pictures show some of the different options that will be available on the Main page for specific situations and resource types.

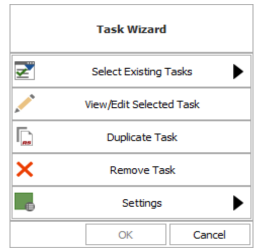
If a model hasn't been defined...



If a task hasn't been defined...



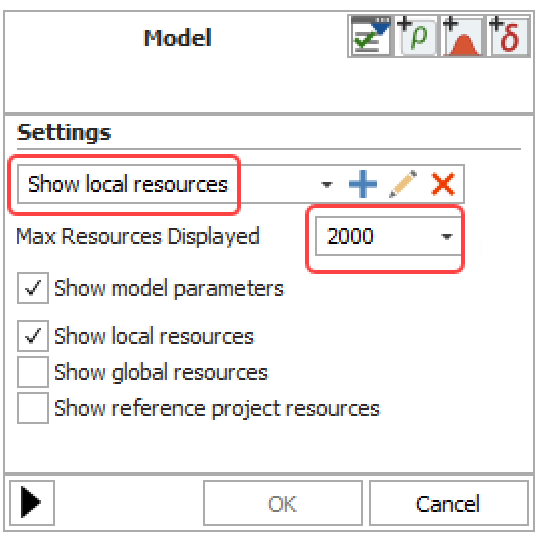
If a task has already been defined...

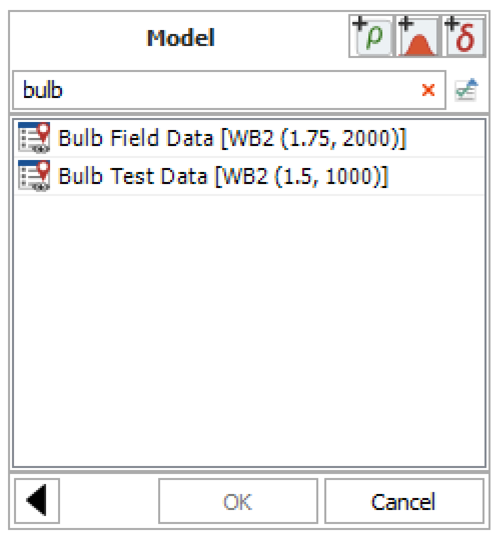


Selecting an Existing Resource

The Select page of the wizard displays a list of existing resources that meet the criteria specified on the Settings page. You can then further limit the list by typing inside the text box.

As an example, the following pictures show a situation in which the settings are configured to display up to 2,000 models (more resources take longer to load) from the current project only (called *local resources*), and the user has selected to see only the ones in which the name includes “bulb.”





In addition, for model resources only, you must specify whether you want to display the model parameters in the list (which also takes longer to load).

To change the initial criteria for the list, click the **Back** arrow and then click the **Settings** button on the main page.

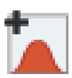
Alternatively, if you need more information about the available resources and/or a wider range of filtering tools, you can click the icon to open the Select Resource window.

Creating a Resource

In most wizards, you can click the **Create New** button or icon to create a new resource.

In the model wizard, click the button or icon for the type of model you want to create:

 New Constant

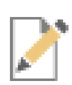
 New Distribution

 New Dynamic

 New Profile (available only when creating a cost or duration model for use in a spare part pool)

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 in the field or in the wizard.



Any changes you make will apply everywhere that the resource is used. This includes analyses in other ReliaSoft applications.

Removing a Resource

If a resource has already been assigned, you can remove it from this location by clicking the Remove button or icon.

The resource will remain in the database for use in other locations. If you want to completely remove a resource from the database, you'll need to use the Resource Manager or Select Resource window.

Universal Reliability Definitions (URDs)

A universal reliability definition (URD) describes a set of reliability and maintenance characteristics for a particular component/assembly. Like any other resource, you can create or edit URDs while you're performing a relevant analysis, and from the Resource Manager.

Note: Depending on where you are using a URD, only those properties that are relevant are applied. For example, if you apply a URD to a block in an analytical diagram in BlockSim, only the failure model associated with the URD will be applied to the block.

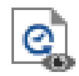
For a new resource, a name will be proposed automatically based on the default naming criteria established for the current database (see Default Name Formats window). You can replace this with your own name, if desired. Remember that the name and identifiers are the primary way in which your team will be able to find the resources you need for your analyses.

When editing the URD's properties, you can choose or create:

- A **model** to describe the behavior associated with the URD. This can be a reliability model, a probability of failure model or an event occurrence model.
- A **corrective task** that describes the maintenance action taken to restore a failed component to operational status.
- Zero, one or many **scheduled tasks** that describe the preventive maintenance, inspections and/or on condition maintenance tasks. The order in which the tasks are displayed reflects the priority with which tasks that are scheduled in the same way (see Task Scheduling) will be performed; for example, if there are two tasks scheduled based on interval and they conflict, the task higher in the priority list will be performed and the lower priority task will be disregarded. You can use the up and down arrows in the cells in this column to move the tasks up and down the priority list.

Additionally, the following are shown:

- **Identifiers** contains additional identifying information that can be used to search for this resource.
- **History** provides information about when the record was created and last updated. 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 record.



- **Watch** allows each individual user to subscribe to receive an alert (via e-mail, SMS text message or portal message) when the resource is changed.

Trace Usage

For existing resources, the link at the bottom of the window indicates how many times the resource is currently being used. If you need more information, click the link or the icon to open the Dependency Viewer.

Models

In ReliaSoft applications, *models* can represent probabilities, durations or costs, either fixed or time-dependent. They are used by other resources to represent the reliability of a component, the duration of a task, the expected cost of a repair and many other characteristics.

Like any other resource, you can create or edit models while you're performing a relevant analysis, and from the Resource Manager. In addition, you can publish models that are based on an existing analysis (e.g., a life data analysis in Weibull++, a diagram in BlockSim, etc.).

This topic describes the properties for both types of models. The interface will vary based on the particular situation.

Naming the Resource

For a new resource, a name will be proposed automatically based on the default naming criteria established for the current database (see Default Name Formats window). You can replace this with your own name, if desired. Remember that the name and identifiers are the primary way in which your team will be able to find the resources you need for your analyses.

Associated Analysis

In published models only, the **Associated Analysis** area identifies the analysis that the model is based on. If the published model reflects the latest analysis results, the status is "Synchronized." If the analysis has been modified since the model was last published, the status is "Out of Sync."

Tip: If you have the ability to open the original analysis (i.e., if the required ReliaSoft application is activated on your computer and your user account has permissions to access the analysis), the **Source** will be configured as a link and there will also be a **Data Source** button at the bottom of the window.

Model Category

The category determines where and how a model can be used. If you are using a resource wizard to create a new model, it will be assigned automatically based on what's relevant for the current field. If you are publishing a model or creating it from the Resource Manager, you must select the appropriate option. Once a model has been created, you cannot change its category.

- **Reliability**, **Probability of Failure** and **Event Occurrence** models represent a likelihood of occurrence.
- **Duration** models represent a length of time.
- **Cost per Unit Time** models are used for costs that accrue over time (e.g., the crew charges $50 per hour).
- **Cost** models are used for costs that don't depend on time (e.g., the part costs $100 or the crew charges $50 per service call in addition to their hourly rate).

Model Type

- A **distribution** model represents behavior that varies based on factors such as time and/or applied stress. For a published model, the inputs and parameters will be specified automatically based on the associated analysis. For manually created models, you can:
  - Select a distribution from the drop-down list and then enter the required parameter(s).

    What's Changed? Starting in Version 2020, two new distributions are available for use in models: The triangular distribution takes three parameters (lower limit, mode, upper limit) defining the shape of the *pdf*. The uniform distribution takes two parameters (lower limit, upper limit) defining the bounds of the *pdf*. For both distributions:

- Lower limit >= 0
- Upper limit >= 0
- Upper limit >= lower limit

For the triangular distribution:

- Mode >= 0
- Mode >= lower limit
- Mode <= upper limit

- Use the Quick Parameter Estimator (QPE) to estimate the parameters of a distribution based on what you know about the behavior.

Tip: A published model includes one additional parameter that is not relevant for manually created models. **PNZ** stands for *percent non-zero*. 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 does include zero failure times (such as out-of-the-box failures, for example). In such cases, the parameters are calculated based on the non-zero failure times, and then the PNZ value is used as a multiplier when calculating certain metrics (e.g., reliability, unreliability).

- A **constant** model represents a fixed probability (e.g., 0.9), duration (e.g., 2 hours) or cost (e.g., $10 or $10 per hour).
- A **dynamic** model represents a fixed probability, duration or cost, based on a specified variable. The variable can then be programmatically varied during simulation in one of two ways, thereby changing the value of the model for successive simulation runs.
  - For event analysis flowcharts, the variable can be varied using the 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.
- A **profile** model represents a changing duration or cost over time, based on a specified profile. Profile models are available only for the cost and duration model categories. The cost or duration of the model will be programatically varied based on an input time parameter used in the profile. When creating a profile model, you must specify a time unit; this will be used for the

profile’s segment times and, if applicable, for the profile’s output duration.

Profile models can be used for all direct costs and acquisition times for spare part pools. The execution time of the task using the spare part pool will be used as the input time parameter for the profile. Profile models are not available for selection anywhere other than in the relevant fields within spare part pools.

Note that models that are in use cannot be changed to profile models.

Additional Tools

Additionally, the following are shown:

- **Identifiers** contains additional identifying information that can be used to search for this resource.
- **History** provides information about when the record was created and last updated. 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 record.

- **Trace Usage.** For existing resources, the link at the bottom of the window indicates how many times the resource is currently being used. If you need more information, click the link or the icon to open the Dependency Viewer.

Publishing Models

This topic provides general information about how to create and work with *published models*, which are model resources that have been published from and continue to be associated with an existing analysis. For information about additional options and requirements in specific applications, see:

- "Publishing Fitted and Analytical Models" in the BlockSim documentation.
- "Publishing Models from R-DOE Results" in the Weibull++ documentation.
- "Publishing Models from Analysis Results" in the Weibull++ documentation (for growth data folios).
- "Publishing from Failure Rate Predictions" in the Lambda Predict documentation.

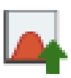
A ReliaSoft desktop application does not automatically update the published model when something changes in the original analysis, but users have the option to republish the model at any time. For any other analysis that uses the model, the change will be reflected the next time that other analysis is recalculated/resimulated.

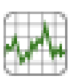
Publishing Tools

For most ReliaSoft desktop applications, the following model publishing tools are found on the Publishing page of the control panel. In Lambda Predict, they are located on the Publishing tab of the Properties panel.

 **Publish Model** publishes the analysis to a model, making it accessible to all ReliaSoft applications.

- If the analysis has never been published, this will create a new model and you will be prompted to specify the name and category.
- If the analysis has already been published, this will update the existing model and change the model's status from "Out of Sync" to "Synchronized."

 **Publish to Existing Model** allows you to select an existing model (using the Select Resource window) and replace it with an association to the current analysis.

 **Trace Usage** opens the Dependency Viewer, 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 user who is considering whether to use the model in a particular analysis.

 **Remove Association** removes the link between the current analysis and the published model. The model will continue to be a resource in the database but it can no longer be synchronized with the analysis.

Published Model Summary

After the model has been published, the summary table in the control panel (or Publishing tab in Lambda Predict) will display the model's name, along with the following details:

- When the **Status** changes from "Unpublished" to "Published," the label becomes a link that opens the model's properties window.
- **Linked?** indicates whether the published model has been used. If the model is in use, you can click the link in this field to open the Dependency Viewer.
- **Synchronized?** displays "Synchronized" if the published model reflects the latest results from the associated analysis. If that analysis has been modified since the model was last published (e.g., if more data has been added, an analysis setting has changed, etc.), the status will display as "Out of Sync."
- The **Created By** and **Modified By** fields display the names of the users who created and last updated the model. Click either of the links to see the dates and times.

Model Basis and Status

When a model has been published from an existing analysis, its properties window will include the following additional information:

Associated Analysis	
Model Name	Weibull_Folio1_Data1
Source	Folio1/Data1
Application	Weibull++
Type	Life Data Folio
Status	Synchronized

- **Source** is the ReliaSoft analysis that the model was published from.
- **Application** is the ReliaSoft application that is required to view/edit the original analysis.
- **Type** is the kind of folio or diagram that the model is based on. For example, this might be a life data folio in Weibull++, an analytical diagram in BlockSim, a failure rate prediction in Lambda Predict, etc.
- **Status** displays "Synchronized" if the published model reflects the latest results from the associated analysis. If that analysis has been modified since the model was last published (e.g., if more data has been added, an analysis setting has changed, etc.), the status will display as "Out of Sync."

Tip: If you have the ability to view the original analysis (i.e., if the required ReliaSoft application is activated on your computer and your user account has permissions to access the analysis), the **Source** will be configured as a link and there will also be a **Data Source** button at the bottom of the window.

Tasks

To properly analyze repairable systems, we first need to understand how components in these systems are restored (i.e., the maintenance activities that are performed on the components). In general, maintenance is defined as any action that restores failed units to an operational condition or keeps non-failed units in an operational state. For repairable systems, maintenance plays a vital role in the life of a system. It affects the system's overall reliability, availability, downtime, cost of operation, etc.

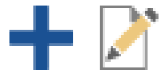
In ReliaSoft applications, maintenance activities are represented using tasks, which are resources that can be shared among analyses and can be managed via the Resource Manager. There are two basic kinds of tasks, which comprise four task classes:

- *Corrective tasks* are the action(s) taken to restore a failed component to operational status. These cannot be scheduled, as the component's exact failure time is not known before it happens.
- *Scheduled tasks* can be performed on a known schedule, based on time, component condition or other factors. These include:
  - Preventive tasks
  - Inspection tasks
  - On condition tasks

Tasks are assigned to URDs, which are in turn used to represent a set of properties that can be applied to standard blocks in RBDs and to events in fault trees.

What's Changed? Starting in Version 2019, crews are assigned to tasks as part of teams. This allows you to require multiple crews to complete a task (e.g., if a task requires both a mechanic and an electrician).

The Maintenance Task window allows you to create, view and edit all classes of maintenance tasks. It can be accessed by clicking the **Create New** or **View/Edit** icon in the Task wizard, which is accessed from **Task** fields in properties windows (e.g., the **Corrective Maintenance Task** field in the Universal Reliability Definition window).



It can also be accessed from the Corrective Tasks and Scheduled Tasks pages of the Resource Manager by choosing **Home > Edit > Add**, by selecting a task and choosing **Home > Edit > View** or by double-clicking a task.

For a new resource, a name will be proposed automatically based on the default naming criteria established for the current database (see Default Name Formats window). You can replace this with your own name, if desired. Remember that the name and identifiers are the primary way in which your team will be able to find the resources you need for your analyses.

The following options must be configured for all classes of tasks. Configuration options that are specific to particular task classes are presented in the corresponding sections.

- **Basic Task Properties**
  - **Task duration** allows you to assign a model, which may represent a fixed time or a distribution, to describe the duration of the task. You can choose an existing model or create a new one. If no model is assigned, it is assumed that the task has a duration of zero (i.e., immediate repair).
  - **Teams for task** allows you to choose or create one or more teams that can perform the task. Each team is made up of one or more crews; all crews assigned to the team are considered to be required (i.e., if any one of the crews assigned to a team is unavailable, the team is unavailable). You can add, remove or edit the crews assigned to any team. You can also remove a team from the task, thereby quickly removing all of its crew associations. An empty team is available at the end of the team list, allowing you to define a new team.

    If multiple teams are assigned to the task and a team is needed, they will be checked for availability in the order in which they are displayed in this list. Use the **Priority Up** and **Priority Down** arrows that appear when you click a team name in the list to move it up and down in the list. If a team is needed and all teams are engaged, the team with the shortest wait time (based on the longest wait time of its constituent crews) is chosen to perform the task.

    If no team is assigned, it is assumed that the work will be done by some undefined team that is always available.
- **Restoration** These properties allow you to specify the degree to which the block will be restored after the performance of the task. In the **How much does this task restore the item?** field, you can specify that the item will be returned to as good as new condition (i.e., full

repair, equal to a restoration factor of 1), to the same condition it was in when it failed (i.e., "as bad as old" or minimal repair, equal to a restoration factor of 0), or you can choose **Partial restoration**. This option provides you with the ability to model maintenance involving "used parts" or imperfect maintenance. The **Restoration amount** field will appear; starting in Version 2019, you can specify the amount of restoration achieved by the task by entering the value as a decimal from 0 to 1. If you specify anything other than 0 (0%) or 1 (100%), you will need to specify what the restoration effect applies to (i.e., the restoration type). You can select:

- **Only damage accumulated since last repair**: Each repair will remove only the damage since the last repair (i.e., the nth repair cannot remove the damage incurred before the (n-1)th repair). Note that in this context any task is considered a repair and any damage that has occurred since the last event (corrective task, preventive task, inspection or on condition task) will be reduced.
- **All accumulated damage**: Each repair can reduce any damage accumulated up to that failure.

For simulation, the application uses the restoration factor to determine the new age of the block after the maintenance action.

For example, consider an automotive engine that fails after 6 years. If the engine is rebuilt and the rebuilding task has a 50% restoration factor:

- If **Only damage accumulated since last repair** is selected, the initial rebuild has the effect of rejuvenating the engine to a condition as if it were 3 years old.

  The engine fails again after 3 years (when it again reaches the effective "age" of 6 years), but the rebuild this time affects only the age accumulated after the first rebuild. Thus the engine has an effective age of 4.5 years after the second rebuild (3 + 3 x (1 - 0.5) = 4.5).

  After the second rebuild, the engine fails again after a period of 1.5 years (when it again reaches the effective age of 6 years) and a third rebuild is required. The effective age of the engine after the third rebuild is 5.25 years (4.5 + 1.5 × (1 - 0.5) = 5.25).
- If **All accumulated damage** is selected, the initial rebuild has the effect of rejuvenating the engine to a condition as if it were 3 years old.

  The engine fails again after 3 years (when it again reaches an effective age of 6 years) and another rebuild is required. This rebuild also rejuvenates the engine by 50%, thus making it effectively 3 years old again.

  After the second rebuild, the engine fails again after a period of 3 years (when it again reaches the effective age of 6 years) and a third rebuild is required. The effective age of the engine after the third rebuild is 3 years.

Compare the following tables to see how the two options differ.

Only Damage Accumulated Since Last Repair

Time	Time Since Last Repair	Effective Age Before Repair	Effective Age After Repair
Start = 0	0	0	0
6 years	6	6	3
9 years	3	6	4.5
10.5 years	1.5	6	5.25

All Accumulated Damage

Time	Time Since Last Repair	Effective Age Before Repair	Effective Age After Repair
Start = 0	0	0	0
6 years	6	6	3
9 years	3	6	3
12 years	3	6	3

The ReliaWiki resource portal has more information on restoration factors at: http://www.reliawiki.org/index.php/Repairable_Systems_Analysis_Through_Simulation.

What's Changed? In versions prior to 2019, the **Restoration amount** was specified as a percentage rather than a decimal, and was entered either manually or using a slider bar.

- **Additional Costs to Consider** allows you to choose or create models to represent costs that are always associated with the task. **Cost per task** uses a cost model, and **Downtime rate** uses a cost per unit time model. If no models are assigned, it is assumed that there are no additional costs.
- **Identifiers** contains additional identifying information that can be used to search for this resource.
- **History** provides information about when the record was created and last updated. 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 record.

- **Watch** allows each individual user to subscribe to receive an alert (via e-mail, SMS text message or portal message) when the resource is changed.
- **Trace Usage.** For existing resources, the link at the bottom of the window indicates how many times the resource is currently being used. If you need more information, click the link or the icon to open the Dependency Viewer.

Corrective Tasks

Corrective maintenance consists of the action(s) taken to restore a failed component to operational status. Corrective maintenance is performed at unpredictable intervals because a component's failure time is not known *a priori*.

Corrective tasks:

- Always bring the block down.
- May bring the system down.
- Require spare parts.

In addition to the common task properties, the following options are used to configure corrective tasks in the Maintenance Task window:

- **Task Scheduling** allows you to specify when the corrective task will be performed.
  - If you choose **Upon item failure**, the task will be initiated upon failure of the component.
  - If you choose **When found failed during an inspection**, the task will be initiated if the component is found to be failed at the next scheduled inspection. This is useful in the case of "hidden failures" (i.e., failures that are not apparent until an inspection is performed) and in cases where the component is not considered mission critical and repair can wait until the next scheduled maintenance.
    - If a preventive task takes place before the next inspection, then preventive maintenance (not corrective maintenance) will be performed to restore the block. The downtime, etc. will depend on the preventive task properties.
    - If a corrective task is performed upon inspection and an inspection finds the block failed, then the total downtime includes the full inspection duration followed by the corrective task duration.

  If you have selected to perform a corrective task upon inspection and an inspection or preventive task does not occur after the failure, then the block will never be restored. This could happen under either of the following circumstances:

  - Neither an inspection nor a preventive task is specified for the block.

    and/or

  - The conditions have not been met to perform an inspection or preventive task.

- **Basic Repair Properties > Spare Part Pool** allows you to choose or create the spare part pool(s) that will be used in performing the task. A spare part pool describes the conditions that determine whether a spare part will be available when needed and specifies the time and costs associated with obtaining the spare part. If a spare part pool is not assigned, it is assumed that unlimited free spares are always immediately available.

  Starting in Version 2019, you need to specify a quantity of parts requested from the pool by the task. The default value in the **Quantity requested by task** field is 1.

  You can assign multiple spare part pools to a task in Version 2019 and later. In this case, the task is assumed to require all requested parts from all requested pools. In other words, the pools are in an AND relationship, not an OR relationship.

For corrective and preventive tasks, the simulation requests a team as soon as a task is initiated; however, the team does not begin performing the task unless/until all required spare parts are available.

The total time for the task consists of:

- The time to complete corrective and preventive maintenance (based on the corrective task or preventive task properties).

  PLUS

- The longer of the following times:
  - The time to obtain all spare parts (based on the spare part pools).

    OR

  - The time to obtain an available crew (based on crew availability) and any logistical delay time associated with the crew.

For example, consider a block that fails at 10 hours (on the system clock). Its corrective task will take 5 hours, the time required to obtain all spares will be 48 hours and the team consists of a single crew with a logistical delay time of 8 hours. In this case:

- All spares will arrive by 58 hours (on the system clock).
- The team will be requested at 10 hours and will arrive at 18 hours (on the system clock).
- The maintenance will be completed at 63 hours (on the system clock).
- The total time for the corrective maintenance will be 53 hours (48 for the spares plus 5 for the task).
- The team's crew will have traveled for 8 hours, waited 40 hours for the spare parts and performed the task for 5 hours for a total of 53 hours (including waiting for the spare parts).

If the spare parts are available without delay and all other conditions are the same, then:

- The team will be requested at 10 hours and will arrive at 18 hours (on the system clock).
- The maintenance will be completed at 23 hours (on the system clock).

- The total time for the corrective maintenance will be 13 hours (8 for the team plus 5 for the task).
- The team's crew will have traveled for 8 hours and performed the task for 5 hours, for a total of 13 hours.

- **Task Consequences**
  - **Does this task bring the system down?**: By default, corrective tasks will not bring the system down unless having the block down brings the system down based on the reliability-wise configuration in the diagram. If you answer **Yes**, the task will bring the system down even if the task has a zero duration. This forces the task to be included in the count of system downing events, regardless of the task's duration.
  - **Does this task bring the item down?**: It is assumed that a block will always be down when a corrective task is performed, even a task with a zero duration; thus, this option cannot be changed.

Preventive Tasks

Preventive maintenance is the practice of repairing or replacing components or subsystems before they fail in order to promote continuous system operation or to avoid dangerous or inconvenient failures. The schedule for preventive maintenance is based on observation of past system behavior, component wearout mechanisms and knowledge of which components are vital to continued system operation. In addition, cost is always a factor in the scheduling of preventive maintenance. In many circumstances, it is financially more sensible to replace parts or components at predetermined intervals rather than to wait for a failure that may result in a costly disruption in operations.

Preventive tasks:

- Always bring the block down.
- May bring the system down.
- Require spare parts.

There are restrictions on the tasks that can be performed simultaneously on the same block. Please refer to Multiple Tasks on the Same Block in the BlockSim documentation.

In addition to the common task properties, the following options are used to configure preventive tasks in the Maintenance Task window:

- **Task Class** allows you to specify the kind of maintenance that the task performs: preventive, inspection or on condition. The options available in the Maintenance Task window will vary depending on your choice in this field.

- **Task Scheduling** allows you to specify the circumstances under which the task will be performed. (See Task Scheduling.)
- **Basic Task Properties > Spare Part Pool** allows you to choose or create the spare part pool (s) that will be used in performing the task. A spare part pool describes the conditions that determine whether a spare part will be available when needed and specifies the time and costs associated with obtaining the spare part. If a spare part pool is not assigned, it is assumed that unlimited free spares are always immediately available.

  Starting in Version 2019, you need to specify a quantity of parts requested from the pool by the task. The default value in the **Quantity requested by task** field is 1.

  You can assign multiple spare part pools to a task in Version 2019 and later. In this case, the task is assumed to require all requested parts from all requested pools. In other words, the pools are in an AND relationship, not an OR relationship.

  For corrective and preventive tasks, the simulation requests a team as soon as a task is initiated; however, the team does not begin performing the task unless/until all required spare parts are available. For a complete explanation of how this affects the total time for the task, see Corrective Tasks.
- **Task Consequences**
  - **Does this task bring the system down?**: By default, preventive tasks will not bring the system down unless having the block down brings the system down based on the reliability-wise configuration in the diagram. If you answer **Yes**, the task will bring the system down, even if the task has a zero duration. This forces the task to be included in the count of system downing events, regardless of the task's duration.
  - **Does this task bring the item down?**: It is assumed that a block will always be down when a preventive task is performed, even a task with a zero duration; thus, this option cannot be changed.
  - **If bringing the item down causes the system to go down, do you still perform the task?**: If you answer **Yes**, the task will be performed even if doing so brings the system down, either because the task itself brings the system down or because the task brings the block down and the block being down causes the system to go down. If you answer No and the task brings the system down, the task will never be performed during simulation unless the system is already down for another reason. For instance, a preventive task that is specified to be performed upon system down will be performed even if it brings the system

down regardless of your answer here. This is because the system is already down, which is what triggered the task in the first place.

A preventive task that does not bring the system down at the preventive maintenance time will still be factored into the simulation even if its duration will bring the system down at a later time.

For a preventive task that is scheduled to occur based on item age and for which you have answered **No** to this question, if the task is going to bring the system down, then it will not take place. If, however, the block reaches the age again (after restoration by a corrective action, inspection or another type of preventive maintenance) and this time it will not bring the system down, then it will be performed.

- **Perform this task even if the item failed before this task was scheduled to occur?**: If you answer **Yes**, the task will be performed even if the item has already failed. This option is available only for the following tasks:
  - Preventive tasks and inspection tasks (including the inspection portion of on condition tasks) that are scheduled based on item age. In this case, the item's "clock" is not stopped upon failure, and the item's "virtual age" is used to trigger the task.
  - Inspection tasks (including the inspection portion of on condition tasks) that are scheduled upon system down or events in a maintenance group.
  - The inspection portion of on condition tasks scheduled based on calendar age.

  Note that this setting is not relevant when a task is scheduled to be performed as part of a maintenance phase in a phase diagram; all such tasks are performed regardless of whether the item has failed.

- **RCM** properties are text-based properties that are used to keep track of details that may be helpful in reliability centered maintenance analysis, but are not used in reliability/maintainability simulations. These properties are shown only if they are enabled for the project via the interface style settings in RCM++.
  - **Status**: The status of the task (choose from a drop-down list). This setting could be used if you want to keep track of all tasks that have been considered, regardless of whether they end up in the actual maintenance plan (e.g., recommended, rejected, assigned).
  - **Proposed Interval**: The interval that was initially proposed for the task. This may be different from the interval that is actually assigned to the task. For example, you may wish to use this property if the team originally suggests a particular interval for the task

(perhaps the calculated optimum interval) but then decides to assign a different interval (perhaps an interval that is more convenient for packaging a group of tasks).

- **Reference Document**: A reference to another document that provides more detailed information about the task (e.g., procedure instructions).
- **Condition**: A description of the condition that indicates that a failure will occur (e.g., a threshold for a measurement of wear, vibration, etc). Typically, this field is used for on condition maintenance tasks.
- **Zone**: The zone of the system in which the task will be performed. Typically, this field is used for aircraft MSG-3 analyses.
- **Access**: The access that will be required in order to perform the task. Typically, this field is used for aircraft MSG-3 analyses.

Note: A preventive task with a restoration factor of 0 will generate a new failure with the current age.

Inspection Tasks

Inspections are used in order to uncover hidden failures (also called "dormant failures"); they are also used as part of on condition tasks to detect impending failures so that preventive maintenance can be performed.

In general, no maintenance action is performed on the component during an inspection unless the component is found failed, in which case a corrective maintenance action is initiated. However, there might be cases where a partial restoration of the inspected item would be performed during an inspection. For example, when checking the motor oil in a car between scheduled oil changes, one might occasionally add some oil in order to keep it at a constant level. This sort of restoration is not considered to be preventive maintenance; the deciding factor is that inspection tasks do not use spare parts. Note that an inspection task that triggers a corrective task will not restore the failed block. Only the corrective task will restore the block.

Inspection tasks (including the inspection and associated minor work):

- May bring the block down.
- May bring the system down; if the task brings the system down, it also brings the block down.
- Do not use spare parts.

In addition to the common task properties, the following options are used to configure inspection tasks in the Maintenance Task window:

- **Task Class** allows you to specify the kind of maintenance that the task performs: preventive, inspection or on condition. The options available in the Maintenance Task window will vary depending on your choice in this field.
- **Task Scheduling** allows you to specify the circumstances under which the task will be performed. (See Task Scheduling.)
- **Task Consequences**: An inspection task will not bring the block down unless one of the following options is selected.
  - **Does this task bring the system down?**: If you answer **Yes**, when the task is performed, the system will be down, even if the task has a zero duration. If not selected, the task will bring the system down only if it brings the block down and having the block down brings the system down based on the reliability-wise configuration in the diagram. If this option is selected, even tasks with a zero duration will bring the system down. This forces the task to be included in the count of system downing events, regardless of the task's duration.
  - **Does this task bring the item down?**: If you answer **Yes**, when the task is performed, the block will be down. Even tasks with a zero duration will bring the block down in this case.
  - **Perform this task even if the item failed before this task was scheduled to occur?**: If you answer **Yes**, the task will be performed even if the item has already failed. This option is available only for the following tasks:
    - Preventive tasks and inspection tasks (including the inspection portion of on condition tasks) that are scheduled based on item age. In this case, the item's "clock" is not stopped upon failure, and the item's "virtual age" is used to trigger the task.
    - Inspection tasks (including the inspection portion of on condition tasks) that are scheduled upon system down or events in a maintenance group.
    - The inspection portion of on condition tasks scheduled based on calendar age.

    Note that this setting is not relevant when a task is scheduled to be performed as part of a maintenance phase in a phase diagram; all such tasks are performed regardless of whether the item has failed.
- **RCM** properties are text-based properties that are used to keep track of details that may be helpful in reliability centered maintenance analysis, but are not used in reliability/maintainability simulations. These properties are shown only if they are enabled for the project via the interface style settings in RCM++.

- **Status**: The status of the task (choose from a drop-down list). This setting could be used if you want to keep track of all tasks that have been considered, regardless of whether they end up in the actual maintenance plan (e.g., recommended, rejected, assigned).
- **Proposed Interval**: The interval that was initially proposed for the task. This may be different from the interval that is actually assigned to the task. For example, you may wish to use this property if the team originally suggests a particular interval for the task (perhaps the calculated optimum interval) but then decides to assign a different interval (perhaps an interval that is more convenient for packaging a group of tasks).
- **Reference Document**: A reference to another document that provides more detailed information about the task (e.g., procedure instructions).
- **Condition**: A description of the condition that indicates that a failure will occur (e.g., a threshold for a measurement of wear, vibration, etc). Typically, this field is used for on condition maintenance tasks.
- **Zone**: The zone of the system in which the task will be performed. Typically, this field is used for aircraft MSG-3 analyses.
- **Access**: The access that will be required in order to perform the task. Typically, this field is used for aircraft MSG-3 analyses

To understand how inspections work during simulation, you should be aware of the following:

- Multiple non-downing inspections cannot occur on the same block at the same time.
- A non-downing inspection with a restoration factor greater than 0 restores the block based on the age of the block at the beginning of the inspection (i.e., the task duration is not restored).

On Condition Tasks

On condition maintenance relies on the capability to detect failures before they happen so that preventive maintenance can be initiated. If, during an inspection, maintenance personnel can find evidence that the equipment is approaching the end of its life, then it may be possible to delay the failure, prevent it from happening or replace the equipment at the earliest convenience rather then allowing the failure to occur and possibly cause severe consequences. In BlockSim, on condition tasks consist of an inspection task that triggers a preventive task when an impending failure is detected during inspection.

For on condition tasks, the Inspection properties allow you to configure the inspection portion of the task, and the options are identical to those available for configuring inspection tasks. The On Condition Task (Upon Detection) properties allow you to configure the preventive portion of the task. The options are the same as those available for configuring preventive tasks, except that the

Task Scheduling properties are not available. Instead, Failure Detection properties are used to specify the "warning period" that spans from the time when a potential failure can first be detected to the time when the failure occurs.

To define the Failure Detection properties, first specify when the task is likely to detect an imminent failure. If you choose **When a certain percentage of the life of the item has been consumed**, you must then specify the percentage of the item's life that must have elapsed in order for approaching failure to be detected. This is called the *failure detection threshold* (FDT). For example, if the FDT is 0.9 and the item will fail at 1,000 days, the approaching failure can begin to be detected at 900 days. If you choose **Within a fixed time frame prior to failure**, you must then specify the amount of time before a failure when the approaching failure can be detected by inspection. This is called the *P-F interval*. For example, if the P-F interval is 200 days and the item will fail at 1,000 days, the approaching failure can begin to be detected at 800 days.

On condition tasks can be set to perform the preventive portion of the task even if the item failed before the approaching failure could be detected. If the inspection portion of the task is scheduled based on item age, you can answer **Yes** to the **Perform this task even if the item failed before this task was scheduled to occur?** option. (For more about this option, see Inspection Tasks.)

Note the following simulation assumptions regarding on condition tasks:

- An inspection that finds a block at or beyond the failure detection threshold or within the range of the P-F interval will trigger the associated preventive task as long as preventive maintenance can be performed on that block.
- If a non-downing inspection triggers a preventive maintenance action because the failure detection threshold or P-F interval range was reached, no other maintenance task will be performed between the inspection and the triggered preventive task; tasks that would otherwise have happened at that time due to system age, system down or group maintenance will be ignored.
- A preventive task that would have been triggered by a non-downing inspection will not happen if the block fails during the inspection, as corrective maintenance will take place instead.
- If a failure will occur within the failure detection threshold or P-F interval set for the inspection, but the preventive task is only supposed to be performed when the system is down, the simulation waits until the requirements of the preventive task are met to perform the preventive maintenance.
- If the on condition inspection triggers the preventive maintenance part of the task, the simulation assumes that the maintenance team will forego any routine servicing associated with the inspection part of the task. In other words, the restoration will come from the preventive main-

tenance, so any restoration factor defined for the inspection will be ignored in these circumstances.

Task Scheduling

The following information applies to the task scheduling properties for preventive tasks, inspection tasks and the inspection portion of on condition tasks.

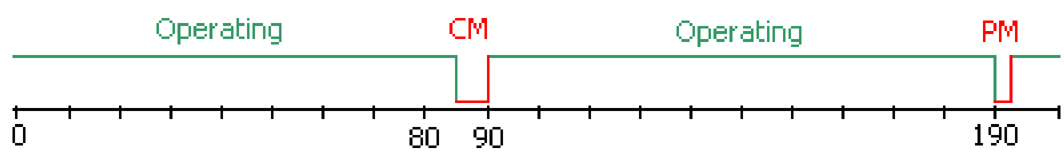
Click the arrow in the **When is this task performed?** field to define the criteria for performing the task. The remaining fields in the task scheduling properties will depend on your selection in this field. The task can be performed:

- **At certain intervals**. If you select this option, you must then specify:
  - Whether the interval is fixed or dynamic.
  - Whether the interval is based on the item's age (taking duty cycle into account) or on calendar time.
- **Upon certain events**. If you select this option, you can choose for the task to be performed:
  - Whenever the system is down for any reason. No further task scheduling properties are required for this option.
  - Based on events in a maintenance group. The task will be performed when user-specified events occur for either some or all blocks in one or more user-specified maintenance groups. The item that the task is assigned to does not need to be part of the selected maintenance group(s).
  - When a maintenance phase in a phase diagram starts. Please note that if the task is selected to be performed only at the start of a maintenance phase and the item that the task is associated with is not included in the maintenance phase's associated maintenance template, the task will never be performed. No further task scheduling properties are required for this option.
  - Based on events in a flow group (see Flow Groups in the BlockSim documentation). This setting is appropriate only for tasks used in process flow simulation diagrams. The task will be performed when flow either reaches/exceeds its threshold or drops to or below its threshold in one or more user-specified flow groups. The block(s)/item(s) affected by the task may or may not belong to any of the flow groups.

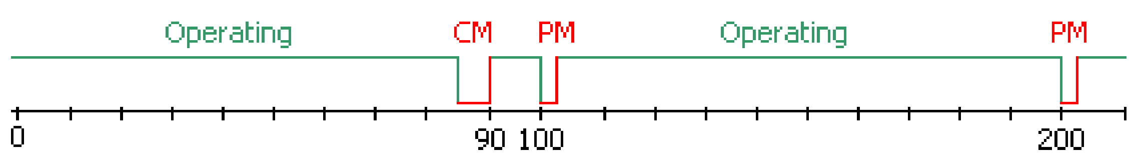
The remaining task scheduling properties for each type of schedule are given next. For any interval that you specify, you will be asked to specify the units used.

- **Fixed interval based on either item age or calendar time**: Enter the interval when the task will be performed (e.g., if you enter 100, the task will be scheduled for 100, 200, 300, etc.).

- Item age refers to the accumulated age of the block, which gets adjusted each time the block is repaired (i.e., restored). If the block is repaired at least once during the simulation, this will be different from the elapsed simulation time. For example, if the restoration factor is 1 (i.e., "as good as new") and the assigned interval is 100 days based on item age, then the task will be scheduled to be performed for the first time at 100 days of elapsed simulation time. However, if the block fails at 85 days and it takes 5 days to complete the repair, then the block will be fully restored at 90 days and its accumulated age will be reset to 0 at that point. Therefore, if another failure does not occur in the meantime, the task will be performed for the first time 100 days later at 190 days of elapsed simulation time.



- Calendar time refers to the elapsed simulation time. If the assigned interval is 100 days based on calendar time, then the task will be performed for the first time at 100 days of elapsed simulation time, for the second time at 200 days of elapsed simulation time and so on, regardless of whether the block fails and gets repaired correctively between those times.



If the task is performed at fixed intervals, you can select the **Override task scheduling properties with a task package** option to assign the task to a task package, which is a group of tasks that are performed together at scheduled intervals. (Note that if a task has been assigned to a task package, the task scheduling properties will be disabled and displayed in italics.)

- **Dynamic interval based on either item age or calendar time**: The task will be performed at variable user-specified intervals. This can be used, for example, to schedule maintenance to be performed with increasing frequency as an item gets older. The **Dynamic intervals** field shows the number of intervals specified. Click the button (**...**) to open the Intervals window, which allows you to enter the item age intervals when the task will be performed. The intervals are the actual time between maintenance (i.e., they are not cumulative). Note that the last value entered in the Intervals window will repeat, if necessary, until the end of the simulation time.

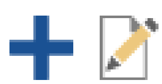
- **Based on events in a maintenance group**: Specify the events that can trigger the task (i.e., a block fails; a corrective, preventive or inspection task starts; a block is restored to operation), and choose the maintenance group(s) in which the event(s) must occur in order to trigger the task.
- **Based on events in a flow group**: Specify the events that can trigger the task (i.e., flow rate through any connector exceeds or drops below a certain level), and choose the flow group(s) in which the event(s) must occur in order to trigger the task.

Note: For preventive and inspection tasks that are scheduled based on item age, the **Perform this task even if the item failed before this task was scheduled to occur?** property determines whether actual age or "virtual age" is considered. If you answer **Yes**, the task will be performed even if the item has already failed. In other words, the item's "clock" is not stopped upon failure, and the item's virtual age is used to trigger the task. (For more about this option, see Preventive Tasks.)

Crews

Crews are resources that can be shared among analyses and can be managed via the Resource Manager. You can assign the crews to maintenance tasks. Starting in Version 2019, crews are assigned to maintenance tasks as part of teams. This allows you to require multiple crews to complete a task (e.g., if a task requires both a mechanic and an electrician). A team can also consist of a single crew.

The Crew window allows you to create, view and edit crews. It can be accessed by clicking the **Create New** or **View/Edit** icon in the Crew wizard, which is accessed from **Crew** fields in properties windows (e.g., under **Teams for task** in the Maintenance Task window).



It can also be accessed from the Crews page of the Resource Manager by choosing **Home > Edit > Add**, by selecting a crew and choosing **Home > Edit > View** or by double-clicking a crew.

For a new resource, a name will be proposed automatically based on the default naming criteria established for the current database (see Default Name Formats window). You can replace this with your own name, if desired. Remember that the name and identifiers are the primary way in which your team will be able to find the resources you need for your analyses.

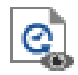
The following settings are available to configure the crew:

- **Crew**
  - **Direct cost** allows you to choose or create a model to represent the direct cost per unit time to engage the crew (e.g., the hourly charge for the time that the crew spends performing the task). This field uses a cost per unit time model. If no model is assigned, it is assumed that there is no cost.
  - **Cost per incident** allows you to choose or create a model to represent the cost per incident to engage the crew (e.g., if you have to pay a fixed fee instead of or in addition to the hourly rate). This field uses a cost model. If no model is assigned, it is assumed that there is no cost.
  - **Is there a limit to the number of tasks this crew can perform at the same time?**: If you answer **Yes**, you must specify the limit in the **Number of tasks** field. For example, if the crew can only fix 3 components at any given time and 4 components fail at the same time, then the crew will not be able to fix the fourth component until one of the other components has been restored.
  - **Logistic Delay** allows you to choose or create a model to describe the delay time before the crew can start the task. If no model is assigned, it is assumed that the crew can start the task immediately without any delays (i.e., immediate repair). The team with the highest priority is always called upon first, regardless of logistic wait times for the team's crews. If a team is needed and all teams are engaged, the team with the shortest wait time (based on the longest wait time of its constituent crews) is chosen to perform the task.

    For a given simulation, a crew's logistic time is constant across that one simulation for the task. It is taken randomly from its distribution or from its fixed time.

    For on condition tasks, if the same crew performs the inspection and preventive maintenance and there are logistic delays associated with the crew, the logistic delays will be factored into the simulation for both parts of the task.
  - If **Include logistic delay in cost** is selected, then the time spent waiting for the crew (including logistic delay and any delay incurred due to limits on the number of tasks the crew can perform simultaneously) will be included when calculating crew costs, based on the direct cost specified for the crew.
  - If **Include part delay in cost** is selected, then the time spent by the crew waiting for spare parts will be included when calculating crew costs, based on the direct cost specified for the crew.
  - **Identifiers** contains additional identifying information that can be used to search for this resource.

- **History** provides information about when the record was created and last updated. 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 record.



- **Watch** allows each individual user to subscribe to receive an alert (via e-mail, SMS text message or portal message) when the resource is changed.

- **Trace Usage.** For existing resources, the link at the bottom of the window indicates how many times the resource is currently being used. If you need more information, click the link or the icon to open the Dependency Viewer.

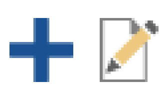
Spare Part Pools

Spare part pools are resources that can be shared among analyses and can be managed via the Resource Manager. A spare part pool describes the conditions that determine whether a spare part will be available when needed and specifies the time and costs associated with obtaining the spare part. You can assign the pools to maintenance tasks.

What's Changed? Starting in Version 2019, tasks can request multiple spare parts from a pool. All requested parts must be delivered before the task can be performed.

In addition, multiple spare part pools can be assigned to tasks. In this case, the task is assumed to require all requested parts from all requested pools.

The Spare Part Pool window allows you to create, view and edit spare part pools. It can be accessed by clicking the **Create New** or **View/Edit** icon in the Spare Part Pool wizard, which is accessed from **Spare Part Pool** fields in properties windows (e.g., in the Maintenance Task window).



