Fatigue Module

User's Guide

COMSOL

Fatigue Module User's Guide

Contact Information

Visit the Contact COMSOL page at www.comsol.com/contact to submit general inquiries or search for an address and phone number. You can also visit the Worldwide Sales Offices page at www.comsol.com/contact/offices for address and contact information.

If you need to contact Support, an online request form is located on the COMSOL Access page at www.comsol.com/support/case. Other useful links include:

- Support Center: www.comsol.com/support
- Product Download: www.comsol.com/product-download
- Product Updates: www.comsol.com/support/updates
- COMSOL Blog: www.comsol.com/blogs
- Discussion Forum: www.comsol.com/forum
- Events: www.comsol.com/events
- COMSOL Video Gallery: www.comsol.com/videos
- Support Knowledge Base: www.comsol.com/support/knowledgebase

Part number: CM023201

Contents

Chapter 1: Introduction

About the Fatigue Module   **6**

Overview of the User's Guide   **12**

Chapter 2: Fatigue Modeling

Modeling with the Fatigue Module   **16**

Chapter 3: The Fatigue Interface

The Fatigue Interface   **28**

Domain, Boundary, Edge, Point, and Pair Nodes for the Fatigue

1

Introduction

This guide describes the Fatigue Module, an optional add-on package for COMSOL Multiphysics® designed to assist you to evaluate fatigue lives of components subjected to repetitive loading, where the stress and strain history has been computed using a physics interface in the Structural Mechanics Module, the Rotordynamics Module, or the Multibody Dynamics Module.

This chapter introduces you to the capabilities of this module. A summary of the physics interfaces and where you can find documentation and model examples is also included. The last section is a brief overview with links to each chapter in this guide.

In this chapter:

- About the Fatigue Module
- Overview of the User's Guide

About the Fatigue Module

These topics are included in this section:

- What Can the Fatigue Module Do?
- The Fatigue Module Interface
- Common Physics Interface and Feature Settings and Nodes
- Where Do I Access the Documentation and Application Libraries?

What Can the Fatigue Module Do?

The Fatigue Module is intended for fatigue analysis of structures. The term *fatigue* is used to describe the phenomenon where a component fails after repeated loadings and unloadings, even though the magnitude of each individual load is smaller than the ultimate stress of the material. The Fatigue Module is an add-on module to the Structural Mechanics Module.

Using the Fatigue interface, you can compute the risk of fatigue cracks occurring in a structure where the stress and strain state during a single load cycle has been computed. One of the Solid Mechanics; Shell; Plate; Membrane; Solid Rotor; Solid Rotor, Fixed Frame; or Multibody Dynamics interfaces is used along with the Fatigue interface to accomplish this analysis.

The stress results upon which the fatigue evaluation is based can come from different study types:

- Stationary (using load cases or parametric sweep)
- Time-dependent
- Frequency domain
- Random vibration

Eight basic types of evaluation can be performed:

- Stress-life analysis for *high-cycle fatigue* (HCF). The result is a lifetime prediction in terms of the number of cycles to fatigue.
- Strain-life analysis for *low-cycle fatigue* (LCF). The result is a lifetime prediction in terms of the number of cycles to fatigue.

- Stress-based analysis for *high-cycle fatigue* (HCF). The result is a usage factor, which tells you how close to the fatigue limit the load cycle is.

- Strain-based analysis for *low-cycle fatigue* (LCF). The result is a lifetime prediction in terms of the number of cycles to fatigue.

- Energy-based analysis when the dissipated energy controls crack formation and growth. The results are a lifetime prediction in terms of the number of cycles to fatigue and a dissipated fatigue energy density.

- Cumulative damage analysis for variable load fatigue. The results are usage factor, which tells you how close to the fatigue limit the load cycle is, counted stress cycles, showing the stress level distribution of the applied load, and the relative usage factor, showing the contribution to the overall fatigue usage from each stress level.