It can also be accessed from the Spare Part Pools page of the Resource Manager by choosing **Home > Edit > Add**, by selecting a spare part pool and choosing **Home > Edit > View** or by double-clicking a spare part pool.

For a new resource, a name will be proposed automatically based on the default naming criteria established for the current database (see Default Name Formats window). You can replace this with your own name, if desired. Remember that the name and identifiers are the primary way in which your team will be able to find the resources you need for your analyses.

The following settings are available to configure the spare part pool:

Tip: In Version 2019 and later, profile models can be used to define any acquisition time and non-holding costs. This new model type allows you to model changes in cost and duration over time. You can also still use all other model types for these fields.

- **Spares**
  - **Direct cost per dispensed item** allows you to specify the direct cost of each spare part in the pool.
  - **Spare acquisition type**: You can select **Unlimited spares** to indicate that an unlimited number of spare parts exist in the pool (i.e., if a spare part is required, it will always be available). If you select **Limited number of spares**, the Pool Restock properties and the Emergency Spare Provisions properties will become available. In addition, you will need to define the following properties:
    - **Initial stock level** is the number of spare parts in the pool at the start of simulation.
    - **Holding Cost ($/Hour in pool)** allows you to choose or create a model to represent the cost of holding a spare part in the pool. The unit used is the default unit for the database. This field uses a cost per unit time model. If no model is assigned, it is assumed that there is no cost. (To determine which unit is defined as the default, choose **File > Manage Database > Unit Settings**. The default unit is the one that is selected in the Use as Default column.)
    - If you select the **Pool has maximum capacity** check box, you will need to specify the maximum number of items that can exist in the spare part pool at any given time. The restock options will add parts only up to this maximum capacity. For example, suppose that the maximum capacity is 20 units and the restock quantity is 5 units. If the current stock level is 18 when a restock is triggered (either on a scheduled interval or because the stock fell to a specified level), then only 2 units will be added to the spare part pool, even though the restock options call for the addition of 5 units at each restock.

If you select **Fixed probability of stockout**, the Emergency Spare Provisions properties will become available. In addition, you will need to enter the probability of running out of stock as a decimal value.

- **Logistic Time for Spare Acquisition** allows you to choose or create a model to describe the amount of time required to obtain the part when it is required for maintenance, assuming that the part is in stock. This field uses a duration model. If no model is assigned, it is assumed that the spare part is available for use immediately.

- The **Pool Restock** properties allow you to define how the spare part pool will be restocked. These properties are available only if you have selected the **Limited number of spares** option in the **Spare acquisition type** field. You can select either or both of the following restock schemes:

  - If the **Scheduled restock** option is selected, the pool will be restocked at fixed intervals. The following properties will be available:

    - **Restock every** allows you to specify the time interval at which the pool will be restocked.

    - **Number added per restock** allows you to specify the number of parts that are added to the pool at every scheduled interval.

  - If the **Restock as needed** option is selected, the pool will be restocked when the stock drops to a specified quantity. This option is available only if the initial stock level entered is at least 1. The following properties will be available:

    - **Restock when stock drops to** allows you to specify the number of parts in the pool that will trigger the restock (e.g., when the stock drops to 5, order more parts).

    - **Number added per restock** allows you to specify the number of parts that are added to the pool when the restock condition has been met.

    - **Required time for stock arrival** allows you to choose or create a model to describe the amount of time required for the new parts to arrive after the restock has been initiated. This field uses a duration model. If no model is assigned, it is assumed that the parts arrive immediately.

    - **Off-site spare part pool** allows you to assign an off-site spare part pool to the current pool. If specified, this pool is considered to be the source for restocking the current pool, which allows you to describe properties for the next level of spare parts that can be accessed when needed. If no off-site spares pool is specified, the parts are assumed to be ordered and no properties are described for the source.

Tip: Circular references between spare part pools are not permitted. In other words, if Pool 1 calls Pool 2 as its off-site spare part pool, and Pool 2 calls Pool 1 as its off-site spare part pool, BlockSim will display a message notifying you of the problem upon simulation.

- The **Emergency Spare Provisions** properties allow you to define the circumstances when the spare part pool is empty but a spare part is required for a task. These properties are available only if you have selected either the **Limited number of spares** option or the **Fixed probability of stockout** option in the **Spare acquisition type** field. Selecting the **Can obtain emergency spares if needed** option indicates that a spare part can be obtained if required and the spare part pool is empty; otherwise, the repair will be delayed until more spare parts arrive in the spare part pool. Selecting this option makes the following properties available:
  - **Number added per emergency** allows you to specify the number of parts that are added to the pool when an emergency acquisition occurs.
  - **Additional costs for emergency spares** allows you to choose or create a model to represent a per incident cost for emergency acquisition of spares (e.g., there is a $50 surcharge when emergency spares are required). This field uses a cost model. If no model is assigned, it is assumed that there is no cost.
  - **Required time for emergency spare** allows you to choose or create a model to describe the amount of time required for the emergency spare parts to arrive after the emergency acquisition has been initiated. This field uses a duration model. If no model is assigned, it is assumed that the parts arrive immediately.
  - **Off-site spare part pool** allows you to assign an emergency off-site spare part pool to the current pool. This allows you to create another level of spare parts that can be accessed for the emergency acquisition.
- **Identifiers** contains additional identifying information that can be used to search for this resource.
- **History** provides information about when the record was created and last updated. 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 record.

- **Watch** allows each individual user to subscribe to receive an alert (via e-mail, SMS text message or portal message) when the resource is changed.

- **Trace Usage.** For existing resources, the link at the bottom of the window indicates how many times the resource is currently being used. If you need more information, click the link or the icon to open the Dependency Viewer.

Metrics

A *metric* is a resource that provides a flexible way to display any numerical value of interest, and track how it changes over time. This can be used for tracking and displaying a variety of Key Performance Indicators (KPIs) and other values, including:

- Reliability metrics (e.g., reliability, BX% life, etc.) calculated from a model resource.
- Any result from a simulated BlockSim diagram.
- The completion percentage and resource usage for Project Planner gates.
- The reliability or availability calculated from an FMRA in XFMEA/RCM++.
- Any other user-defined value that you wish to display and track.

Like any other resource, you can create or edit metrics from the Resource Manager, and while you're performing a relevant analysis. (See Showing Metrics in Folios/Diagrams, Using Metrics in Project Planner Gates and Pushing Metrics from an FMRA.)

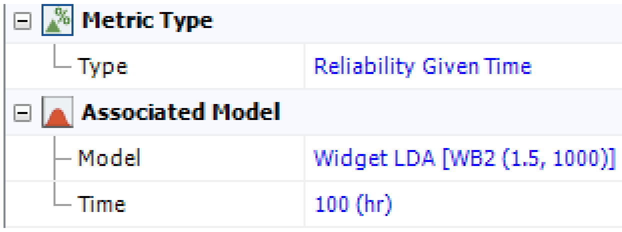
If your organization chooses to implement an SEP web portal, it's easy to choose specific metrics that you want to access quickly from any web-enabled device.

Note: Metrics are always local resources. They cannot be made global. Metrics in a reference project are local to that project and can't be selected for use in other projects.

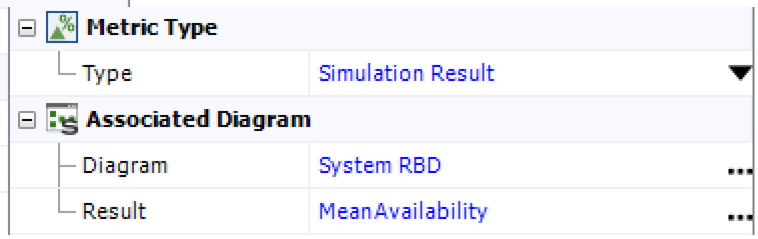
Metric Types

The following pictures demonstrate the four basic options for specifying the value that will be displayed and tracked.

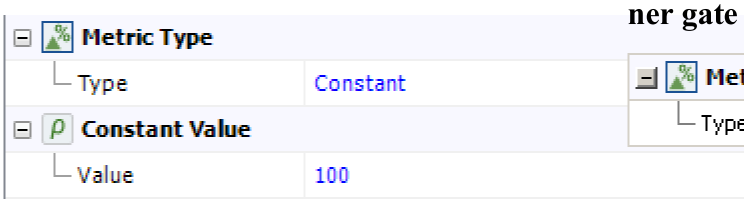
Calculate from a model resource



Obtain from a BlockSim simulation diagram



Enter any user-defined value

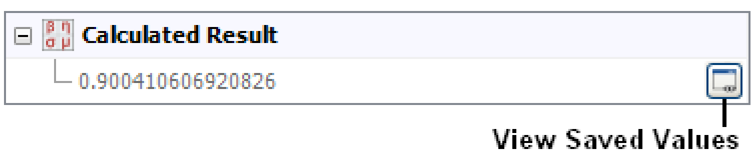


The value is "pushed" from a Project Planner gate or FMRA

Calculated Result and Saved Values

The **Calculated Result** field shows the latest calculated value for the metric. This will be updated automatically if the associated analysis or model changes, and the latest result will be stored in an array of saved values.

You can then click the **View Saved Values** icon to view the history of how the metric has changed over time.



If you change any of the properties that affect the way the result is calculated, you will be prompted to decide whether to clear the array of values that were saved before the change. For example:

- If you change the metric from Reliability to Probability of Failure, or from 1,000 hours to 500 hours, you will probably want to clear the array so the timeline of saved values is not misleading.
- Alternatively, if you change the metric's associated model from prototype1 to prototype2 to prototype3 at different stages of development, you may wish to keep the saved values after each change so the timeline shows the improvement across all three stages.

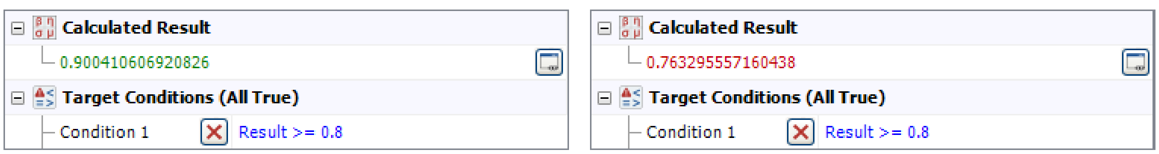
In the Metric Results Viewer, you can also choose to clear the entire history or delete only a particular saved value.

The results viewer also allows you to copy the values to the Clipboard so you can paste into another application. For example, if the metric stores the calculated reliability at different points in time, you may wish to analyze the data with a reliability growth model in a growth data folio.

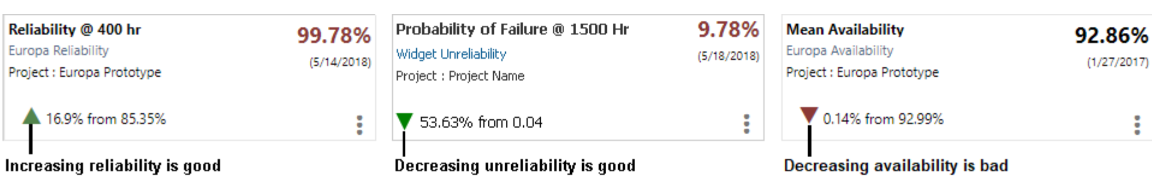
- **Copy > All rows (dates)** copies the exact dates shown in the table.
- **Copy > All rows (cumulative days)** replaces the dates with the cumulative number of days since the date of the last saved value. For example, if the first date is 1/1/2019 and the second date is 12/31/2019, the cumulative days would be 0 for the first date and 365 for the second date.

Target Conditions and Result Indicator

The **Target Conditions** feature color-codes the current value of the metric. If all specified conditions are met, then the metric's value will be displayed in green. If one or more conditions are not met, then it will be in red. For example, in the following pictures, the result is green if the estimated system reliability is at least 80%, and red if it is not.



The **Result Indicator** feature color-codes the indicator that shows whether the value has increased or decreased since the prior saved value. If the direction of the change is desirable, the indicator will be green; if it is undesirable, the indicator will be red. As an example, the following pictures are from the SEP.



Identifiers, History, Watch and Trace Usage

As with most other resources:

- **Identifiers** contains additional identifying information that can be used to search for this resource.

- **History** provides information about when the record was created and last updated. 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 record.

- **Watch** allows each individual user to subscribe to receive an alert (via e-mail, SMS text message or portal message) when the resource is changed.

- **Trace Usage.** For existing resources, the link at the bottom of the window indicates how many times the resource is currently being used. If you need more information, click the link or the icon to open the Dependency Viewer.

Showing Metrics in Folios/Diagrams

The Publishing page of the control panel shows all of the metric resources that are associated with the current analysis. (The metrics can be created and edited either from the Resource Manager or the control panel.) This applies for:

- Data sheets in Weibull++ that have a published model.

- Analytical or simulation diagrams in BlockSim that have a published model for the entire system (either a fitted model or an analytical model).

- Simulation diagrams in BlockSim that have simulation results.

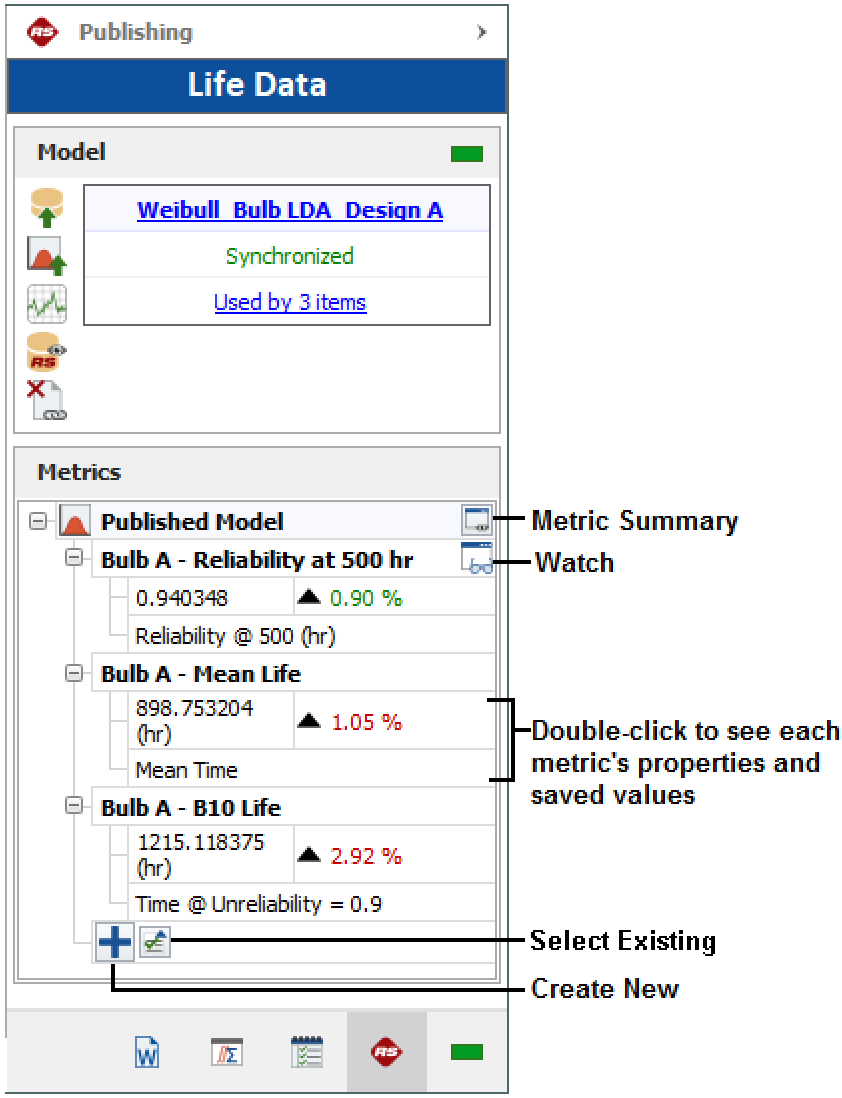
This makes it easy to see the metrics that have been created for each analysis (which may also be displayed in relevant Project Planner gates or in the SEP web portal) and to see the history of how the analysis results changed over time. It also provides a convenient way to create new metrics for the analysis, if desired.

As an example, the following picture shows a typical scenario for Weibull++. In Weibull++, the control panel shows all metrics that are associated with the model published from the current data sheet. In BlockSim, it shows all metrics that are associated with a model published from the diagram and/or obtained from the diagram's simulation results.

- Double-click a metric to see its properties and history of saved values.

- A **Watch** icon will be shown beside any metric for which you have subscribed to receive an alert when the resource is changed.

- Click the **Metric Summary** icon to see a quick report of all the metrics displayed in the current branch.
- Click the **Create New** icon to create a new metric.



Associating Existing Metrics with a Different Data Sheet

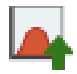
When an analysis changes (e.g., new data, after a design change, etc.), you may choose to simply update the existing data sheet and recalculate, or keep the original analysis unchanged and perform the updated analysis in a new data sheet.

If you decided to create a new data sheet, there are two ways to ensure that the metrics will reflect the complete history from both analyses:

PUBLISH TO THE EXISTING MODEL FROM THE NEW DATA SHEET

If you want the published model to reflect the latest analysis results, you can simply transfer both the model and the associated metrics to the new data sheet. To do that:

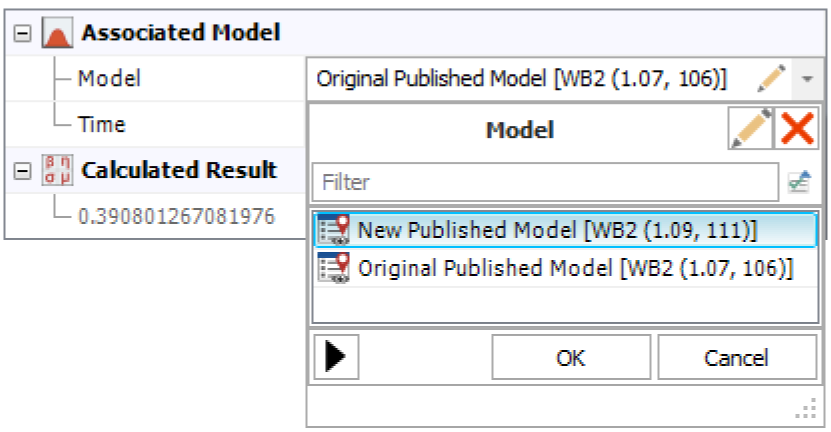
1. Calculate the new data sheet and then go to the Publishing page of the control panel.
2. Click **Publish to Existing Model** and then select the model from the original data sheet.



CHANGE THE MODEL FOR THE FOR THE EXISTING METRICS

If you want to keep the original published model associated with the original data sheet, you can publish a new model for the new data sheet and then transfer the metrics to the new model. To do that:

1. Publish a new model for the new data sheet.
2. From either the original data sheet's control panel or the Resource Manager, open each metric that was associated with the original data sheet and change the **Associated Model**.



As a shortcut, you can click the **Select Existing** icon on the new data sheet's control panel and choose one or more existing metrics in the Select Resource window. It is important to keep in mind that when you do this, you are changing the associated model for each selected metric.

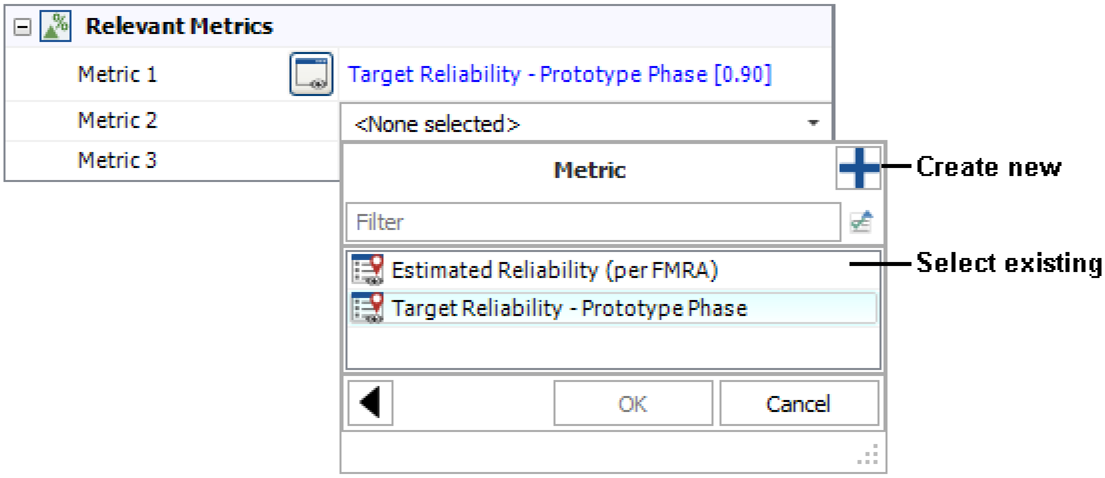
Using Metrics in Project Planner Gates

There are two ways to utilize metrics in a Project Planner gate:

Relevant Metrics

For any gate in a project plan, you can choose to display up to three metrics that are relevant to this stage of the project (e.g., the target reliability, the latest calculated reliability, etc.).

Use the fields under the **Relevant Metrics** heading to select an existing metric or create a new one.



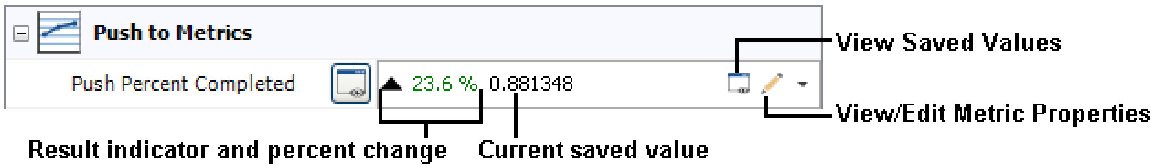
Push to Metrics

For any gate in a project plan, you can also choose to push any of the following values to a metric resource. This makes it easy to see the history of all saved values, and enables users to subscribe to receive alerts when the metric changes. You can also choose to show this information in the SEP web portal if it is implemented for the current database.

- **Percent Completed** is the percentage of the total duration of dependent actions/gates that is complete. For example, if Action 1 (duration = 3 days) is complete and Action 2 (duration = 1 day) is incomplete, the progress is 75%.
- **Actual Man Hours %** is the percentage of planned hours that have been used to complete the gate.
- **Actual Costs %** is the percentage of the budget that has actually been used to complete the gate.

Use the fields under the **Push to Metrics** heading to create a new metric or select an existing metric that is not already assigned to receive a pushed value. (Note that only unused metrics are shown in this list.)

When you first assign a metric in one of these fields, the current value will be automatically saved to the metric. Subsequently, the value will be recalculated each time an associated action or gate is changed. If the value is found to have changed, it will be saved to the metric. To see the current saved value, click inside the field. This will also display a result indicator and the **View Saved Values** icon, which you can click to see a history of all saved values.

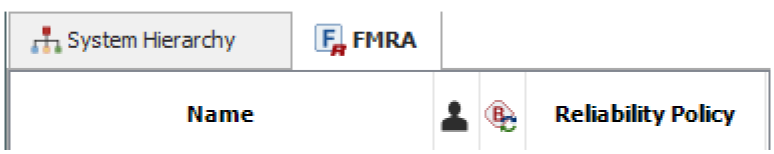


Pushing Metrics from an FMRA

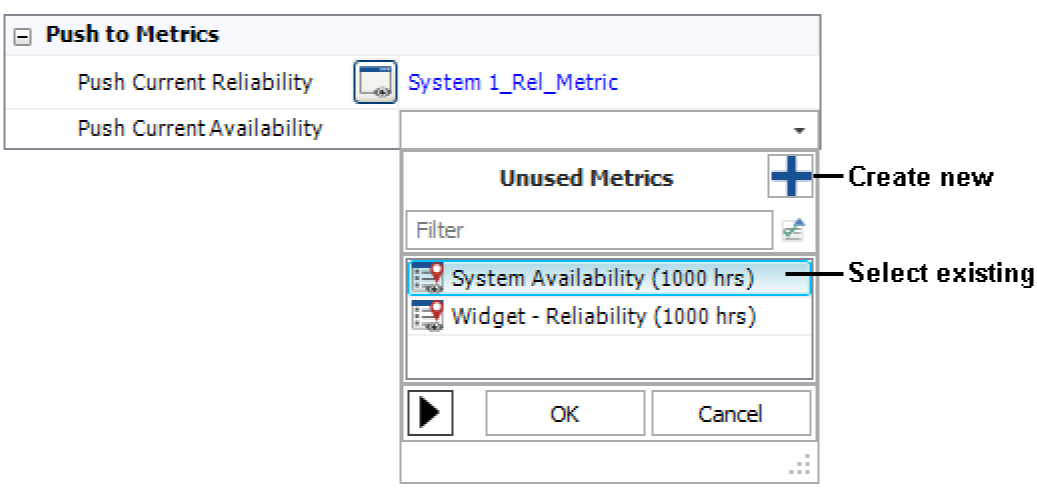
The **Push to Metrics** fields on the Properties tab of the Analysis panel in XFMEA/RCM++ enable you to "push" the reliability and availability values that were calculated/simulated from the Failure Modes and Reliability Analysis (FMRA) to metric resources.

This makes it easy to see the history of all saved values while you're working in the FMRA, and enables users to subscribe to receive alerts when the metric changes. If desired, you can also choose to show this information in a specific Project Planner gate (under the **Relevant Metrics** heading), and in the SEP web portal if it is implemented for the current database.

1. Go to the FMRA tab in the system panel. (See Enabling and Viewing the FMRA in the XFMEA/RCM++ documentation.)

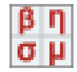


2. Select any item in the FMRA hierarchy and use the fields under the **Push to Metrics** heading to create a new metric or select an existing metric that is not already assigned to receive a pushed value.



3. Calculate and/or simulate the FMRA:

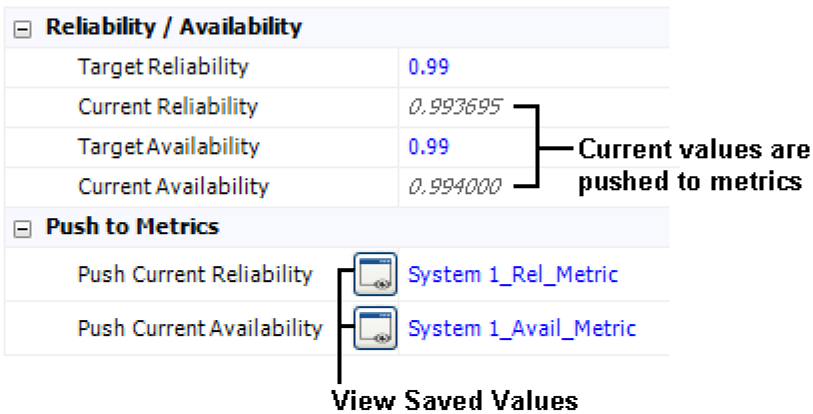
   a. To calculate the reliability, choose **FMRA > Calculations > Calculate (Reliability)**.

   b. To simulate the availability, choose **FMRA > Calculations > Simulate (Availability)**.

4. Each time you calculate/simulate the FMRA, the latest value(s) will be automatically saved to the metric(s). To see the history of all saved values, click **View Saved Values**.



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 in BlockSim. You can use variables:

- In event analysis flowcharts, to act as a constant (i.e., maintain a fixed numerical value during simulation).
- In event analysis flowcharts, to temporarily store the output value of a block during simulation. See the next section for more information.
- In event analysis flowcharts, 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, 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.
- In process flow simulation diagrams, to define a throughput amount.
- As inputs for certain fields in simulation RBDs via simulation worksheets, allowing programmatic variation of the field value during simulation.

Using Variables to Store Output Values in Event Analysis Flowcharts

You can use variables to hold numerical values passed to them during simulation. In event analysis flowcharts, 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:

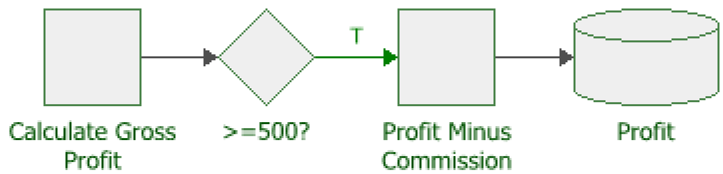
- **Reset After Each Simulation** (i.e., reset after a single pass through the flowchart)
- **Reset After Each Run** (i.e., reset after each set of simulations)
- **Reset After Analysis** (i.e., reset after each set of runs)
- Never Reset

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 can use a result storage block to store it.

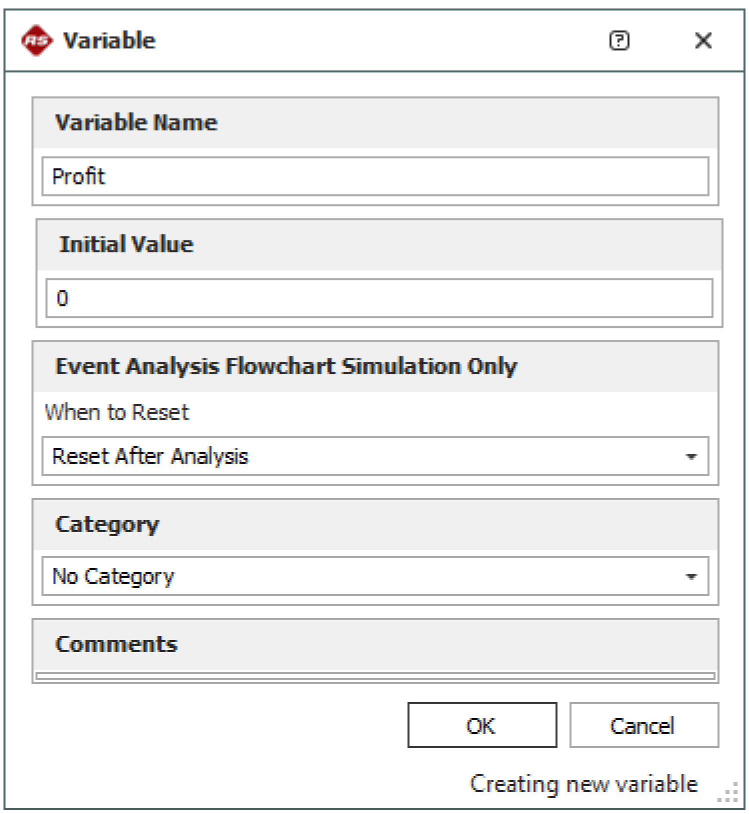
Example

The following example demonstrates two things: a) how a variable may be used to store an output value and b) how a variable may be used in an equation.

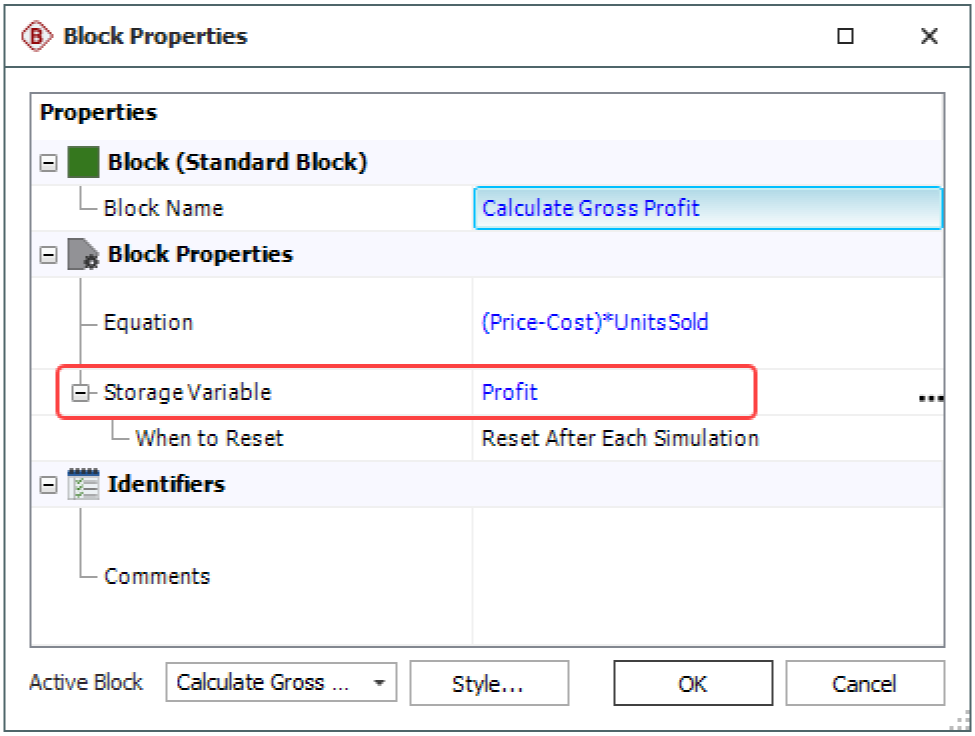
In the following configuration, the first block calculates the gross profit from a sale, and then stores its output into a variable called "Profit." The conditional block determines whether the gross profit exceeds 500. If the output is true, the third block deducts the sale's commission from the profit, and overwrites the stored value with the new output.



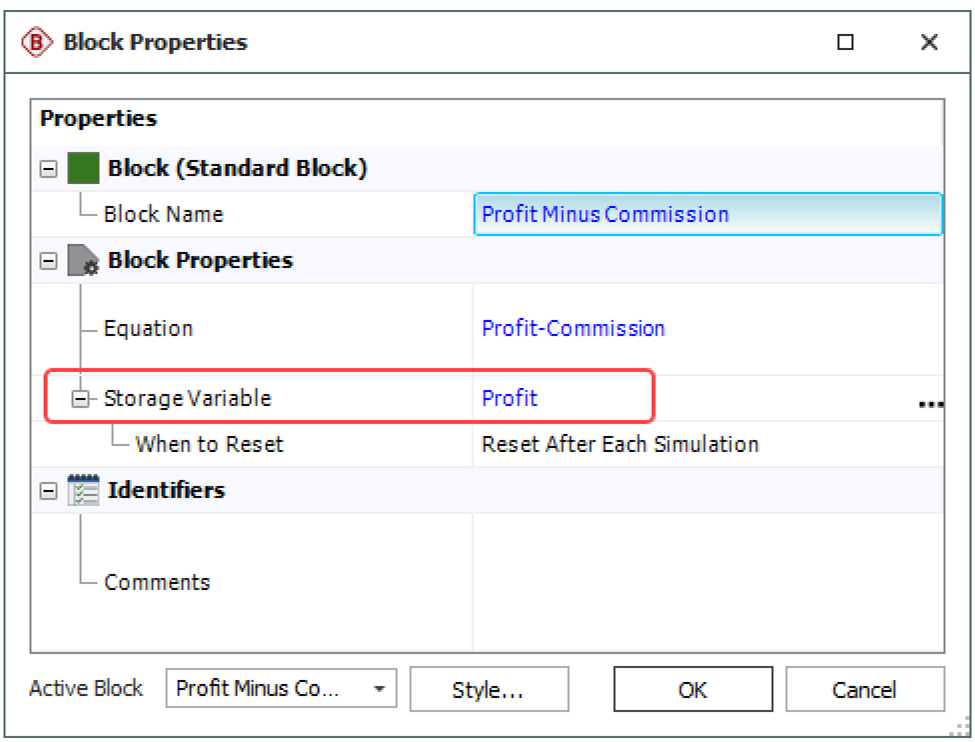
The following picture shows the properties window of the variable called "Profit." The initial value of this variable is set to 0.



The following picture shows the Block Properties window of the first block in the example. The output of the equation is stored in the variable.



The following picture shows the Block Properties window of the third block in the flowchart. The equation shows that the commission is deducted from the existing value in the variable, and then the variable is overwritten with the new result. At the end of the simulation, the result storage block in the flowchart stores the final value of the variable, and then the variable resets to 0.



Profiles

A profile is a resource that allows you to represent a value that varies with time. It consists of a basic pattern that either repeats as a cycle or occurs once and then continues from its last defined setting.

There are several types of profiles:

- Stress profiles are used in life-stress data folios. These profiles can be created manually, or by importing data from nCode GlyphWorks. *For information about how to use stress profiles, see Time-Dependent Stress Profiles in the Weibull++ documentation.*
- Throughput profiles are used in process flow simulation (PFS) diagrams (if supported by your license). *For information about how to use throughput profiles, see Process Flow Simulation Diagrams in the BlockSim documentation.*
- Starting in Version 2019, cost profiles and duration profiles are used in profile models, which are available for use in spare part pools.

Creating Profiles Manually (All Types)

Like any other resource, you can create or edit profiles from the Resource Manager, or while you're performing a relevant analysis. To create a profile:

1. Specify the type of profile that you are creating.
2. Define a pattern for how the stress, throughput level, cost or amount of time (i.e., duration) 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 a value expressed in one of two ways:

   - a constant stress, throughput, cost or duration value, or
   - a function that takes a time value and returns a stress, throughput, cost or duration value.

   These segments are defined in the spreadsheet, 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)**, **Throughput T(t)**, **Cost C(t)** or **Duration D(t)** is the value or function for the segment. If the stress, throughput level, cost or duration will stay constant during a segment, enter a constant value (e.g., 30). If the value will change during a segment, enter the it as a function of time (e.g., at time = 20, the function $t + 20$ will return a value of 40). When entering a function, you must use t or T as the time variable.

Tip: Because units are not defined in the profile, it is important to apply stress and throughput profiles only to folios or diagrams that are intended to use the same stress/throughput and time units. In the Profile window, you may want to use the Comments page as a reminder of which units are applicable to that profile.

For cost and duration profiles, the time units are defined within the model where the profile is used. This setting applies to the time segments within the profile and, for duration profiles, to the duration values.

For example, suppose you are creating a stress profile where 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 spreadsheet 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 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.

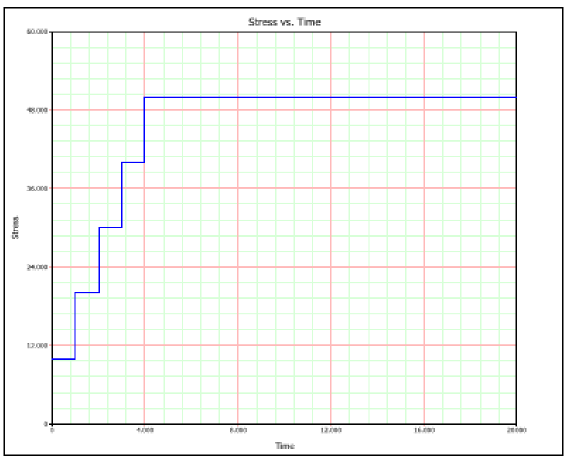
Note, too, that the columns have been renamed to reflect the units for the profile.

3. 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).

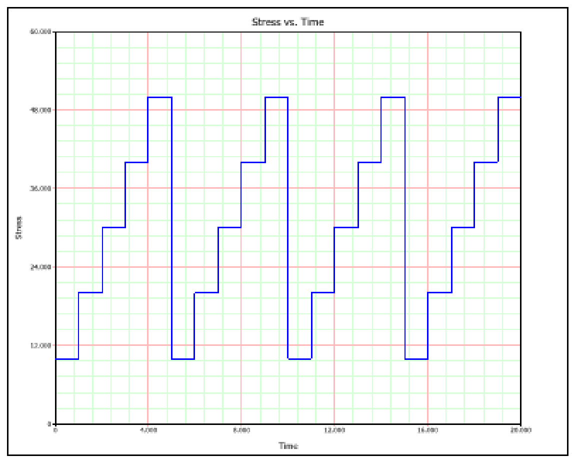
   If you select **Continue with last value**, all times after the last segment will use the value/-function defined in the last segment.

   If you select **Repeat cycle**, the entire pattern of segments will be treated as a repeating cycle.

   For example, suppose you defined a stress 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

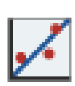


Repeat cycle

4. Before you can use the new profile in your analysis, you must save any changes you have made and validate the current profile settings. To do this, click the **Validate Profile** icon.

   After you save the changes in your profile, the **Profile Summary** area will appear. Click the **Detailed Summary** icon  to open the Results window, which shows the current profile in a worksheet that you can copy or print.

   To view a stress vs. time or throughput vs. time plot of the profile, click the **Plot** icon. (See Plot Utilities for general information on plots.)



Note: Clicking the **Validate Profile** icon will not automatically update your 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 profile.

5. Click **OK** to save your changes.

Importing Data from nCode GlyphWorks to Create Stress Profiles

You can import nCode GlyphWorks time series data from .S3T files for use as stress profiles. The stress values in the .S3T file must be greater than zero in order for the data to be imported.

To import data from an nCode .S3T file, first create a new profile from the Resource Manager (**Home > ReliaSoft > Resource Manager**). In the Profile window, click the **Import from GlyphWorks** icon on the control panel.

This launches the import wizard, which guides you through the steps required to import data from the file. You can then edit the resulting imported data, if desired. Note that if you update the .S3T file in GlyphWorks, the associated profile in Weibull++ will not be updated automatically. You must re-import the data to reflect the changes.

Tip: Because stress units (e.g., volts) and time units (e.g., hours) are not defined in the profile, it is important to apply profiles only to folios that are intended to use the same stress and time units. In the Profile window, you may want to use the Comments page as a reminder of which units are applicable to that profile.

Maintenance Groups

Maintenance groups are resources that are available for use throughout the project and can be managed via the Resource Manager. A maintenance group is a set of blocks or system hierarchy items where some event within the group can trigger either maintenance or state changes for one or more blocks or items, either within the group or outside of it. You can use a maintenance group:

- In BlockSim: To turn a block on or off. State change triggers are used to activate or deactivate a block when items in one or more specified maintenance groups go down or are restored. The block whose state is being changed may or may not belong to any of the maintenance groups.

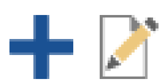
This allows you to model a cold standby configuration (i.e., one where the component cannot fail when in standby) without using a standby container, which may be useful if you are using a parallel or complex configuration, as blocks can be connected only in series in standby containers.

- In BlockSim or RCM++: To trigger a scheduled task (i.e., a preventive task, inspection or on condition task). You can set the task to be performed based on events in one or more maintenance groups. Triggering events within the maintenance group can include block/item failure; start of corrective, preventive or inspection tasks; and/or block/item restoration. The block (s)/item(s) affected by the task may or may not belong to any of the maintenance groups. For example, if you want to perform preventive maintenance on Component A every time you perform corrective maintenance on Component B, then you can assign Component B to Maintenance Group 1 and then set the preventive task assigned to Component A to be performed upon the start of corrective maintenance within Maintenance Group 1. See Task Scheduling for more information on how tasks can be scheduled.

Note: When you specify a maintenance group for a subdiagram block, it applies at the subdiagram level rather than to the individual blocks within the subdiagram; for example, the subdiagram block is considered "down" only if some event within it causes the entire subsystem modeled by the diagram to go down.

Creating and Editing Maintenance Groups

The Maintenance Group window allows you to create, view and edit maintenance groups. It can be accessed by clicking the **Create New** or **View/Edit** icon in the Maintenance Group wizard, which is accessed from the **Maintenance Group** field in Block Properties windows in BlockSim, and in the FMRA operation properties in RCM++.



It can also be accessed from the Maintenance Groups page of the Resource Manager by choosing **Home > Edit > New**, by selecting a maintenance group and choosing **Home > Edit > View** or by double-clicking a maintenance group.

In BlockSim, it can also be accessed via the icons in the Maintenance Group Manager.

At the top of the window, you can specify the maintenance group name. You can replace the default name with your own name, if desired. To change the default names, choose **File > Manage Database > Default Name Formats**. (In a secure database, this is available only to users with the "Manage other database settings" permission.)

You can also enter comments about the maintenance group in this window.

For existing resources, the link at the bottom of the window indicates how many times the resource is currently being used. If you need more information, click the link or the icon to open the Dependency Viewer.

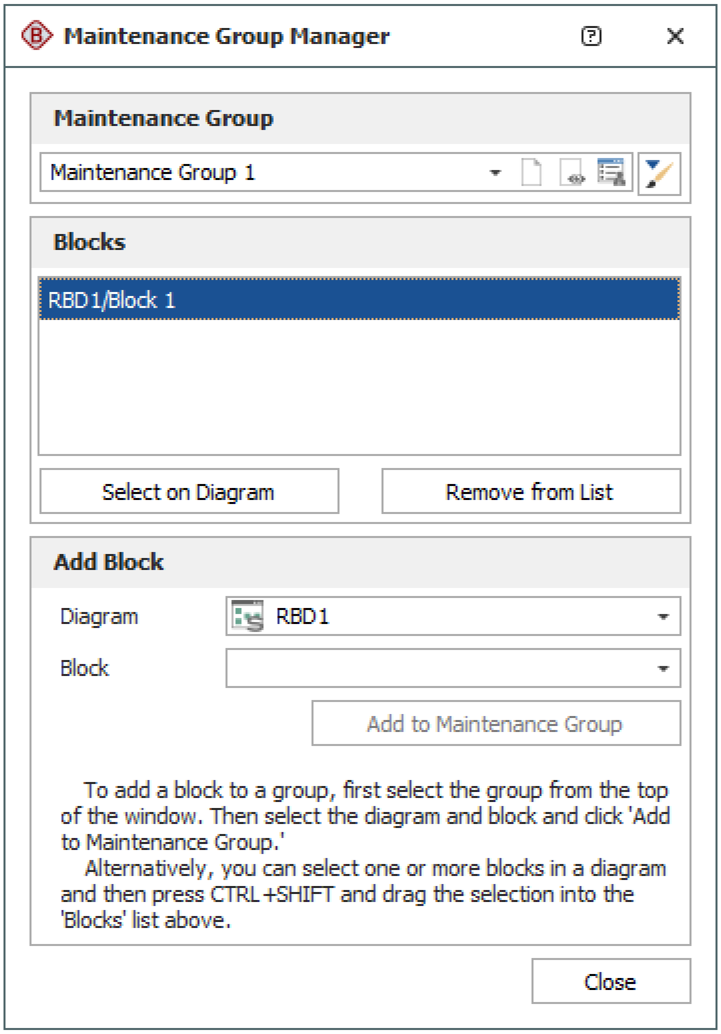
Assigning Maintenance Groups

In RCM++, system hierarchy items can belong to maintenance groups. To assign an item to a maintenance group, select the item in the FMRA and go to the **Operation** heading in the Properties tab. See Setting the Operation Properties for an FMRA in the XFMEA/RCM++ documentation.

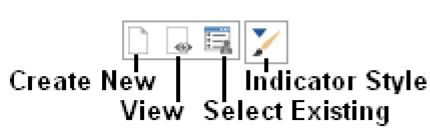
In BlockSim, certain types of blocks can belong to maintenance groups. You can assign an individual block to a maintenance group in the Block Properties window, or you can use the Maintenance Group Manager to define the group membership. Blocks that belong to a maintenance group have a circle at the upper left corner of the block. You can change the size of the indicator via the relevant Block Corner Indicators page of the Diagram Style window. You can change the color used for each maintenance group via the Maintenance Group Manager.

Maintenance Group Manager

In BlockSim only, the Maintenance Group Manager allows you to add blocks to or remove blocks from a maintenance group without having to go into each block's properties individually. This window can be accessed by choosing **Project > Data Management > Maintenance Group Manager** or, when you are viewing the Maintenance Groups page of the Resource Manager, by choosing **Home > Actions > Maintenance Group Manager**.



The **Maintenance Group** area allows you to choose an existing maintenance group from the drop-down list. In addition, you can click the **Select Existing** icon to open the Select Resource window. If there is not an existing maintenance group that meets your needs, you can create a new one by clicking the **Create New** icon to open a window that allows you to specify a name for the new maintenance group, as well as any comments. Click the **View** icon to view the characteristics of the selected maintenance group.



You can also click the **Indicator Style** icon to modify the appearance of the corner indicator that will appear on each block that belongs to the maintenance group, including:

- The indicator's background color, fill color and fill style. The background color is the underlying color applied to the indicator; the fill color is applied over the background color in the pattern specified by the fill style.
- The style (e.g., solid, dash, etc.), color and thickness of the indicator border.

You can change the size of the indicator via the relevant Block Corner Indicators page of the Diagram Style window.

The **Blocks** area displays all of the blocks in the maintenance group, in the format [Diagram Name]\[Block Name].

- To add a block to the maintenance group, you can select the diagram and block in the **Add Block** area and click the **Add to Maintenance Group** button. You also can select the block(s) in the diagram, press **CTRL+SHIFT** and drag the block to the list in the **Blocks** area of the Maintenance Group Manager.
- To view a block in the maintenance group, select the block in the list and click the **Select on Diagram** button. The diagram will be displayed with the block selected.
- To remove a block from the maintenance group, select the block in the list and click the **Remove from List** button.

Mirror Groups

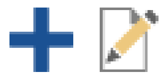
Mirror groups are resources that are available for use throughout the project and can be managed via the Resource Manager. You can use a mirror group:

- In BlockSim: To place the same block in more than one location (i.e., to represent a single component more than once in a diagram or in multiple diagrams within a project). See Mirroring (Using Blocks in Multiple Locations) in the BlockSim documentation.
- In the FMRA in BlockSim or XFMEA/RCM++: To place the same cause in more than one location in the FMRA hierarchy, in order to consider common cause failures. See Using Mirror Groups in an FMRA in the XFMEA/RCM++ documentation.

Task Packages

Task packages are resources that can be shared among analyses and can be managed via the Resource Manager. They represent groups of tasks that are performed together at scheduled intervals, for the most efficient allocation of resources and downtime management.

The Task Package window allows you to create, view and edit task packages. It can be accessed by clicking the **Create New** or **View/Edit** icon in the Task Package wizard, which is accessed from the **Task Package** field that appears when you select the **Override task scheduling properties with a task package** option for a preventive, inspection or on-condition task that is scheduled to be performed at fixed intervals.



It can also be accessed from the Task Packages page of the Resource Manager by choosing **Home > Edit > Add**, by selecting a task package and choosing **Home > Edit > View** or by double-clicking a task package.

For a new resource, a name will be proposed automatically based on the default naming criteria established for the current database (see Default Name Formats window). You can replace this with your own name, if desired. Remember that the name and identifiers are the primary way in which your team will be able to find the resources you need for your analyses.

The following settings are available to configure the task package:

- In the **Perform every** field, enter the length of the interval at which the task package will be performed. You must specify units for this value.
- The **Auto-Packaging Intervals** fields allow you to specify a range of scheduled intervals that will automatically be incorporated into the task package if the user clicks the **Auto-Package Tasks** button at the bottom of the Maintenance Task Packaging window in RCM++. The units for these fields are the same as those specified for the **Perform every** field. For example, if you enter **150** hours in the **Start time** field and **200** hours in the **End time** field, then tasks that are scheduled to be performed at fixed intervals from 150 to 200 hours can automatically be included in the task package to be performed at the scheduled time of the task package rather than at their individually scheduled times.
- **Identifiers** contains additional identifying information that can be used to search for this resource.
- **History** provides information about when the record was created and last updated. 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 record.

- **Watch** allows each individual user to subscribe to receive an alert (via e-mail, SMS text message or portal message) when the resource is changed.
- **Trace Usage.** For existing resources, the link at the bottom of the window indicates how many times the resource is currently being used. If you need more information, click the link or the icon to open the Dependency Viewer.

Action Resources

Throughout the ReliaSoft desktop applications, your team can use actions to track specific assignments that need to be performed. These versatile resources can be used multiple times in different locations, if appropriate. Specifically, actions can be created, displayed and/or managed:

- via Project Plans or the Resource Manager in all ReliaSoft desktop applications except MPC.
- via My Portal or Actions Explorer in all ReliaSoft desktop applications.
- via the SEP web portal (if it is implemented for your database).
- via FMEAs or test plans in XFMEA/RCM++.

For specific instructions regarding how to create or edit an action in each location, please consult the documentation for that particular feature. This section focuses on general considerations that will be applicable regardless of where/how the action is being used.

Tip: If an action is used in an FMEA, the properties window will be different depending on the current context. If you're using XFMEA/RCM++ and you open the action from the FMEA or My Portal, the window includes additional features that provide quick access to associated FMEA records (failures, causes, etc.) and the FMEA change log (if activated). These features are not available if you open the action from another location (e.g., Resource Manager, Project Planner, Actions Explorer) or in another application, because the associated FMEA can't be opened in those situations.

Actions as Resources

As with any other 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, a test plan or a Project Plan, the resource will remain in the project unless/until an authorized user deletes it from the database (via My Portal, Resource Manager or Actions Explorer).

Note: Actions are always local resources. They cannot be made global. Actions in a reference project are local to that project and can't be selected for use in other projects.

Configurable Settings

Some properties for action records can be configured via interface style settings. See Configurable Settings.

Show/Hide Properties for Individual Records

You can also use the following ribbon commands to show/hide certain features for a particular action record.

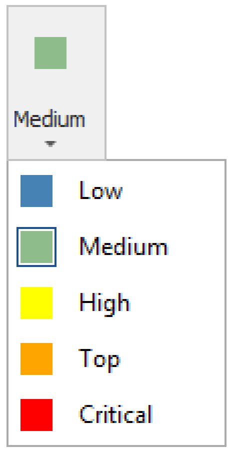
Generic vs. **Detailed**: A detailed action contains additional properties for describing a specific test that needs to be performed (Specifications, Requirements and Reports). These properties may be of interest in test plans in XFMEA/RCM++. (See Test Plans in the XFMEA/RCM++ documentation.) Most actions, unless they are used in test plans, do not need to display these fields.

Show Resources: The resources fields (Team, Facility, Material and Additional Costs) are used for calculating costs and resource utilization. They are always enabled but you can toggle this command to hide or display them for any particular action record. The **Check Utilization** command provides a summary of the resources that have already been assigned. (See Costs and Man Hours.)

Show Review/Approval: If the Reviewer property is enabled in the configurable settings for the current project, you can toggle this command to hide or display these fields for any given action. The **Review Action** command will be enabled if you are assigned to review the action and it is ready to be reviewed. (See Review and Approve Actions.)

Priority

The color-coded **Priority** command provides a consistent scale for prioritizing actions across all projects in the database. (The same options are also available for gates in the Project Planner.)

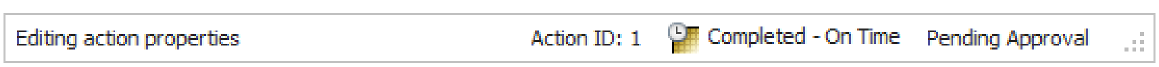


This is intended to be used instead of the configurable **Action Priority** field from prior versions, which can be different for different projects. When you convert from a Version 8/9 database, the software will attempt to match any priorities that were assigned with the old configurable drop-down list. For example, if you were using the default "High," "Medium" and "Low" labels that were shipped with Version 8/9 libraries, they will automatically be mapped to the corresponding options in the new ribbon command.

Action Status and Review Status

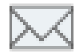
The action's status bar displays two complementary status indicators. The first one is based on the actual and expected start date and completion date. It indicates that the action is not started, in progress or complete and whether it is on time or running late. (See Action Timeline and Status.)

The second one is based on whether a reviewer has been assigned in the Review/Approval section and, if so, the status of that review.



E-mail and Alerts

The **E-mail** command will be enabled if an authorized user has defined a valid SMTP server. This allows you to manually send an e-mail containing the current action details to any valid e-mail address.



If a valid SMTP server has been defined within the database, your team can also choose to use automated alerts (via e-mail or SMS text message). Each individual user can subscribe to "watch" specific action resources, and the database can also be configured to auto-subscribe users assigned to specific roles for the action. (See Watches and Alerts.)

Click **Action Monitors** to specify individual users and/or groups of users who will always receive alerts for the record, regardless of whether they have personally subscribed to a “watch.” (See Action and Gate Monitors.)

Add to Outlook Calendar

The **Add to Outlook Calendar** command will be enabled if Microsoft Outlook is installed on your computer.

It launches Outlook’s interface for creating a new calendar event and automatically populates the subject and date. You can modify the details as needed 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.

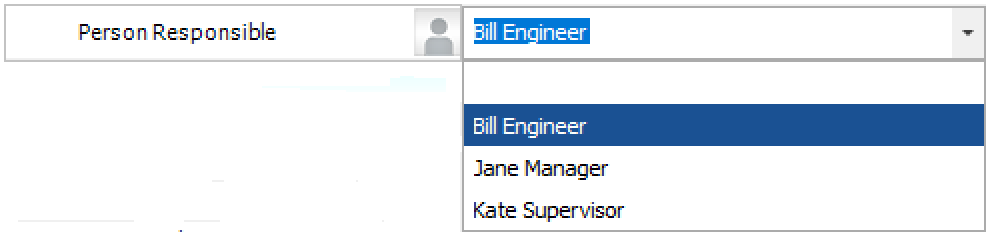
Person Responsible and Resources

For each action resource, you can assign both an individual database user who is primarily responsible for completing the action and a team of users who are also involved. You also have the option to track the percentage of their available time that is planned to be utilized for the activity.

Tip: If a user is assigned as person responsible and also belongs to the team, the costs and utilization values will be summed. For example, the application assumes that the utilization is X% as person responsible plus an additional Y% as a member of the team.

Person Responsible

The **Person Responsible** field can be enabled or disabled, depending on the configurable settings for the current project. If you want to use personalized alerts, track resource utilization etc., you must select a database user. Alternatively, you can type a text description that will be displayed in the FMEA spreadsheet.

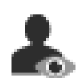


Tip: If you want to change the person responsible for multiple actions simultaneously (e.g., if a user changes roles and his/her actions need to be reassigned), use the Actions Explorer.

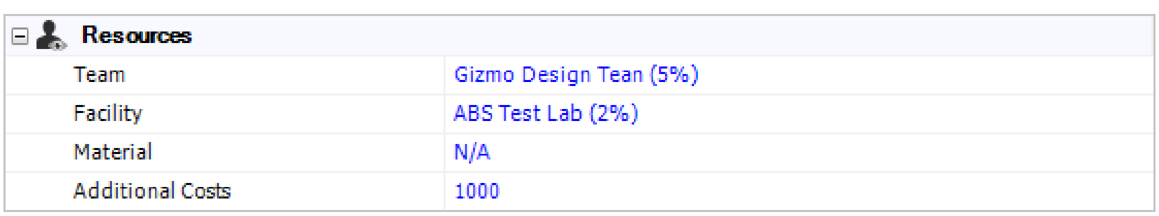
If the **Set action creator as 'Person Responsible' by default** check box is selected in the Database Settings window, then the person who created the action is automatically shown in the **Person Responsible** field, but that can be changed.

Resources

The **Resources** fields are always enabled but you can toggle the **Show Resources** command to hide or display them for any particular action record.

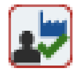


You can select from the teams, facilities and materials resources that have been predefined within the database (or create them on-the-fly if you have the required permissions). In a secure database, only users with the "Manage project planning resources and working days" permission will be able to create and edit these resources. (See Project Planning Resources.)



Check Utilization

If you need to view the utilizations of users and facilities for the current action, click **Check Utilization**.



Action Timeline and Status

For all action resources, the status is determined automatically based on the actual and expected start/completion dates. If an action is included in the Project Planner, its *expected timeline* can automatically shift in response to delays in prior activities. If the action is not included in the plan, the expected dates are always identical to the originally planned dates.

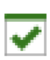
The following status indicators are displayed in the status bar of the action properties window and in other relevant locations. (Note that the status bar now also displays a separate review status to indicate whether a user has been assigned to review/approve the action and, if so, the status of the review. See Review and Approve Actions.)

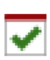
 **Not Started**: The action hasn't started, but it can still start on time.

 **Past Start Date**: The action hasn't started, and it is too late to start on time.

 **In Progress**: The action has started.

 **Past Due**: The action hasn't completed, and it is too late to complete on time.

 **Completed - On Time**: The action completed on time.

 **Completed - Late**: The action completed late.

Costs and Man Hours

For action resources, you can now track the planned vs. actual usage for both costs and man hours.

The **Planned Cost** and **Actual Cost** fields can be enabled or disabled, depending on the configurable settings for the current project. If the properties are enabled, the planned costs will always be calculated automatically based on the action duration, person responsible and project planning resources. For the actual costs, you can choose to enter manually or use the automatic calculations.

Enter Manually

If you prefer to enter the actual costs/hours instead of using the automatically calculated values, select **Yes** in the **Manually enter costs/man hours?** field.

Manually enter costs/man hours?	No	Manually enter costs/man hours?	Yes
Actual Cost	*240.00*	Actual Cost	475
Actual Man Hours	*2.4*	Actual Man Hours	5

Let the software calculate **Enter the values yourself**

Calculate Automatically

The automatic calculations are based on:

- The Duration (the number of working days from the start date to the due/completion date).
- The Person Responsible and/or the Team:
  - The % utilization is set in each action record.
  - The Hours per Day and Cost Category are set in each user account record.
- The Facility (e.g., test lab or other facility needed to complete the action):
  - The % utilization is set in each action record.
  - The Max Hours per Day and Cost Category are set in each facility properties record.
- The Materials (e.g., test samples or other material needed to complete the action):
  - The Quantity and Cost Category are set in the material properties record.
- Any Additional Costs that are defined directly in each action record.

The following simple examples demonstrate how each type of cost is calculated.

PERSONNEL COSTS

Action Duration:	5 working days
Joe Engineer:	Works 8 hours per day (5 days x 8 hours = 40 hours)
	Time is billed at $100 per hour (no "per instance" cost)
	Assigned as **Person Responsible** with 5% utilization (40 hours x .05 = 2 hours)
	Also belongs to the assigned **Team** with 1% utilization (40 hours x .01 = .4 hours)

The calculated values for Joe's work on this action are 2.4 man hours and 2.4 x $100 = $240.

FACILITIES COSTS

Action Duration:	5 working days
ABC Test Lab:	Is available up to 8 hours per day (5 days x 8 hours = 40 hours)
	The utilization for this action is 20% (40 hours x .20 = 8 hours)
	Costs $500 "per instance" plus $50 per hour

The calculated cost for using this facility to complete the action is $500 + (8x$50) = $900.

MATERIALS COSTS

Gizmo Prototype:	Costs $5000 to produce the batch of test units (the "per instance" cost)
	The test requires 10 samples
	Each unit costs $500 (the "direct cost per hour or unit")

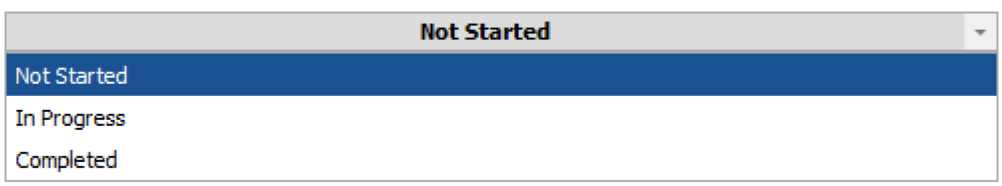
The calculated cost for the materials required to complete the action is $5000 + (10x$500) = $10,000.

Action Status Updates

For all action resources, the **Progress (Status Updates)** area displays any notes that have been added to report progress. The software automatically adds these updates under certain conditions (e.g., upon review/approval), and you can also create updates manually when needed.

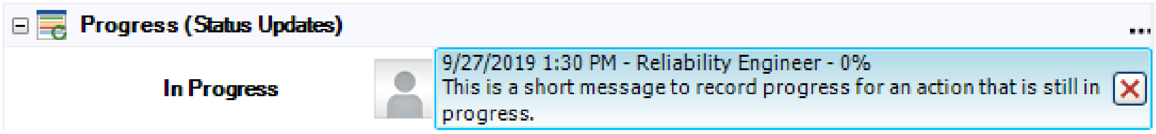
To add a new note, click **Add Status Update** in the action's ribbon or click the heading and then click the **Add** icon that becomes visible.

The type that you select in the Add Status Update window can affect the actual start and completion dates of the action, and also update the action status. (See Action Timelines and Status.)



- **Not Started** allows you to record comments without entering a start date. If an actual start date has already been entered, the update will remove the date and change the action's status back to *Not Started*.
- **In Progress**
  - If an actual start date is not already defined, sets the current date as the **Actual Start Date** and allows you to specify the **% Completed**. This will change the action's status to *In Progress*.
  - If the action is already in progress, allows you to add another note and update the completion percentage.
- **Completed** sets the current date as the **Completion Date**. This will change the action's status to either *Completed-On Time* or *Completed Late*.

To delete an existing note, click inside the field and then click the **Delete** icon that becomes visible. ***There is no undo for delete***.



Review and Approve Actions

Action resources can be marked for review and approval. If the Reviewer property is enabled in the configurable settings for the current project, you can click **Show Review/Approval** in the action ribbon to hide or display these fields for any given action.

(Note that if an action has been assigned for review or is already reviewed, you can no longer hide the fields.)

Assigning a Reviewer

If an action has not yet been reviewed, any user who is able to edit the record can assign or change the reviewer at any time.

In a secure database, the Reviewer drop-down list contains all of the database users who have the "Approve actions" permission.

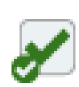
- If a user has been assigned to review the action but an actual completion date is not entered, the review status will be *Reviewer Assigned.*
- When the completion date is entered and the action is ready for review, the status will be *Pending Approval*.

Tip: If you want to change the assigned reviewer for multiple actions simultaneously (e.g., if a user changes roles and his/her actions need to be reassigned), use the Actions Explorer.

Reviewing and Approving the Action

When you are assigned to review/approve an action that has been completed, the record will be highlighted for you in My Portal (and in the SEP web portal if implemented), and you may also receive automated alerts (via e-mail and/or SMS) if applicable.

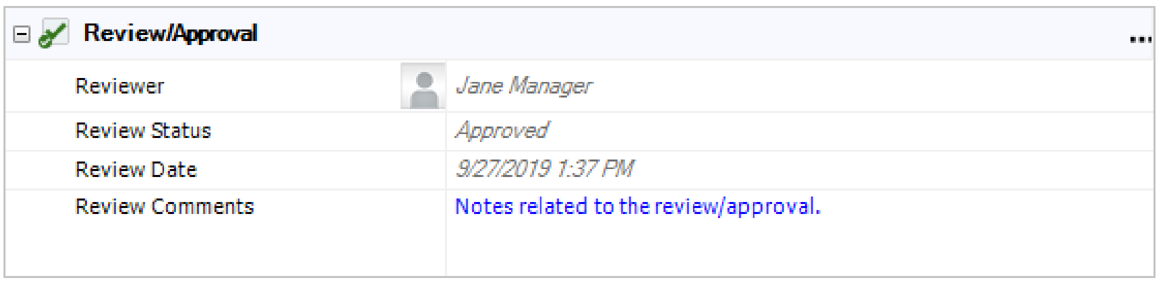
To record your decision, open the action record and choose **Review Action**.



In the Review Action window, you can choose one of the following options:

- **Approve action** assigns the review date/time and marks the status *Approved*.
- **Reject and re-open action** removes the completion date and marks the status *Rejected and in progress*. The team can perform any rework that is needed and resubmit the action for another review.
- **Reject and close action** marks the status *Rejected and closed*.

You can enter your comments directly in this window, or enter/update them later in the comments field.



Configurable Settings for Actions and Controls

Some of the properties in action records and in control records are configurable via the interface style settings that have been defined for the current project. The interface style is used in XFMEA/RCM++ to customize a variety of analysis fields to fit the organization's particular preferences and needs. (See About Configurable Settings in the XFMEA/RCM++ documentation.)

You can view/edit the configurable settings for actions and controls from any ReliaSoft desktop application by clicking **Configurable Settings** of the action properties window or the control properties window. (In secure databases, this is available only if the user is the project owner, or has the "Edit project properties" permission.)

Any changes to these settings will apply to all records of that type in the current project.

Starting in Version 2020, two new configurable drop-down lists are available for actions. These fields are intended to allow users to follow the *AIAG & VDA FMEA Handbook* released in 2019.

- The options in the **Action Type** drop-down list depend on the current project's interface style. For the "DFMEA: AIAG & VDA-1" and "PFMEA: AIAG & VDA-1" profiles, the default values are:

- Detection actions will help to enhance, optimize or otherwise improve the ability detect the failure and/or cause before it reaches the end user.
- Prevention actions will help to improve the ability to prevent the failure and/or cause from occurring.

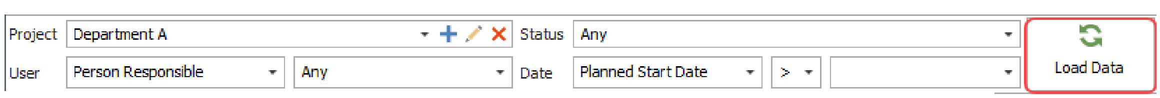
- **Status Detail** is used to manually record the current status of the action. For the "DFMEA: AIAG & VDA-1" and "PFMEA: AIAG & VDA-1" profiles, the default values are:
  - Open
  - Decision Pending
  - Implementation Pending
  - Completed
  - Not Implemented

Actions Explorer

The Actions Explorer is available in all ReliaSoft desktop applications except MPC. You can use this flexible tool to explore all of the action resources that are stored in the current database.

To access the utility, choose **Home > ReliaSoft > Actions Explorer**.

To reduce the amount of time required to populate the grid, first use the Project, Status, User and/or Date filters to limit your search, then click **Load Data**.



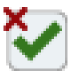
Add, Edit and Delete Actions

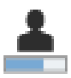
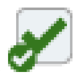
When actions are loaded in the grid, you can use the following ribbon commands to add, edit and delete actions. (In a secure database, the ability to add or edit an action is only available for users with the "Create/edit/delete local resources" or "Create/edit/delete local resources" permission in the relevant project.)

To delete or edit multiple actions at the same time, press **CTRL** or **SHIFT** while clicking rows in the grid. Selected rows will be highlighted.

 **Add Action** creates a new action resource in the project that currently has focus.

 **Edit Action** allows you to view and edit all of the properties of the action that is currently selected.

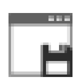
 **Delete Action** deletes all of the actions that are currently selected. ***There is no undo for delete***.

 **Assign Person Responsible** and  **Assign Reviewer** allow you to set (or change) the person responsible or reviewer for all of the actions that are currently selected.

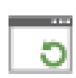
Save and Apply Views

Once you have customized the grid to suit a particular purpose (using the built-in filter, column configuration and grouping features described below), you can save the preferences as a *view* that can be used again whenever you need it. A view is saved in the database and available only to the user who created it.

To create a view, first configure the grid to suit your particular needs and then click **Save View**.



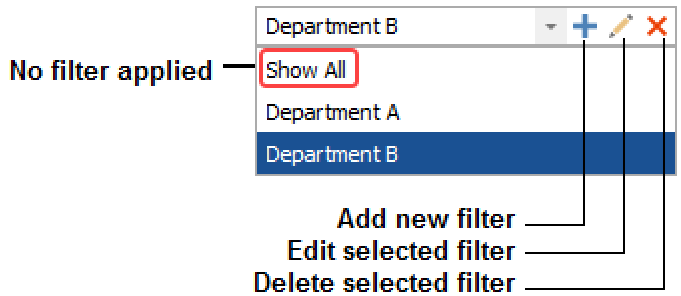
To quickly apply these same preferences again at any time, click **Apply View** and select one of the saved views from the list.



Note: The view does not affect filtering that has been applied from the Project, Status, User and/or Date filters, or from the Relevance Panel.

Project, Status, User and Date Filters

The Actions Explorer can utilize the same project filter that is available in many other locations throughout ReliaSoft desktop applications.



You can also filter by status (e.g., In Progress, Completed, etc.), user (e.g., Person Responsible or Reviewer) and date (e.g., Planned Start Date, Actual Start Date, etc.).

After you clear or change these filters, you must click **Load Data** again to update the grid.

Relevance Panel

The relevance panel provides a quick way to filter the actions based on how they are relevant to you (based on your personal username).

- **I am responsible for** – you are assigned in the **Person Responsible** field.
- **I am a team member in** – you belong to the team assigned in the **Team** field.
- **I need to review/approve** – you are assigned in the **Reviewer** field.
- **I am monitoring** – you are assigned in the Action Monitors window, or you have personally subscribed to "watch" the action.
- **I am the creator** – you are listed in the **Created By** field.
- **All actions** – displays all actions, regardless of whether they're relevant to you.

Tip: If an action is relevant to you in more than one way, it will display if any applicable relevance check box is selected. For example, if you are both the person responsible and the creator, the action will display if you have selected "I am responsible for", "I am the creator" or "All actions."

Built-in Find/Filter, Configuration and Grouping Tools

The Actions Explorer offers the same filter, column configuration and grouping tools that are built in to other utilities that use a similar grid (e.g., the Resource Manager). For details about how to use each feature, see:

- Finding and Filtering Records
- Configuring Columns
- Grouping Panel

Controls

Controls are resources used in FMEAs that are the methods or actions currently planned or already in place to reduce or eliminate risk. Controls can be managed via the Resource Manager.

As with any other resource:

- Any changes to the control properties will be reflected in all locations where the resource is used.
- If you remove a control from an FMEA, it will remain in the project unless/until an authorized user deletes it from the database.
- If a control resource is not currently used in an FMEA, you can delete it via the Resource Manager. (In a secure database, this is possible only if the user a) is the project owner, b) is the control creator, or c) has the applicable "create/edit/delete resources" permission.)

For existing resources, the link at the bottom of the window indicates how many times the resource is currently being used. If you need more information, click the link or the icon to open the Dependency Viewer.

Note: Controls are always local resources. They cannot be made global. Controls in a reference project are local to that project and can't be selected for use in other projects.

For specific instructions on using controls in FMEAs, see the "Controls in FMEAs" topic in the XFMEA/RCM++ documentation.

Configurable Settings

Some properties for controls can be configured via interface style settings. See Configurable Settings.

Resource Manager

The Resource Manager allows you to create, view, edit and delete resources. It also helps you to trace where each resource is used, and to create global resources that are available to all projects in the database.

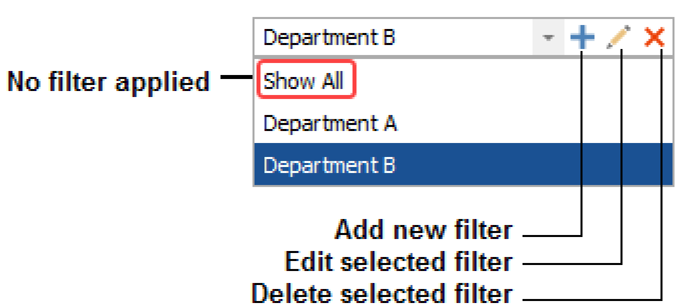
To open the Resource Manager, choose **Home > ReliaSoft > Resource Manager**.

To use the utility, choose the type of resource that you want to work with in the navigation panel on the left side of the window. A table that displays the resources of the selected type will be shown on the right side of the window.

Tip: When you need to create or edit a large number of resources, you can also use ReliaSoft APIs to import the data from an Excel file or other data source. For example, the URDExample.xlsm file at C:\Users\Public\Documents\ReliaSoft\Examples21\API\VBA enables you to quickly create/update multiple URDs with some basic properties (models and corrective tasks). If you have the necessary software coding experience, you can expand this tool or create your own custom applications to meet specific needs.

Resource Filter

The Resource Manager can utilize the same resource filter that is available in many other locations throughout ReliaSoft desktop applications.



Built-in Find/Filter, Configuration and Grouping Tools

The Resource Manager offers the same filter, column configuration and grouping tools that are built in to other utilities that use a similar grid (e.g., the Analysis Explorer, Actions Explorer, etc.). For details about how to use each feature, see:

- Finding and Filtering Records
- Configuring Columns
- Grouping Panel

Creating, Viewing, Editing and Deleting Resources

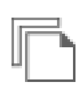
You can:

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

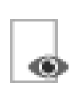
  

  To add multiple resources at the same time, specify the quantity in the **Home > Edit > Number to Add** field and then choose **Home > Edit > Batch Add**. You will specify the properties for one of the new resources, and the additional resources will be duplicates of that. You can then edit each resource individually, or you can go to the Batch Properties Editor to edit them in a spreadsheet format.

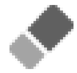
- Duplicate an existing resource by selecting the row in the table and choosing **Home > Edit > Duplicate**. You can choose to duplicate just the selected resource or, for local resources only, you can choose **Cascade Duplicate** to duplicate the resource and all resources assigned to it, at all levels (e.g., cascade duplicating a corrective task will also duplicate the task's duration model, the assigned crew and all models assigned to the crew, etc.). Note that only local resources will be duplicated in a cascade duplication; if the original uses any global or reference resources at any level, the copy will also use those original global or reference resources rather than creating copies of them.

- View or edit an existing resource by double-clicking the row in the table or by selecting the row and choosing **Home > Edit > View**.

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

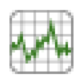


You cannot delete a resource that is currently being used in any other resource or analysis.

Note: A published model cannot be edited or deleted from the Resource Manager. If changes are needed, you must update the original analysis and then republish the model. If you wish to delete the model, you must first remove the association with the original analysis and also make sure the model is not being used. (See Publishing Models.)

Tracing Where Resources Are Used

With a single resource selected, you can choose **Home > Selected Resource > 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.



If you need to know which resources are not currently being used (e.g., because you want to delete obsolete records), choose **Home > Display > Show Only Unused**.

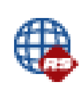
Local and Global and Reference Resources

As explained in Local, Global and Reference Resources, the availability of resources for use is determined by their scope. *Local resources* are available only in the current project. *Reference resources* are available for any project within the database, depending on the user's permissions within the reference project. *Global resources* are available for use in any project in the database.

The far left column in the Resource Manager displays an icon that indicates whether each resource is local, global or a reference resource.

Local Resource

Global Resource

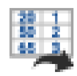


Reference Resource

You can use the **Home > Display > [Show Local/Show Global/Show References]** commands to toggle the types of resources shown in the Resource Manager.

Exporting Information from the Resource Manager

To send the contents of the Resource Manager to Excel, or to a built-in ReliaSoft Workbook or spreadsheet if that's available in the current application, choose **Administration > Output > Transfer Report**.



You can send either the current item (i.e., the table for the currently displayed page) or the full report (i.e., the tables for all pages in the Resource Manager).

Removing Unused and Duplicate Resources

The Resource Manager provides several tools to make it easy to delete unused resources and merge duplicate resources.

Finding and Removing Unused Resources

To find unused resources of the selected type, choose **Home > Display > Show Only Unused**.

Once you have found the unused resources, you can decide which ones to delete.

You can return to viewing all resources of the selected type by choosing the **Show All** command.

Finding Duplicate Resources

You may sometimes have duplicate resources within a project (i.e., several models that all represent the identical distribution, several tasks that are identical, etc.) within a project. For example, this might happen if you have imported diagrams that all reference the same model, but the imports happened at separate times and therefore the model had to be imported again each time.

To find duplicate resources, choose **Home > Display > Show Only Duplicates**.

In the window that appears, select the properties that must match in order for the resources to be considered duplicates. For example, if you are looking for duplicate URDs, you might select to compare the model, corrective task and scheduled tasks, but not the URD name. Once you have selected the columns to compare, click **OK** to show the duplicate resources.

You can merge multiple resources of the same type into one resource, either automatically or manually. Note that models must be of the same category in order to be merged, and you cannot merge published models. When you merge resources, any folios that you may have open will be closed prior to the merge in order to prevent data inconsistencies.

IMPORTANT: Merging cannot be undone, and only the information from the "target" resource (i.e., the resource that you merge the others into) will be retained.

Automatically Merging Duplicates

To merge resources automatically, choose **Administration > Cleanup > Merge All Duplicates**.

This will find all sets of duplicates within the selected resource type (e.g., three identical Weibull models and 2 identical lognormal models); and for each set, it will merge the duplicates into the first instance found. After you confirm that this is what you want to do, you will be prompted to select the properties that must match in order for the resources to be considered duplicates. The process begins as soon as you select at least one column and click **OK**. It cannot be undone.

Selecting Resources to Merge

To merge resources manually, select the resources and choose **Administration > Cleanup > Merge & Delete**.

The window shows a list of the resources you selected to merge and allows you to choose which one will be retained (i.e., the one that the other selected resources will be merged into). After you click **OK**, this resource will be used in all places where the other selected resources were previously used.

Select Resource Window

The Select Resource window allows you to choose an existing resource to use at your current location. You will see this window in situations like the following:

- When you click the **Select Existing** icon in any resource wizard.

- When you are adding an existing action in an FMEA or in the Project Planner, or an existing control in an FMEA.

The Select Resource window is similar to the Resource Manager, but it shows only the resource type that you are currently working with. The functionality of the window will vary slightly depending on where you have opened the window from.

- If you can select only one resource (e.g., in a resource wizard), each row in the table will contain a radio button that allows you to select the resource for use. If you can select multiple resources (e.g., adding existing actions or controls in an FMEA), each row will have a check box instead.
- Actions, controls and metrics are always local, so the commands to show global or reference resources will not be available in these cases.
- The ability to apply an item filter is available only for resources that have a full set of identifiers. This excludes actions and controls.
- Some resources may not be available for selection. For example:
    - In an FMEA, you can't select an action or control that is already assigned to the current cause.
    - In the project planner, an action cannot be added if it is already in use in the project plan or if its start date is either before or more than a year after the gate's start date.
    - A metric can't be selected to have a value pushed to it if it's already in use somewhere else, or if it's a type that is not compatible with the current use.

Starting in Version 2020, the following changes have been made to make the window more intuitive to use:

- When you click **OK**, only the resources that are selected and currently visible are added to the desired location. That is, if you have selected a resource and then applied a filter that causes that resource to be hidden from view in the Select Resource window, it will not be added. Note that this is true regardless of whether the window is in single selection (radio buttons) mode or in multiple selection (check boxes) mode.

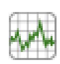
- The selection status of resources is preserved while filters change. If you have selected a resource and then applied a filter that hides the resource from view, if you subsequently remove the filter, the resource that you had selected continues to be selected.
- The **Select All** check box in the column heading selects only visible resources.

Taken together, these changes allow you to change your filters multiple times in order to select various resources. You can then clear the filters and all selected resources will be added to the desired location.

Trace Usage

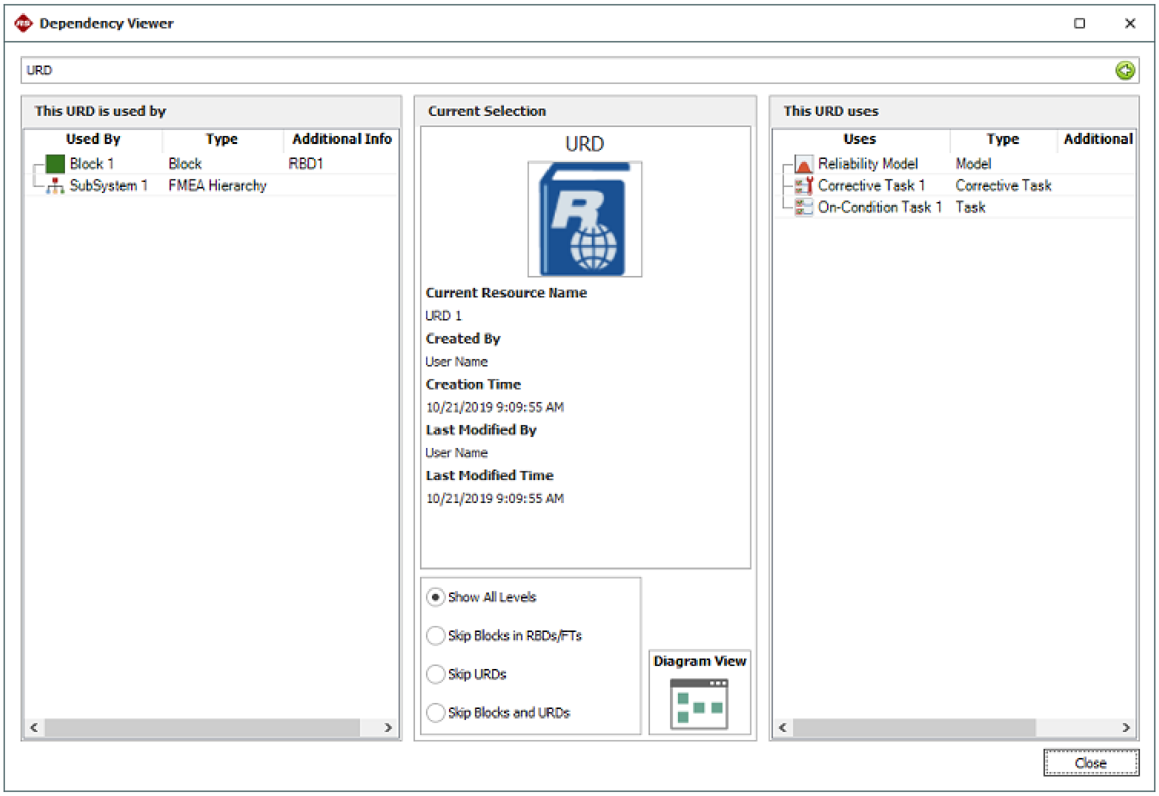
Dependency Viewer

Because resources can be used multiple times, it is important to know where a given resource is used before making changes to it. The Dependency Viewer, which is accessed by clicking the **Trace Usage** icon in each resource properties window or in the Resource Manager, provides information on where the resource is used and what additional resources, if any, it uses.



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

The items that use the selected resource, if any, will be displayed in the **This [resource] is used by** area on the left. If it uses any resources, those resources will be displayed in the **This [resource] uses** area on the right. For example, in the picture shown next, the selected resource is a URD. It is used by Block 1 in a BlockSim diagram and by SubSystem 1 in an FMEA analysis. The URD uses an assigned model and two tasks.



You can edit any resource shown in the Dependency Viewer by right-clicking it and choosing **Edit Item** on the shortcut menu. The resource's properties window will open.

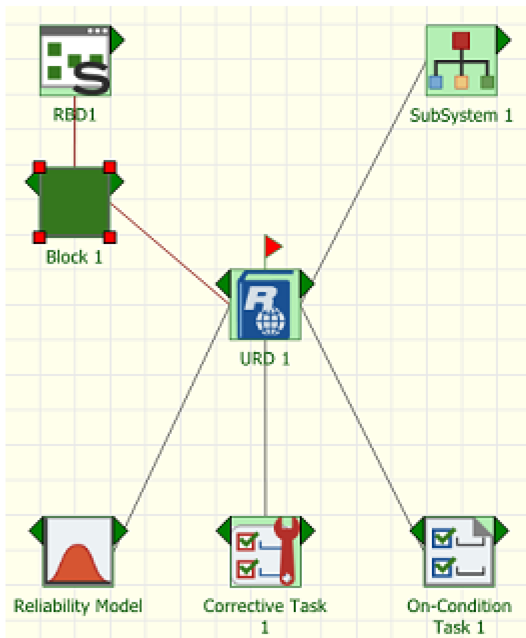
You can double-click an item in either area to make it the current selection in the window. For example, you could double-click a task used by the URD to see the model(s), crew(s) and/or spare part pool(s) that the task uses.

As you change selected items, each previous item that you selected will be added to the "path" of recent selections at the top of the window; click **Back** to go back through the path to the previously selected items. For example, the following picture shows a path in which the user started with the URD, then proceeded to its corrective maintenance task, the crew for that task and the logistic delay for that crew (not shown in the path because it is the current selection). The button now provides a quick way to go back through those same resources in reverse order if needed.

You can click the **Diagram View** button to open the diagram view of the Dependency Viewer. The currently selected item will be the main block in the diagram that is created.

Choosing What You See

In the **Current Selection** area, you can choose what is shown in the **This [resource] is used by** area. You have the option to skip the URD and/or the block that the resource is used by. For example, the image below shows the connections among analyses and resources that we have been discussing. If you have selected to show all levels, you can trace from Reliability Model to URD 1 to Block 1 to RBD1. But if you have selected **Skip Blocks and RBDs**, it just shows that Reliability Model is used by RBD1.