- Harmonic vibration analysis, based on the results from a frequency domain sweep. The result is a usage factor, which tells you how large fraction of the fatigue life that has been consumed by the cycles in the frequency sweep.

- Random vibration fatigue analysis, based on results from a random vibration analysis where the loading is represented by a PSD.

The Fatigue Module Interface

The Fatigue Module contains only one physics interface, Fatigue

PHYSICS INTERFACE	ICON	TAG	SPACE DIMENSION	AVAILABLE STUDY TYPE
🖥 **Structural Mechanics**				
Fatigue[1]	◇	ftg	3D, 2D, 2D axisymmetric	fatigue; stationary
[1] Requires the addition of the Structural Mechanics Module.				

🔍 • The Fatigue Study

Common Physics Interface and Feature Settings and Nodes

There are several common settings and sections available for the physics interfaces and feature nodes. Some of these sections also have similar settings or are implemented in the same way no matter the physics interface or feature being used. There are also some physics feature nodes that display in COMSOL Multiphysics.

In each module's documentation, only unique or extra information is included; standard information and procedures are centralized in the *COMSOL Multiphysics Reference Manual*.

> In the *COMSOL Multiphysics Reference Manual* see Table 2-4 for links to common sections and Table 2-5 to common feature nodes. You can also search for information: press F1 to open the **Help** window or Ctrl+F1 to open the **Documentation** window.

Where Do I Access the Documentation and Application Libraries?

A number of online resources have more information about COMSOL, including licensing and technical information. The electronic documentation, topic-based (or context-based) help, and the Application Libraries are all accessed through the COMSOL Desktop.

> If you are reading the documentation as a PDF file on your computer, the blue links do not work to open an application or content referenced in a different guide. However, if you are using the Help system in COMSOL Multiphysics, these links work to open other modules, application examples, and documentation sets.

THE DOCUMENTATION AND ONLINE HELP

The *COMSOL Multiphysics Reference Manual* describes the core physics interfaces and functionality included with the COMSOL Multiphysics license. This book also has instructions on how to use COMSOL Multiphysics and how to access the electronic Documentation and Help content.

Opening Topic-Based Help

The Help window is useful as it is connected to the features in the COMSOL Desktop. To learn more about a node in the Model Builder, or a window on the Desktop, click

to highlight a node or window, then press F1 to open the Help window, which then displays information about that feature (or click a node in the Model Builder followed by the **Help** button (?). This is called *topic-based* (or *context) help*.

Win	To open the **Help** window: • In the **Model Builder**, **Application Builder**, or **Physics Builder**, click a node or window and then press F1. • On any toolbar (for example, **Home**, **Definitions**, or **Geometry**), hover the mouse over a button (for example, **Add Physics** or **Build All**) and then press F1. • From the **File** menu, click **Help** (?). • In the upper-right corner of the COMSOL Desktop, click the **Help** (?) button.

Mac **Linux**	To open the **Help** window: • In the **Model Builder** or **Physics Builder**, click a node or window and then press F1. • On the main toolbar, click the **Help** (?) button. • From the main menu, select **Help>Help**.

Opening the Documentation Window

Win	To open the **Documentation** window: • Press Ctrl+F1. • From the **File** menu, select **Help>Documentation** (▮).

Mac **Linux**	To open the **Documentation** window: • Press Ctrl+F1. • On the main toolbar, click the **Documentation** (▮) button. • From the main menu, select **Help>Documentation**.

THE APPLICATION LIBRARIES WINDOW

Each model or application includes documentation with the theoretical background and step-by-step instructions to create a model or application. The models and applications are available in COMSOL Multiphysics as MPH-files that you can open for further investigation. You can use the step-by-step instructions and the actual models as templates for your own modeling. In most models, SI units are used to describe the relevant properties, parameters, and dimensions, but other unit systems are available.

Once the Application Libraries window is opened, you can search by name or browse under a module folder name. Click to view a summary of the model or application and its properties, including options to open it or its associated PDF document.

🔍	The Application Libraries Window in the *COMSOL Multiphysics Reference Manual*.

Opening the Application Libraries Window
To open the **Application Libraries** window (▥):

Win	From the **File** menu, select **Application Libraries**. To include the latest versions of model examples, from the **File>Help** menu, select (▦) **Update COMSOL Application Library**.

Mac Linux	Select **Application Libraries** from the main **File** or **Windows** menus. To include the latest versions of model examples, from the **Help** menu, select (▦) **Update COMSOL Application Library**.

CONTACTING COMSOL BY EMAIL

For general product information, contact COMSOL at info@comsol.com.

COMSOL ACCESS AND TECHNICAL SUPPORT

To receive technical support from COMSOL for the COMSOL products, please contact your local COMSOL representative or send your questions to support@comsol.com. An automatic notification and a case number will be sent to you

by email. You can also access technical support, software updates, license information, and other resources by registering for a COMSOL Access account.

COMSOL ONLINE RESOURCES

COMSOL website	www.comsol.com
Contact COMSOL	www.comsol.com/contact
COMSOL Access	www.comsol.com/access
Support Center	www.comsol.com/support
Product Download	www.comsol.com/product-download
Product Updates	www.comsol.com/support/updates
COMSOL Blog	www.comsol.com/blogs
Discussion Forum	www.comsol.com/forum
Events	www.comsol.com/events
COMSOL Application Gallery	www.comsol.com/models
COMSOL Video Gallery	www.comsol.com/video
Support Knowledge Base	www.comsol.com/support/knowledgebase

Overview of the User's Guide

The *Fatigue Module User's Guide* gets you started with fatigue evaluation using COMSOL Multiphysics. The information in this guide is specific to this module for fatigue analysis. Instructions how to use COMSOL in general are included with the *COMSOL Multiphysics Reference Manual*.

📄	For all other physics and theory, also see the *Structural Mechanics Module User's Guide* (or search online).

💡	As detailed in the section Where Do I Access the Documentation and Application Libraries? this information can also be searched from the COMSOL Multiphysics software **Help** menu.

TABLE OF CONTENTS AND INDEX

To help you navigate through this guide, see the Contents and Index.

MODELING

The Fatigue Modeling chapter discusses

- The Underlying Stress Analysis
- Adding Fatigue Interfaces
- The Fatigue Study
- Stress-Life Results
- Strain-Life Results
- Stress-Based Results
- Strain-Based Results
- Energy-Based Results
- Cumulative Damage Results
- Harmonic Vibration Results
- Matrix Histogram Plot

THE FATIGUE INTERFACE

The Fatigue Interface describes both the interface and its theory, which includes topics such as

- Defining Fatigue
- Phenomenology and Testing
- Definitions of Fatigue Quantities
- Design Strategies, Critical Plane Methods
- Energy Dissipation
- Stress-Life Models
- Strain-Life Models
- Stress-Based Fatigue Models
- Strain-Based Fatigue Models
- Energy-Based Fatigue Models
- Cumulative Damage Model
- Harmonic Vibration Model

2

Fatigue Modeling

The Fatigue Module has the tools to perform fatigue evaluation based on the stress and strain results computed in other physics interfaces in the Structural Mechanics Module or the Multibody Dynamics Module.

This chapter introduces you to Modeling with the Fatigue Module and describes how you add fatigue evaluation to a COMSOL Multiphysics® model.

Modeling with the Fatigue Module

In this section:

- The Underlying Stress Analysis
- Adding Fatigue Interfaces
- The Fatigue Study
- Stress-Life Results
- Strain-Life Results
- Stress-Based Results
- Strain-Based Results
- Energy-Based Results
- Cumulative Damage Results
- Harmonic Vibration Results
- Random Vibration Results
- Materials
- Matrix Histogram Plot

🔍	• The Fatigue Interface • Theory for the Fatigue Interface

❗	The links to nodes or theory described in the *Structural Mechanics Module User's Guide* (or any external guide) do not work in the PDF, only from the on line help in COMSOL Multiphysics.