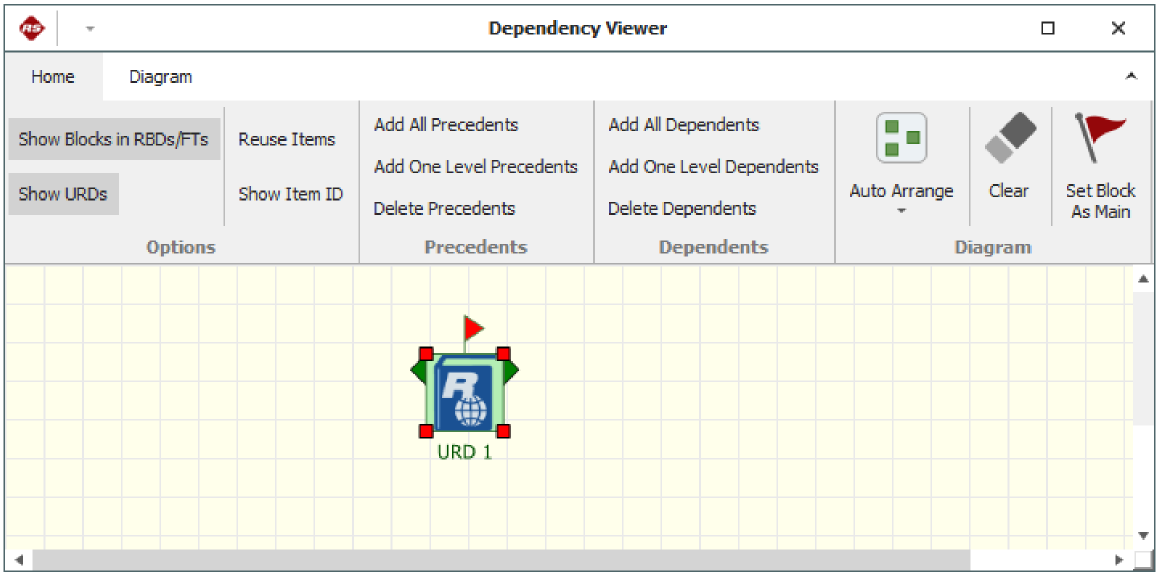


Note that blocks are always skipped in event analysis flowcharts (i.e., only the flowchart will be shown as the precedent for a resource used by any block within the flowchart).

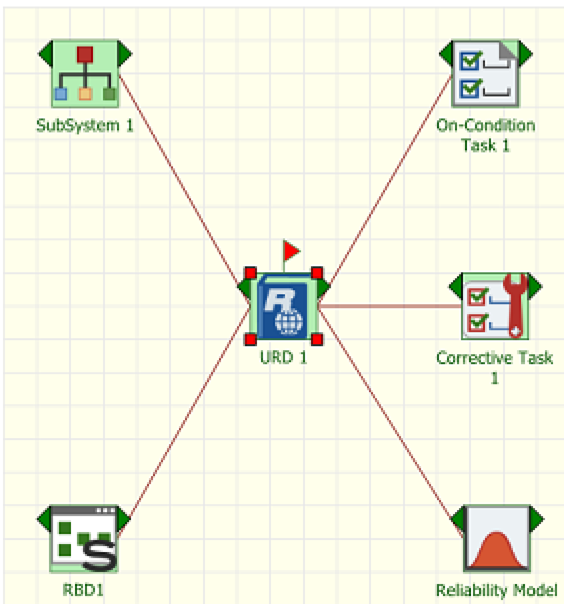
Dependency Viewer - Diagram View

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

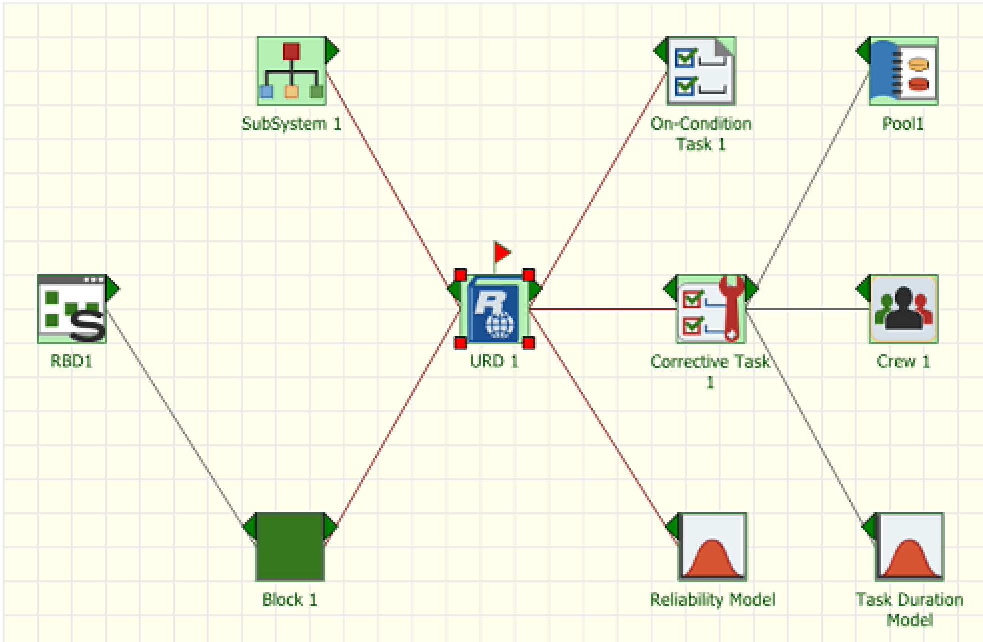
To understand how to use this view, consider the same example shown in the Dependency Viewer topic. Clicking the **Diagram View** button for that example will yield a diagram view that shows only the main item (in this case, the URD), which is marked with a flag:



To view a graphical representation of the information that was shown in the Dependency Viewer, choose **Add One Level Precedents** and **Add One Level Dependents**. This will show the items that use the resource (i.e., its precedents), and the resources that it uses, if any (i.e., its dependents).



If you want to see all levels, choose **Add All Precedents** and **Add All Dependents**:

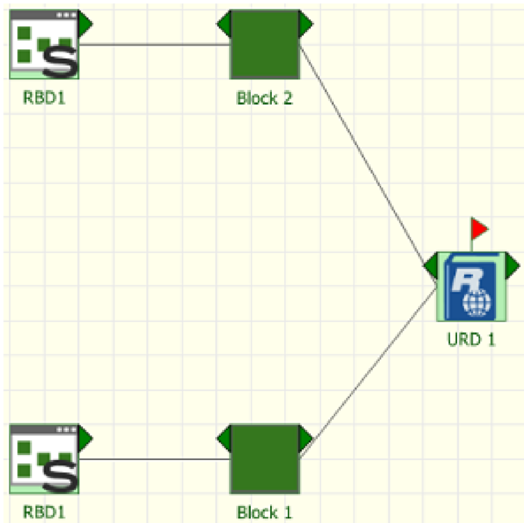


If you want to add or remove one branch at a time, click the green arrows on the sides of the blocks. Alternatively, if you want to remove one branch, select the block and choose **Delete Precedents** or **Delete Dependents**.

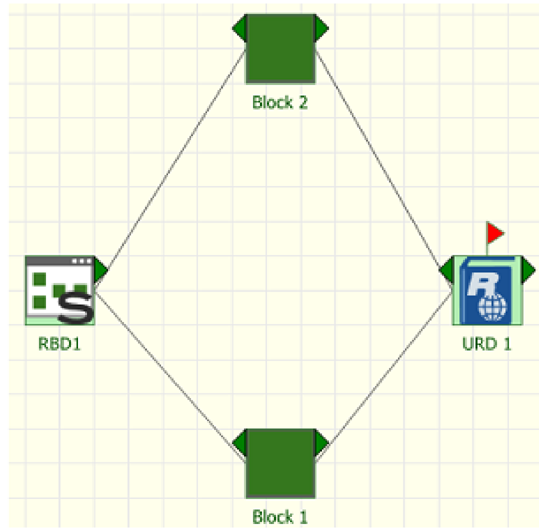
To set a different item as the main item in the diagram, select it and choose **Set Block as Main**. This serves the same purpose as changing the selected block in the Dependency Viewer.

Use the commands in the Options group of the ribbon's Home tab to specify how the diagram will be constructed:

- Select **Reuse Items** to show each item only once in the diagram. For example, if you are viewing a URD that is used by two blocks in the same diagram and you choose **Add All Precedents**, the Reuse Items command will affect the display as shown next:



Without Reuse Items Selected



With Reuse Items Selected

This command must be selected before the diagram is created, or you must clear the current diagram to apply it.

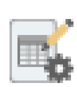
- Use the **Show Blocks in RBDs/FTs** and **Show URDs** commands to determine whether blocks and/or URDs are shown as intermediate steps between resources and the analyses in which they are used.
- Select **Show Item ID** to display the internal ID assigned to each resource, block and/or diagram. These IDs can be used to differentiate among items that share the same name.

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

Batch Properties Editor

The Batch Properties Editor is designed to support batch editing of most types of local resources in the current project. In BlockSim, it also allows you to edit the blocks used in the diagrams. It enables you to edit the properties in a convenient grid, rather than opening the properties window for each individual resource/block. This utility is available for all desktop applications except MPC.

To open the Batch Properties Editor, choose **Home > ReliaSoft > Batch Properties Editor**.



To use the utility, choose the type of resource or diagram that you want to work with in the navigation panel on the left side of the window. The right side of the window shows the resources or blocks you can edit. For information on configuring the table and limiting the items shown, refer to the Configuring Columns and Finding and Filtering Records topics.

Tip: If you change the information in one column, it may affect the information required in another column. For example, if you are editing a model and you change the Model Type column from "Weibull-2" to "Weibull-3," then the Parameter 3 column will change from "N/A" to requiring a numerical input for the third parameter for the Weibull distribution (i.e., gamma).

The changes you make in the Batch Properties Editor will be saved when you close the window. You can also choose **Home** > **Display** > **Refresh** to commit your changes immediately and refresh the Batch Properties Editor to display the most current information, including recent changes by other users.

Note the following:

- **Gray Background**. If the property does not apply to the corresponding resource or block, "N/A" will appear in the column and the cell will have a gray background. Properties that cannot be edited via the Batch Properties Editor, such as the model category, will also have a gray background.
- **Required Properties**. Depending on the type of item you are editing, certain properties may be required (e.g., if you are editing a model using a 2-parameter distribution, both parameters must be populated). In these cases, you can edit the properties, but you cannot delete them. This also means that the Cut command will function as a Copy command for these cells.
- **Published Models**. Models created by publishing results from an analysis in a ReliaSoft application cannot be edited in the Batch Properties Editor. To edit a model that was published from an analysis, you must return to the original data source, make the necessary changes, recalculate and republish the model.
- **Model Parameters**. Since the required parameters for a model will vary depending on the model type, the way you use the parameter columns in the Batch Properties Editor will vary. The parameter columns in the Batch Properties Editor for each model type are presented in the table shown next.

Model Type	Parameter 1	Parameter 2	Parameter 3	Parameter 4	Parameter 5	Parameter 6	Parameter 7	Parameter 8	Parameter 9	Parameter 10	Parameter 11	Parameter 12
Fixed Reliability	Reliability											
Fixed Unreliability	Unreliability											
Fixed Duration	Duration											
Weibull-2	Beta	Eta										
Weibull-3	Beta	Eta	Gamma									
Mixed Weibull-2	Population 1 Beta	Population 1 Eta	Population 1 Portion	Population 2 Beta	Population 2 Eta	Population 2 Portion						
Mixed Weibull-3	Population 1 Beta	Population 1 Eta	Population 1 Portion	Population 2 Beta	Population 2 Eta	Population 2 Portion	Population 3 Beta	Population 3 Eta	Population 3 Portion			
Mixed Weibull-4	Population 1 Beta	Population 1 Eta	Population 1 Portion	Population 2 Beta	Population 2 Eta	Population 2 Portion	Population 3 Beta	Population 3 Eta	Population 3 Portion	Population 4 Beta	Population 4 Eta	Population 4 Portion
Normal	Mean	Std										
Lognormal	Log-Mean	Log-Std										
Exponential-1	Mean Time or Lambda											
Exponential-2	Mean Time or Lambda	Gamma										
Generalized Gamma	Mu	Sigma	Lambda									
Gamma	Mu	K										
Logistic	Mu	Sigma										
Loglogistic	Mu	Sigma										
Gumbel	Mu	Sigma										
Arrhenius-Weibull	Beta	B	C									
Arrhenius-Lognormal	Log-Std	B	C									
Arrhenius-Exponential	B	C										
Eyring-Weibull	Beta	A	B									
Eyring-Lognormal	Log-Std	A	B									
Eyring-Exponential	A	B										
IPL-Weibull	Beta	K	n									
IPL-Lognormal	Log-Std	K	n									
IPL-Exponential	K	n										

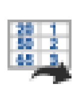
Spreadsheet Editor

You can choose the **Spreadsheet Editor** command on the ribbon to open the Spreadsheet Editor, which displays the resources or blocks in a spreadsheet-based view. This enables you to use the capabilities inherent in a spreadsheet (e.g., batch copying, formulas, etc.) to make editing multiple items easier. When you are finished editing, choose **Close** > **OK** to save your changes and close the Spreadsheet Editor or **Close** > **Cancel** to close the Spreadsheet Editor without saving your changes.

Exporting Information from the Batch Properties Editor

To print the contents of the Batch Properties Editor, choose **Administration** > **Output** > **Print** command on the ribbon.

To send the contents of the Batch Properties Editor to Excel, or to a built-in ReliaSoft Workbook or spreadsheet if that's available in the current application, choose **Administration** > **Output** > **Transfer Report**.



You can send either the current item (i.e., the table for the currently displayed page) or the full report (i.e., the tables for all pages in the Batch Properties Editor).

Tip: When you need to create or edit a large number of resources, you can also use ReliaSoft APIs to import the data from an Excel file or other data source. For example, the URDExample_V10.xlsm file at C:\Users\Public\Documents\ReliaSoft\Examples10\API enables you to quickly create/update multiple URDs with some basic properties (models and corrective tasks). If you have the necessary software coding experience, you can expand this tool or create your own custom applications to meet specific needs.

Project Planner

IN THIS CHAPTER

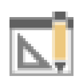
The Project Planner provides extensive project planning and management capabilities, including the ability to create timelines, evaluate progress, track resource usage and estimate costs. Any project in a ReliaSoft desktop application can be associated with a project plan.

The Project Planner allows you to create, edit or view the plan in any ReliaSoft desktop application except MPC. If the SEP web portal has been implemented for an enterprise database, you can also access a streamlined view of the plan from any web-enabled device.

What's Changed? Starting in Version 2020, the Project Planner is disabled by default. You can enable it for the current database via the Other Settings page of the Database Settings window.

Creating a Project Plan

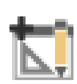
If the Project Planner is enabled for the database, each project in the database can have a project plan. To view the plan from within an open project, choose **Project > Management > Project Planner**. (In a secure database, any user with "read" access to the project will be able to view the plan and update any actions that are assigned to them. However, only users with the "Create/edit project plans" permission will be able to create or edit the plan.)



A blank plan will be created automatically the first time you open the Project Planner in a given project. By default, the project plan shows a sample starting gate, which records the date you opened the project plan for the first time. You can edit the sample gate and start building the plan from there.

Importing or Starting Again from a Blank Plan

Alternatively, you can start again from a blank plan or import an existing project plan from another project. In the Project Planner, choose **Project Planner > Plan > Create Project Plan**. (In a secure database, this command is available only to users with the “Create/edit project plans” permission.)



Then choose one of the following options. In both cases, you will need to enter the date the plan will begin in the **Set Start Date** field at the bottom of the window.

- Choose **Blank** to start again from a blank plan. This will delete all of the current gates (***there is no undo for delete***). It will also remove any actions that are assigned to the plan. The action resources will remain in the database unless/until an authorized user deletes them from the database via My Portal, Resource Manager or Actions Explorer.

  OR

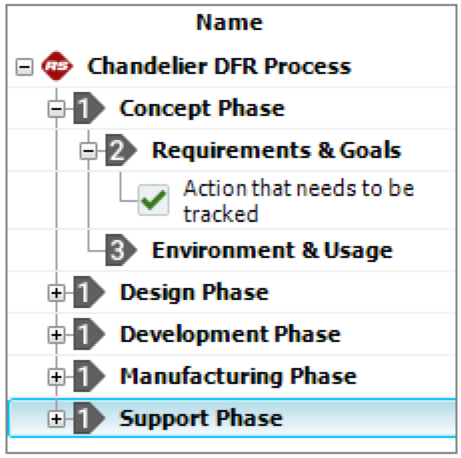
- Choose **Import from another project** to import an existing plan. You can import from another project in the current database or from any other existing standard or enterprise database.

  Choose a source database and source project for the import. The **Plan Preview** area shows the gates and actions (if any) in the project that is currently selected. By default, the import includes both gates and actions. If you want to import gates only, clear the **Include actions** check box at the bottom of the window.

Tip: If you want to use any of the Design for Reliability (DFR) project planning "starter" templates that were installed with XFMEA/RCM++/RBI Version 9, you can either import directly from one of your own projects (converted to Version 2021), or use the "Project_Planner_Templates_Rev1" database (choose **File > Help > Open Examples Folder** then open the file in the "Project Planner" folder). To preserve the integrity of this database, a copy will be created.

Project Plan Hierarchy

In the Project Planner, the plan consists of gates arranged in a tree-based hierarchy to represent the different phases of a process. You can also include action resources that represent specific assignments that need to be performed. For example, the configuration below shows a plan with five top-level gates (Concept Phase, Design Phase, etc.), where each gate has its own sub-gates and actions.



Certain properties of higher-level gates (e.g., status and dates) are determined by the properties of their dependent gates and actions. For example, the "Concept Phase" above has two direct dependents. If these are marked as complete, then "Concept Phase" is automatically marked complete. (See Gates in the Project Plan for details on how gates inherit their properties.)

In this way, the planner makes it easy for you to monitor how the progress of individual activities relates to the completion of the entire process.

Project Planner Columns

To hide or display Project Planner columns, right-click the column headings then click **Customize Columns**. (You can also change the column order by dragging and dropping column headings into the desired positions.) These settings are stored per computer/username. Any project that you open on this computer will have the same columns displayed, but other users may have different display preferences.

- **General** columns display graphics/text to summarize the status, as well as the priority level and percent completion.
- **Planned Timeline** columns display information about the original plan, including planned start/due dates and resource usage. You can also display the percentage of the budget that is allocated to different project planning resources (i.e., teams, facilities and materials).
- **Expected Timeline** columns display expected start/completion dates given any delays that may have occurred for prior activities, as well as estimates of the costs/man hours to date. (See Project Planner Timelines.)
- **Actual Timeline** columns display actual start/completion dates, as well as the costs/man hours that were used for completed gates/actions.

- **Delta Values** columns compare two different values, such as the difference between an action's actual and planned costs.
- **Relevant Metrics** display values for up to three selected metrics. (See Using Metrics in Project Planner Gates.)
- **Notes** display other applicable information, including descriptions, completion notes and whether an attached file exists.

Project Planner Timelines

For gates in the Project Planner, and for any action resource that is included in a project plan, three different timelines are used to track progress:

- The **Planned** timeline consists of the originally planned start/completion dates.
- The **Actual** timeline records when a gate or action has actually started or completed.
- The **Expected** timeline shows the start/completion dates that are now expected given the current situation. For project plans that use precedents, the expected dates for an activity automatically shift in response to delays in prior activities.

All of these timelines can be shown in gate/action properties and in the Project Planner hierarchy columns. The expected dates are used to determine the status (e.g., past start date, past due, etc.) and are also shown on the summary panel.

Planned Dates

For actions and for gates with no specified precedent, the planned start/completion dates are entered directly in the Gate Properties window.

For gates with precedents, the planned start date is automatically set to the working day following the precedent's due date (i.e., planned completion date). By using precedents in your plan, you guarantee that there will be no gaps between one gate's due date and the next one's start date.

In addition, the software uses the following rules to make sure a gate's planned timeline is consistent with that of its dependent gates and actions:

- If a gate's planned start date is changed (e.g., because of changes to a prior gate's planned duration), the planned dates of all dependents will be shifted in the same way.
- You cannot change a dependent's planned start date to precede that of its parent gate.
- If a dependent's due date is changed so that it comes after its parent gate's due date, the parent gate's due date will be changed to match it.

Actual Dates

Actual start/completion dates are entered directly for actions and for gates with no dependents. For gates that have dependents:

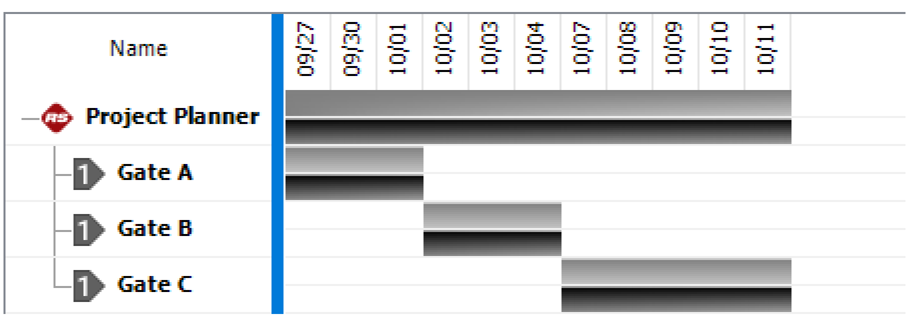
- The parent gate's actual start date is automatically set to match the earliest actual start date of its dependents.
- Once all the dependents have completion dates, the latest of those dates is set as the parent gate's actual completion.

Expected Dates

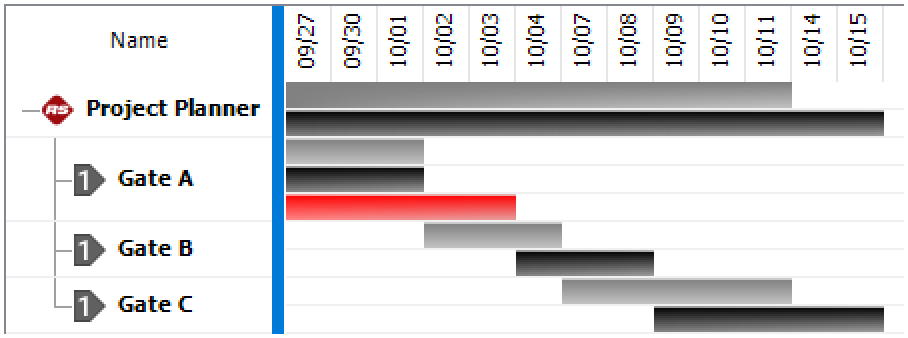
By default, expected start/completion dates are equal to the planned dates. However, if the gate/action is part of a plan that uses precedents, the two timelines can differ.

Specifically, when precedents are used, the expected dates will automatically shift in response to recorded delays in prior gates (while the planned dates remain unchanged). For example, if Gate A is the precedent of Gate B, and Gate A completes two days late, then the expected dates for Gate B will be its planned dates + two days. The same will apply to any other gates with precedents that trace back to Gate A.

As an example, consider the following Gantt chart for a plan that has not yet started. The planned and expected timelines (shown in gray and black, respectively) are identical at first.



However, if Gate A completes late (shown in red), the expected timeline shifts for the remaining gates. (The overall plan bars at the top also indicate that the expected duration for the entire plan has now increased by two days.)



Project Planner Gates

Each gate in the Project Planner has its own properties and a status that is determined based on its expected start/completion dates. Additional information about progress and resource usage is available in the summary panel.

To view the properties of a gate, double-click it, or select it and choose **Project Planner > Gate > Edit Gate**.

Gate Properties

- The **Gate** fields provide general information about the gate, including its **Name**, **Description** and:
  - **Deliverables** are what will be produced and delivered as a result of completing the gate.
  - **Priority** is a ranking of the gate's importance ("Low," "High," etc.).
- The **Precedent Gate** field identifies the gate that must be completed before the current gate can start, if any. If you choose to use precedents in a project plan:
  - The precedent gate must be on the same level in the hierarchy as the current gate. (If the **Use Precedence** option is selected, the precedent will be suggested automatically for each new gate you create but you can change it manually if desired.)
  - When a gate has a precedent, its planned start date will be automatically set to the working day after the precedent's due date. In addition, delays in the precedent will automatically shift the expected timeline for future gates.
- The **Planned Timeline** consists of the originally planned start/completion dates.

- The **Expected Timeline** shows the start/completion dates that are now expected given the current situation. For project plans that use precedents, the expected dates for an activity automatically shift in response to delays in prior activities. (See Project Planner Timelines.)
- The **Actual Timeline** records when a gate or action has actually started or completed. If the gate has dependents, these dates are automatically inherited in the following way:
  - The actual start date is the earliest actual start date from the dependents.
  - The actual completion date is the latest actual completion date from the dependents.
- The **Relevant Metrics** fields let you display up to three relevant metrics in the gate. The **Push to Metrics** fields let you "push" the gate's percent completed and actual vs. planned resource usage to metrics so they can be tracked and displayed in other locations. (See Using Metrics in Project Planner Gates.)
- The **History** fields display information about when the gate was created and last modified. 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 item.
- **Watch** allows each individual user to subscribe to receive an alert (via e-mail, SMS text message or portal message) when the gate is changed. (See Subscribing to a Watch.)
- **Gate Monitors** allows you to specify individual users and/or groups of users who will receive alerts for the gate, regardless of whether they have personally subscribed to a watch. (See Action and Gate Monitors.)

Gate Status

The gate status (which is displayed in the summary panel and in the status bar) is determined based on the same logic used for action statuses, which takes into account the expected start/completion date and any actual dates. For example, a gate's status is "Past Start Date" if it hasn't started and it's too late to start by the expected date, while "Past Due" means it is too late to complete on time.

Actions in Project Plans

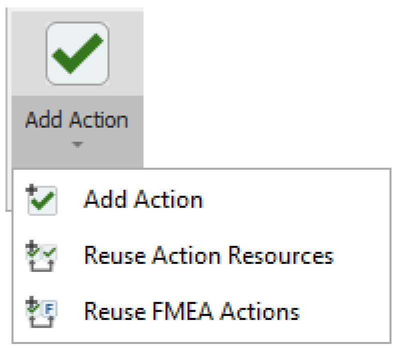
You can add actions to gates in the Project Planner in order to track specific assignments that need to be performed as part of specific phases of the plan. These are the same action resources that can be included in My Portal and SEP web portal, as well as FMEAs or Test Plans in XFMEA/RCM++.

When you add an action to a gate in the Project Planner, the software will:

- Use timelines in the action that automatically shift in response to delays in prior activities, which will affect how the action's status is determined.
- Display the action's plan summary in the Action properties window.
- Use the action's status, percent complete and resources to "roll up" to the gate it is assigned to.

Adding Actions to the Plan

With a gate selected in the Project Planner, choose **Project Planner > Action > Add Action**. (In a secure database, this is only available to users with the "Create/edit project plans" permission.)



- **Add Action** creates a new action resource.
- **Reuse Action Resources** allows you to select existing action(s) from the project that a) are not already used in the current plan and b) have dates that fit within the current gate. Specifically, the action's start date can't be before or more than a year after the gate's start date. (See Select Resource window.)
- **Reuse FMEA Actions** allows you to select one or more XFMEA/RCM++ system hierarchy items from the current project that have FMEAs. You can then select to use existing action(s) from those particular FMEAs in the project plan.

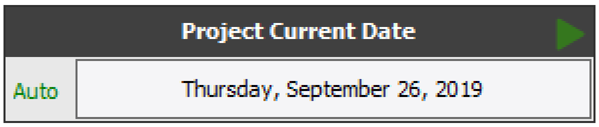
Project Plan Summary Panel

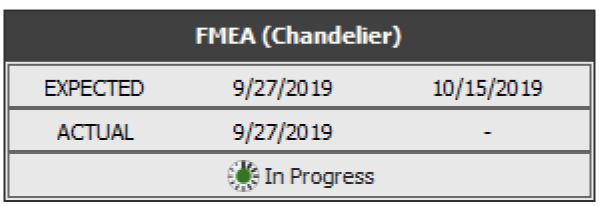
The project plan summary panel displays information about start/completion dates, status, progress and resource usage for an entire project plan, and for individual gates and actions.

The summary panel is accessible from multiple locations across the ReliaSoft desktop application:

- From the Project Planner, you can select a gate or action in the hierarchy to view its summary, or you can select the project at the top of the hierarchy to view a summary of the entire plan.
- The gate/action properties window shows the summary for that gate/action.

- The Plan Summary Page in My Portal displays a summary of the entire plan for the project that currently has focus.
- These summaries are also displayed in the SEP web portal, if it is implemented for an enterprise database.

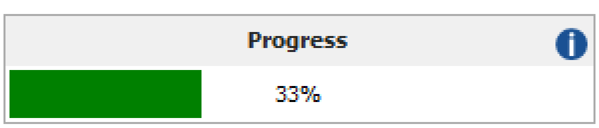
<table>
<tr>
<td>


</td>
<td>In most cases, this shows the current date. If you are working in the Project Planner and you want to see what the statuses would be on a different date, click the green "play" arrow ( ) and then select a different date from the calendar. When you’re ready to return to the current date, click the green "pause" bars ( ).</td>
</tr>
<tr>
<td>


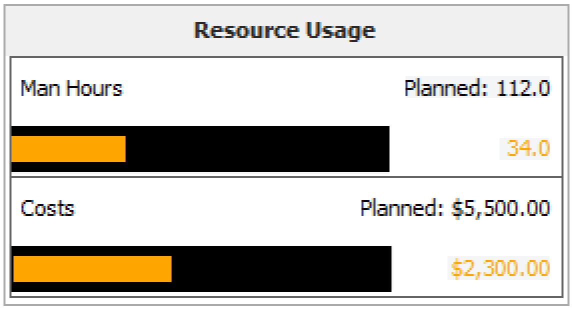
</td>
<td>This bar displays the name of the project, gate or action that the summary applies to, followed by:

• The EXPECTED start date and completion date. (This will be the same as the planned dates unless a prior delay has caused the expected dates to shift.)

• The ACTUAL start date and completion date (if the action is in progress or complete).

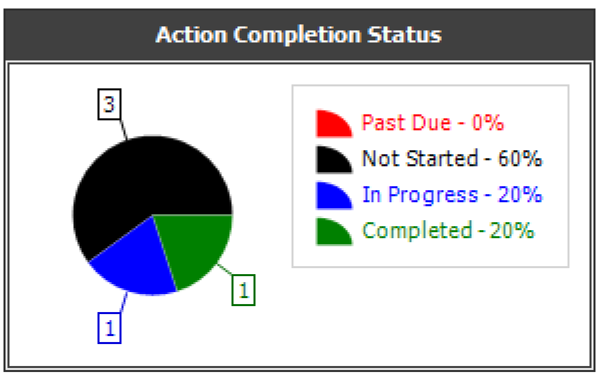
• The current status. (See Action Timeline and Status.)</td>
</tr>
<tr>
<td>


</td>
<td>For actions, the progress bar shows the percentage entered by the user. For gates, it shows the percentage of the total duration of dependent actions/gates that is complete. For example, if Action 1 (duration = 3 days) is complete and Action 2 (duration = 1 day) is incomplete, the progress is 75%.</td>
</tr>
</table>



For resource usage, the black bars show the planned man hours and costs. The colored bars show either the expected usage to date (estimated based on the number of days in progress), or the actual usage if complete. (See Costs and Man Hours.)

- Yellow = In progress and usage to date is still within plan
- Orange = In progress and usage to date has exceeded plan
- Green = Complete and actual usage was within plan
- Red = Complete and actual usage exceeded plan



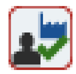
The pie chart will be visible only for projects or gates that have at least one action assigned (either directly or via dependent gates).

- Not Started = Actions that have not started and can complete on time
- In Progress = Actions that are in progress and can complete on time
- Completed = Actions that have a completion date
- Past Due = Actions that can no longer complete on time

Check Utilization

The Check Utilization window lets you examine the utilizations for individual users and facilities across one or more actions. It displays the total utilization for each resource on each specific date, so you can easily identify dates when the resource is over-utilized and make any adjustments that may be needed.

- To view utilizations for a specific action, open the action and choose **Tools > Check Utilization**.
- To view utilizations across multiple actions, open the Project Planner and choose **Project Planner > Tools > Check Utilization**.

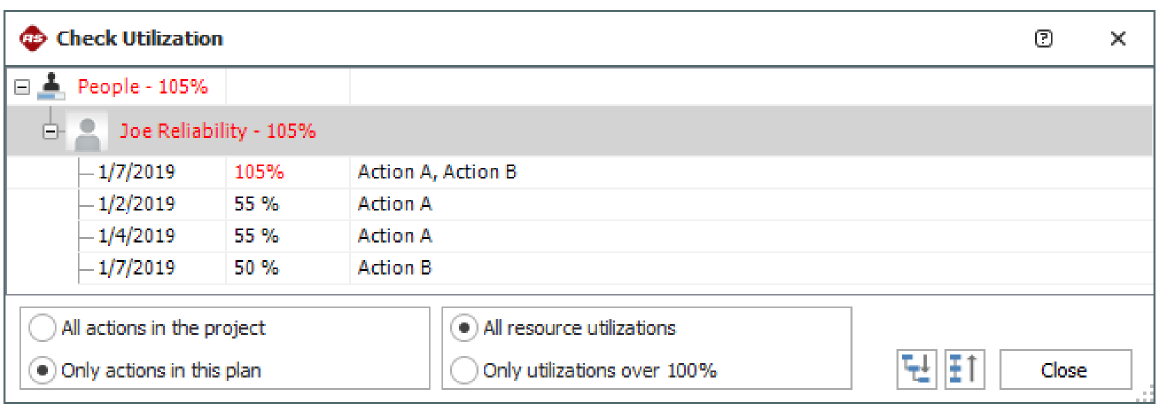


If you open the utility from the Project Planner ribbon, there are two options. By default, it shows **Only actions in this plan**. Alternatively, you can choose to see utilizations from **All actions in the project** (even if they are not used in the plan).

Regardless of whether you're checking a single action or the entire project plan, the utility shows **All resource utilizations** by default. Alternatively, you can choose to see **Only utilizations over 100%**.

Over-utilized resources (i.e., those with a utilization that exceeds 100%) are shown in red.

Double-click a resource to view its total utilization for each specific date, as well as the action(s) that use the resource on each date. For example, according to the window shown next, Joe Reliability's highest utilization for a single day exceeds 100%. By double-clicking his name, you can see that he is assigned to two actions that overlap on 1/7/2019, and as a result his combined utilization for that day is 105%.



Project Planner Gantt View

The Project Planner also includes a Gantt chart that makes it easy to visualize the timelines in your project and examine how different planning resources are utilized over time.

To access this view, click **Gantt** at the bottom of the Project Planner.

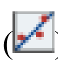
- Use the **Gantt Type** drop-down list on the control panel to specify what to display:
  - **Project Plan** shows timelines for gates and actions.
  - **Personnel Utilization** and **Facility Utilization** show the utilizations for the person responsible, team and/or facilities. (See Project Planning Resources.)
- In the **Gantt Display** area, select the part of the plan you want to view in the **Filter** drop-down list. For example you could view a specific gate (and its dependents) or a specific user (and all their utilizations). If you want to change the widths of columns that contain dates, use the **Fit Mode**.
- In the **Date Selections** area:
  - Select to view the entire plan or focus only on a specific date range.
  - Specify whether each column represents a specific number of **Days**, or **Other** increments such as weeks, months, quarters, etc.
- In the **Date Display** area:
  - If each column represents a single day, select whether to display **Non-Working Days**. If displayed, those columns will have a gray background. (See Working Days/Holidays.)
  - Use **Show Planned**, **Show Expected** and **Show Actual** to specify which timeline bars will be displayed.
    - To specify the colors that will be used in both the project plan Gantt chart and the timeline plot, choose **View > Settings > Set Colors**. (See Set Colors for Project Planner.)
  - For the planned and actual timeline bars in the project plan chart, you can also choose whether to **Show Percent Completed**.
- If you are using one of the utilization charts and the date columns each represent more than a single day, use the **Calculate Utilization As** area to specify how to calculate the utilization. This can be:
  - The **Max** (highest) daily utilization among all days in the interval. For example, if the interval is 3 days and the utilizations were 15%, 0% and 10%, the max will be 15%.
  - The **Average** across only the days in the interval that have utilizations. For example, if the interval is 5 days but the resource was only utilized on 3 of those days, the average will be the total utilization/3 days.

- The **Overall Average** across all working days in the interval. For example, if the interval is 5 working days, the overall average will be the total utilization/5 days even if the resource was not utilized every day.

Project Planner Plot View

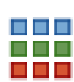
The Project Planner includes a plot view that makes it easy to visualize the timelines in your project.

To access this view, click **Plot** at the bottom of the Project Planner.

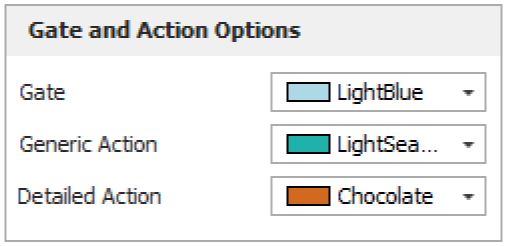
- Use the **Filter Options** area to select whether to filter the data by **Gate** or by using a **Date** range. You can also choose to view data for **Top Level Gates** only.
- In the Display Options area:
  - **Show Planned**, **Show Expected** and **Show Actual** determine which timelines will be shown in the plot.
    - To specify the colors that will be used in both the project plan Gantt chart and the plot, choose **View > Settings > Set Colors**. (See Set Colors for Project Planner.)
  - **Display Actions** shows the dependent actions for each gate.
  - **Show Project Item** shows the “project” gate at the top of the hierarchy.
  - **Show First Gate on Top** determines whether the gate with the earliest start date will show at the top or bottom of the chart.
  - **Auto Refresh** refreshes the plot automatically whenever changes have been made. If this option is cleared, you can click the **Redraw Plot** icon () to refresh the display.
  - **Keep Aspect Ratio** maintains the proportional relationship between the width and height of the plot image.

Set Colors

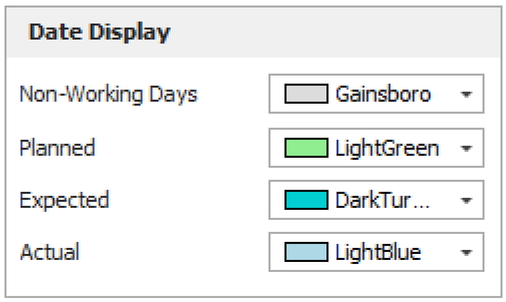
To open the Set Colors window for the Project Planner, choose **View > Settings > Set Colors**.



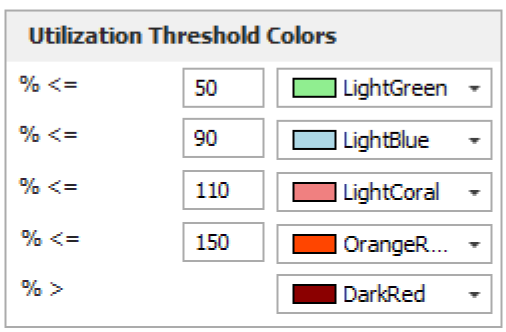
- The **Gate and Action Options** apply to the Gantt chart (if viewing the plan), plan hierarchy and plot.



- The **Date Display** options apply only to the Gantt chart and plot.



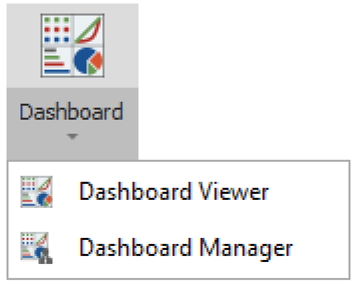
- The **Utilization Threshold Colors** apply only when using the Gantt chart to view team or facility utilization.



Project Planner Dashboards

You can use the flexible Dashboard utility for presenting data from any Project Planner in the database.

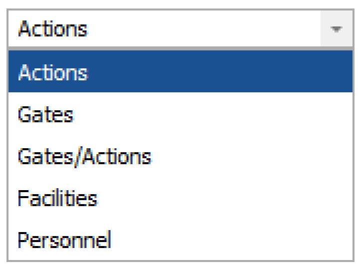
To access this feature, open the planner and choose **Project Planner > Tools > Dashboard**.



As with any other dashboard, you can use the **Dashboard Viewer** to select any of the layouts that have been predefined for this type of data. In a secure database, only users with the "Manage dashboard layouts" permission can use the **Dashboard Manager** to create or edit layouts.

Data Source Drop-Down List

When you are creating a dashboard layout based on project plan data, the drop-down list at the top of the Data Source Panel gives the following options:



- **Actions** returns data only from actions in the current plan.
- **Gates** returns data only from gates in the current plan.
- **Gates/Actions** returns data from both actions and gates.
- **Facilities** returns data from the facilities assigned to actions.
- **Personnel** returns data from the person responsible and/or team assigned to actions.

Each data source allows you to choose relevant fields from the record properties, summary panel and/or project plan hierarchy.

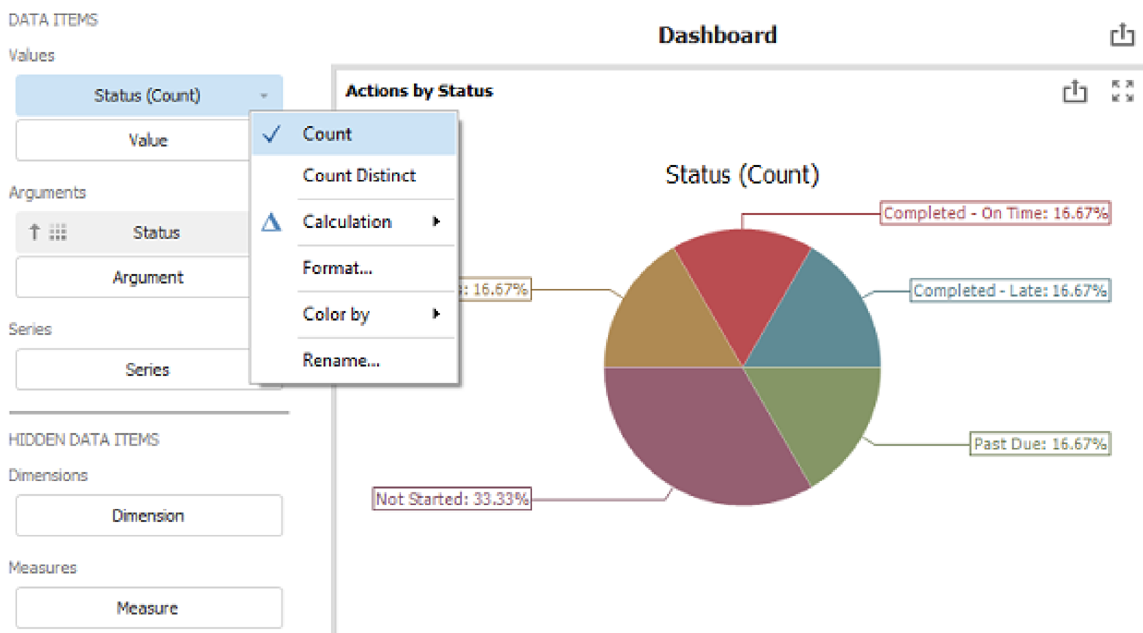
Note that all of the data fields used within a particular dashboard item must come from the same source. For example, you can't combine fields from the "Actions" data source and the "Gates" data source in the same chart. If you want to create a chart that includes both actions and gates, use the fields from the "Gates/Actions" data source instead.

Examples for Different Field Types

The icon for each field indicates the data type: text, number or date. There are many possible ways to use these fields in your own customized dashboard layouts. The following simple examples demonstrate basic applications for each data type.

Example Using Counts

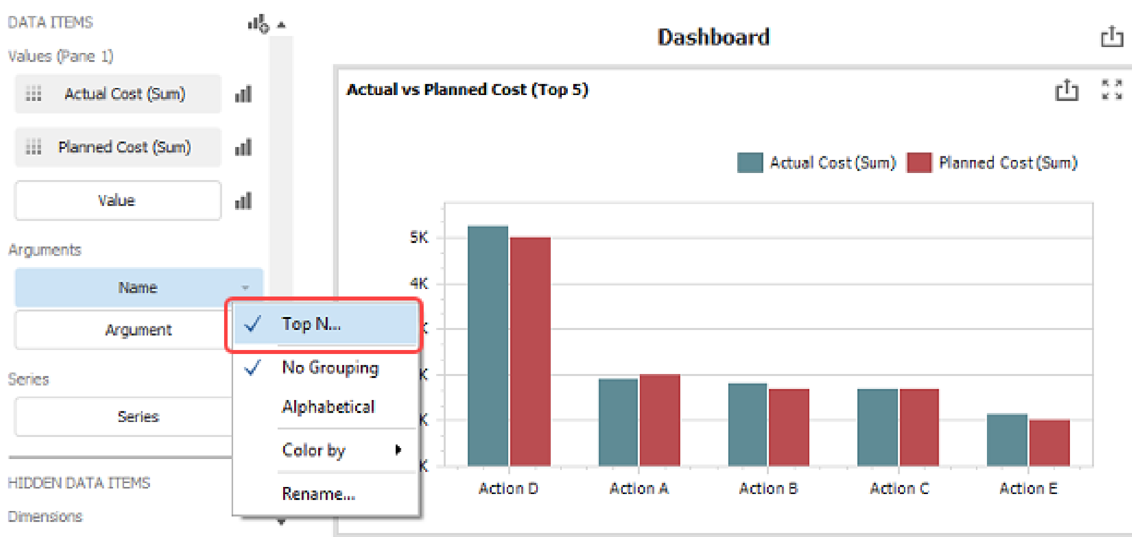
This pie chart uses the **Count** value for a text field to show the actions in the current project plan broken down by status. The argument determines the slices shown in the pie.