The Underlying Stress Analysis

The Fatigue Module provides the ability to postprocess stress and strain results, which have been computed using one of these physics interfaces:

- Solid Mechanics
- Shell
- Plate

- Membrane
- Multibody Dynamics
- Solid Rotor
- Solid Rotor, Fixed Frame

Since fatigue is caused by the variation in stress and strain, it is necessary to have at least two sets of results from a stress analysis available for postprocessing. If the unloaded state is a part of the load history, then the trivial stress free state must be computed as well. Fatigue analysis can be done on results from the following types of studies:

- A Stationary study with at least two load cases
- A Stationary study using the parametric solver, which has at least two parameter values
- A Time Dependent study, either direct or modal, having results at least in two time instances
- A Frequency Domain study, either direct or modal.
- A Frequency Domain, Modal Reduced-Order Model study, with PSD loading.

In all cases, it is one full load cycle that is used for fatigue analysis. Often, several load cycles need to be computed before a steady state is reached. From that point, each consecutive cycle repeats the stress-strain response and such a cycle is representative for the fatigue life.

!	If the Fatigue interface has already been added before computing the stresses, then the physics interface should be disabled in the **Physics and Variables Selection** section of the study step for the stress analysis.

▥	For an example about how to set up an analysis having a shake-down to a stable cycle, see *Elastoplastic Low-Cycle Fatigue Analysis of Cylinder with a Hole*: Application Library path **Fatigue_Module/Strain_Based/ cylinder_with_hole_plastic**.

Adding Fatigue Interfaces

You can add one or more Fatigue interfaces to your model and each can contain a combination of subnodes for fatigue evaluation — Stress-Life, Strain-Life, Stress-Based, Strain-Based, Energy-Based, Cumulative Damage, or Harmonic Vibration. The fatigue evaluation can be done on any geometrical dimension, from the domain level to the point level, even if the stress analysis is performed on a higher geometrical dimension.

In most cases, fatigue cracks do appear at the surface of the structure. Cracks that are initiated inside the volume usually only occur when two objects are in contact, so that there are significant compressive stresses at the surface. For the fatigue evaluation you can thus in most cases select boundaries instead of domains. This approach results in significant performance gains.

The Fatigue Study

The **Fatigue** study ( ) is a dedicated study step for fatigue evaluation. It processes a load cycle and evaluates a fatigue criterion specified in The Fatigue Interface. Before a Fatigue study can be calculated, a load cycle must be simulated. This is done in a Stationary, Time Dependent, or Frequency Domain study step, where the structural

response to several loading events is analyzed. The following selection must be made in the **Fatigue** study step:

- In the **Study Settings** section, the setting for **Include geometric nonlinearity** should match the setting that was used in the simulation of the load cycle.

- In the **Values of variables not solved for** section, **Settings** should be **User controlled**.

- From the **Method** list, select **Solution**.

- From the **Study** list, select the study containing the load cycle to be used in the fatigue analysis.

- Depending on the type of study, you will get a different type of list to choose from; time steps, parameter values, or load cases. From the list, select the appropriate option for the load cycle definition. The option **All** evaluates fatigue using all load events of the study selected in the **Study** list. Use options **From list** or **Manual** to evaluate fatigue on only a part of a load history. With the option **From list**, select also the load cases, time steps, and so on, that define the load cycle. Alternatively, you can use the option **Manual** to select also the load cases that define the load cycle. Here the load cases have an ordering number with the first load case being one, second load case being two and so on.

> ⚠️ If large deformations, such as rigid body rotations, are modeled during the load cycle simulation, it is important to select **Include geometric nonlinearity** also in the **Fatigue** study step.

Fatigue analysis can also be performed using a **Stationary** study step. The evaluation options are then somewhat limited as compared to a **Fatigue** study step, and the following settings must be made for the study step:

- Make sure that only Fatigue interfaces are selected in the **Physics and Variables Selection** section.

- In the **Values of variables not solved for** section, **Settings** should be **User controlled**.

- From the **Method** list, select **Solution**.

- From the **Study** list, select the study containing the stress cycle to be used in the fatigue analysis.

- From the list of solution steps, select **All**. This means that you do not have the option to select only part of the load history for the fatigue analysis when using **Stationary** study step.

You can add as many Fatigue interfaces as needed and solve them in the same study.

- Applying Loads in the *Structural Mechanics Module User's Guide*

  In the *COMSOL Multiphysics Reference Manual:*

- Introduction to Solvers and Studies
- Fatigue and Stationary (studies)
- Using Load Cases
- Values of Dependent Variables
- Physics and Variables Selection

Stress-Life Results

In a stress-life evaluation, the result is a field named *interface*.ctf, which is the number of *cycles to failure*.

When plotting fatigue lives it is often useful to use a logarithmic scale.

When the S-N Curve Model, or the Basquin Model is evaluated, the computed number of cycles to failure is limited as follows: for small loads, when the number of cycles to failure becomes very large, the **Cycle cutoff** parameter, N_{cut}, provides an upper bound for the value of ctf.

- The Fatigue Interface
- Theory for the Fatigue Interface
- Results Analysis and Plots in the *COMSOL Multiphysics Reference Manual*

Strain-Life Results

In a strain-life evaluation, the result is a field named *interface*.ctf, which is the number of *cycles to failure*.

> 💡   When plotting fatigue lives it is often useful to use a logarithmic scale.

When the E-N curve Model, or the Combined Basquin and Coffin Manson Model is evaluated, the computed number of cycles to failure is limited as follows: for low loads, when the number of cycles to failure becomes very large, the **Cycle cutoff** parameter, N_{cut}, provides an upper bound for the value of ctf.

When the Coffin-Manson Model is evaluated, the accumulated effective inelastic strain range, *interface*.Dei, is also calculated.

> 🔍
> - The Fatigue Interface
> - Theory for the Fatigue Interface
> - Results Analysis and Plots in the *COMSOL Multiphysics Reference Manual*

Stress-Based Results

In a stress-based evaluation, the result is a field named *interface*.fus. This is a *fatigue usage factor*. If fus is below 1 then the stresses are below the fatigue limit as specified by the **Limit factor** *f* in the criterion. Because all the criteria are linear with respect to the stress and strain amplitude, fus can be considered the reciprocal of a safety factor.

> 🔍
> - The Fatigue Interface
> - Theory for the Fatigue Interface
> - Results Analysis and Plots in the *COMSOL Multiphysics Reference Manual*

Strain-Based Results

In a strain-based evaluation, the result is a field named *interface*.ctf, which is the number of *cycles to failure*.

	When plotting fatigue lives it is often useful to use a logarithmic scale.

When the Smith–Watson–Topper (SWT) Model, the Wang–Brown Model or the Fatemi–Socie Model is evaluated, the computed number of cycles to failure is limited. For low loads, when the number of cycles to failure becomes very large, the **Cycle cutoff** parameter, N_{cut}, provides an upper bound for the value of ctf. If the load is so high that the computed number of cycles is less than 1, the value of ctf is set to 0.5.

	• The Fatigue Interface
	• Theory for the Fatigue Interface
	• Results Analysis and Plots in the *COMSOL Multiphysics Reference Manual*

Energy-Based Results

In an energy-based evaluation, the result is a field named *interface*.ctf, which is the number of cycles to failure. This result is based on the dissipated energy density during one fatigue cycle. Therefore, the dissipated fatigue energy density range, *interface*.DWd, is calculated. With the Darveaux Model, the dissipated averaged fatigue energy density range, *interface*.DWa, is also available.