Tip: If you want to see both the quantity and the percentage, choose **Design > Data Labels > Argument, Value And Percent**.

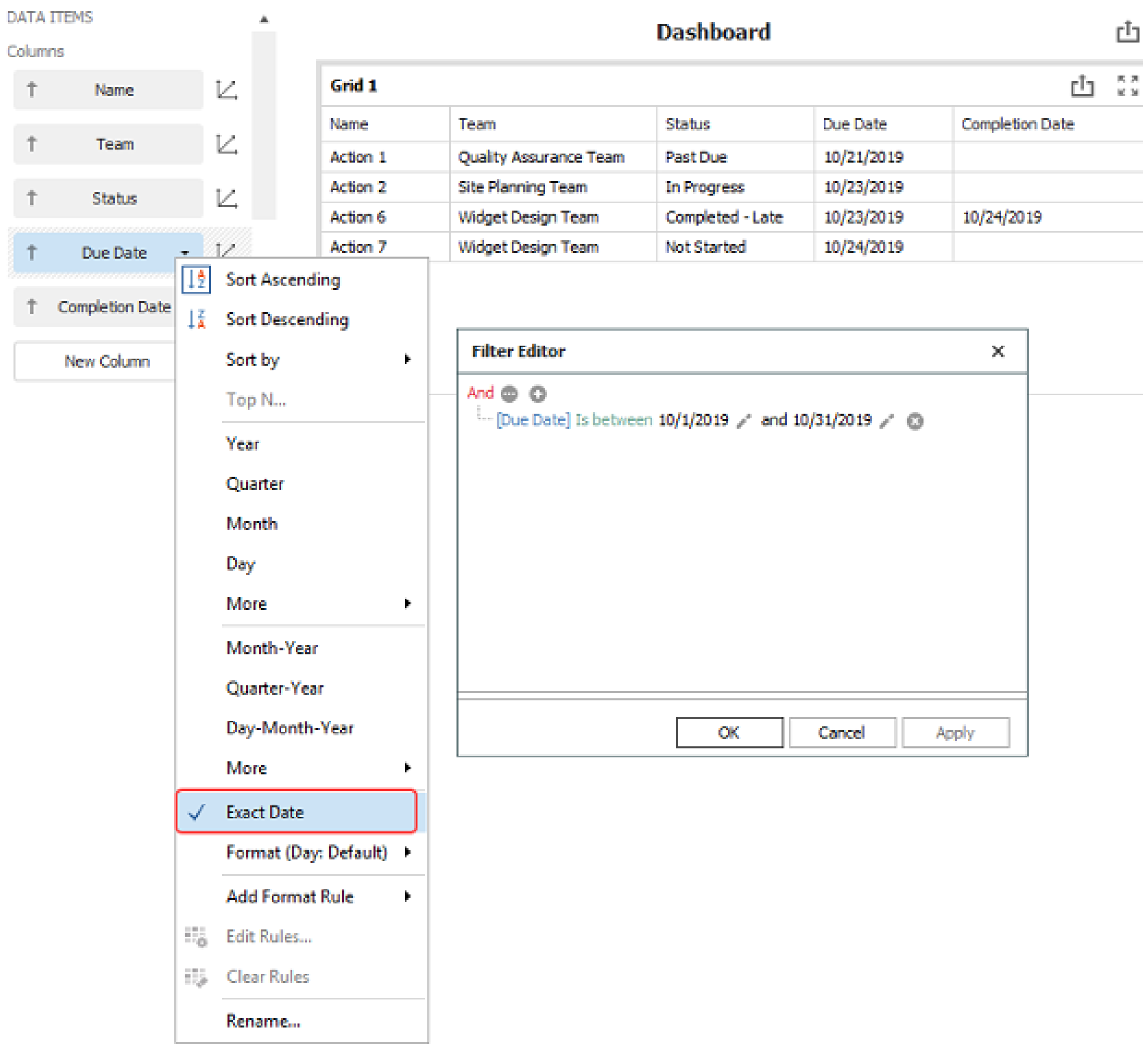
Example Using Numbers

This bar chart uses the **Sum** values for two number fields to show the top 5 most costly actions in the current project plan, with comparison to the planned cost. The arguments determine how the bars are labeled, grouped and sorted.



Example Using Dates

This grid uses the **Exact Date** for two date fields to show a list of all actions in the current plan that are due in the current month. (To set the criteria that filters the records shown in the grid, right-click inside the chart and choose **Edit Filter**.)



Project Planner Ribbon

Project Planner Tab

The Project Planner tab in the Project Planner ribbon contains the following commands.

Project Planner

 **Refresh** updates the Project Planner hierarchy, if necessary. For example, if multiple users are accessing the same plan simultaneously, this will update the plan to reflect any changes made by other users.

Create Project Plan opens the Create Project Plan window so you can start again from a blank plan, or import an existing plan from another project.

Delete Entire Plan deletes every gate and removes every action from the plan.

Item Properties allows you to view or edit the Identifiers defined for the current plan. These identifiers can be used in the Analysis Explorer to explore all of the different types of analyses (including project plans) in the current database.

Clipboard

Paste pastes the contents of the Clipboard into the level below the selected gate (or on the same level of the selected action). If you want to paste gates only (without sub-gates and actions), use the Paste Without Actions command.

Cut cuts the selected gate/action and any dependents to the Clipboard. You can paste this information into another Project Planner within the same database.

Copy copies the selected gate/action and any dependents to the Clipboard. To copy everything, use the **Copy Entire Plan** command. You can paste this information into another Project Planner within the same database.

Delete deletes the selected gate/action and any dependents.

Gate

Add Gate

- **Add Same Level Gate** adds a gate on the same level as the selected gate.
- **Add Next Level Gate** adds a gate one level below the selected gate.
- **Add Gate** adds a top-level gate to the end of the plan.
- **Insert Gate** inserts a gate at the selected location.

Use Precedence, when selected, automatically sets a new gate's precedent to the prior gate, which allows you to take advantage of expected timelines and ensure that there are no gaps between gates. If you clear this option, the default precedent will be "None."

Edit Gate opens the Gate Properties window.

Promote and **Demote** moves the selected gate to the next higher or lower level in the plan. An item can be demoted only if there is another item on the same level and above the selected item that it can be demoted under.

Action

Add Action

- **Add Action** creates a new action resource.
- **Reuse Action Resources** allows you to select existing action(s) from the project that a) are not already used in the current plan and b) have dates that fit within the current gate. Specifically, the action’s start date can’t be before or more than a year after the gate’s start date. (See Select Resource window.)
- **Reuse FMEA Actions** allows you to select one or more XFMEA/RCM++ system hierarchy items from the current project that have FMEAs. You can then select to use existing action(s) from those particular FMEAs in the project plan.

Edit Action opens action properties window. (See Actions in the Project Planner.)

Tools

Remove Gaps in Dates automatically assigns precedents to every applicable gate. This ensures that there will be no gaps in the planned or expected timelines, and it allows the expected dates for an activity to automatically shift in response to delays in prior activities. (See Project Planner Timelines.) You can choose to remove gaps in the **Entire Plan** or just for the gates that fall under the selected **Branch**.

Check Utilization opens the Check Utilization window so you can examine the utilizations for individual users and facilities across multiple actions (e.g., is a particular user assigned to too many actions at the same time, or is a particular test facility already booked for a particular timeframe).

Expand All and **Collapse All** expands or collapses the gates in the project plan hierarchy.

Dashboard

- **Dashboard Viewer** opens the Dashboard Viewer so you can view dashboards based on predefined layouts.
- **Dashboard Manager** opens the Dashboard Layout Manager so you can create, edit and delete the predefined layouts that will be available for any user to view for any project plan in the database.

Excel Import/Export

- **Export View to Excel** exports the entire hierarchy and all displayed columns to an Excel sheet in a format that's suitable for presentation.
- **Import Hierarchy** and **Export Hierarchy** use an Excel format that's suitable for recreating the plan hierarchy in another project plan using information about precedents, outline level, etc.

Copy Current View copies the currently visible contents of the plan hierarchy or Gantt chart to the Clipboard as a graphic that can be pasted to other applications.

View Tab

The View tab in the Project Planner ribbon contains the following commands.

View

Color Code Deltas applies highlight colors to any "Delta Values" columns for dates or durations that are displayed in the project plan hierarchy (e.g., the difference between the planned duration and the actual duration, or between the expected completion date and the actual completion date). Highlights are applied on a gradient from blue (ahead of schedule) to red (significant delay).

Fit All Columns automatically resizes all columns to fit their own data.

Settings

Customize Columns allows you to hide or display specific columns in the plan hierarchy.

Save View and **Apply View** create or apply a view that stores your preferences for which columns to display in the plan hierarchy and which colors to use for gates, actions, timelines and utilizations. Each view is saved per computer/username.

Set Colors allows you to select the colors to use for gates, actions, timelines and utilizations.

Zoom

Normal Zoom sets the degree of magnification to 100%

Zoom In and **Zoom Out** increase or decrease the font size by 25% increments.

Custom Zoom allows you to specify the zoom percentage.

Plot Utilities

IN THIS CHAPTER

ReliaSoft applications offer a variety of plotting utilities for visual presentation of your analysis results. The following topics provide general information about built-in utilities, where the types of plots/charts available are predefined based on the analysis method and type of data you are working with. For more information about specific plot types, please consult the documentation for the relevant analysis.

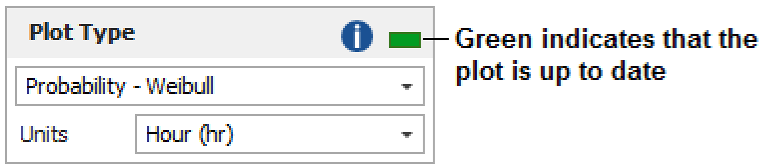
Tip: In addition to the built-in plotting utilities described here, you can also create your own custom plots/charts in the ReliaSoft Workbooks and Dashboards.

Basic Plot Features

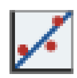
This topic describes some basic features and capabilities that are available in most built-in plot utilities in ReliaSoft desktop applications (i.e., utilities where the types of plots available are predefined based on the analysis method and type of data you are working with). For information about creating your own custom charts in ReliaSoft Workbooks or Dashboards, see Custom Charts or Dashboard Layout Designer.

Redraw Plots

A plot needs to be refreshed whenever its data source has been recalculated or when its inputs or settings have been modified. This ensures that the plot reflects the most current results. In most plot sheets, the control panel displays a status light that indicates whether the plot needs to be refreshed. A green light indicates that the data and the plot are in sync, while a red light indicates that the plot is out of sync with the latest analysis.



You can manually update a plot to reflect any changes by clicking the **Redraw Plot** icon in the plot's control panel.



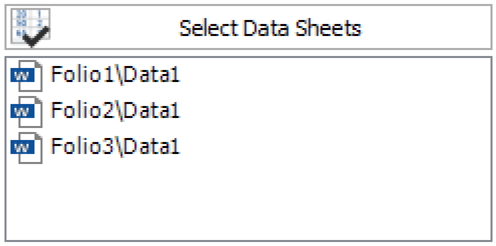
Alternatively, a plot can be refreshed automatically whenever changes have been made by selecting the **Auto Refresh** check box on the plot's control panel.

Identify a Plot's Source Analysis

For most plots, the control panel displays a blue link that displays the plot's source analysis, as shown in the following example for a Weibull++ plot. You can click the link to open the folio and view the data set.

In some specialty plots (side-by-side plot, life comparison tool, etc.), you can select the source analysis for the plot by clicking the icon on the plot's control panel or ribbon.

If the plot contains results from more than one analysis (e.g., overlay plots), the control panel includes a button that allows you to add or remove analyses from the plot, as shown in the example below for Weibull++ data sheets. In addition, you can click and drag the data sheets to the desired order.



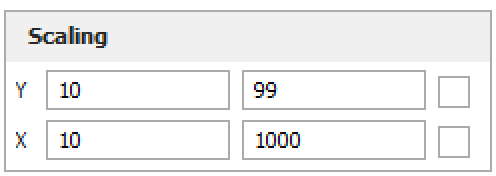
Show or Hide Plot Items

When applicable, a plot may provide options for showing or hiding certain plot items such as data points, lines, probability scales, etc. The available options depend on the plot type and analysis. To view the options for the plot you are working with, choose **Plot > Actions > Show/Hide Plot Items** or right-click the plot and choose **Show/Hide** Items on the shortcut menu.

Select the check box for each item you want to appear in the plot, or clear the item's check box to hide the item from view.

Scaling

When applicable, the plot control panel may display X and Y scaling boxes that show the minimum and maximum values for the x- and y-axes. You can clear the check boxes and click inside the fields to manually edit the values, or select the check boxes to have the application choose the appropriate values for the range, based on the data.



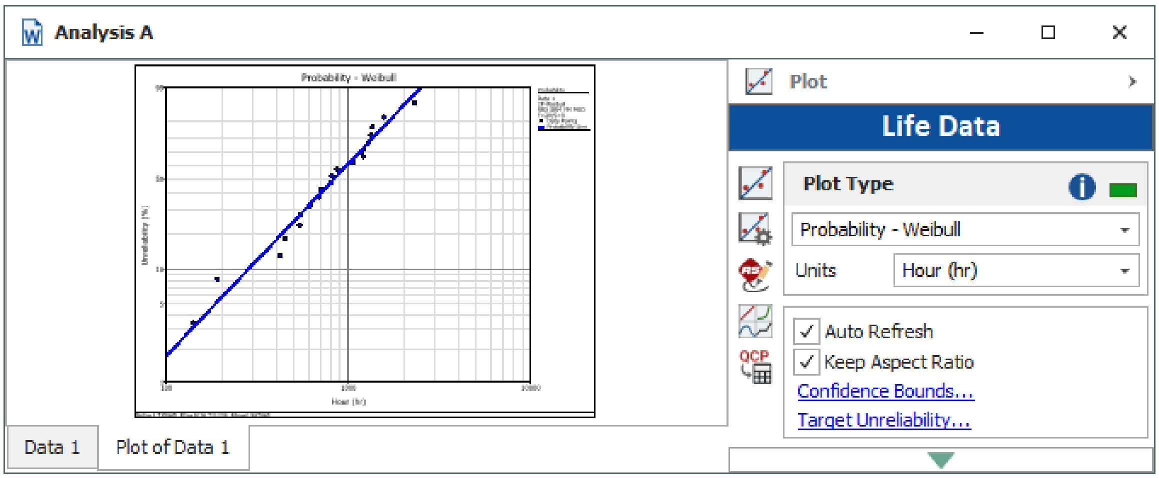
Zoom

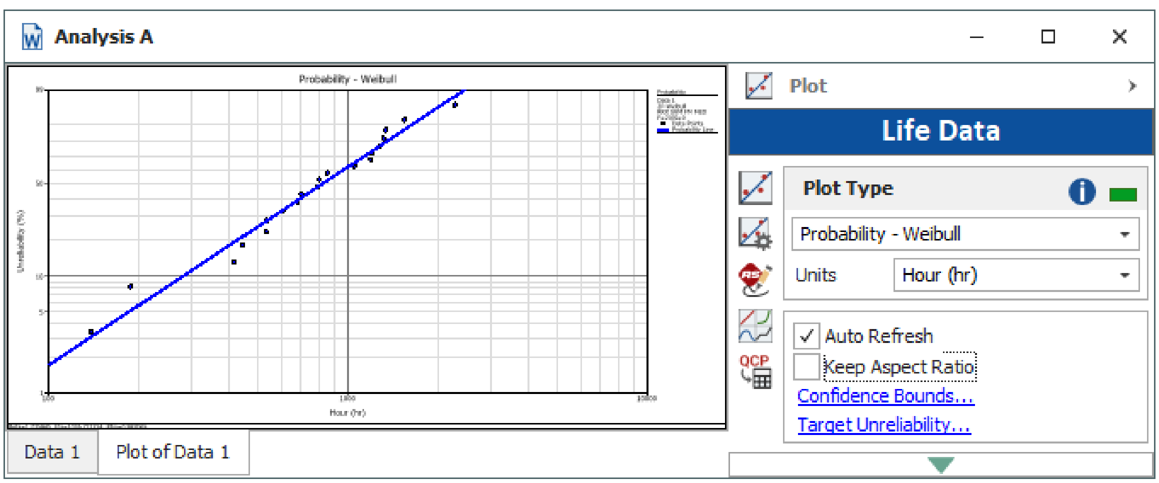
The zoom function is available for plots that allow both X and Y scales to be adjusted. To zoom in or out of a plot, click inside the plot, point the mouse pointer over the area of interest, and then rotate the mouse wheel. Press **ESC** to return to the original scale.

For non-3D plots, you can also select a specific area to zoom in by holding the **CTRL** + **SHIFT** keys while clicking and dragging the mouse over the desired area. You can customize the color and line style of the zoom rectangle via the Zoom page of the Plot Setup.

Aspect Ratio

Plots are automatically resized whenever you resize the plot sheets. To maintain the proportional relationship between the width and height of the plot image, select the **Keep Aspect Ratio** check box on the plot's control panel (as shown in the first example below). Clear the check box if you wish to stretch the dimensions of the plot graphic to fill the plot sheet (as shown in the second example below).



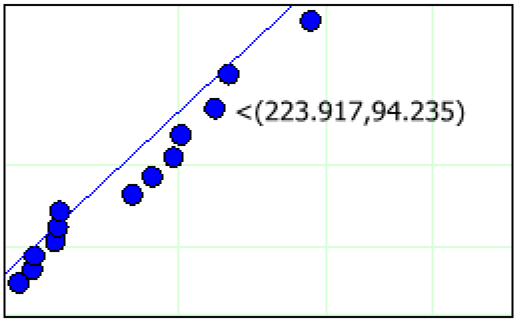


Show Coordinates

To show the coordinates for a location on a plot, press **SHIFT** and click the plot (no need to hold down the mouse button). To stop showing the coordinates and return the pointer to its normal mode, click the plot again.

To track the coordinates on a plot line, click the line. A crosshair shows the current location on the line. The crosshair will stay on the line and track the coordinates as you move the pointer.

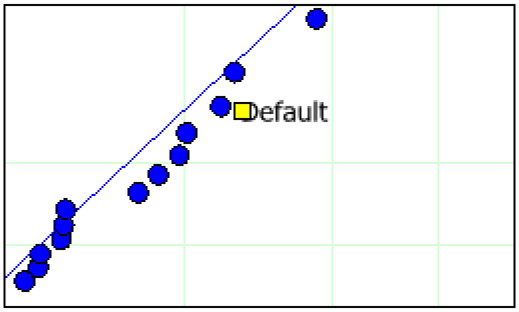
When applicable, you can add a label that displays the coordinates of a location by pressing **CTRL + ALT** and then clicking the location you want labeled.



Add Custom Labels

When applicable, you can add a custom label to a plot by pressing **CTRL** and then clicking the plot. A label named "Default" will appear in the plot. Select the label to edit the text directly in the plot or double-click it to open the Edit Label window, which gives you the option to select a font style or format.

You can then move the label by clicking it and dragging its yellow handle to the desired location on the plot. (For more extensive annotations, see ReliaSoft Draw.)

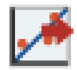


Move Plot Items

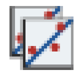
If your plot has lines or points that are obscuring one or the other, you can move an item by pressing **ALT** and clicking the item. The Move Plot Item window will appear, giving you the option to move the selected item either to the front or back of the obscuring graphic.

Export or Copy Plot Graphics

You can save a plot graphic as a *.wmf, *.png, *.gif, or *.jpg file for use in other applications by clicking the **Export Plot Graphic** icon on the plot's control panel, or by choosing [**Plot/3D Plot**] > **Actions** > **Export Plot Graphic** if applicable.



In addition, you can save a plot graphic to the Clipboard by clicking the **Copy Plot Graphic** icon on the plot's control panel or by choosing [**Plot/3D Plot**] > **Actions** > **Copy Plot Graphic** if applicable.

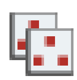


When saving a 3D plot graphic to the Clipboard, the image will be in *.bmp format. For all other types of plots there are three choices: If you will be pasting copied plots into any one of the spreadsheets built in to ReliaSoft desktop applications (e.g., ReliaSoft Workbooks or General Spreadsheets), choose **Metafile Optimized for ReliaSoft Spreadsheet**. If you will be pasting them into external applications, choose **Bitmap** or **Metafile Optimized for External Use**.

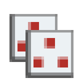
Export or Copy Plot Data

You can export the data from a plot to the Clipboard so that you can paste it into another application. The information depends on the plot type and on the application you are using.

- In Weibull++ or BlockSim, choose **Plot** > **Actions** > **Copy Plot Data**.

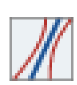


- In XFMEA/RCM++ or Lambda Predict, click the **Copy Plot Data** icon on the plot's control panel.



Setting Confidence Bounds

If your analysis includes calculations for confidence bounds, you can display the bounds in the associated plot by clicking the **Confidence Bounds** link on the plot control panel or by choosing **Plot** > **Confidence Bounds** > **Confidence Bounds**.



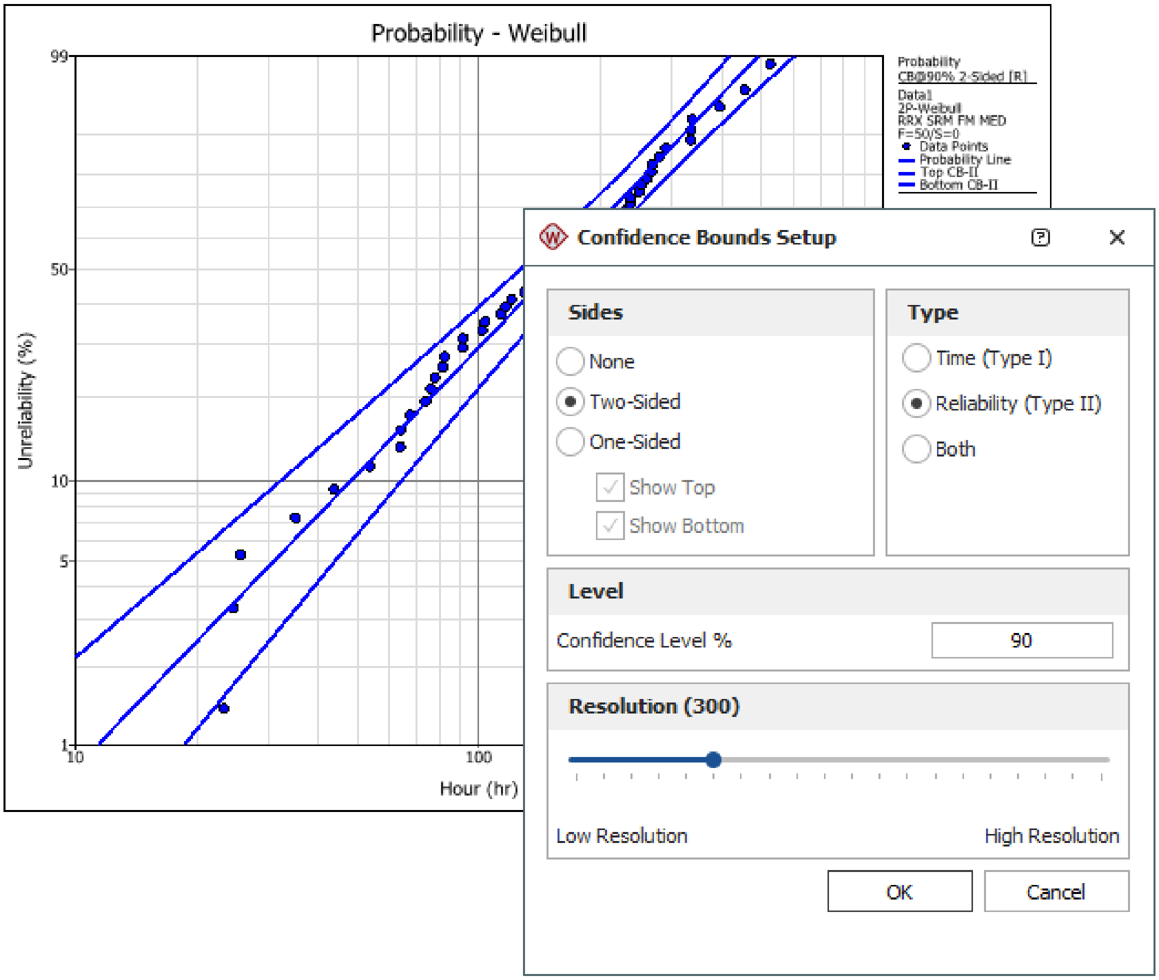
Depending on the type of data you are working with, you may be presented with different options for setting up the confidence bounds on the plot. (The ReliaWiki resource portal provides information on the background theory of confidence bounds at: http://www.reliawiki.org/index.php/Confidence Bounds.)

In the Confidence Bounds Setup window, you will need to:

1. Select which lines to show on the plot (e.g., two-sided, bottom one-sided, etc.)
2. Select which type of bounds to show on the plot. This may be the bounds on the time estimate (Type I), bounds on the reliability/unreliability estimate (Type II: Reliability); or when applicable, bounds on functions such as number of failures, growth potential, etc. (Type II: Function).
3. Enter the percent confidence level.

To change the level of detail at which the bounds are plotted, 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 next to the field name. Note that higher resolutions will take longer to plot.

To hide the confidence bounds, choose **Plot > Confidence Bounds > Hide Confidence Bounds**.

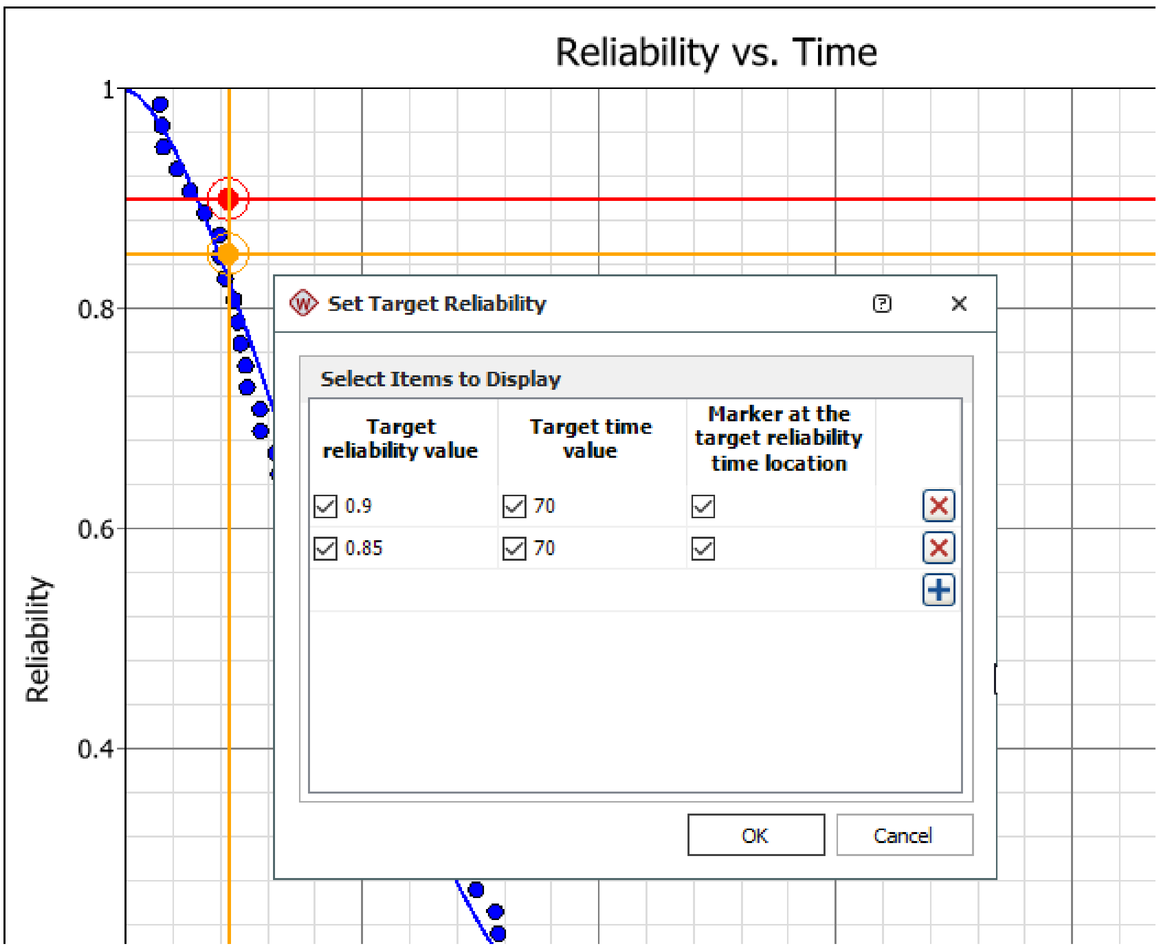

Probability - Weibull
Unreliability (%)
Hour (hr)
Probability
CB@90% 2-Sided [R]
Data1
2P-Weibull
RRX SRM FM MED
F=50/S=0
Data Points
Probability Line
Top CB-II
Bottom CB-II
Confidence Bounds Setup
Sides
None
Two-Sided
One-Sided
Show Top
Show Bottom
Type
Time (Type I)
Reliability (Type II)
Both
Level
Confidence Level %
90
Resolution (300)
Low Resolution
High Resolution
OK
Cancel

Setting Target Markers

When applicable, the plot control panel may display a link for **Target Reliability**, **Target Unreliability**, **Target MTBF/FI** or **Target Availability**. These links give you the option to display markers for target values on the plot.

You can select to show a specified target value (displayed as a horizontal line), target time value (displayed as a vertical line), and/or insert a marker at the point where the two target values intersect. Below is an example for a plot that shows the target reliability.

Click the **Add** or **Remove** icon to manage the list of target values to be displayed. You can add up to 5 sets of target values/markers to a plot.

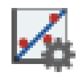


Plot Setup

The Plot Setup utility provides advanced settings for customizing the look of a plot to meet your needs and preferences. It allows you to: a) make changes and apply them to the current plot; b) save the changes and use them as default settings for all new plots; c) restore previously saved default values; or d) reset all settings to the shipped defaults.

Tip: You can quickly set some basic default settings for all new plots created in the current ReliaSoft desktop application via the Plots page of the Application Setup (**File > Application Setup**). The Application Setup also allows you to set some default analysis settings that may affect the appearance of plots (varies depending on the ReliaSoft desktop application).

To open the Plot Setup, click the **Plot Setup** icon on the plot's control panel.



Alternatively, you can double-click the item on the plot you wish to edit (e.g., points, lines, etc.) and the Plot Setup will open and display automatically the options associated with that item.

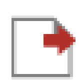
To apply your changes to the current plot, click **OK**. To save your changes as the new defaults, or restore previously saved defaults, or reset to shipped defaults, click the **Defaults** button. (See Plot Defaults Window for more information.)

What's Changed? Starting in Version 2019, many plot options that had also been available from the Application Setup window are now available only with the Plot Setup utility.

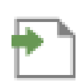
Importing and Exporting Plot Settings

Starting in Version 2019, you have the option to import and export plot settings to an XML file. This allows you to define the settings for one plot and then use them in other plots.

- To export the current settings, click **Export**.



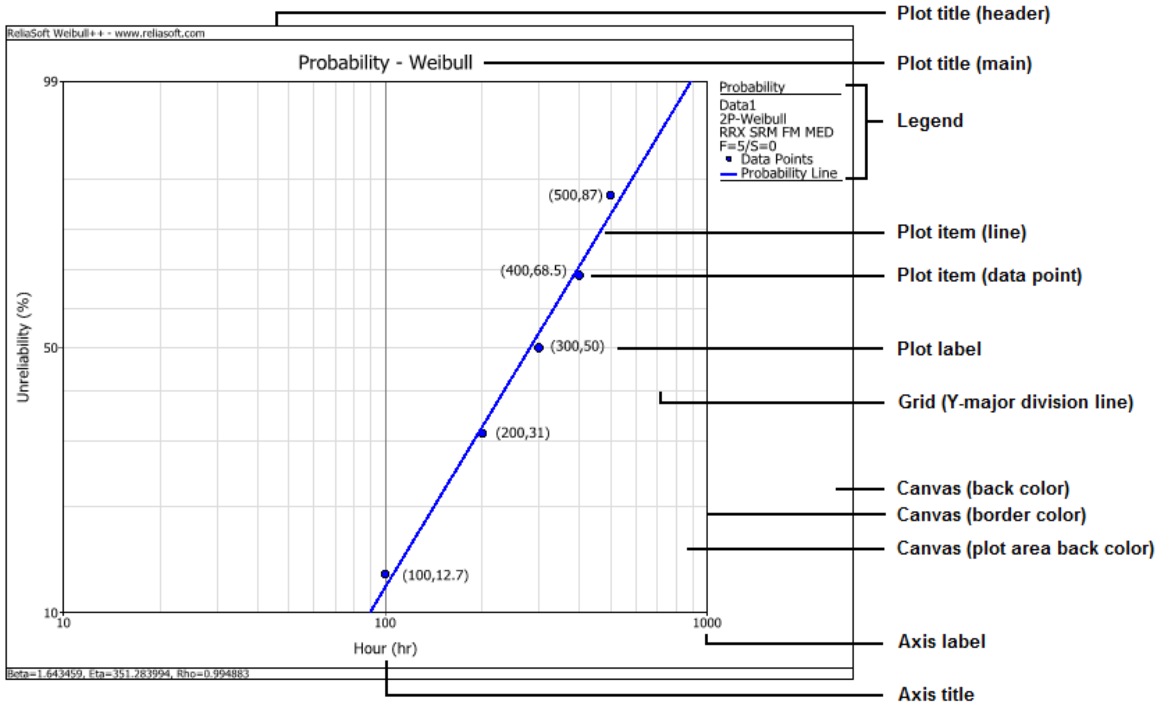
- To import settings from an XML file, click **Import**.



Plot Elements

The Plot Setup is divided into pages that contain related settings for particular plot elements. The available pages/elements will depend on the type of plot you are working with.

As an example, the following picture identifies some of the basic plot elements that can be configured via the Plot Setup.



Plot Setup: 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. If you type new text in an input box, either alone or in addition to the asterisk, the text you type will be displayed on the plot. For example, if you add **(Helicopter)** to the **Main Title** field, the default main title will be displayed, followed by "(Helicopter)." The changes you make will be applied regardless of plot type, so the main title of a Failure Rate vs Time plot would then be "Failure Rate vs Time (Helicopter)" and the main title of a System Failures plot would be "System Failures (Helicopter)." (This assumes that you have not changed the default main titles from the ones shipped with the application.)

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 **Set 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 **Set Image** button. The image will also appear at the left side of the header or footer text. To remove the image, click **Clear Image**.

Plot Setup: 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.

- **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 type new text in an input box, either alone or in addition to the asterisk, the text you type will be displayed on the plot. For example, if you add **(Hours)** to the **Y-Axis Title** field, the default y-Axis title will be displayed, followed by "(Hours)." The changes you make will be applied regardless of plot type, so the y-axis title of a Static Reliability Importance plot would then be "Static Reliability Importance (Hours)" and the y-axis title of a Throughput vs. Time plot would be "Throughput(t) (Hours)." (This assumes that you have not changed the default axis titles from the ones shipped with the application.)

- **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.

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 display of numerical values shown on the axes. The available options depend on the setting of the **Use Significant Figure Formatting** check box.

  If selected:

  Use the significant figures fields to set the maximum number of digits that are displayed (e.g., the number 1.23456789 displays as 1 with 1 significant figure while it displays as 1.23457 with 6 significant figures). If the value is large and contains more significant figures than the set value, then the number is simply rounded to the nearest whole number.

  If not selected:

  The mathematical precision sets the number of decimal places that a number will display. 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 greater than 1,000 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 reenter the settings.

Plot Setup: 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. 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.)
  - **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.
  - **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.

Plot Setup: 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.)
- 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.

- For bar and pie charts in BlockSim only, **Color Spectrum** allows you to set the color limits for the plot.
    - You can set both lower and upper color limits by dragging the markers on the scale between 0% and 100%.
    - **Show all values (0-100%) in legend** if selected, the legend will show the full range of values; if not selected, the legend will show only the values between the upper and lower limits that you have specified.

  The preview shows how the specified settings will appear in the legend.

Plot Setup: 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.

Plot Setup: 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 reenter the settings.

Plot Setup: Bars Page

The Bars page allows you to customize the appearance of the bars in the plot. This page is available only when you are working with a bar chart.

- **Bar Border Lines** allows you to hide or show the border line for the bars, as well as to select a color, a thickness and a style for the border.
- **Bar Orientation** allows you to select how you want the plot laid out. You can choose to use vertical bars or horizontal bars.

Plot Setup: Slices Page

The Slices page allows you to customize the appearance of the slices in the plot. This page is available only when you are working with a pie chart or tableau 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).

Plot Setup: 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 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. You can adjust the thickness of a grid line by entering a positive integer in the input box. Adjust the thickness of the line by entering a positive integer in the input box. Change the style of each line such as solid, dashed, etc., by clicking the **Line Style** box.
- **Point Settings**
  - **Color, Shape, Size and Fill** allows you to specify the appearance of the point body. To change the color of the point, click the **Color** box. You can select a point shape from the drop-down list, select the size of the point (with 1 being the smallest point) and select a fill style from the drop-down list.

- **Border Color, Thickness and Style** allows you to modify the appearance of the border of the points. To change the color of the border, click the **Color** box. Adjust the thickness of the border by entering a positive integer in the input box. Change the style of each line such as solid, dashed, etc., by clicking the **Line Style** box.

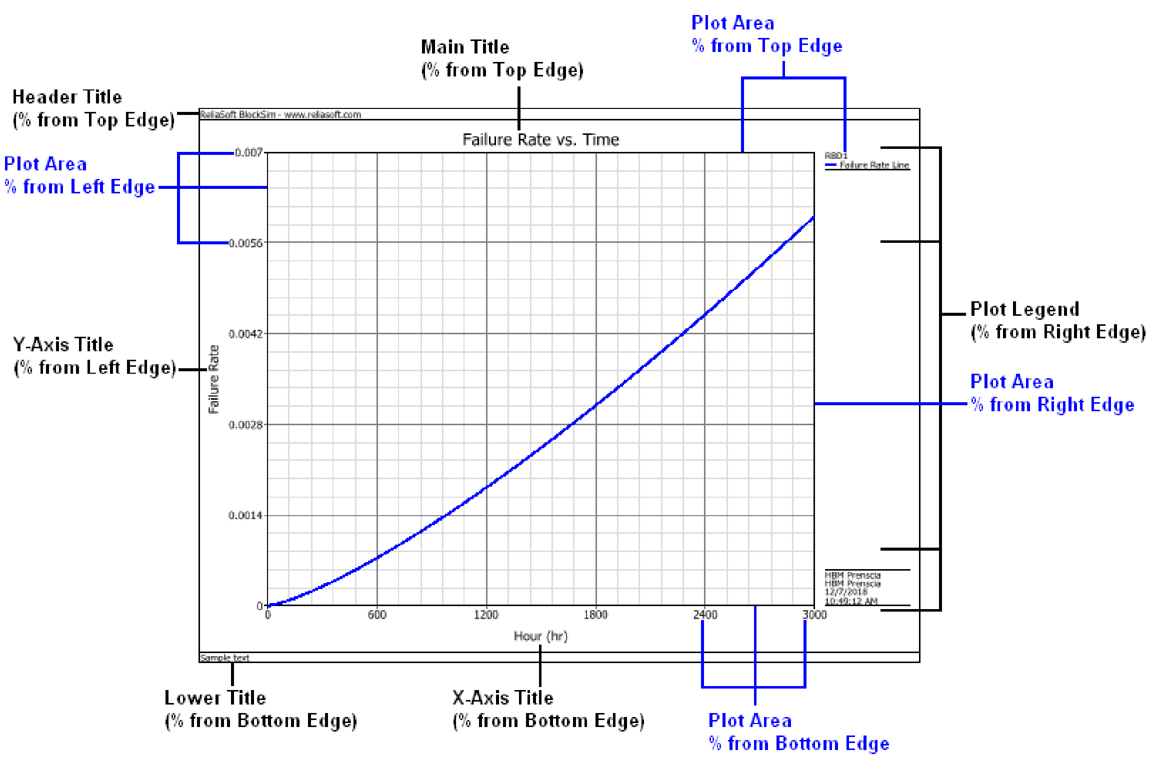
Certain point shapes are drawn using only the border color. These include minus, pike, plus and x-cross.

Plot Setup: 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 BlockSim, but the settings are the same for all ReliaSoft desktop applications.)



Plot Setup: Zoom Page

The Zoom page allows you to customize the options used when zooming in on a plot by using the mouse wheel. Zooming works only for plots with scales. For more information, see Basic Plot Features.

Allow Zoom if selected, allows the plot zoom feature to be used.

- **Factor** use this option to set the amount of the magnification each time you use the mouse wheel on the plot. The lower the number, the greater the increase in the magnification.
- Use the **Border Color, Thickness and Style** option to set the appearance of the zoom box. 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.
- **Fill Style** allows you to select a style of hatching to be used on the zoom box.
- **Fill Color** allows you to specify a foreground color to be used on the zoom box. This color is not used if the selected fill style does not use a background color (i.e., solid or transparent fill style).
- **Background color** allows you to specify a background color to be used on the zoom box. This color is not used if the selected fill style does not use a background color (i.e., solid or transparent fill style).

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 a Failure Rate vs. Time plot while changing the titles for a System Failures plot.
- 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.
- 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.

Plot Defaults Window: Titles Text 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.

Plot Defaults Window: Plot Items Display Page

Plot Item Type allows you to select the style of plot that you want to specify default settings for. The settings you specify are retained in the background when you choose a new item in this list, so you can specify settings for each type and save all of them, if desired.

The options on this page are identical to those on the Plot Items page of the Plot Setup window for the corresponding plot style.

ReliaSoft Draw

ReliaSoft Draw (RS Draw) is a metafile graphics editor that allows you to annotate and customize your plots. 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 one of the following formats: *.rdc, *.jpg, *.gif, *.png or *.wmf.

RS Draw is available by clicking the icon on the plot control panel.

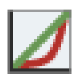
See ReliaSoft Draw Help for information on the features available in ReliaSoft Draw. If you do not currently have Internet access, this link will not work, but you can browse to the local copy of the help file by opening the Help\RSDraw\index.htm file in your ReliaSoft application directory.

Overlay Plots

Available in Weibull++ and BlockSim, overlay plots give you the ability to display results from multiple analyses 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.

Creating an Overlay Plot

Before using an overlay plot, you must first analyze the data sets (or simulate the diagrams) you wish to include in the plot. Then to add an overlay plot to the project, choose **Home** > **Insert** > **Overlay Plot**.

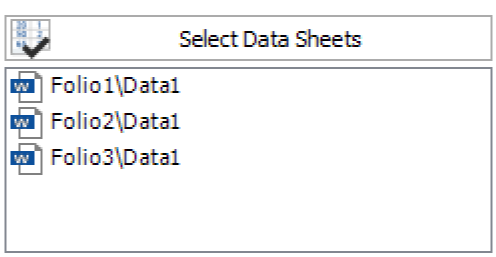


In the window that appears, select the data sheets/diagrams you want to include in the plot (up to a maximum of 20) and click **OK** to create the plot sheet. The plot sheet will be saved automatically under the **Multiplots** heading in the current project explorer.

Tip: In Weibull++, you can add additional plot sheets to a folio by choosing [**Life Data/Life-Stress Data/Growth Data**] **> Folio Sheets > Insert Additional Plot**. The additional sheets can function as overlay plots to display results from multiple data sheets in the current folio on a single plot.

Adding or Removing Analyses

To add or remove analyses from an overlay plot, click the button on the control panel, as shown in the example below for data sheets.



If you later delete an analysis that is referenced by the plot, the plot will remain available and will continue to show the results of the analysis until you redraw the plot (by clicking the **Redraw Plot** icon on the control panel).

Note that importing or exporting an overlay plot will automatically import/export the associated source analyses. (See Importing/Exporting Project Items or Resources.)

Available Plot Types for Overlay Plots

The available plots depend on the analysis or type of data you are working with.

For Weibull++

Probability plots, contour plots and life-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.