	When plotting fatigue lives it is often useful to use a logarithmic scale.

	• The Fatigue Interface
	• Theory for the Fatigue Interface
	• Results Analysis and Plots in the *COMSOL Multiphysics Reference Manual*

Cumulative Damage Results

In a cumulative damage evaluation, the result is a field named `interface.fus`. This is the fatigue usage factor, which describes the accumulated relative damage. Values of `fus` below 1 indicate that the stress history experienced does not lead to fatigue.

The **S-N curve** relates the applied stress to the permissible number of cycles to fatigue. For high loads, the stress that corresponds to 0.1 cycles is taken as the highest stress that can be experienced in the material. If the loading results in a stress which exceeds this limiting value, an error message displays stating that static failure is expected and no solution is provided. For low loads the **Cycle cutoff** parameter, N_{cut}, is used to obtain the lower stress limit, which can be seen as the endurance limit. If stresses are below this value they are considered not to be damaging and they do thus not contribute to `fus`.

Two additional variables describe the stress history in the point having the highest fatigue usage: `interface.csc` and `interface.rus`. The first variable defines the number of *counted stress cycles* in the load history within a certain stress amplitude and mean stress range. The second variable defines the *relative usage factor*, which is a ratio between fatigue usage of a stress cycle in a certain range and the fatigue usage factor.

In cases where the fatigue usage factor is 0 in the entire structure, the applied load is nondamaging. Since, in this case, no most damaging point exists, the figures with counted stress cycles and relative usage factor are empty. The figure with the fatigue usage factor shows a zero value, however.

The default is that `csc` and `rus` are calculated in the point that has the highest fatigue usage. In order to compute these results in a specific point, add a point in the specific coordinate and perform cumulative damage evaluation on this point.

- The Fatigue Interface
- Phenomenology and Testing
- Results Analysis and Plots in the *COMSOL Multiphysics Reference Manual*

Harmonic Vibration Results

In a harmonic vibration evaluation, the result is a field named *interface*.fus. This is the fatigue usage factor, which describes the accumulated relative damage. Values of fus below 1 indicate that the stress history experienced does not lead to fatigue.

The **S-N curve** relates the applied stress to the permissible number of cycles to fatigue. For high loads, the stress that corresponds to 0.1 cycles is taken as the highest stress that can be experienced in the material. If the loading results in a stress which exceeds this limiting value, an error message displays stating that static failure is expected and no solution is provided. For low loads the **Cycle cutoff** parameter, N_{cut}, is used to obtain the lower stress limit, which can be seen as the endurance limit. If stresses are below this value they are considered not to be damaging and they do thus not contribute to fus.

🔍	• The Fatigue Interface
	• Phenomenology and Testing
	• Results Analysis and Plots in the *COMSOL Multiphysics Reference Manual*

Random Vibration Results

In a random vibration evaluation, the results are fields named *interface*.dfus, *interface*.fus, and *interface*.life. These are, respectively, the fatigue usage factor per unit time, the fatigue usage factor, and the fatigue life. Values of fus below 1 indicate that the stresses experienced over the duration of the event do not lead to fatigue.

To evaluate the risk for fatigue, each stress level encountered during the random vibration event must be related to the permissible number of cycles to fatigue. In the case where the **S-N curve** fatigue model is used, a stress that corresponds to 0.1 cycles is taken as the highest stress that can be experienced in the material. If a stress during the random vibration event exceeds this limiting value, an error message displays stating that static failure is expected and no solution is provided. For low loads the **Cycle cutoff** parameter, N_{cut}, is used to obtain the lower stress limit, which can be seen as the endurance limit. If stresses are below this value they are considered not to be damaging and they do thus not contribute to fus. If a part of the structure experiences stresses that are low enough that no damage results, this corresponds to infinite life.

The **Life cutoff** parameter, L_{cut}, can be used to specify a finite approximation to infinite life. This way, results from computations of fatigue life can be more easily visualized.

@	• The Fatigue Interface • Phenomenology and Testing • Results Analysis and Plots in the *COMSOL Multiphysics Reference Manual*

Materials

@	The material property groups (including all associated properties) can be added to models from the **Material** page. See Materials in the *COMSOL Multiphysics Reference Manual*.

Matrix Histogram Plot

Use the **Matrix Histogram** plot in 2D ( ) when you have a precomputed matrix that you want to visualize as a 2D histogram. In a fatigue analysis use it to display the stress cycle distribution of the rainflow counting, for example. It shows the number of cycles with a certain stress amplitude and a certain mean stress in the load history. You can also use this plot to visualize how the damage is distributed between the different stress levels. If a large fraction of the total damage is caused by loads that occur only a few times, the statistical sample of the loads may be too small to reach any good conclusions.

@	Matrix Histogram and in Results Analysis and Plots the *COMSOL Multiphysics Reference Manual*

3

The Fatigue Interface

The Fatigue Module provides a collection of fatigue models for evaluating structural integrity of components subjected to repetitive loads. The fatigue cycle is computed in a structural simulation.

In this chapter:

- The Fatigue Interface
- Theory for the Fatigue Interface

The Fatigue Interface

The **Fatigue (ftg)** interface (⌃), found under the **Structural Mechanics** branch (⊟) when adding a physics interface, is used for fatigue evaluation based on stress, strain, or energy results computed by other structural mechanics interfaces. Using this physics interface, you can compute the risk of fatigue cracks to occur as an effect of repeated loading in a structure where the stress and strain state has been computed using one of the following physics interfaces — Solid Mechanics; Shell; Plate; Membrane; Multibody Dynamics (which requires the Multibody Dynamics Module); Solid Rotor; and Solid Rotor, Fixed Frame (the two latter require the Rotordynamics Module). The stress and strain state can be either the result of a single, deterministic load cycle, the response from a frequency sweep, or it can be a random (nondeterministic) load cycle, based on results from a random vibration analysis.

- A deterministic load cycle can be computed using a sequence of stationary load cases, a parametric solution, or a time-dependent solution. It is possible to perform rainflow analysis of stresses based on computed or measured load histories.

- In a frequency-domain study, the response at number of frequencies can be computed. Then, the fatigue damage in a vibration test where the excitation frequency is continually increased can be simulated.

- A random load cycle is computed in a random vibration analysis, where the structure has been subjected to power spectral density spectrum (PSD) type loading.

You can perform the following evaluations:

- Stress-Life analysis for high-cycle fatigue. The result is a lifetime prediction in terms of the number of cycles to fatigue.

- Strain-Life analysis for low-cycle fatigue. The result is a lifetime prediction in terms of the number of cycles to fatigue.

- Stress-Based analysis for high-cycle fatigue. The result is a usage factor, which tells you how close to the fatigue limit the load cycle is.

- Strain-Based analysis for low-cycle fatigue. The result is a lifetime prediction in terms of the number of cycles to fatigue.

- Energy-Based analysis when the dissipated energy controls crack formation and growth. The results are a lifetime prediction in terms of the number of cycles to failure and a dissipated fatigue energy density.

- Cumulative Damage analysis for variable load history. The results are a usage factor, a stress distribution, and a fatigue usage distribution. The last two parameters present an overview of the stress variation in the load history and how damaging different cycles are. The usage factor is a relation between an accumulated damage and a damage that causes fatigue. The distribution variables are visualized with the Matrix Histogram Plot in 2D.

- Harmonic Vibration analysis for frequency domain results. Typically, a frequency sweep is made, and it is assumed that the rate of change in frequency is small enough to make the response at any frequency to be considered as steady state. The result is a usage factor, which is a relation between an accumulated damage and a damage that causes fatigue.

- Random Vibration analysis for structures undergoing random vibrations. The results from this analysis type is a lifetime prediction in terms of the fatigue usage factor during a time period, the damage accumulated per unit time, or the total time to failure.