In DOE design folios, plots that are intended to be viewed as singular plots will not be available for inclusion in overlay plots. These include the comparison chart, Pareto chart, interaction matrix, term effect plot, cube plot, residual histogram, residual autocorrelation plot, and Box-Cox transformation plot. In addition, only plots that are common to the type(s) of design you have selected will be available in the overlay plot. If a plot type is unavailable for any of the included design types, it will be unavailable in the overlay plot.

For growth data, only plots that are common to the analyses you have selected will be available in the overlay plot. If a plot type is unavailable for any of the included analyses, it will be unavailable in the overlay plot. It is possible for the selected analyses to not have any common plot types.

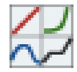
For BlockSim

You can compare data from analytical and simulation diagrams in the same plot, but keep in mind that analytical diagrams always show the reliability over time (does not account for repairs) while simulation diagrams always show the point reliability over time (may include repairs).

Side-By-Side Plots

Available in Weibull++, side-by-side plots give you the ability to display different plots for a single data set all in a single window for easy comparison.

To add a side-by-side plot to a project, choose **Home > Insert > Side-by-Side Plot**.



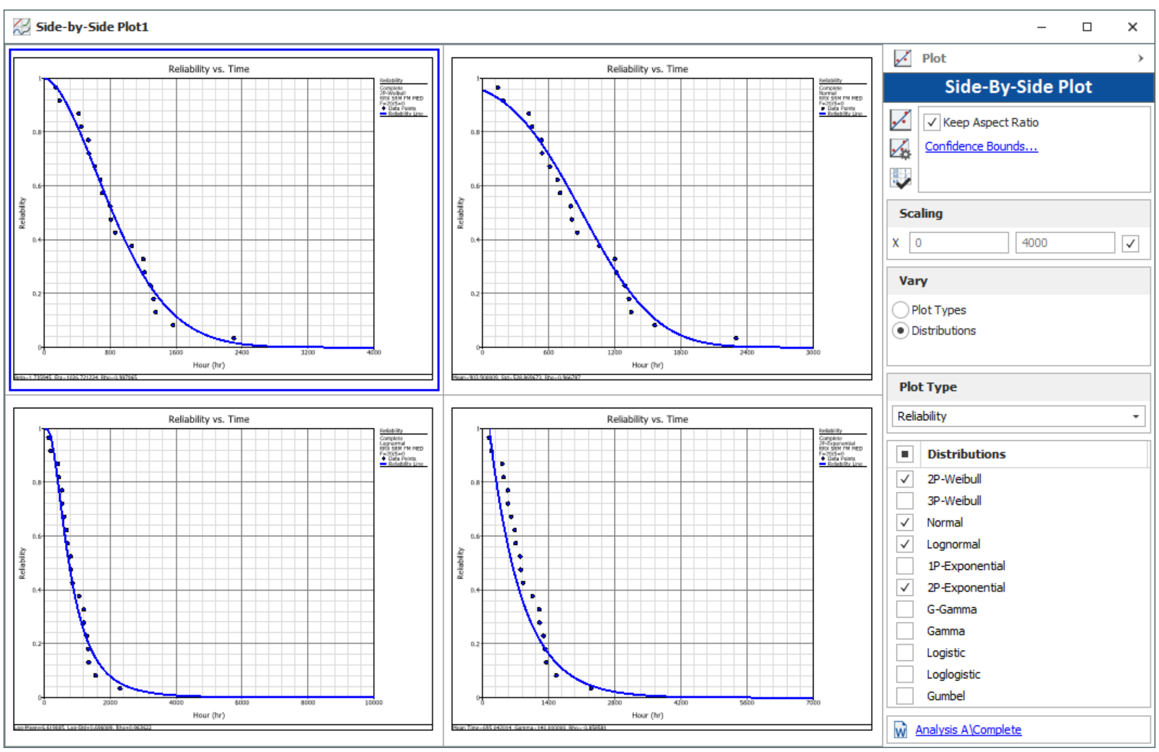
In the window that appears, select the analysis/data sheet you want to plot and click **OK** to create the plot sheet.

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.

Choosing Plots to Display

The control panel provides options for selecting the type of plots to display. For most analyses, the **Vary** area will be available, which contains two options:

- The vary **Plot Types** option allows you to create different plots for the same data set. For example, you might wish to display both the Reliability vs. Time plot and the Failure Rate vs. Time plot of the data set.
- The vary **Distributions** or **Models** option allows you to compare how different distributions or models fit a particular data set. For example, you might wish to display a reliability plot for different life distributions, as shown next.



3D Plots

Available in Weibull++ and for two-way sensitivity analysis in event analysis flowcharts in BlockSim, 3D plots give you the ability to graph functions with three variables, such as the reliability at a given time and stress level. In addition, Weibull++ offers the option to create 3D overlay plots, which are similar to regular overlay plots but displayed in 3D space.

Creating 3D Plots

Before creating a plot, you must first analyze the data sets (or in event analysis flowcharts, you must first run the simulation for the two-way sensitivity analysis).

To add 3D plots to a project, right-click the **Multiplots** folder in the current project explorer and choose the type of plot:

 3D Plot

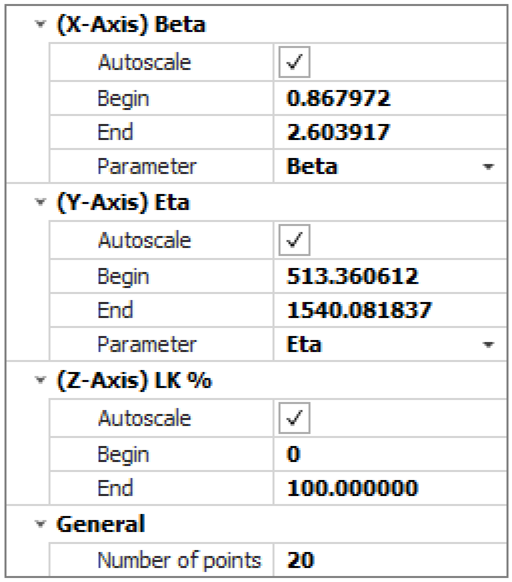
 3D Overlay Plot (Weibull++ only)

In both cases, the plots are automatically saved and stored in the **Multiplots** heading in the current project explorer.

Defining the X, Y and Z Axes

The control panel in the 3D plot folio contains basic settings for defining the X, Y and Z axes. (For advanced customization options, see 3D Plot Setup.)

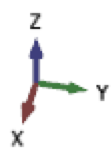
The **Begin** and **End** fields, as shown next, display the minimum and maximum values for the axes. You can click inside these fields to manually edit the values, or select the **Autoscale** check box to have the application automatically choose the appropriate values for the range.



Whenever applicable, the **Parameter** fields will be displayed, as shown in the example above. Click these fields to change which parameter lies on which axis. The name of the parameter associated with the axis will be displayed next to that axis's label.

Rotating 3D Plots

The XYZ indicator shown at the lower-left of a 3D plot represents the plot's horizontal rotation (*azimuth*) and vertical rotation (*elevation*) in 3D space. Any changes to a plot's rotation are reflected in the XYZ indicator, helping you to visualize the plot's orientation on the screen.



There are several ways to rotate or manipulate a 3D plot, using the mouse, keyboard shortcuts or the view cube.

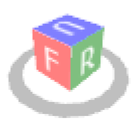
Using a Mouse and Keyboard Shortcuts

If you want to	Use a mouse	Use keyboard shortcuts
Rotate the plot or change the viewing angle	Click anywhere in the plot area, and then press and hold the left mouse button while moving the pointer.	Press the Up, Down, Left or Right arrow keys.

Move the plot without changing its rotation	Click anywhere in the plot area, and then press and hold the right mouse button while moving the pointer.	Hold the SHIFT key while pressing the Up, Down, Left, or Right arrow keys.
Zoom in or out	Rotate the mouse wheel. Alternatively, you can choose **3D Plot > Display > Zoom In** or **Zoom out**.  or 	Press the W or S keys.
Automatically center the plot on canvas	Choose **3D Plot > Display > Center on Canvas**. 	N/A

Using the View Cube

The View Cube helps you to easily orient a 3D plot using predefined viewpoints. The cube is located at the lower-right of the 3D plot.



The letters on the cube represent a particular viewpoint. For instance, clicking the "F" side of the cube orients the plot such that you are looking directly at the plot's front side (e.g., Y and Z plane).

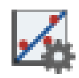
- **F**ront
- **B**ack
- **L**eft
- **R**ight
- **U**p
- **D**own

You can rotate the cube in the same way you would rotate any 3D plot (i.e., press and hold the left mouse button while moving the pointer anywhere on the screen; or press the Up, Down, Left or Right arrow keys on your keyboard).

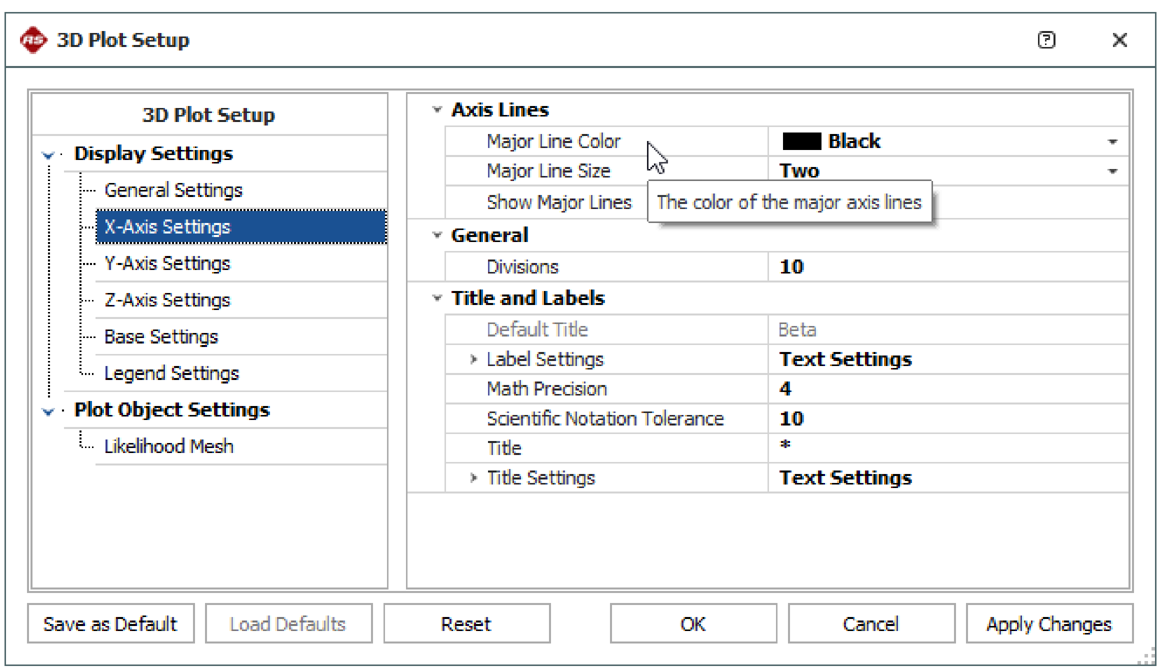
3D Plot Setup

The 3D Plot Setup allows you to customize the look of 3D plots to meet your needs. It gives you full control over the settings used for individual 3D plots and the default settings for all new 3D plots.

To access the 3D Plot Setup, choose **3D Plot** > **Actions** > **Plot Setup**.



Alternatively, you can double-click the item on the plot you wish to edit. This will automatically open the 3D Plot Setup and display the options associated with that item. The available options in the 3D Plot Setup will vary, depending on the plot type. You can move the mouse pointer over an option to display its definition.



To apply your changes to the current plot only, click **OK**.

To save your changes and use them as the default setting for all new 3D plots, click the **Save as Default** button. This will overwrite the previous default settings.

To restore the previously saved defaults, click the **Load Defaults** button. This is an easy way to undo changes you have made in the 3D Plot Setup and re-apply default settings.

To reset all settings to the default values that were shipped with the application, click the **Reset** button.

ReliaSoft Workbooks

IN THIS CHAPTER

The ReliaSoft Workbook is a custom reporting tool that is built directly in to many ReliaSoft applications, including Weibull++ and BlockSim. It combines two reporting modules – Spreadsheets and Word Processing – into the same flexible interface, thereby replacing the Analysis Workbook and Word Report Template functionality from prior versions.

Tip: If you convert a project from Version 10 or earlier, any Analysis Workbooks and Word Report Templates will be automatically converted to ReliaSoft Workbooks.

If your organization has implemented an SEP web portal for an enterprise database, you can choose to publish selected ReliaSoft Workbook reports to be accessed from any web-enabled device. See Publishing to SEP.

ReliaSoft Workbook Wizard

To add a new ReliaSoft Workbook in an existing project, choose **Home > Insert > ReliaSoft Workbook**.

This launches the wizard, which gives you the opportunity to assign the first associated data source and/or select a saved template, if desired. (Note that the examples shown here are from Weibull++ but similar functionality is available in BlockSim.)

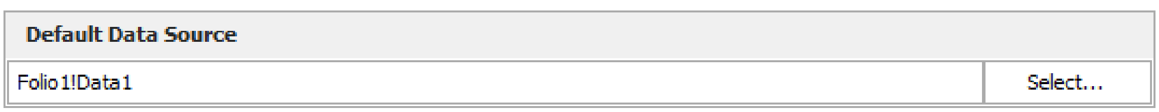
If you prefer to start with a blank report:

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:

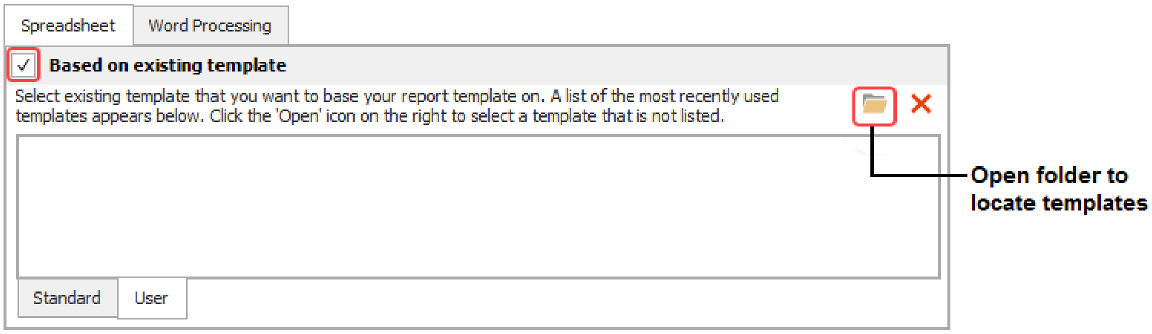
Click **Select** and choose one of the available analyses (i.e., a data sheet in Weibull++ or a diagram in BlockSim). When you return to the wizard, click **OK** to create the report.

Note that you will be able to associate more data sources after the report is created, and change any of those assignments at any time. (See Associated Data Sources.)



If you want to use a saved template:

Click **Spreadsheet** or **Word Processing**. 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. (See ReliaSoft Workbook Templates.)



Associated Data Sources

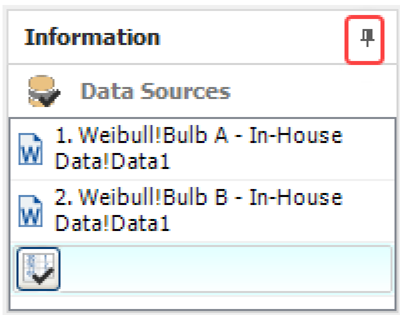
In ReliaSoft Workbooks in Weibull++ and BlockSim, you have the option to associate multiple data sources that can be used for any function that obtains data or results from an existing analysis. (To obtain results from a DOE analysis in Weibull++, see DOE Analysis Report.)

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.

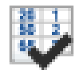
Although you will be prompted to associate the first data source when you create the report, you can add or change associated data sources for an existing report at any time.

Associating Data Sources

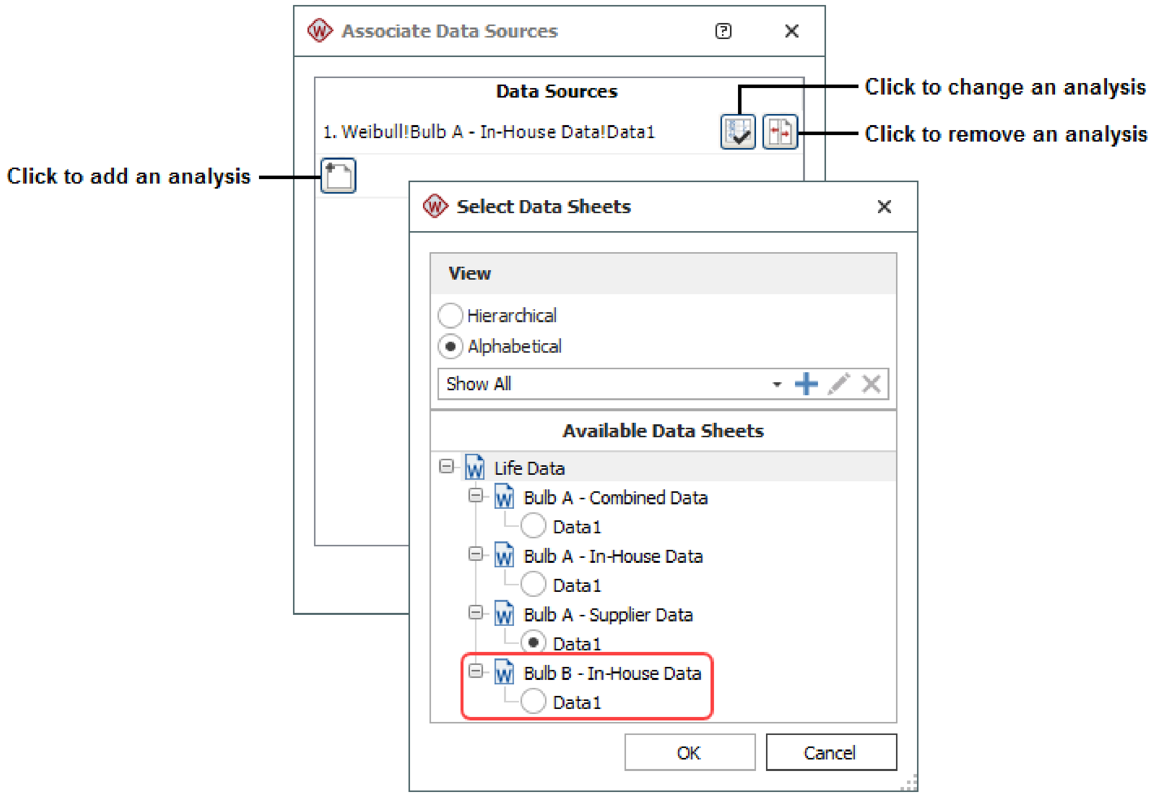
For ReliaSoft Workbooks, the Information panel (located on the right side of the window) shows which analyses are currently assigned as default data sources. To have the panel always display, click the pushpin icon into the vertical position.



To add or remove analyses, choose **Home > Report > Associate Data Sources** or click the icon in the Information panel.



The following picture shows the windows for Weibull++. Similar functionality is available in BlockSim.



Note that:

- In **Weibull++** you can select any data sheet from a folio that contains a life data analysis or a growth data analysis:
  - Life data folio
  - Life-stress data folio
  - Non-parametric LDA folio
  - Warranty analysis folio
  - Degradation analysis folios (the results shown in the report will be based on the analysis of

the extrapolated failure/suspension times)

- Growth data 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 terms of the System Base Unit (SBU) for the database. However, in Weibull++, 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:

- Convert the data in the original data source. For example, if one data set is in "hours" and another is in "days," you could use the Change Units feature to automatically convert the data in the first analysis from "hours" to "days."

  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.

- Manually adjust the functions so they return results in the same units. For example, if the data set is in "days" and you want the results to display the B10 life in "hours," you could adjust the function by adding a conversion ratio. For example, if 1 day is equivalent to 24 hours, you could edit the function as follows: `=(TIMEATPF(Weibull!Week!Data1,0.1))*(24).`

ReliaSoft Workbook Templates

In ReliaSoft Workbooks, saved templates make it easy to reuse the same report multiple times — in different analysis projects and with different data sets.

Note: For DOE analyses, instead of saved templates for the entire workbook, you can use saved profiles in the DOE Report Generator.

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. Note that the templates are module-specific (i.e., compatible with either the spreadsheet module or the word processing module).

Saving Your Own Templates

It is easy to create a template from any ReliaSoft Workbook that is currently open.

Select either the spreadsheet or the word processing module, then choose **Document > Document > Save 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.

Using a Template

You can select a template when you create a new ReliaSoft Workbook. (See ReliaSoft Workbook Wizard.)

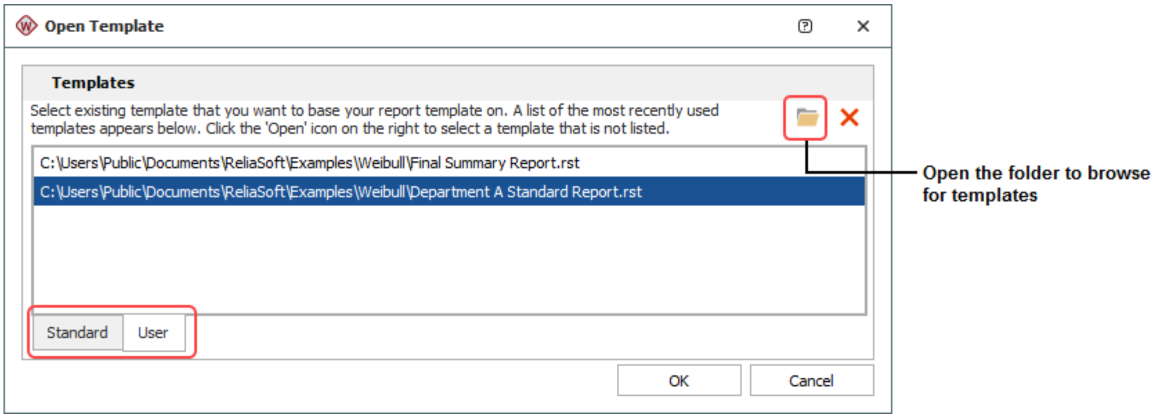
Alternatively, you can use the steps below to apply or change the template for an existing workbook at any time.

IMPORTANT: This will replace any existing content in the current module, and the change can't be undone.

1. Select either the spreadsheet or word processing module and then choose **Document > Document > Open Template**.

2. Select a template from either tab then click **OK** to apply the template to the new or existing report.
   - The **Standard** tab displays the templates that are installed with the software and stored in the applicable sub-folder under C:\Users\Public\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.



Spreadsheet Module

The spreadsheet module in ReliaSoft Workbooks provides functionality similar to Microsoft Excel (with built-in functions and complete in-cell formula support), and it can be used to integrate data and/or results from multiple analyses at the same time.

Send to Excel

To export all sheets in the spreadsheet module to a Microsoft Excel file, choose **Home > Report > Send to Excel**.

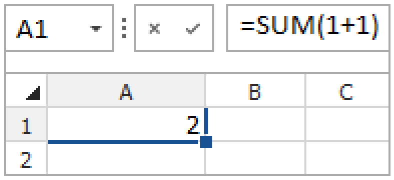
Inserting Functions

- To build and insert functions that utilize a referenced analysis (data source), see Function Wizard - Data Sources.
- To add math, date, logic and other functions, see Function Wizard - Formulas.

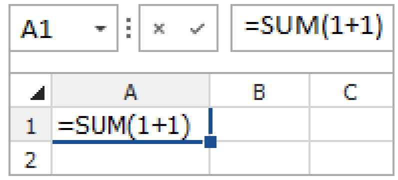
Showing Formulas

To display the formulas instead of the calculated results in the worksheet cells, choose **Formulas > Formula Auditing > Show Formulas**.

You can toggle the formula display on and off.



Result displayed in A1

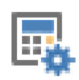


Formula displayed in A1

Recalculate Formulas

By default, the spreadsheet automatically recalculates all formulas whenever you open the workbook or when the cells that a formula depends on have changed. However, if your spreadsheet contains a large number of formulas, the recalculation process may take more time and every change may require you to wait several seconds or minutes for the application to recalculate all values. In this case, you can control when calculation occurs by changing the setting to manual calculations.

To use manual calculations, choose **Formulas > Calculation > Calculation Options**, and then choose the **Manual** option. To change the settings back to automatic calculations, choose the **Automatic** option.



To recalculate all formulas when in Manual mode, choose **Formulas > Calculate > Calculate Now**.

The **Calculate Now** command can also be used to refresh the spreadsheet. A spreadsheet needs to be refreshed whenever its referenced data source(s) has been recalculated. This ensures that the spreadsheet reflects the most current results.

Adding Custom Charts

To create your own custom chart, first select the cells that contain the relevant data, then go to **Document > Charts** and choose a chart type and style. (See Custom Charts in ReliaSoft Workbooks.)

Defining Names

To create variable names that reference specific spreadsheet cells, see Defined Names.

Word Processing Module

The word processing module in ReliaSoft Workbooks offers 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.

This module has two tabs located at the bottom of the window: Design and Review. Use the Design tab to add text, functions and plot holders. Use the Review tab to see the results before you generate the final report.

To export the report to Microsoft Word, choose **Send to Word**.

Using the Function Wizard

To use the Function Wizard to build and insert functions that utilize a referenced analysis (data source), choose **Home** > **Report** > **Function Wizard**. (See Function Wizard - Data Sources.)

Using the Plot Wizard

To use the Plot Wizard to generate a variety of plots based on a referenced analysis, choose **Home > Report** > **Plot Wizard**. These are the same types of plots that are generated in an analysis folio or diagram plot sheets. (See Plot Wizard.)

Using Spreadsheet References

To reference cells from the spreadsheet module (within the same workbook), choose **Home** > **Report** > **Spreadsheet Reference** and select the desired cells. The link between the word processing and spreadsheet modules is dynamic; when data in the spreadsheet is changed, the word processing module is automatically updated. (See Spreadsheet References.)

Formulas and Functions

Function Wizard - Data Sources

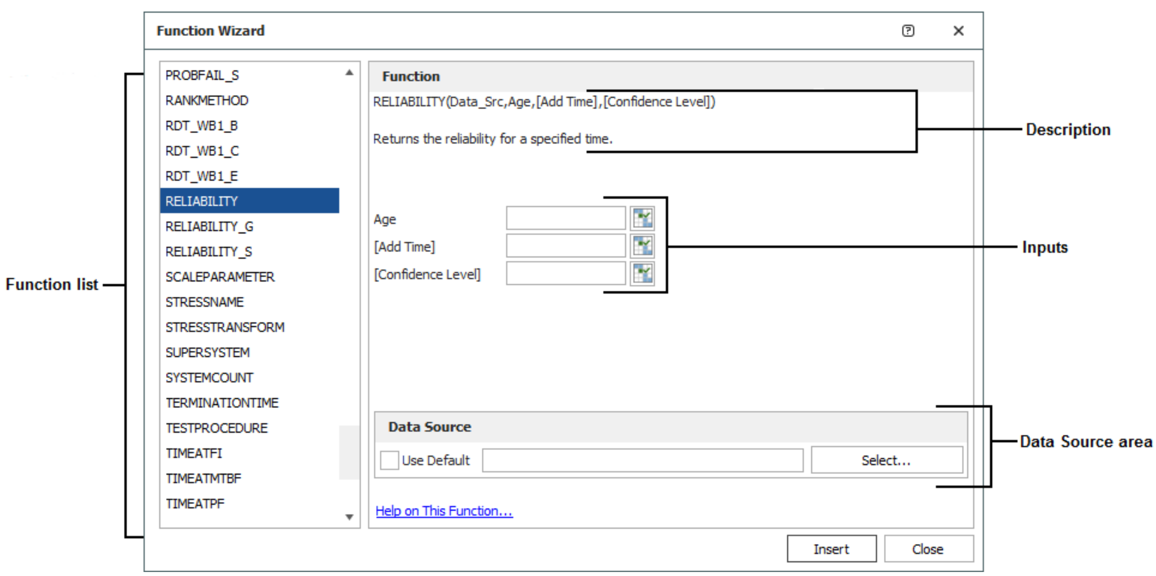
In both the spreadsheet module and word processing module for ReliaSoft Workbooks in Weibull++ and BlockSim, you can build functions that return results based on an analyzed folio or diagram.

Note: To insert math, date, logic and other functions into the spreadsheet module of a ReliaSoft Workbook, see Function Wizard - Formulas.

Using the Function Wizard

To open the Function Wizard in ReliaSoft Workbooks, choose **Home > Report > Function Wizard**.

Select a function from the navigation panel and enter any required inputs. The following picture shows the most complex configuration as an example. After entering the inputs, click **Insert** to place the function into the report at the current cursor location. You can move and/or modify the function expression after it has been inserted.



For spreadsheet functions:

- The bracketed parameters indicate that the input is optional. In the example above, the `Add Time` and `Confidence Level` parameters are optional.
- You can use cell references as inputs. 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.)
- You can use variable names as inputs. (See Defined Names.)

- You can 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 the word processing module:

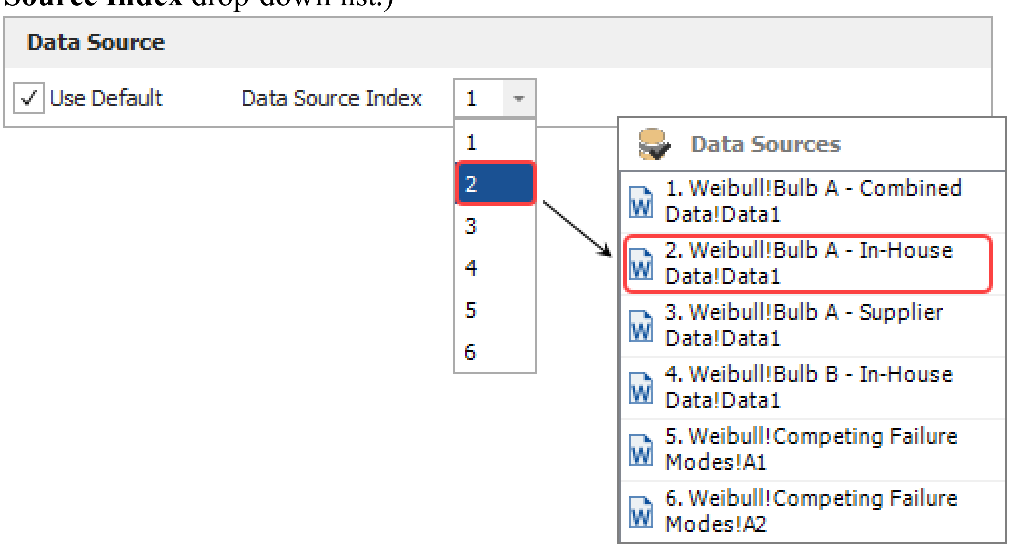
- The brackets are part of the function field and are not optional.
- The functions will not return any results until you either switch to the **Review** tab or generate the report in Microsoft Word.

Selecting a Data Source (if Applicable)

There are two ways to specify the data source:

- **Use the Data Source Name** (spreadsheets module only): This approach will return results based on a specific data sheet or diagram.In the Function Wizard, clear the **Use Default** check box and click the **Select** button to choose the data source, as shown next. In this example, the function will return results based on the data sheet called "Data1" in the Weibull++ life data folio called "Folio1."

- **Use a Data Source Index** (spreadsheet and word processing modules): This approach allows you to use the report as a template and return results based on a given data set.First, you must add data sources to the ReliaSoft Workbook (see Associated Data Sources). Then, in the Function Wizard, click the **Data Source Index** drop-down list and choose the index number of the desired data source. (In spreadsheets, select the **Use Default** check box to access the **Data Source Index** drop-down list.)



Function Wizard - Formulas

In the spreadsheet module for ReliaSoft Workbooks in Weibull++ and BlockSim, you can insert functions that perform math, date, logic and other operations.

Using the Function Library

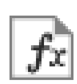
The quickest way to add a function is to select it from the appropriate function category drop-down list, then select the cells of interest (hold down the **CTRL** key to select nonconsecutive cells) and press **ENTER**. You can move and/or modify the function expression after it has been inserted.

For example, to calculate the average reliability for the following units, you would choose **Formulas > Function Library > AutoSum Average**. Select cells B3 through B9 and press **ENTER**. Note that the cell references can be relative (B3:B9) or absolute ($B$3:$B$9). (See Referencing a Cell.)

	A	B
1		
2	**Unit**	**Reliability at 700 Hours**
3	1	0.86359
4	2	0.81894
5	3	0.83176
6	4	0.85132
7	5	0.79121
8	6	0.83992
9	7	0.80279
10	Average:	=AVERAGE(B3:B9)

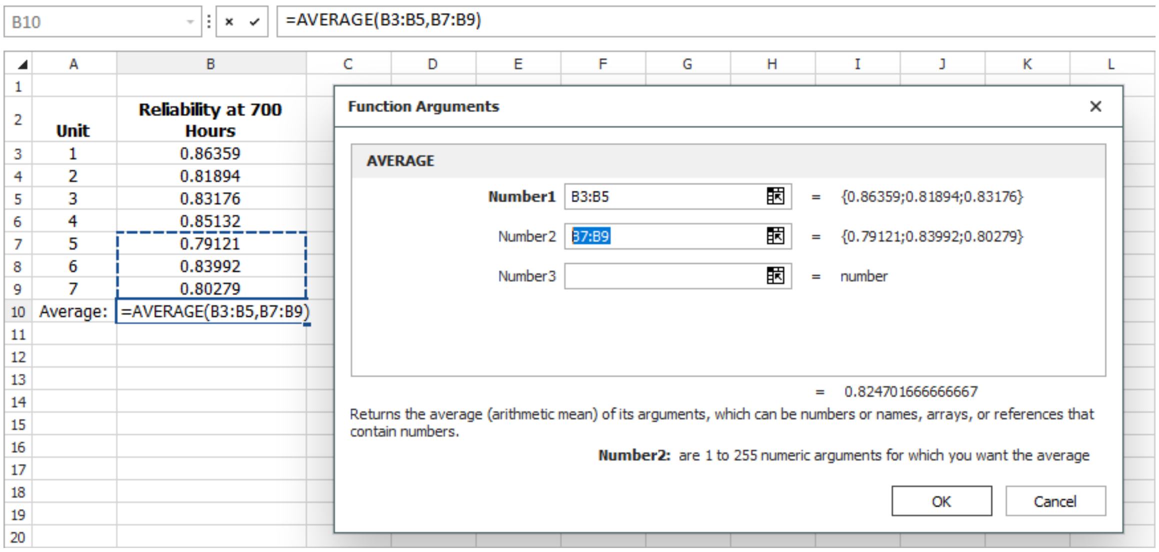
Using the Function Wizard

To use the Function Wizard, choose **Formulas > Function Library > Insert Function**.



Select a function from the drop-down list and click **OK**. Then in the Function Argument window, enter the input by either selecting the cells in the sheet or typing them directly into the appropriate field. For information relating to entering text as an input, cell references, and working with date and time functions, see Data Entry Tips for Functions.

For example, to exclude the value from unit 4 (failure due to operator error) from the average, you would select cells B3 through B5 for **Number 1**, and B7 through B9 for **Number 2**.



Data Entry Tips for Functions

This topic provides some data entry tips for using spreadsheet functions in ReliaSoft Workbooks and general spreadsheets. 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 design folios, 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). For that type of analysis, you can use the DOE Analysis Reports feature to obtain results from a design, multiple linear regression or one-way ANOVA folio, or any of the measurement systems analysis folios.

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(Weibull!Folio1!Data1)` – and situations where you need enter a time or date value in one of the accepted text formats – `DAY("22-Aug-2020")`.

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("Weibull!Folio1!Data1";A4)`).

Referencing a Cell in the Same Sheet

If you want to use another cell in the same sheet, 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).

- A **relative reference** points to a cell based on its relative position to the current cell (e.g., B2). 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. Absolute references are designated by placing a dollar sign ($) in front of the row and/or column that is to be absolute. For instance, $B$2 is an absolute reference that points to the cell located in Column B, Row 2 regardless of the position of the cell containing the reference.

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. (See Defined Names.)

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!$B$2` is a reference to the cell located in Column B, Row 2 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 workbook.

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++ 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++ life data 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.

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 ("8/22/2019"). For example, `=DAY("8/22/2019")` returns 22.
- Day-Month-Year ("22-Aug-2019"). For example, `=MONTH("22-Aug-2019")` 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(43466)` returns 2019.

- 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.

D2 =DATE(A2,B2,C2)

	A	B	C	D	E
1	**Year**	**Month**	**Day**	**Serial #**	
2	2019	8	22	43699	
3					

  - The `DATEVALUE` function requires you to enter the date in an accepted text format. For example, `=DATEVALUE(8/22/2019)` returns 43699.

Finally, you can also use the results of other functions within a date function. For example:

- To return the month from today’s current date, use: `=MONTH(TODAY())`
- To return the day of the week for a date that is specified in three separate cells (A2=Year, B2=Month and C2=Day), use: `=WEEKDAY(DATE(A2,B2,C2))`

D2 =WEEKDAY(DATE(A2,B2,C2))

	A	B	C	D	E
1	**Year**	**Month**	**Day**	**Weekday**	
2	2019	8	22	5	
3					
4	(Thursday is the 5th day of the week.)				
5					

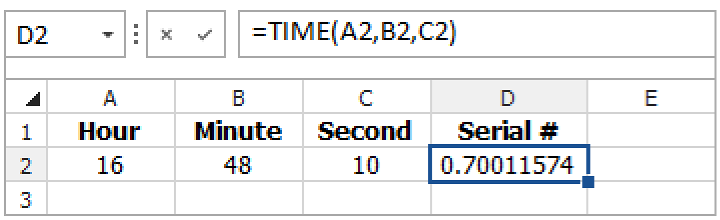
Working with Time Functions

When using one of the spreadsheet time functions (`HOUR`, `MINUTE` and `SECOND`) to enter a time, you can use one of the following valid text formats:

- Hour:Minute[:Second] [AM/PM]. For example, `=HOUR("4:48:10 PM")` returns 16 (the hour using the 24 hour system).
- Month/Day/Year Hour:Minute[:Second] [AM/PM]. For example, `=MINUTE("8/22/2019 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.



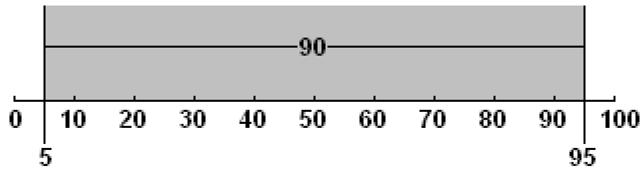
  - The `TIMEVALUE` function requires you to enter the time as text in one of the accepted text formats. For example, `=TIMEVALUE(4:48:10 PM)` returns 0.70011574.

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.

Returning Confidence Bounds

For functions that return confidence bounds, note that the Function Wizard for data sources 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 a spreadsheet, 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%

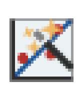
  Note that the above function expressions for the confidence bounds uses two commas (,,) to indicate that another optional input ([Add Time]) was intentionally left blank. For more information about the syntax for functions containing multiple optional inputs, see Data Entry Tips for Functions.

- In a word processing module for ReliaSoft Workbooks, 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.

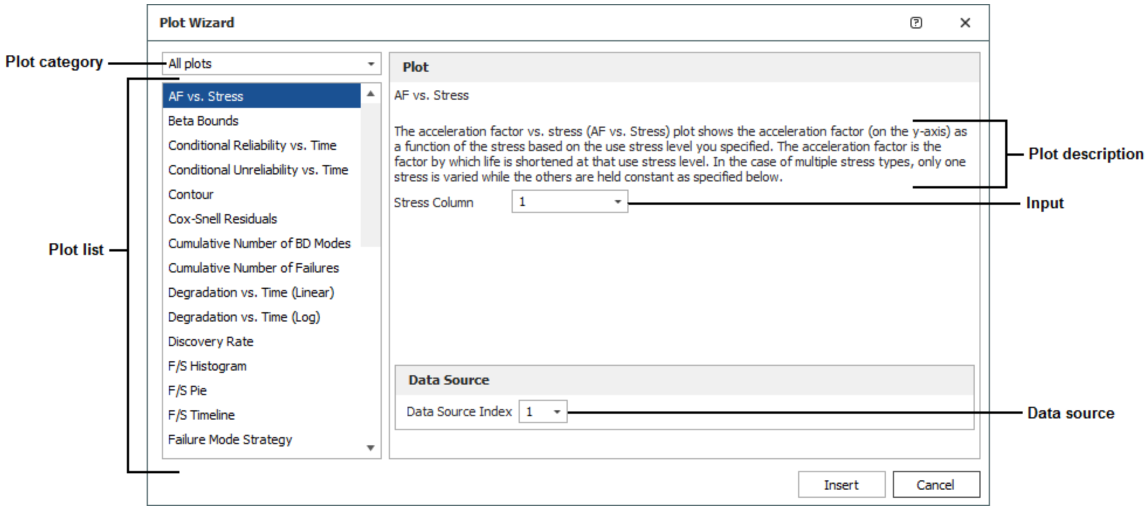
Plot Wizard

In the word processing module for ReliaSoft Workbooks in Weibull++ and BlockSim, you can use the Plot Wizard to generate 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, using the Plot Wizard 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. (For DOE design folios, you can use the DOE Analysis Reports feature to insert plots from a design, multiple linear regression or one-way ANOVA folio, or any of the measurement systems analysis folios.)

To open the Plot Wizard, choose **Home > Report > Plot Wizard**.



The following picture shows the plot wizard in Weibull++. Similar functionality is available for 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

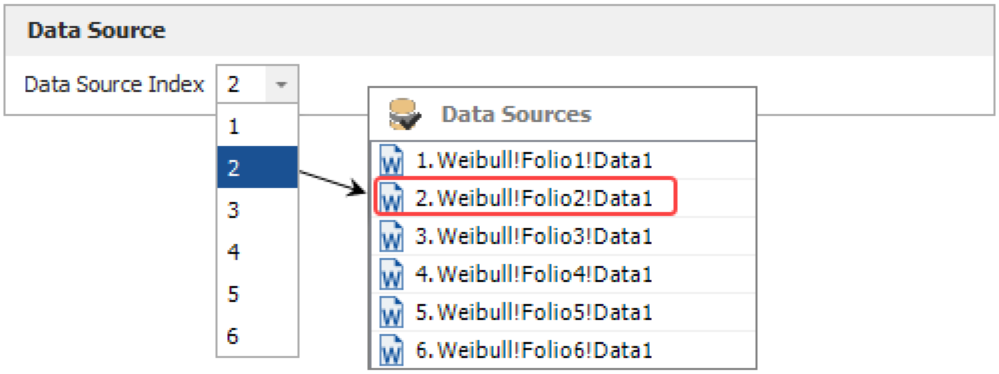
Select the Plot Type

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 life-stress plots that require you to select a specific stress column).

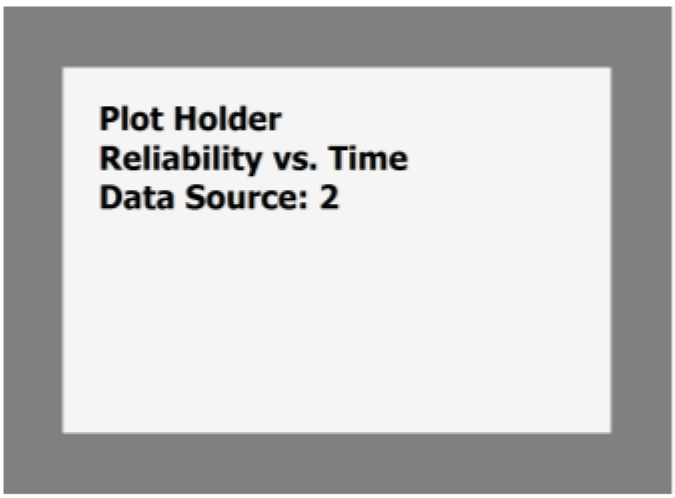
Select the Data Source

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.

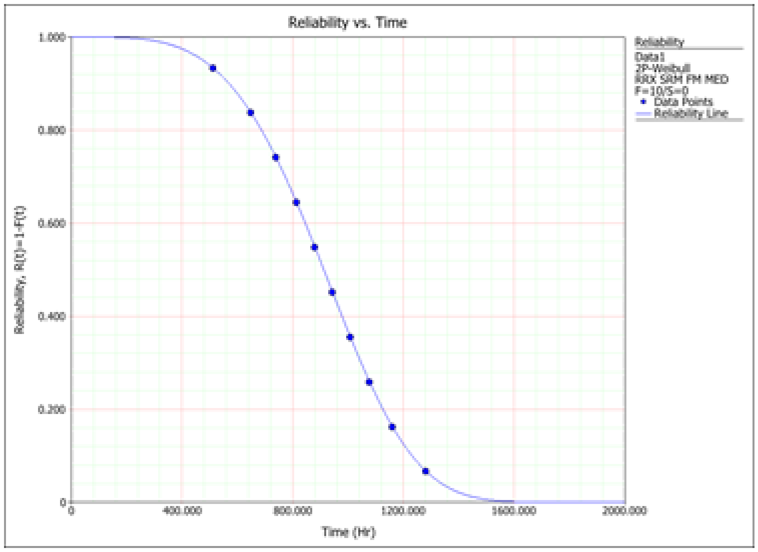


Insert the Plot Holder

When the plot is fully defined, click **Insert** to place it at the current cursor location. You can move and/or resize the plot holder after it has been inserted. The plot holder shown next will return the reliability vs. time plot for whatever data source is currently second in the list of associated data sources.



Plot holders will not return any results until you view it in the Review tab or generate the report (**Home > Report > Send to Word**).



Custom Charts

In the spreadsheet module for ReliaSoft Workbooks, you can use the **Document > Chart** commands to insert your own custom charts. The charts can be placed anywhere in the spreadsheet, and are dynamic (automatically update when the data used to create the chart is changed).

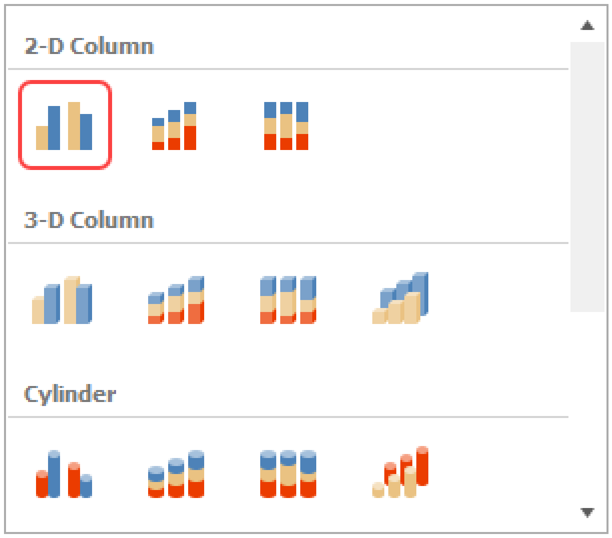
Adding a Chart

1. Select the cells that contain the data of interest. In the example data below, the headings are included in the selection and will be used as the legend labels.

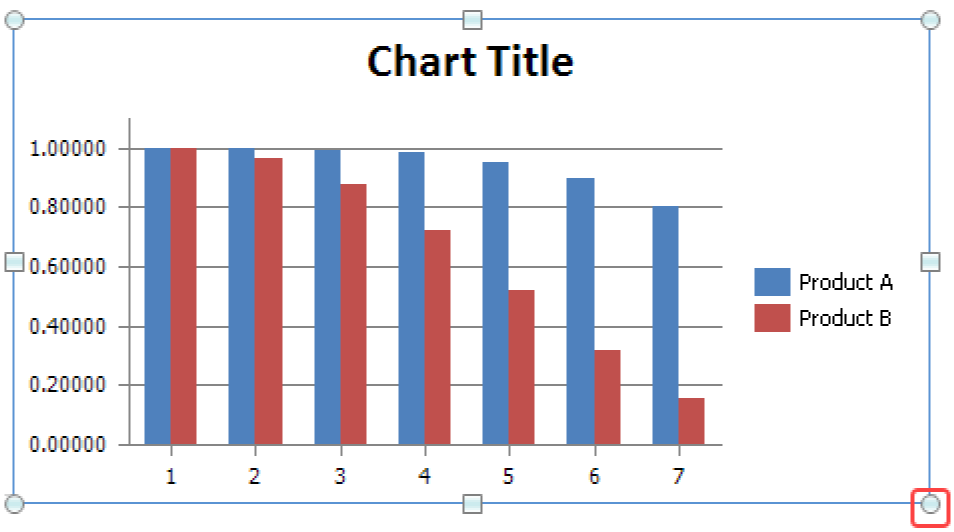
	Reliability	
Time	Product A	Product B
100	0.99996	0.99586
200	0.99920	0.96401
300	0.99506	0.87716
400	0.98217	0.72349
500	0.95220	0.52071
600	0.89495	0.31434
700	0.80119	0.15282

Headings are used as legend labels

2. Choose **Document > Charts**. Choose the chart type, and then click a chart subtype that you want to use.



3. To move the chart, click and drag the chart to the desired location.
4. To resize the chart, click the chart and then drag the sizing handles to the desired size.



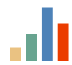
Changing the Data Displayed on an Axis

To change the way the data series is displayed on the axes, click the chart, then choose **Design > Data > Switch Row/Column**.

Changing the Chart Type and/or Data Set

You can change the type of chart and/or the data that is used in a chart at any time. To do this, click the chart and then:

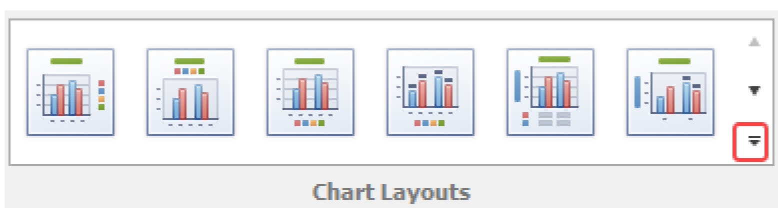
- To change the chart type, choose **Design > Type > Change Chart Type**. Select the desired type.



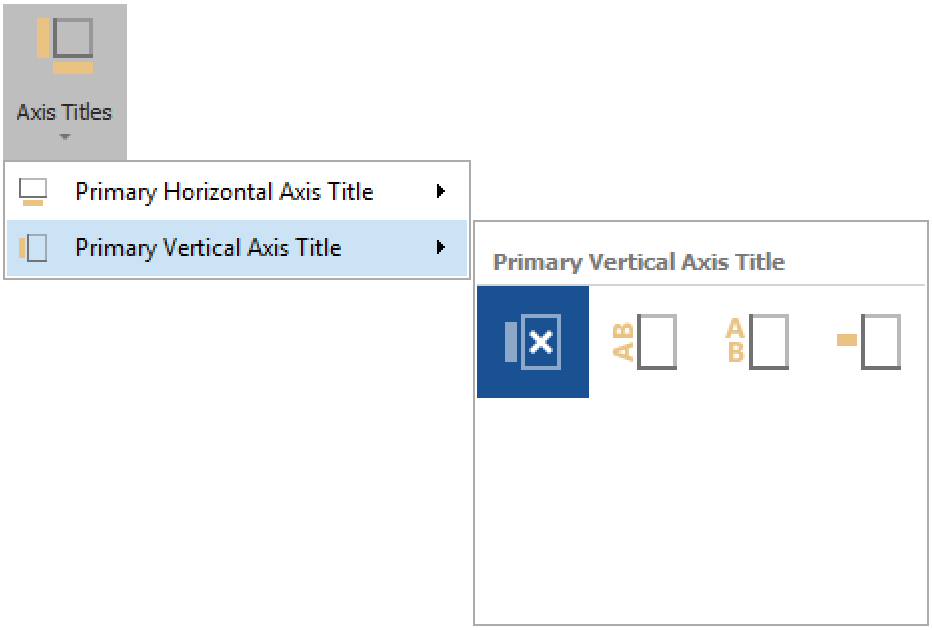
- To change the data set, choose **Design > Data > Select Data**. Select the cells that contain the data of interest.

Applying a Chart Layout and/or Chart Style

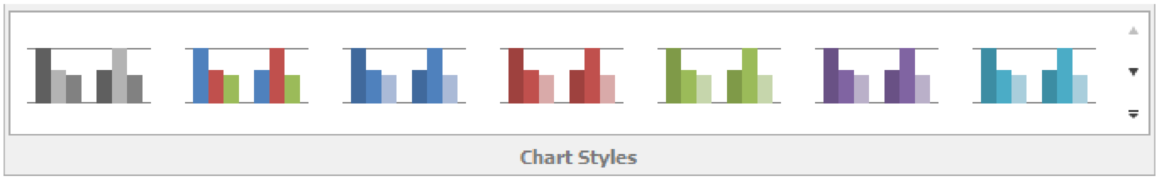
- To apply a predefined chart layout, choose **Design > Chart Layouts**. As an example, the following picture shows some of the options for 2-dimensional bar charts. To see the rest, click the arrow in the bottom-right corner.



If none of the predefined options are suitable for your chart, choose **Layout > Labels**, then select the option you want to customize. The following picture shows the options for the vertical axis.

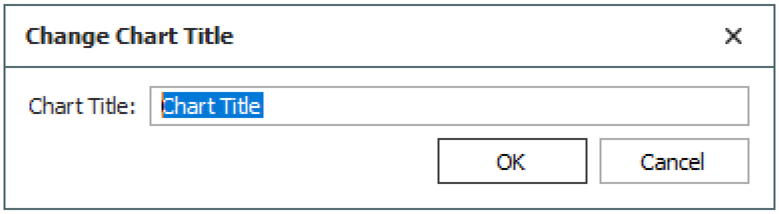


- To apply a predefined chart style, choose **Design > Chart Styles**.



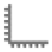
Renaming Chart and Axes Titles

When displaying chart and/or axes titles, you can rename the titles by right-clicking the chart and selecting the appropriate title option from the shortcut menu. For example, choosing **Change Chart Title** opens the following window:



Modifying the Axes and Gridlines

To hide or display axes and/or gridlines, choose **Layout > Axes > Axes** or **Gridlines**.

 or 

- When displaying axes, you can change direction of the x-axis and/or change the scale of the y-axis.
- When displaying gridlines, you can increase or decrease the number of

Defined Names

In the spreadsheet module for ReliaSoft Workbooks and general spreadsheets in Weibull++ folios, you can create variable names that reference specific spreadsheet cells. The names can then be used in any formula or function within the same workbook.

For example, say you want to obtain the reliability of a Weibull++ data set. The operating time is stored in cell B5 of the spreadsheet:

```
=RELIABILITY("Weibull!Bulb!Data1!"$B$5))
```

You can create a defined name for the operating time to make the formula easier to understand and maintain. In this case, the formula might be rewritten as:

```
=RELIABILITY("Weibull!Bulb!Data1!"OpTime)
```

You can then use the defined name `OpTime` in any other formula in the workbook that requires the operating time.

Syntax Rules for Names

- The first character of a name must be a letter or an underscore. The remaining characters can be letters, numbers, periods and underscores. Names are not case sensitive.
- Spaces are not allowed as part of a name. An underscore (_) or period (.) can be used as a word separator.
- Symbols, except for underscores (_), periods (.) and backslashes (\ ), aren't allowed.

Managing Names

To view and manage defined names:

- In ReliaSoft Workbooks, choose **Formulas > Defined Names > Name Manager**.
- In general spreadsheets, choose **Sheet > Format and View > More Settings > Defined Names**.

You can then use the **New**, **Edit**, or **Delete** commands in the window to create or modify the names.

Defining Names

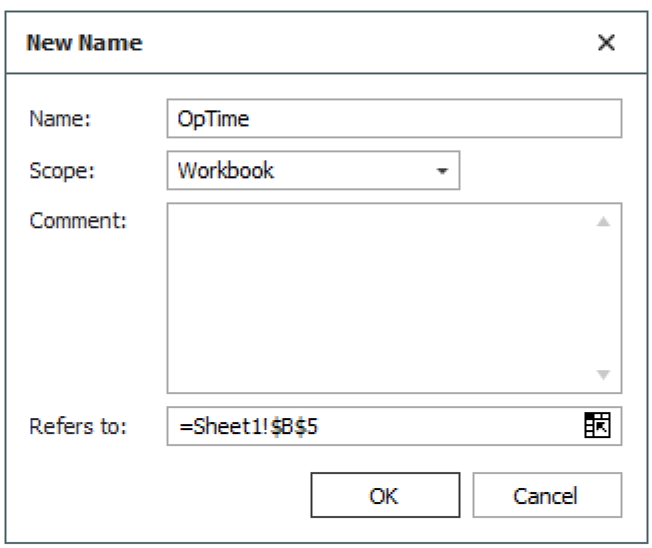
There are several ways to define a name.

Define a Name via the Name Manager

This method applies to both ReliaSoft Workbooks and general spreadsheets.

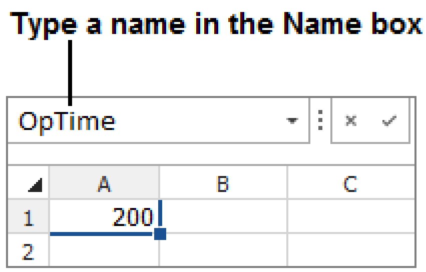
1. Select a cell or range of cells.
2. Create a new name:
   - In ReliaSoft Workbooks choose **Formulas > Defined Names > Define Name** (or choose **Formulas > Defined Names > Name Manager**, then click the **New** button in the window).
   - In general spreadsheets, choose **Sheet > Format and View > More Settings > Defined Names**. Then in the window, click the **New** button.
3. Type a name into the **Name** field. In the **Scope** drop-down list, choose whether this name will be used for the entire workbook or a specific sheet. The **Comment** field is optional.

   The selected cell/range will already be defined in the **Refers to** field, but you can click into the spreadsheet and select different cells, if desired.



Define a Name via the Name Box (ReliaSoft Workbook Only)

This method applies only to the spreadsheet module of ReliaSoft Workbooks. Select the cell or range of cells, click the Name box and type a name. Press **Enter** to create the name.

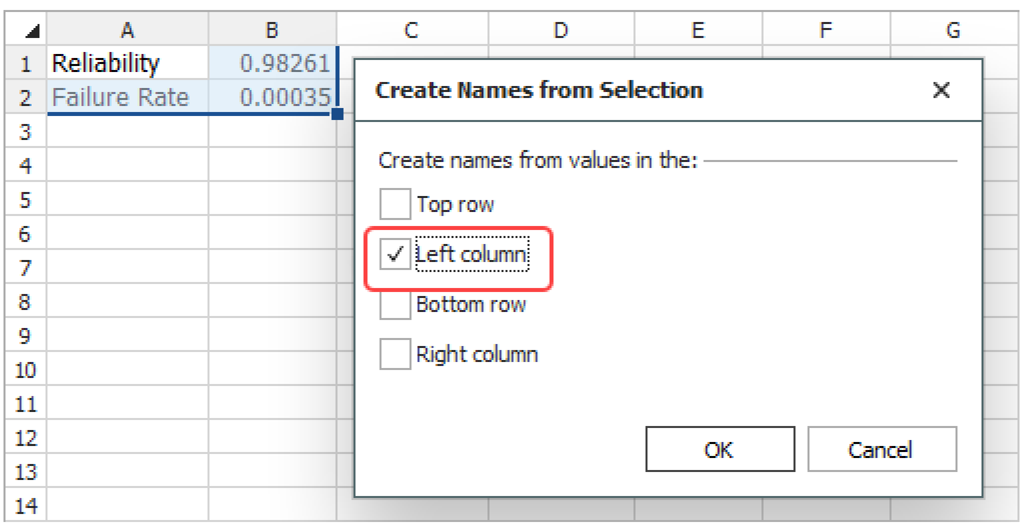


Define a Name via the Selection Command (ReliaSoft Workbook Only)

If the fields are already labeled in the spreadsheet, you can use them to create names.

1. Select the cells of interest (including the row or column labels), then choose **Formulas > Defined Names > Create from Selection**.

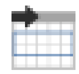
2. In the window, select the location of the labels. Click **OK**.



Spreadsheet References

In ReliaSoft Workbooks, the *spreadsheet reference* feature provides integration between the spreadsheet and word processing modules in the same workbook. You can insert a reference from any cell, or consecutive range of cells, from the spreadsheet into a desired location in the word processing document. If the referenced cells in the spreadsheet are changed, the word processing module is automatically updated. The changes are visible when you use the Review tab to see a preview or generate the Word report.

To insert a spreadsheet reference, place the cursor at the desired location in the word processing document and choose **Home > Report > Spreadsheet Reference**.



Note that you can also use the `Spreadsheet Reference` function in the word processing module’s Function Wizard.)

The Select Cells window shows the contents of the spreadsheet module in the current workbook. Select a cell or range of consecutive cells and click **OK**.

	A	B	C	D	E
1					
2			Reliability		
3		Time	Prooduct A	Product B	
4		100	0.99996	0.99586	
5		250	0.99782	0.92876	
6		400	0.98217	0.72349	
7		550	0.92764	0.41460	
8		700	0.80119	0.15282	
9		850	0.58770	0.03150	
10		1000	0.33340	0.00314	
11					

For the example data shown above, the entry would look like this:

[TBLSSREF(Sheet1!B2:D10)]

DOE Analysis Reports

The DOE Analysis Reports feature allows you to insert selected analysis results and plots into a ReliaSoft Workbook.

This feature is available for any analyzed response in a design, multiple linear regression, one-way ANOVA or measurement systems analysis folio. The report can display any result that's available in the folio's Analysis Summary window for the selected response, as well as any applicable plot.

This topic describes how to build and insert a DOE analysis report, and how to use saved profiles for reports that you need to generate frequently. Note that you can insert multiple reports into the same spreadsheet or word processing document, if desired. Each additional report will be appended to the end.

Inserting a DOE Analysis Report

1. Select either the spreadsheet or word processing module in a ReliaSoft Workbook and choose **Home > Report > DOE Analysis Report**.

2. In the Select Response window, choose an analyzed response to copy the results from and click **OK**.

3. In the DOE Analysis Report window, the **Available Report Items** area (left) shows all of the available results and plots for the selected response. The **Selected Report Items** area (middle) shows the selected results/plots, in the order in which they will appear. You can:

   a. Choose **Profiles > Open Profile** to apply all of the relevant selections from a saved report profile. (See Using DOE Report Profiles.)

   b. Build/modify the list using double-click, drag and drop or the **Add/Remove** and **Up/Down** buttons.

   Note that you can include multiple instances of the same report item, if desired.

4. When you click each selected report item, the **Item Properties** area (right) allows you to view/change the name (heading) that will be used in the report, if desired.

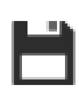
   For plots, you can also click the **Preview Plot** button to see what the plot will look like in the report. Any plot settings you change in the plot preview will be used in the generated report but will not affect the folio the report is based on.

5. Choose **Actions > Generate Report**.

Using DOE Report Profiles

When the DOE Analysis Report window is open, you can save the current settings (i.e., report item selections), or load settings that were previously saved.

- To save all your current report settings for use in future reports, choose **Profiles > Save Profile**.



Then specify the name and location of the DOE report file (*.drt) that will store the settings.

- To overwrite all the settings in the DOE Analysis Report window and use the settings from a DOE report file instead, choose **Profiles > Open Profile**.

Then select the *.drt file that has all the desired settings.

Note: When you load report items from a DOE report file, only the items that apply to the selected folio/response will be included in the report.

Dashboards

IN THIS CHAPTER

The dashboard is a flexible tool for graphical presentation of data. This can include bar and pie charts, gauges, maps, etc.

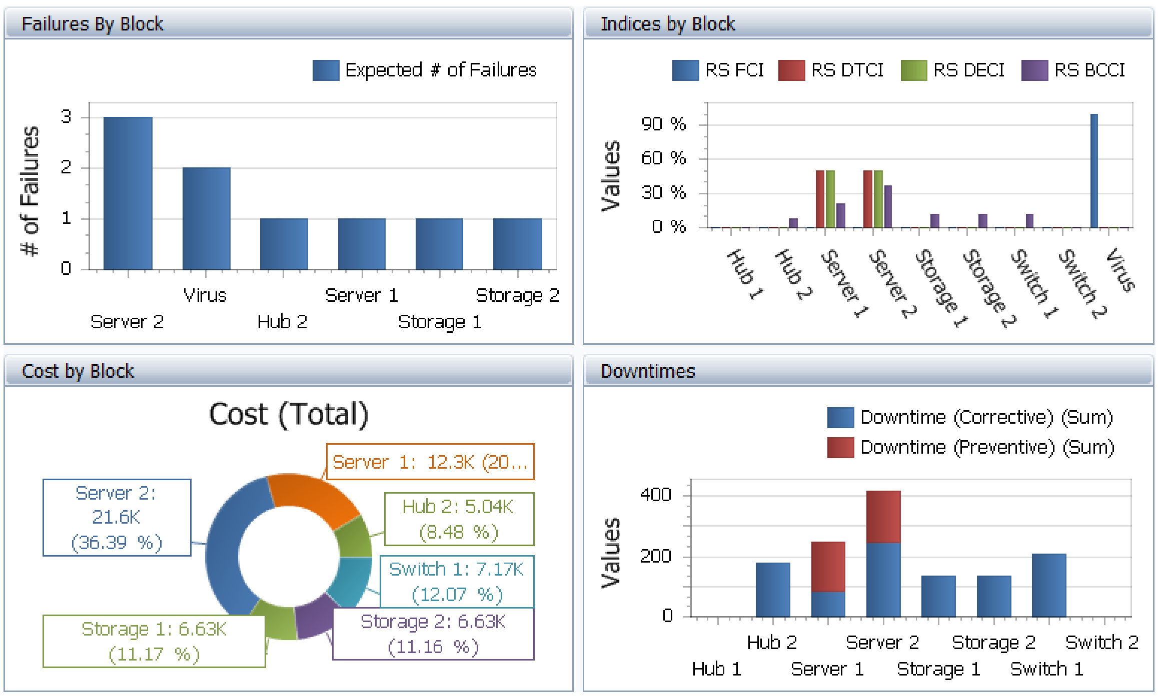
All users can view dashboards based on predefined layouts (see Dashboard Viewer). The viewer is available in the following locations in ReliaSoft desktop applications:

- Project Planner
- Analysis Explorer

- Simulation diagram results in BlockSim (see Simulation Diagram Dashboards in the BlockSim documentation)
- FMEA and FMRA data in XFMEA/RCM++ (see FMEA Dashboards in the XFMEA/RCM++ documentation)
- Reliability Data Warehouse (RDW) data in Weibull++ (see Reliability Data Warehouse in the in Weibull++ documentation)

In a secure database, only users with the "Manage dashboard layouts" permission can manage and create predefined layouts (see Dashboard Layout Manager and Dashboard Layout Designer).

If your organization chooses to implement an SEP web portal for an enterprise database, you can also share selected dashboards to be accessed from any web-enabled device!



Dashboard Viewer

Any user can access the Dashboard Viewer in a variety of locations throughout ReliaSoft desktop applications for graphical presentation of the latest analysis data.

When dashboard functionality is available for the data you're working with (e.g., Project Planner, Analysis Explorer, simulation diagrams in BlockSim, etc.), choose **Dashboard > Dashboard Viewer** to open the viewer.

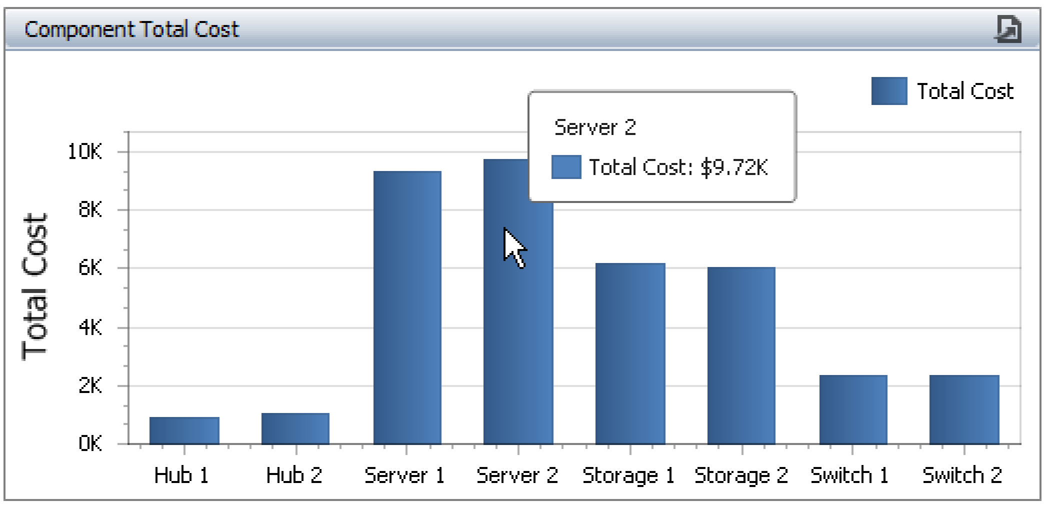
Select any of the predefined dashboard layouts from the drop-down list to see the latest charts for the current data set.

Resizing Panels

To resize the dashboard items vertically and/or horizontally, click between two items and drag the resize cursor (+) to the height and/or width you want. This option is available when a dashboard contains multiple items.

Showing Details in Charts and Pies

To see more details about the underlying data, move the mouse pointer over a bar or slice. For example, when the pointer is over a bar in a chart, the values for the x and y axes will be displayed in the tooltip.

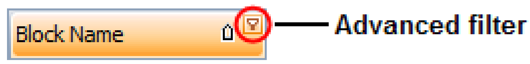


Drill Downs and Master Filters

For information about how to use charts that have been configured for a drill down, see Viewing Drill Downs. For information about how to use charts that have been configured with master filters, see Using Master Filters.

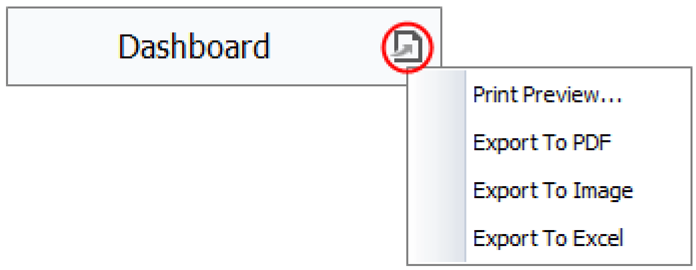
Sorting and Filtering Data in a Grid

- To sort the data in ascending or descending order, click the column heading.
- To filter the data, click the Advanced Filter icon to see a drop-down list of filters that you can apply. The dashboard uses the same advanced filters functionality that is built in to other utilities (e.g., Resource Manager, Analysis Explorer, etc.). (See Finding and Filtering Records.)



Printing or Exporting the Dashboard

To print the entire dashboard, or export it as a *.pdf file, an image or an Excel file, click the **Export To** icon.

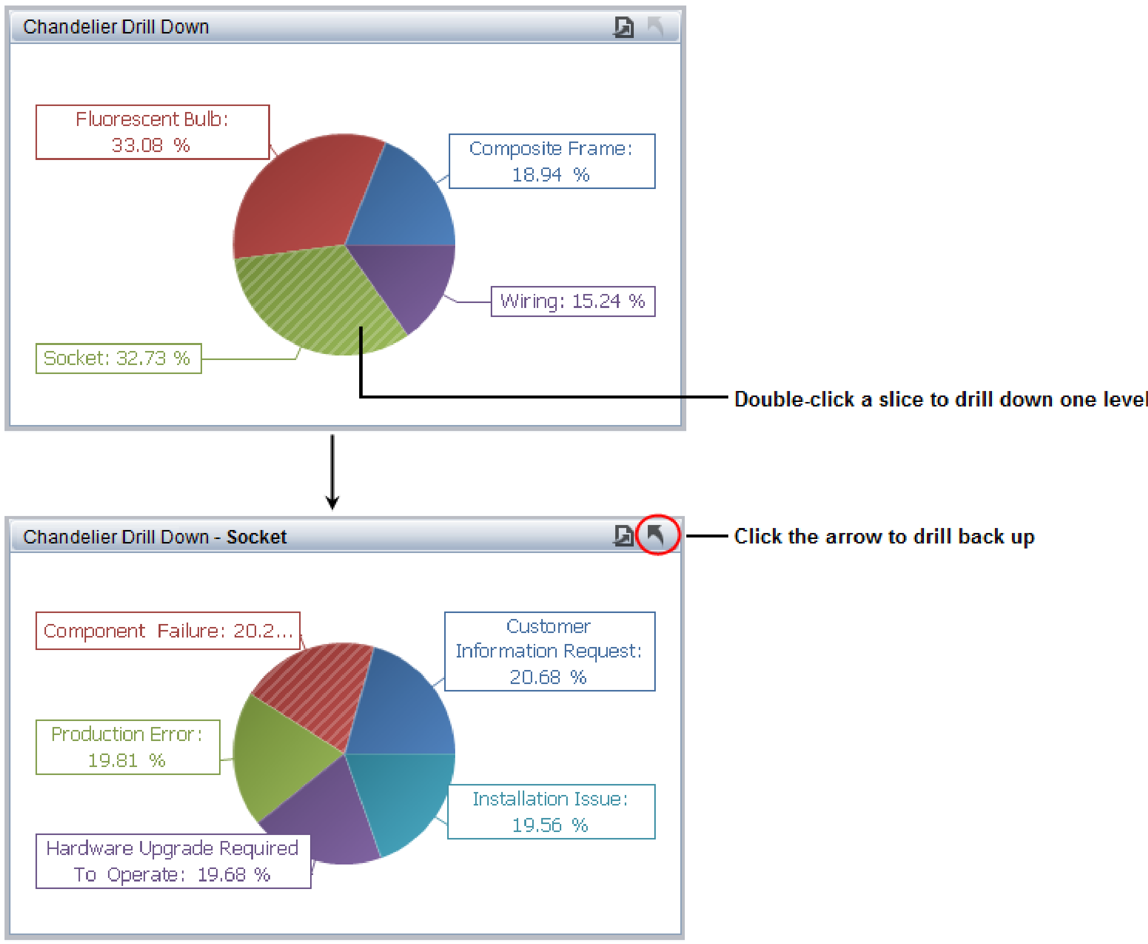


Viewing Drill Downs

An easy way to determine if the dashboard was configured for drill down is to look for the arrow in the caption bar, if the bar is visible.

To drill down, double-click an area of interest.

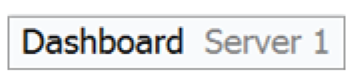
To drill up, click the arrow ( ) until you have reached the desired level or right-click the chart, then choose **Drill Up**.



Using Master Filters

When a dashboard with master filters is created, it can be configured for single or multiple filtering. If you see either of the following, then filtering is enabled:

- For **single** filtering, the dashboard title is followed by light gray text.



- For **multiple** filtering, the **Clear Master Filter** icon displays in the caption bar, if the bar is visible.

Note that if you see both the icon and light gray text, this means that multiple filtering is enabled, but only one item in the filter is selected.

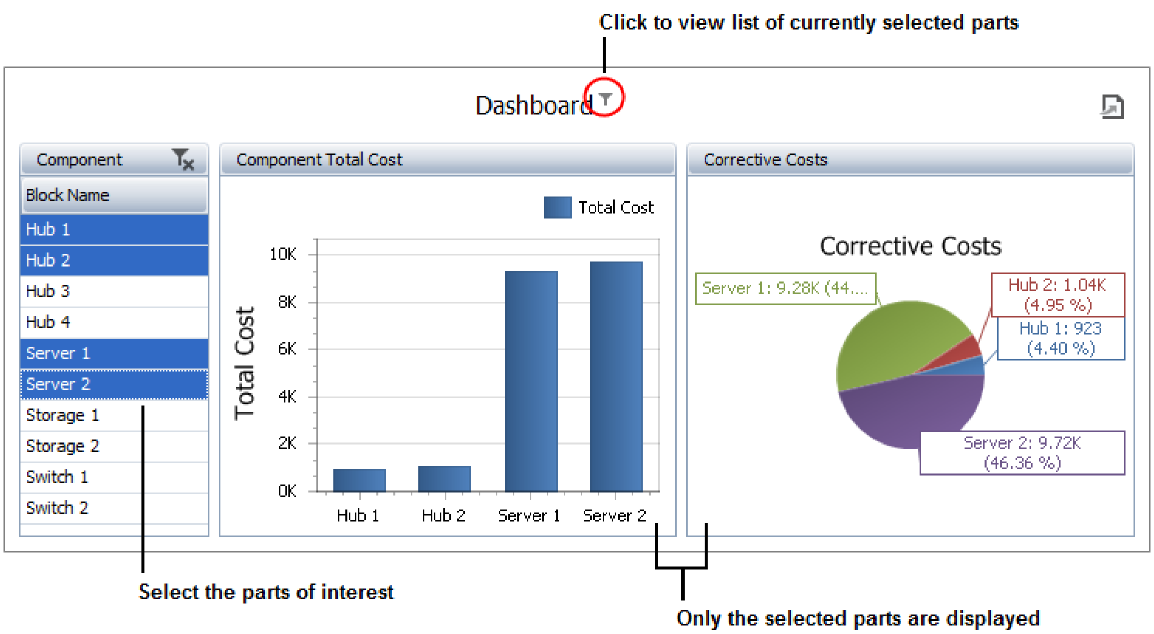
Single Filtering

For single filtering, you can only select one item at a time. You will not be able to clear the view, only select a different item.

Multiple Filtering

For multiple filtering, you can select as many filter items as you want to view. Hold down **SHIFT** while clicking to select consecutive elements or hold down **CTRL** while clicking for non-consecutive items.

In the example shown next, the grid was set up as the master multiple filter. The chart and pie charts display the data for the four parts selected in the grid.

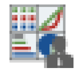


To reset the view, click the **Clear Master Filter** icon or right-click, then choose **Clear Master Filter**.

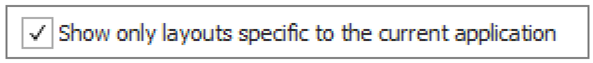
Dashboard Layout Manager

You can use the Dashboard Layout Manager to create, edit and delete the predefined dashboard layouts that will be available for any user to view for a particular data set. (In a secure database, this is available only for users with the "Manage dashboard layouts" permission.)

To open the Dashboard Layout Manager, choose the command from any interface that supports dashboard functionality (e.g., Project Planner, Analysis Explorer, simulation diagrams in BlockSim, etc.). It will display only the layouts for that particular data type.



You can also open the Dashboard Layout Manager from the Backstage view (**File > Manage Database > Dashboard Layout Manager**). It will display all of the layouts for any data type. If you want to see only the layouts specific to the application you are using, select the check box.



Show in Viewer

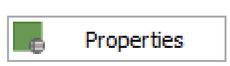
Use the check boxes in the Show in Viewer column to select which of the predefined layouts will be available to all database users via the Dashboard Viewer. If you clear a check box, the layout will not show in the viewer's drop-down list but will remain in the manager's list.

Name	Description	Show in Viewer
System Dashboard	Sample BlockSim simulation dashboard using System results.	☑
Block Dashboard	Sample BlockSim simulation dashboard using Block results.	☑
Crew Dashboard	Sample BlockSim simulation dashboard using Crew results.	☐

Use the up/down arrows to change the order of the layouts that are shown in the viewer's drop-down list.

Layout Properties

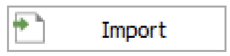
For all dashboard layouts that are selected to show in the Dashboard Viewer, the name that will be visible in the list is set from the Layout Properties window. To view or change these properties, select a row in the table and click the **Properties** button (or right-click and choose **Properties**).



The description and update history are available to help you manage the layouts; that information is not visible to all database users.

Importing Layouts

It is easy to reuse dashboard layouts that were created in other databases. Click the **Import** button and then select the database you want to import from. If you are using the Dashboard Layout Manager in the Backstage view, you can import layouts of any type. If you're using a manager from one of the locations where dashboards are used, you can only import layouts of the current type.



Dashboard Layout Designer

You can use the Dashboard Layout Designer to create or edit each predefined dashboard layout. (In a secure database, this is available only for users with the "Manage dashboard layouts" permissions.)

To access this window, first open the Dashboard Layout Manager and then choose to either create a new layout or edit an existing one.

Separate Layouts for Each Data Type

Each dashboard layout is designed to be used with a specific data type, which determines which fields are available and where the dashboard can be viewed. For information about creating dashboards for each particular type of data, see:

- Project Planner Dashboards
- Analysis Explorer Dashboards
- Simulation Diagram Dashboards (in the BlockSim documentation)
- FMEA Dashboards (in the XFMEA/RCM++ documentation)
- Reliability Data Warehouse (RDW) Dashboards (in the Weibull++ documentation)

Tip: For dashboards that are intended to be viewed in both desktop applications and the SEP web portal, it is recommended to use the **Local Colors** option on the Design tab of the Dashboard Layout Designer to save your preferred colors with the layout. The global colors definition may not be the same in both web and desktop.

Configuring a Dashboard Item

This topic demonstrates some of the basic steps for configuring a dashboard item (bar chart, pie chart, gauges, etc.) in the Dashboard Layout Designer. The simple bar chart in this example is based on block-level simulation results from a reliability block diagram in BlockSim. The data source options will be different for other locations where dashboards are available, see Project Planner, Analysis Explorer, Simulation Diagram Dashboards (BlockSim), FMEA Dashboards (XFMEA/RCM++) or RDW Dashboards (Weibull++).

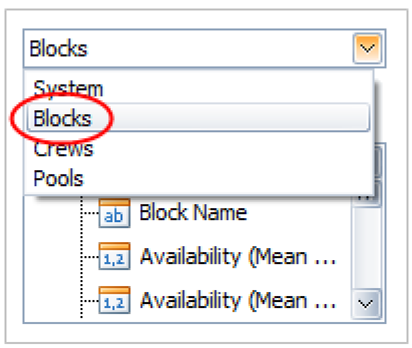
In a secure database, only users with the "Manage dashboard layouts" permissions can create layouts.

Add the Chart

Add a chart to the Layout panel by choosing **Home > Insert > Chart**.

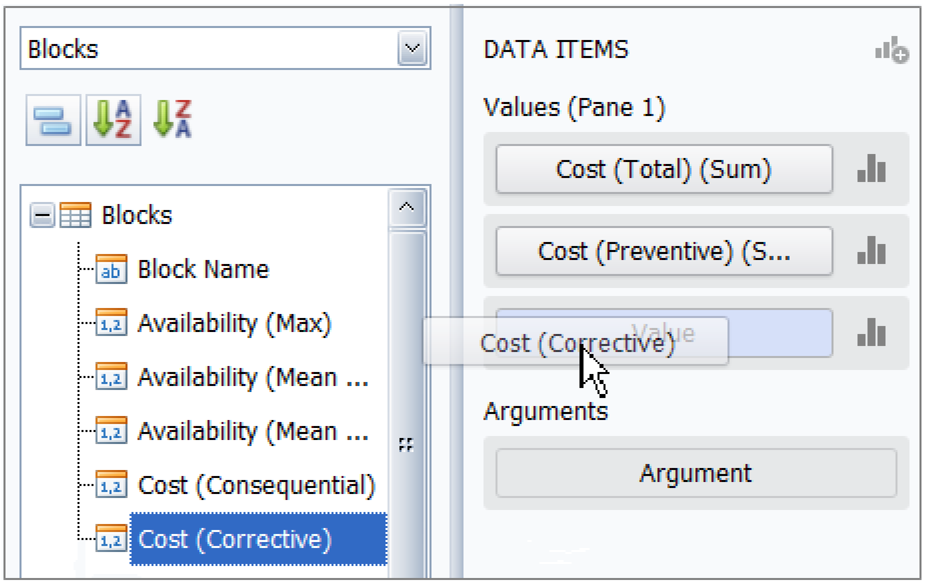
Define the Chart Data

1. Select a data source from the drop-down list in the Data Source panel, if applicable.



2. Choose the data field (or fields) that will be displayed in the y-axis of the bar chart, then drag each field into the **Values** area in the DATA ITEMS panel. For this example, the chart will display total operating cost, cost from preventive maintenance and cost from corrective maintenance.

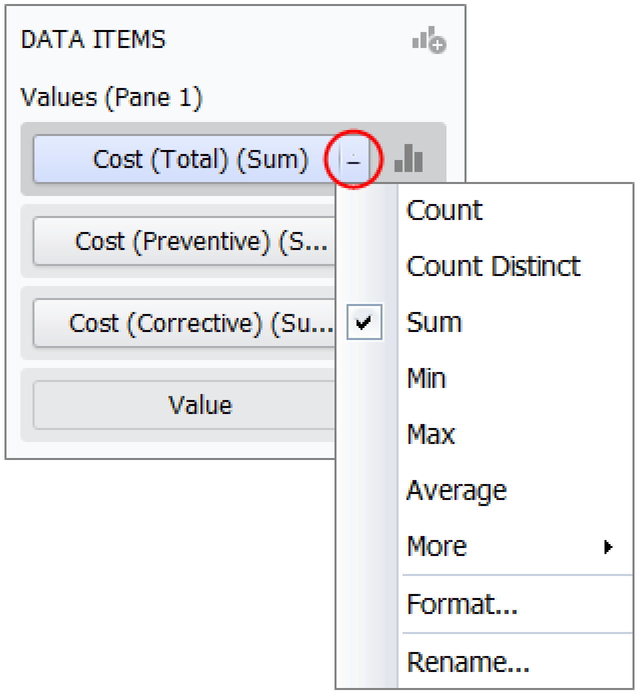
   Note that when different data sources are available, all of the data fields used within a chart must come from the same source.



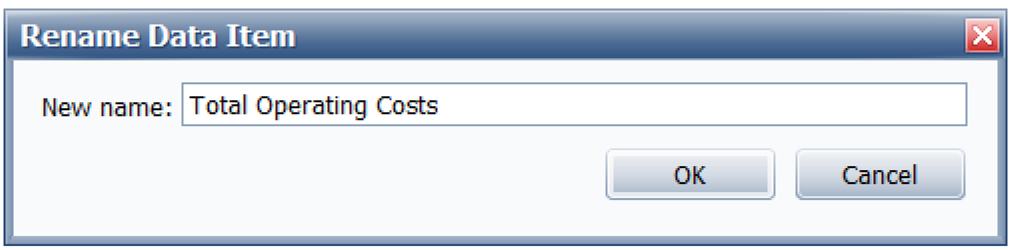
Tip: To replace a data item, drag a new field on top of the current one. To remove a field, drag it anywhere outside of the data item; and to completely clear the DATA ITEMS panel, right-click the chart and select **Remove Data Items**.

3. Use the drop-down list to see which values are available for this data field (e.g., count, sum, average, etc.) and choose one to display in the chart. Note that the same option must be selected for all values in the same chart.

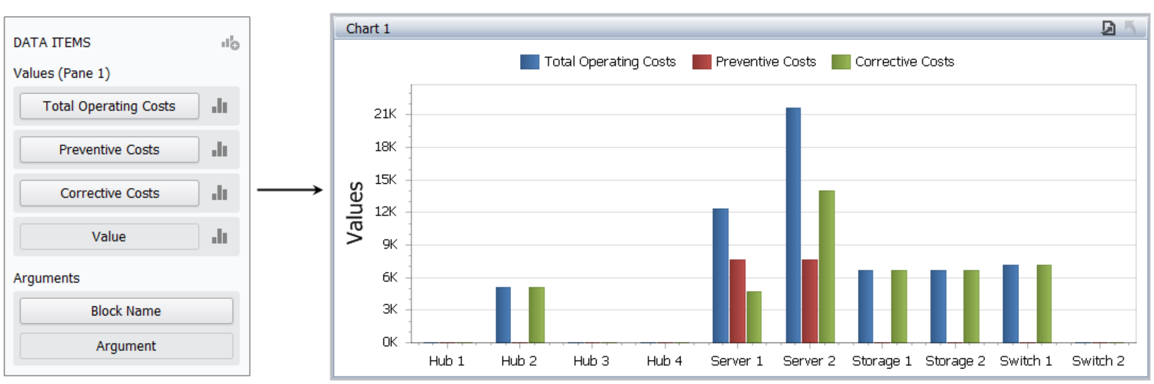
   Because we want the total cost from all blocks that meet the given criteria, we use the default **Sum** for this example.



4. If appropriate, use the **Rename** command to change the name that will be displayed in the chart legend, tooltip, etc.



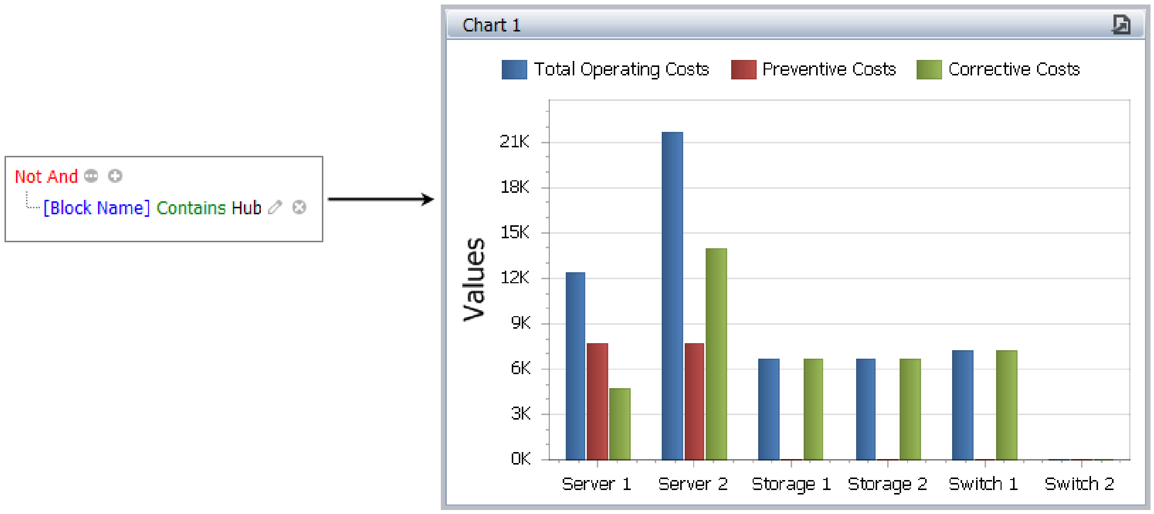
5. Choose the data field that will be displayed in the x-axis, then drag it to the **Argument** bar.



Filter the Data

To filter the data displayed in the chart, choose **Data** > **Filtering** > **Edit Filter**.

Use the Filter Editor to define the criteria. For this example, we’re using the "Not And" operator to exclude any blocks with a name that contains "Hub."

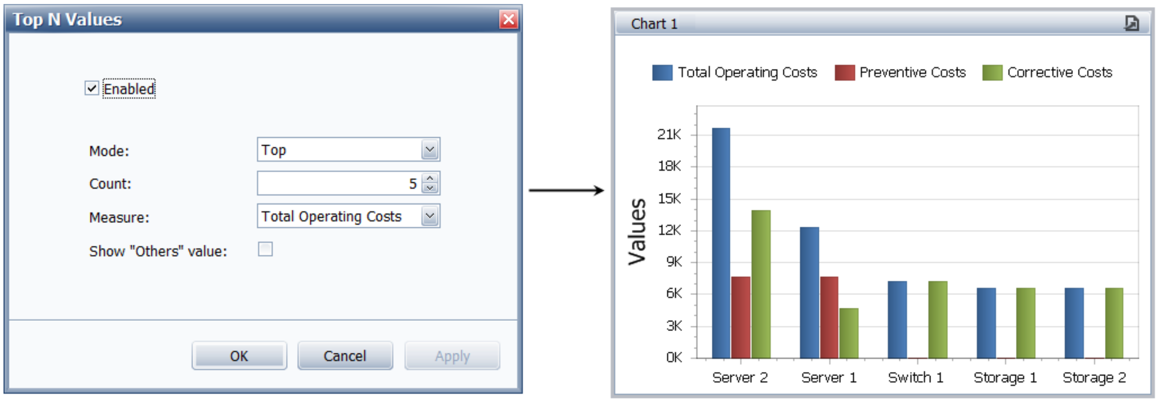


Tip: You can use hidden data items to filter based on fields that are not displayed in the chart.

Sort by “Top N”

To configure the chart to show only the top 5, bottom 10, etc., use the **Top N** command in the Argument data item’s drop-down list.

For this example, we’re showing only the Top 5 blocks, based on total operating cost. The results are displayed from highest to lowest and the order cannot be changed.



Tip: Alternatively, you could use the **Sort by** command to sort by the argument data item — which is labeled (Value) in this menu — or by any of the value data items. Click the Argument bar to toggle the sort between ascending and descending order.

Other Design Options

Click the **Design** tab on the ribbon to access other configurable options for this type of chart. For example:

1. To hide or display the caption bar above the chart, choose **Show Caption**.

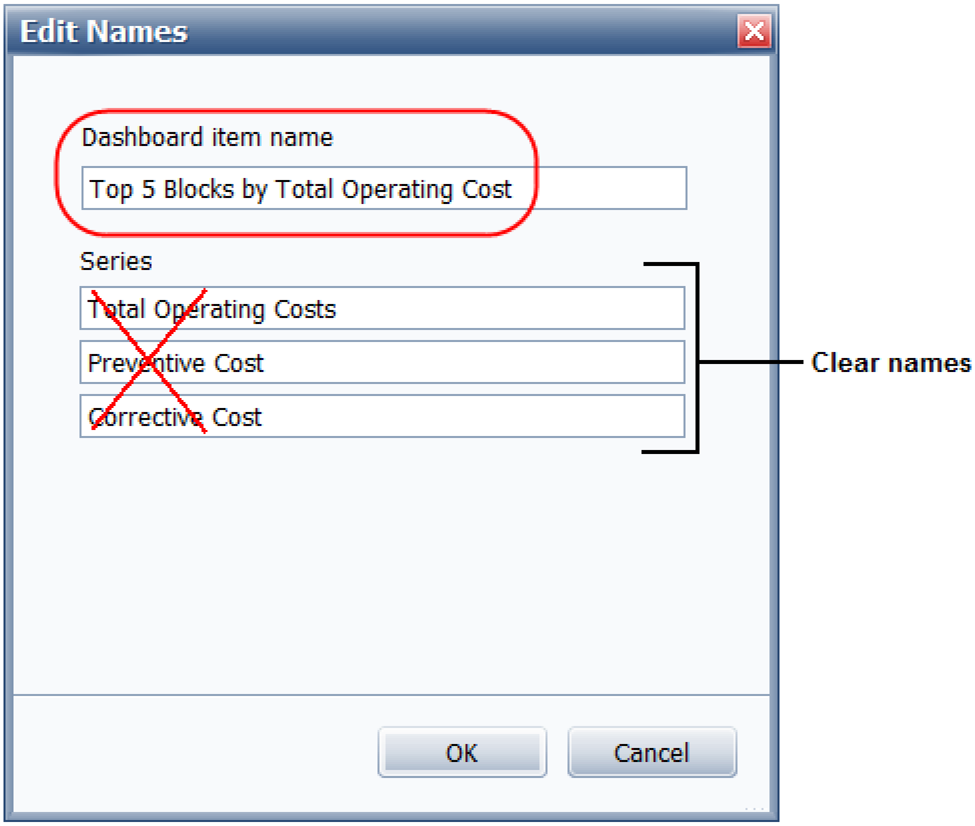
   

   We will keep it displayed for this example.

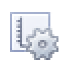
2. To change the title in the caption bar to something more meaningful, choose **Edit Names**.

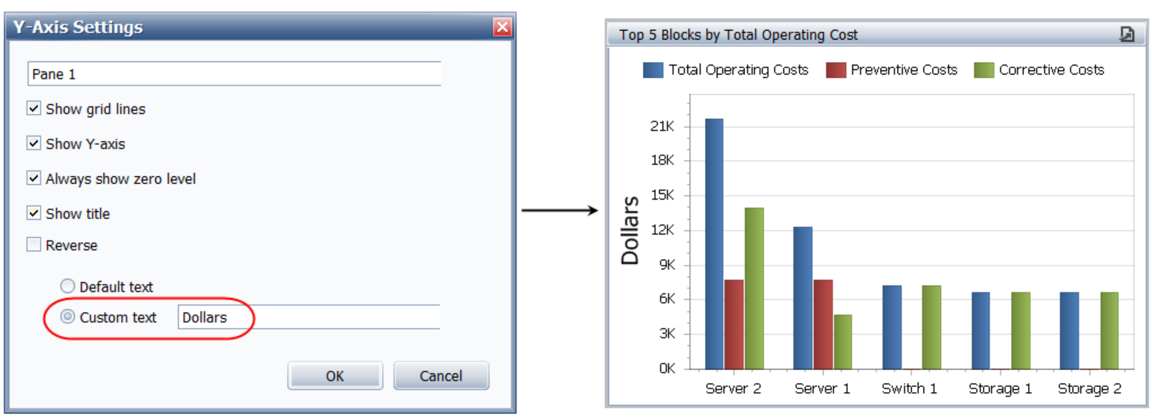
   

   For this example, we will set the Dashboard item name to “Top 5 Blocks by Total Operating Cost” and clear all of the **Series** fields to make sure the latest data item name (set via the **Rename** command in the data item’s drop-down list) will always be used in the chart legend and tooltip.



3. To change the y-axis title to a more meaningful name, choose **Y-Axis Settings**.





Available Dashboard Items

The following dashboard items can be added to any layout. You can add the same type of item more than once, if desired. To insert an item, choose **Home** > **Insert**, then click one of the following options.

Pivot	Chart	Gauges	Choropleth Map	Range Filter	Scatter Chart
Grid	Pies	Cards	Geo Point Map	Text Box	Image

Data Types

There are two types of data that can be used in a dashboard item: measure or dimension.

- **Measure** data is calculated (sum, count, average, etc.). For example, the values on the y-axis of a chart, the size of a slice in a pie or a target value in a card or gauge. Values and targets are forms of measure data.
- **Dimension** data can be sorted but is not calculated. For example, the bars in chart, slices in a pie or text in a grid column. Arguments and series are forms of dimension data.

Data Shaping

You can use the Dimensions and/or Measures bars in the HIDDEN DATA ITEMS panel when you want to perform data shaping (filtering, sorting or Top N) but do not want the data to be displayed in the dashboard item. Data shaping is available for any dashboard item, except text boxes and images.

- The data fields contained in the **Dimensions** bar can be used in Filter Editor window to create filter conditions based on their values. Dimensions in this area have the same sorting functionality that is available in the argument, attribute or series DATA ITEMS area.
- The data fields contained in the **Measures** bar can be used in the Sort by submenu and in the Top N Values window for data sorting. Measures in this area have the same calculation functionality that is available in the values and target DATA ITEMS area.

Configuring a Drill Down

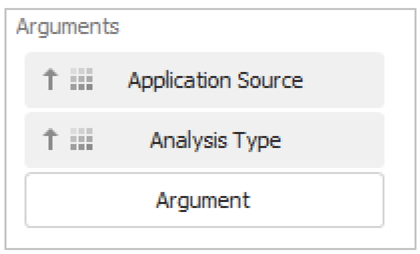
Drill down can be enabled for the following dashboard items:

- Grids: Specify two or more columns
- Charts and pies: Specify two or more arguments and/or series
- Gauges: Specify two or more series
- Cards: Specify two or more series

Configuring a Drill Down Sequence

1. In the Dashboard Layout Designer window, select the chart or grid you want to drill down.
2. Drag the additional data fields into the **Arguments** area in the appropriate order for the desired drill down sequence.

   For example, you can use the following sequence to configure a pie chart in a Analysis Explorer dashboard to first show analyses by application source, then drill down to analysis type.



3. Choose **Data > Filtering > Drill Down**. This command is a toggle; choose the command again to disable the drill down.

When any user views this layout in the Dashboard Viewer, he/she will be able to double-click inside the chart to drill down, and use the arrow () to drill back up. (See Viewing Drill Downs.)

Tip: To make it easy to identify a drill down in the Dashboard Viewer, display the dashboard item's caption bar (**Design > Common > Show Caption**).

Filtering

Using the Filter Editor

You can use the Filter Editor to incorporate filters into dashboard items that contain dimension data. Arguments and series are forms of dimension data.

This type of filtering only applies to the dashboard item you have selected. To filter the data across the entire dashboard, see Configuring a Master Filter.

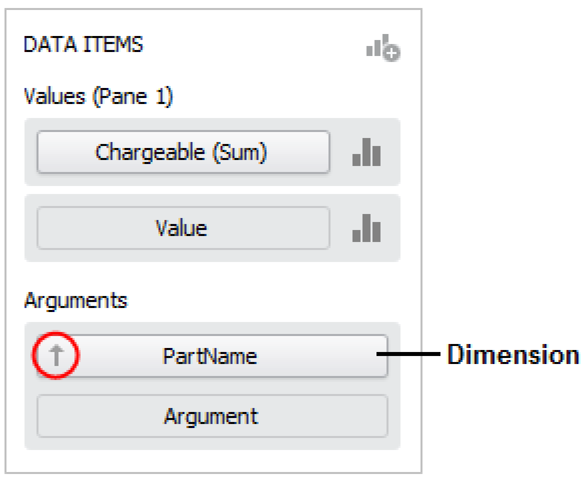
Tip: While you cannot use the Filter Editor to sort non-dimension data, users can apply filters to such data fields within dashboard grids. See Sorting and Filtering Data in a Grid.

DIMENSION DATA LOCATIONS

Dimension data can be used in any or all of the following areas:

- Dimension bar in the HIDDEN DATA ITEMS panel.
- Dimension, argument and/or series bars in the DATA ITEMS panel.

An easy way to determine if a data field is a dimension, argument or series, is to look for the sort arrow to the left of the data field name.

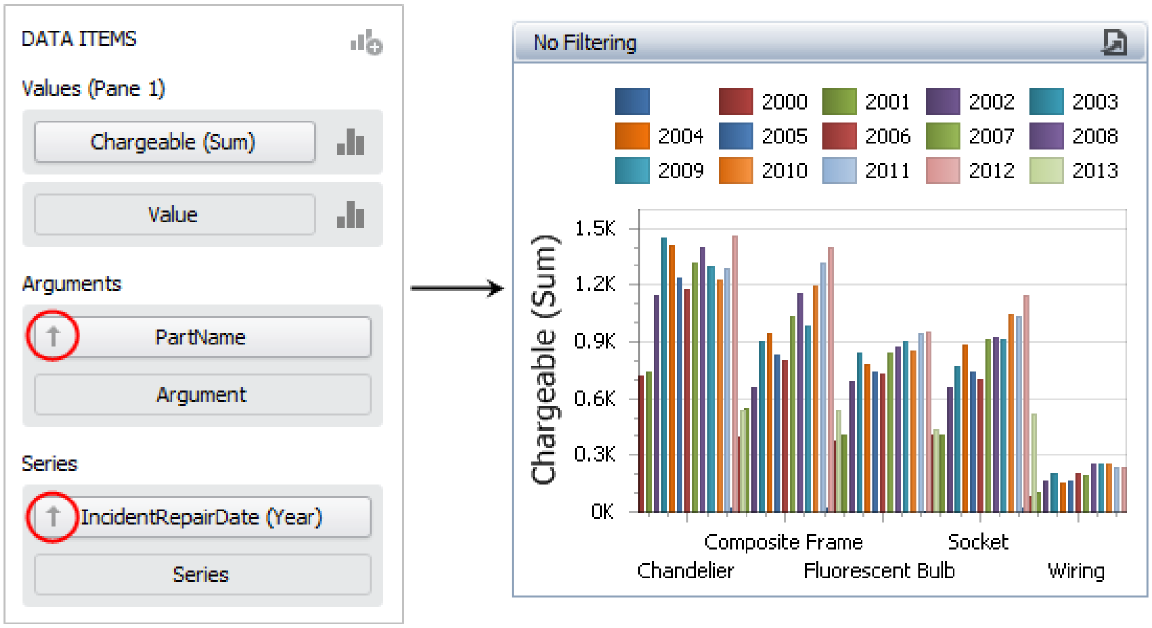


CONFIGURING A FILTER

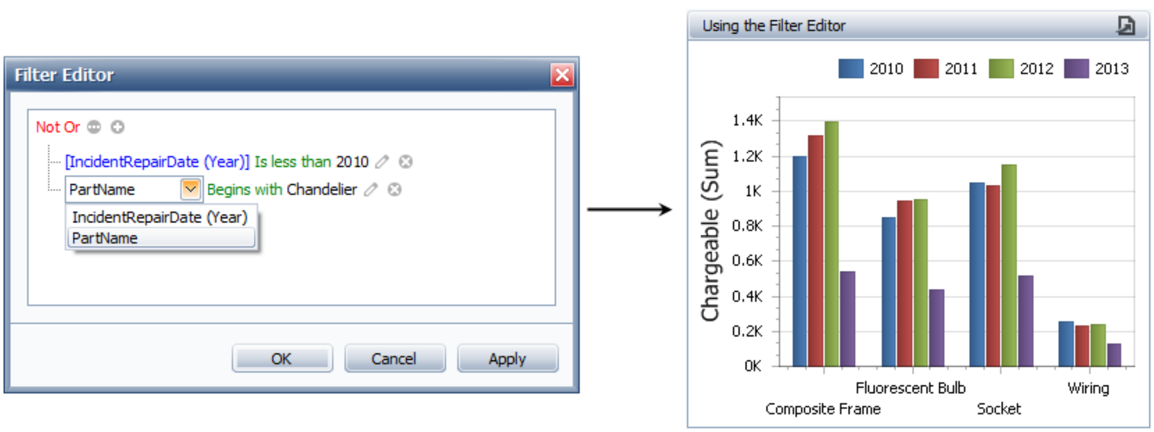
1. In the Dashboard Layout Designer widow, select a dashboard item, then choose **Data > Filtering > Edit Filter**. This command is also available in the shortcut menu when you right-click the dashboard item.

2. In the Filter Editor window, define the criteria. The dashboard uses the same Edit Filter functionality that is built in to other utilities (e.g., Resource Manager, Analysis Explorer, etc.). (See Finding and Filtering Records.)

In the following example, a chart was created for data that was extracted from XFRACAS into the Reliability Data Warehouse (RDW) in Weibull++.



Because we are interested in only the chandelier parts (not the whole unit) and data from 2010 and later, we can apply the following filter to remove the unwanted data:



Configuring a Master Filter

A master filter can be used to filter data across the entire dashboard. Grids, pies, charts, gauges, cards and maps dashboard items can be used as master filters.

There are two types of master filters that can be configured:

- A **Single** filter allows you to select one item at a time. You will not be able to clear the filter, only select a different item.
- A **Multiple** filter allows you to select one or more items at a time. You can also clear the filter to view all items at any time. This is the most flexible option.

For example, suppose your layout contains a grid, a chart and a pie, and the grid is configured as the multiple master filter. When you select items in the grid, only the data for those items will display in the chart and pie.

CONFIGURING A MASTER FILTER

In the Dashboard Layout Designer window, select the dashboard item, then enable the master filtering by choosing **Data > Interactivity > [Single Master Filter/Multiple Master Filter]**. This command is a toggle; choose the command again to disable it.

 or 

When any user views this layout in the Dashboard Viewer, he/she can use the filter to view only the data of interest. (See Using Master Filters.)

Note that you can prevent a specific dashboard item from being affected by master filters, by selecting it in the Layout panel and then choosing **Data > Interactivity settings > Ignore Master Filters**. This command is a toggle; choose the command again to disable it.

Tip: To make it easy to identify that a dashboard item is configured for filtering in the Dashboard Viewer, display the item's caption bar (**Design > Common > Show Caption**).

End User License Agreement

Click Wrap Agreement at time of product installation

BY INSTALLING THIS SOFTWARE OR BY CLICKING THE "I ACCEPT" BUTTON AND ACCEPTING THIS LICENSE IN ACCORDANCE WITH SECTION D (LICENSE ACCEPTANCE PROCEDURES) YOU AGREE TO THE TERMS OF THIS LICENSE WHICH WILL BIND YOU, THE COMPANY ON WHOSE BEHALF YOU ARE ENTERING INTO THIS LICENSE AGREEMENT, AND ITS EMPLOYEES, AS APPLICABLE.

THIS LICENSE AGREEMENT IS A LEGAL AGREEMENT BETWEEN HOTTINGER BRUEL & KJAER UK LTD, A COMPANY INCORPORATED IN THE UNITED KINGDOM, WHOSE REGISTERED NUMBER IS 1589921 AND WHOSE REGISTERED OFFICE IS AT TECHNOLOGY CENTRE, ADVANCED MANUFACTURING PARK, BRUNEL WAY, CATCLIFFE, ROTHERHAM S60 5WG (THE "LICENSOR") AND THE END USER LICENSEE OR ITS REPRESENTATIVE/AGENT AUTHORISED TO BIND THE END USER LICENSEE (REFERENCED HEREIN AS "YOU" OR "YOUR").

PLEASE READ THIS CAREFULLY BEFORE USING THE SOFTWARE PRODUCT

A: BACKGROUND

THIS LICENSE AGREEMENT SHALL BE BETWEEN YOU AND THE LICENSOR REGARDLESS OF WHETHER YOU HAVE PURCHASED THE SOFTWARE PRODUCT DIRECTLY FROM THE LICENSOR OR AN AUTHORISED RESELLER AS SET OUT BELOW.

THE LICENSOR APPOINTS EXPERIENCED PROVIDERS OF SOFTWARE RESALE SERVICES WHO DISTRIBUTE THE SOFTWARE PRODUCT TO END USERS ("AUTHORISED RESELLER"). IF YOU HAVE PURCHASED THE SOFTWARE PRODUCT AND ANY ASSOCIATED SUPPORT AND MAINTENANCE SERVICES FROM AN AUTHORISED RESELLER, SUCH PRODUCTS AND SERVICES WILL BE PROVIDED DIRECTLY TO YOU BY THE RELEVANT AUTHORISED RESELLER AND ALL PAYMENTS SHALL BE MADE TO THE AUTHORISED RESELLER ON THE AUTHORISED RESELLER'S TERMS AND PRICES.

IF YOU HAVE PURCHASED THE SOFTWARE PRODUCT AND SUPPORT AND MAINTENANCE SERVICES DIRECTLY FROM THE LICENSOR, SUCH SUPPORT AND

MAINTENANCE SERVICES WILL BE DEALT WITH IN A SEPARATE AGREEMENT BETWEEN YOU AND THE LICENSOR AND ALL PAYMENTS WILL BE MADE TO THE LICENSOR AND GOVERNED BY THE LICENSOR'S TERMS OF SALE.

B: PROPERTY OF LICENSOR

THE LICENSOR, OR, WHERE APPLICABLE, THE RELEVANT MEMBER OF THE LICENSOR'S CORPORATE GROUP, IS THE ENTIRE LEGAL AND BENEFICIAL OWNER OF CERTAIN SOFTWARE PRODUCT(S) (THE "SOFTWARE PRODUCT") AS SET OUT IN THE QUOTATION OF INFORMATION, WHICH SHALL ALSO INCLUDE YOUR LICENSE KEY, SENT TO YOU BY EMAIL EITHER BY THE LICENSOR OR, IF YOU HAVE PURCHASED THE SOFTWARE PRODUCT FROM AN AUTHORISED RESELLER, BY THE AUTHORISED RESELLER ("QUOTATION").THE LICENSOR IS WILLING TO GRANT YOU A LICENSE TO USE THE SOFTWARE PRODUCT ONLY IF YOU ACCEPT ALL THE TERMS AND CONDITIONS SET OUT BELOW.

C: LICENSE TO BE GRANTED

THE LICENSOR IS PROPOSING TO GRANT YOU ONE OF THE FOUR FORMS OF LICENSES IN RESPECT OF THE SOFTWARE PRODUCT. THE DETAILS OF EACH FORM OF LICENSE ARE SET OUT IN CLAUSES 2.1 AND 3 BELOW. THE FORM OF LICENSE THAT IS PROPOSED TO BE GRANTED TO YOU IS SET OUT IN THE QUOTATION. THE QUOTATION FORMS PART OF THE LICENSE AGREEMENT AND BY ACCEPTING THE TERMS OF THE LICENSE YOU ARE ACCEPTING THOSE TERMS IN THE QUOTATION. IF YOU HAVE NOT RECEIVED THE QUOTATION FROM EITHER THE LICENSOR OR FROM THE AUTHORISED RESELLER (IF YOU HAVE PURCHASED THE SOFTWARE PRODUCT FROM AN AUTHORISED RESELLER) AT THE DATE HEREOF OR IF ANY DETAILS WITHIN THE QUOTATION ARE INCORRECT THEN YOU SHOULD FOLLOW THE "LICENSE REJECTION PROCEDURES" SET OUT AT PARAGRAPH E BELOW AND IMMEDIATELY CONTACT THE LICENSOR OR THE AUTHORISED RESELLER (AS APPLICABLE).

D: LICENSE ACCEPTANCE PROCEDURES

BY CLICKING ON THE ACCEPTANCE BUTTON WHICH FOLLOWS THIS LICENSE AGREEMENT (MARKED "I ACCEPT") IN THE UI, YOU INDICATE ACCEPTANCE OF THIS LICENSE AGREEMENT AND THE LIMITED WARRANTY AND LIMITATION OF LIABILITY SET OUT IN THIS LICENSE AGREEMENT. SUCH ACCEPTANCE IS ON YOUR OWN BEHALF, IF YOU ARE AN INDIVIDUAL, AND ON BEHALF OF ANY

CORPORATE ENTITY WHICH EMPLOYS YOU OR WHICH YOU REPRESENT ("CORPORATE LICENSEE") IN ENTERING INTO THIS LICENSE AND YOU WARRANT THAT YOU HAVE THE NECESSARY AUTHORITY TO BIND THE CORPORATE ENTITY IN THIS RESPECT OR THAT THE AUTHORISED REPRESENTATIVES OF THE CORPORATE ENTITY ARE AWARE OF YOUR ACCEPTANCE OF THIS AGREEMENT ON ITS BEHALF. IN THIS LICENSE AGREEMENT, "YOU" REFERS TO THE LICENSEE, ANY SUCH CORPORATE LICENSEE, OR BOTH, AS A COMMON SENSE INTERPRETATION OF THE RELEVANT SECTION DICTATES.

E: LICENSE REJECTION PROCEDURES

IF YOU DO NOT ACCEPT THESE TERMS AND CONDITIONS, YOU SHOULD CLICK ON THE "I DO NOT ACCEPT" BUTTON IN THE UI, DELETE THE SOFTWARE PRODUCT FROM YOUR COMPUTER AND PROMPTLY (AND IN ANY EVENT, WITHIN 14 DAYS OF RECEIPT) RETURN TO THE RESELLER (A) ANY MEDIA; AND (B) ANY OTHER ITEMS PROVIDED, THAT ARE PART OF THIS PRODUCT.

F: OTHER AGREEMENTS

IF THE LICENSOR HAS EXPRESSLY AGREED WITH YOU IN WRITING THAT YOUR USE OF THE SOFTWARE PRODUCT IS GOVERNED BY A PRIOR EXECUTED LICENSE AGREEMENT WITH THE LICENSOR, SUCH AGREEMENT SHALL APPLY INSTEAD OF THE FOLLOWING TERMS AND CONDITIONS TO THE EXTENT THAT THEY CONFLICT.

YOU SHOULD PRINT A COPY OF THIS LICENSE AGREEMENT FOR FUTURE REFERENCE.

LICENSE AGREEMENT

1. Ownership of Software Product and Copies

1.1 The Software Product and any printed materials or electronic documentation accompanying the Software Product ("**Documentation**") are copyrighted works of authorship, and are also protected under applicable database laws. The Licensor, or, where applicable, the relevant member of the Licensor’s corporate group, retains ownership of the Software Product and all subsequent copies of the Software Product, regardless of the form in which the copies may exist. This License Agreement is not a sale of the original Software Product or any copies.

2. Commercial License

2.1 In consideration of You agreeing to abide by the terms of this License Agreement and payment of the applicable License fees either to the Licensor or, if You have purchased the Software Product from an Authorised Reseller, to the Authorised Reseller (as specified in the Quotation), the Licensor hereby grants to You a personal, non-exclusive, and (unless expressly agreed otherwise by the Licensor in accordance with clause 5.1) non-transferable License to use the Software Product and the Documentation on the terms of this License Agreement. The License shall take one of the following forms, and shall be set out in the Quotation:

(a) Annual License:

(i) Duration – this License shall continue for a period of 12 months ("Initial Period") from the date of delivery of the Quotation (“the Delivery Date”) and, provided that You have paid all applicable renewal fees, from year to year thereafter unless terminated in accordance with any provision of clause 10 below or any other clause of this License Agreement.

(ii) Termination – You must give the Licensor 90 days' written notice prior to the anniversary of the Delivery Date, such notice not to expire before such anniversary date.

(b) Perpetual License:

(i) Duration – this License shall continue for a period of 25 years from the Delivery Date unless terminated in accordance with any provision of clause 10 below or any other clause of this License Agreement.

(ii) Conversion – this License may be converted to a Token Based Annual License, upon application to either the Licensor or the Authorised Reseller (if You have purchased the Software Product from an Authorised Reseller), and payment of an additional fee to the Licensor or the Authorised Reseller(as applicable), which shall be additional to the fees paid or payable in respect of any maintenance and support services provided by either by the Licensor or the Authorised Reseller (if You have purchased the Software Product from an Authorised Reseller). You may reverse this conversion to the same terms as agreed in the Perpetual License at the anniversary of the conversion by giving the Licensor 90 days' written notice prior to the anniversary of the Delivery Date, such notice not to expire before the anniversary of the Delivery Date.

(c) Token Based Annual License (referenced in the Quotation as a Prenscia Access System, CDS, CRS, or CDRS License):

(i) Subject to the terms and conditions of this License Agreement, You may access any of the Software Product in the Quotation at any one time and in any combination up to the number of subscriptions granting You access to the Software Product for simultaneous use by You and purchased by You pursuant to this Agreement ("Access Credits"). You may add additional Access Credits which shall be charged in accordance with either the Licensor's prices or the Authorised Reseller's prices (if You have purchased the Software Product from an Authorised Reseller). Such increase in Access Credits shall apply to the following 12 month period, unless You give the Licensor or the Authorised Reseller (as applicable) written notice at least 90 days in advance of the expiration of the Initial Period, that You do not wish to sustain such increase in Access Credits. You may opt to reduce the number of Access Credits with 90 days' written notice in advance of the 12 month period following from the anniversary of the Delivery Date, provided that the total number of Access Credits does not fall below the minimum threshold set by the Licensor or the Authorised Reseller (as applicable) or below the initial number of Access Credits calculated at the time of conversion of any Perpetual License.

(ii) Duration – this License shall continue for the Initial Period and, provided that You have paid for the applicable Access Credits, from year to year thereafter unless terminated in accordance with any provision of clause 10 below or any other clause of this License Agreement.

(iii) Reporting – in order to serve the Licensor's customers better, the Licensor needs to be able to understand how its customers use the Token Based Annual License. As a condition of Your continued access to the system, You are asked to provide to the Licensor on a quarterly basis, the log file. You will be provided with easy-to-follow instructions on how to do this.

(iv) Termination – You must give the Licensor 90 days' written notice in advance of the expiration of the Initial Period of Your intention to discontinue the Token Based Annual License, such notice not to expire before the anniversary of the Delivery Date, and You must have submitted all reports required of You in clause 2.1(c)(iii) above. Failure to provide such notice and reports will mean that the License shall automatically continue on the same terms until terminated in accordance with this clause.

2.2 As an end user licensee of the Software Product, You may, subject to the following conditions and any conditions specified in the Quotation:

(a) use and copy the Software Product only for use on any computer system that is detailed in the release notes of the respective Software Product and which is licensed, leased and/or controlled by You (the Corporate Licensee) or any member of Your corporate group, which expression includes the Corporate Licensee's majority-owned subsidiaries, any parent company having a majority-owned interest in the Corporate Licensee, and such parent's majority-owned subsidiaries;

(b) load the Software Product into and use it only on computers at the locations specified in the Quotation and which are under Your control; and

(c) copy the Software Product only for back-up and archival purposes and make up to three copies of the Documentation, provided that the original and each copy is kept in Your possession and that Your installation and use of the Software Product does not exceed any restrictions contained in the Quotation.

3. Demonstration License

3.1 If the Licensor has provided You with the Software Product free of charge for evaluation purposes, this clause 3 shall apply to Your use of the Software Product.

3.2 You may use the Software Product on a computer solely for the purposes of evaluating the Software Product.

3.3 The Software Product is provided to You free of charge and on an "AS IS" basis, without any technical support or warranty of any kind from the Licensor including, without limitation, a warranty of satisfactory quality, fitness for purpose and non-infringement.

3.4 This License shall continue for a period of 30 days from the Delivery Date unless terminated in accordance with any provision of clause 10 below or any other applicable clause of this License Agreement or until such License is converted into an Annual License, Perpetual License or Token Based Annual License, if earlier.

3.5 As part of the Licensor's testing and evaluation of the Software Product, You shall share with the Licensor the input and views received and any reports and analysis produced promptly upon receipt of such feedback.

3.6 Clauses 4, 5.1(a), 6.1(b), 6.1(f), 8.1, 8.2, 8.5, 10.1(a) of this License Agreement shall not apply to a Demonstration License.

4. Delivery and Risk

4.1 The Software Product shall consist of one copy of the object code of the Software Product on machine-readable form only, on the disk, dongle or other media on which the Software Product is delivered as described in the Quotation (the "Media"). Risk in the Media shall pass to You on delivery. If any part of the Media shall thereafter be lost, damaged or destroyed, You must contact either the Licensor or the Authorised Reseller from whom You have purchased the Software Product who will arrange for a repair or replacement of such Media at a cost (if any) determined by the Licensor or the Authorised Reseller (as applicable).

5. License Restrictions

5.1 You may not nor permit others to:

(a) transfer the Software Product from one location to another, without the express written consent of the Licensor and provided such transfers do not exceed two in any twelve month period. It will be a condition of the grant of any such consent that the Software Product held on hard drives at the current location must be erased and You must enter into a new License agreement with the Licensor before a Software Product License key is provided to You. The administration charge (currently US $1,500) will be payable by You in respect of such transfer. This charge is only applicable to end user licensees who do not have a support and maintenance services agreement with the Licensor or Authorised Reseller;

(b) exceed the number of concurrent users specified in the Quotation, if any;

(c) sub-License, sell, assign, rent, pledge, lease, transfer or otherwise dispose of the Software Product, on a temporary or permanent basis, without the prior written consent of the Licensor;

(d) translate, reverse engineer, decompile, disassemble, modify or create derivative works based on the Software Product except as permitted by law;

(e) make copies of the Software Product, in whole or part, except for back-up or archival purposes as permitted in this License Agreement;

(f) use any back-up copy of the Software Product for any purpose other than to replace the original copy in the event that it is destroyed or becomes defective;

(g) copy the Documentation (except as provided by this License Agreement);

(h) adapt, modify, delete or translate the Documentation in any way for any purpose whatsoever;

(i) vary, delete or obscure any notices of proprietary rights or any product identification or restrictions on or in the Software Product.

6. Open Source

Notwithstanding the foregoing provisions, You acknowledge that certain components of the Software Product may be covered by so-called open source software licenses (“Open Source Components”), which means any software licenses approved as open source licenses by the Open Source Initiative or any substantially similar licenses, including without limitation any license that, as a condition of distribution of the software licensed under such license, requires that the distributor make the software available in source code format. To the extent required by the licenses covering Open Source Components, the terms of such licenses will apply to such Open Source Components in lieu of the terms of this Agreement. To the extent the terms of the licenses applicable to Open Source Components prohibit any of the restriction in this License Agreement with respect to such Open Source Components, such restrictions will not apply to such Open Source Components. To the extent the terms of the licenses applicable to Open Source Components require Licensor to make an offer to provide source code or related information in connection with the Software Product, such offer is hereby made. Any request for source code or related information should be directed only to Customer Support, Hottinger Bruel & Kjaer UK LTD, AMP Technology Centre, Brunel Way, Catcliffe, Rotherham, South Yorkshire, S60 5WG, United Kingdom. You acknowledge receipt of notices for the Open Source Components for the initial delivery of the Software Product

7. Undertakings

7.1 You undertake:

(a) to ensure that, prior to use of the Software Product by Your employees or agents, all such parties are notified of this License and the terms of this License Agreement and that any such use as aforesaid is in accordance with the terms of this License Agreement;

(b) to replace the current version of the Software Product with any updated or upgraded version or new release provided by the Licensor or the Authorised Reseller (if applicable) under the terms of this License Agreement immediately on receipt of such version or release;

(c) to use the Software Product only as described in the Documentation and subject to the restrictions as set out in clause 5 (License Restrictions above);

(d) to reproduce and include the Licensor's copyright notice (or such other party's copyright notice as specified on the Software Product) on all and any copies of the Software Product, including any partial copies of the Software Product as permitted to be made under the terms of this License Agreement;

(e) to hold all drawings, specifications, data (including object and source codes), Software Product listings and all other information relating to the Software Product confidential and not at any time, during this License or after its expiration, disclose the same, whether directly or indirectly, to any third party without the Licensor's consent and

(f) to effect and maintain adequate security measures to safeguard the Software Product from access or use by any unauthorised person and indemnify the Licensor for losses flowing from Your failure to do so.

8. Intellectual Property Rights

8.1 You acknowledge that all intellectual property rights in the Software Product and the Documentation throughout the world belong to the Licensor, that rights in the Software Product are licensed (not sold) to You, and that You have no rights in, or to, the Software Product or the Documentation other than the right to use them in accordance with the terms of this License Agreement.

8.2 You acknowledge that You have no right to have access to the Software Product in source code form or in unlocked coding or with comments.

8.3 The integrity of this Software Product is protected by technical protection measures ("TPM") so that the intellectual property rights, including copyright, in the software of the Licensor are not misappropriated. You must not attempt in any way to remove or circumvent any such TPM, nor to apply, manufacture for sale, hire, import, distribute, sell, nor let, offer, advertise or expose for sale or hire, nor have in Your possession for private or commercial purposes, any means whose sole intended purpose is to facilitate the unauthorised removal or circumvention of such TPM.

9. Limited Warranty

9.1 Subject to the limitations and exclusions of liability below and providing the Software is used in accordance with the Documentation and on an operating system or computer for which it was designed, the Licensor warrants that (a) the Media on which the Software Product is furnished will be free from material defects under normal use for a period of 90 days from the date of delivery (the "Warranty Period"); and that (b) during the Warranty Period, the Software Product will, when properly used, perform substantially in accordance with the functions described in the Documentation; and (c) that the Documentation correctly describes the operation of the Software Product in all material respects.

9.2 If, within the Warranty Period, You notify the Licensor in writing of any defect or fault in the Software Product in consequence of which it fails to perform substantially in accordance with the Documentation, and such defect or fault does not result from You having amended the Software Product or used it in contravention of the terms of this License Agreement, the Licensor will, at its sole option, i) repair or replace the Software Product, provided that You make available all information that may be necessary to assist the Licensor in resolving the defect or fault, including sufficient information to enable the Licensor to recreate the defect or fault, or ii) terminate this License Agreement immediately by notice in writing to You and the Licensor will refund or if the Software Product has been purchased from an Authorised Reseller will procure that the Authorised Reseller shall refund; any of the fees paid as at the date of termination (less a reasonable sum in respect of Your use of the Software Product to the date of termination) on return of the Software Product and all copies thereof. The Licensor's obligation under this clause 8.2 is subject to Your compliance with clause 6.1(b).

9.3 You acknowledge that the Software Product has not been developed to meet Your individual requirements and that it is therefore Your responsibility to ensure that the facilities and functions of the Software Product as described in the Documentation meet Your requirements.

9.4 You acknowledge that the Software Product may not be free of bugs or errors and You agree that the existence of any minor errors shall not constitute a breach of this License Agreement.

9.5 The Licensor shall not be liable under the said warranty above if the Software Product fails to operate in accordance with the said warranty as a result of any modification, variation or addition to the Software Product not performed by the Licensor or caused by any abuse, corruption or incorrect use of the Software Product, including use of the Software Product with equipment or other software which is incompatible.

9.6 All other conditions, warranties or other terms which might have effect or be implied or incorporated into this License Agreement or any collateral contract, whether by statute, common law or otherwise, are hereby excluded, including, without limitation, the implied conditions, warranties or other terms as to satisfactory quality, fitness for purpose or the use of reasonable skill and care.

10. Exclusion of Liability

10.1 Nothing in this License Agreement shall limit or exclude the liability of either party for death or personal injury resulting from negligence, for fraud or for fraudulent misrepresentation.

10.2 Subject to clause 9.1, the Licensor shall have no liability for any losses or damages which may be suffered by You (or any person claiming under or through You), whether the same are suffered directly or indirectly or are immediate or consequential, and whether the same arise in contract, tort (including negligence) or otherwise howsoever, which fall within any of the following:

(a) loss of income;

(b) loss of business profits or contracts;

(c) business interruption;

(d) loss of the use of money or anticipated savings;

(e) loss of information;

(f) loss of opportunity, goodwill or reputation;

(g) loss of, damage to or corruption of data; or

(h) any indirect or consequential loss or damage of any kind howsoever arising and whether caused by tort (including negligence), breach of contract or otherwise;

provided that this clause 9.2 shall not prevent claims for loss of or damage to Your tangible property that fall within the terms of clause 8 or a claim for a refund of the License fees paid to the Licensor or the Authorised Reseller (as applicable) for the License of the Software Product.

10.3 Subject to clause 9.1 and clause 9.2, the Licensor's maximum aggregate liability under or in connection with this License Agreement, or any collateral contract, whether in contract, tort (including negligence) or otherwise, shall be limited to a sum equal to the fees paid by You to the Licensor or the Authorised Reseller (as applicable) for the License of the Software Product.

10.4 Subject to clause 9.1, clause 9.2 and clause 9.3, the Licensor's liability for infringement of third party intellectual property rights shall be limited to breaches of third party rights subsisting in the UK and USA.

10.5 This License Agreement sets out the full extent of the Licensor's obligations and liabilities in respect of the supply of the Software and Documentation. In particular, there are no conditions, warranties, representations or other terms, express or implied, that are binding on the Licensor except as specifically stated in this License Agreement. Any condition, warranty, representation or other term concerning the supply of the Software and Documentation which might otherwise be implied into, or incorporated in, this License Agreement, or any collateral contract, whether by statute, common law or otherwise, is hereby excluded to the fullest extent permitted by law.

11. Term and Termination

11.1 This License Agreement is effective for the period referred to in clause 2.1 or until otherwise terminated in accordance with that clause. The Licensor may terminate this License Agreement immediately by written notice to You if:

(a) save in respect of a Perpetual License, if You purchase the Software Product directly from the Licensor, You fail to pay any renewal or other fees due by You to the Licensor in respect of this License Agreement;

(b) You fail to comply with any provisions of this License Agreement;

(c) You commit a material or persistent breach of this License Agreement which You fail to remedy (if remediable) within 14 days after the service on You of written notice requiring You to do so;

(d) a petition for a bankruptcy order to be made against You has been presented to the court; or

(e) where You, as the Corporate Licensee, become insolvent or unable to pay its (the company's) debts, enters into liquidation, whether voluntary or compulsory (other than for reasons of bona fide amalgamation or reconstruction), passes a resolution for its winding-up, has a receiver or Administrator manager, trustee, liquidator or similar officer appointed over the whole or any part of its assets, makes any composition or arrangement with its creditors or takes or suffers any similar action in consequence of its debt, or becomes unable to pay its debts or ceases to trade.

11.2 In the event of termination in accordance with clause 10.1, You must immediately pay to the Licensor any sums due to the Licensor under the License Agreement and (at the Licensor's sole option) You must return, destroy or delete all copies of the Software Product from all storage media in Your control and, in the case of destruction, certify to the Licensor that You have done so.

12. Export

12.1 You will comply with all applicable laws, rules, and regulations governing export of goods and information, including the laws of the countries in which the Software Product was created. In particular, You will not export or re-export, directly or indirectly, separately or as a part of a system, the Software Product or other information relating thereto to any country for which an export License or other approval is required, without first obtaining such License or other approval.

13. General

13.1 You agree that the Licensor shall have the right, after supplying undertakings as to confidentiality, to audit any computer system on which the Software Product are installed in order to verify compliance with this License Agreement.

13.2 You agree that the Licensor may use Your company name in Licensor's customer lists and other promotional materials describing Your company as a customer or user of Licensor's software or services, as applicable, unless it receives written notice from You objecting to such use.

13.3 This License Agreement, its subject matter or its formation (including non-contractual disputes or claims) shall be governed by and construed in accordance with English law and submitted to the non-exclusive jurisdiction of the English courts.

13.4 This License Agreement constitutes the complete and exclusive statement of the agreement between the Licensor and You with respect to the subject matter of this License and supersedes all proposals, representations, understandings and prior agreements, whether oral or written, and all other communications between the parties relating to that subject matter except for those expressly referred to in this License Agreement.

13.5 Any clause in this License Agreement that is found to be invalid or unenforceable shall be deemed deleted and the remainder of this License Agreement shall not be affected by that deletion.

13.6 Failure or neglect by either party to exercise any of its rights or remedies under this License Agreement will not be construed as a waiver of that party's rights nor in any way affect the validity of the whole or part of this License Agreement nor prejudice that party's right to take subsequent action.

13.7 This License Agreement is personal to You and You may not assign, transfer, sub-contract or otherwise part with this License or any right or obligation under it without the Licensor's prior written consent.