Deterministic Load Cycles

A fatigue analysis requires a load cycle simulation. In a fatigue study each load event of the load cycle is processed in search for the fatigue controlling quantities. Each fatigue model depends on different quantities and you should consider whether or not a certain fatigue evaluation is suitable for a given load cycle. Load cycles can be roughly be divided into the following cases:

- Proportional load
- Nonproportional load
- Variable amplitude load

Proportional loading is a situation where the orientation of the principal stresses and strains does not change during a load cycle. This can be described by a load event that increases to a certain level followed be a decrease to the initial load level. The Stress-Life Models, and the Strain-Life Models can be used for this type of loading. Also the Stress-Based Fatigue Models, the Strain-Based Fatigue Models, and the Cumulative Damage Model can be used although they are intended for more general types of loading.

In nonproportional loading the directions of principal stresses and strains vary during the load cycle and special methods for the evaluation of the fatigue controlling variables are required. The Stress-Based Fatigue Models, the Strain-Based Fatigue Models can be used for evaluation of this type of loading since they are Critical Plane

Methods and thus examine different orientations in space in order to find a critical plane, on which fatigue occurs. Also the Energy-Based Fatigue Models can be used since they take into account the entire stress and strain state at each load event.

In variable amplitude loading the entire load history, as opposed to a single representative load cycle, must be simulated. For certain cases it is possible to use only a part of the load history for the fatigue evaluation. This part must however be long enough to be representative. Note that peak loads with a low frequency can have a strong influence on the fatigue life. The Cumulative Damage Model can be used for variable amplitude load fatigue evaluation. This model is primarily intended for proportional loading, so if the critical points experience significant nonproportional loading, you need to review the results thoroughly.

Another type of variable amplitude loading occurs in forced vibration by a deterministic excitation frequency. In such an application a structure is first excited for a significant period of time at a given frequency, then the excitation frequency shifts to a new value where the structure continues to vibrate. In such a loading condition the concept of a load block, which is the dynamic load experience at each excitation frequency, as opposed to a load cycle, is more appropriate for load history definition.

Each excitation frequency introduces a repetitive stress history that changes when the excitation frequency changes. The stress response at two different excitation frequencies can be either proportional or nonproportional. For such applications, the Harmonic Vibration Model can be used to evaluate fatigue, since it computes fatigue damage at each load block and sums them to a cumulative value.
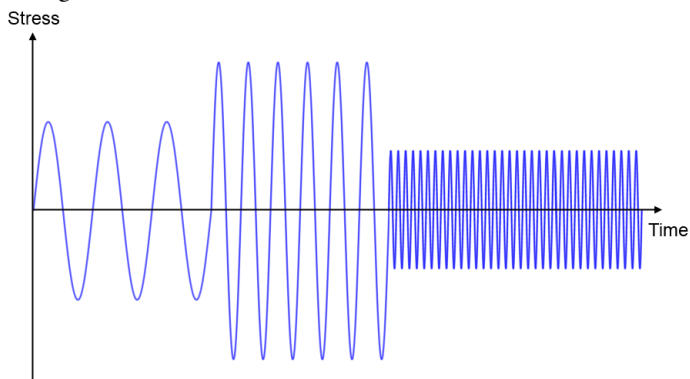
Figure 3-1: Forced vibration stress history.

Random Load Cycles

In random loading, the loads are defined using a statistical representation in the form of a power spectral density (PSD). This is a useful modeling approach in situations where the loads acting on a structure can no longer be expressed deterministically. The computed stress response PSD makes it possible to extract meaningful quantities for fatigue evaluation. Combined with a suitable description of the fatigue properties of the material, for example an S-N curve or the Basquin model, lifetime predictions can be made.

SETTINGS

The **Label** is the default physics interface name.

The **Name** is used primarily as a scope prefix for variables defined by the physics interface. Refer to such physics interface variables in expressions using the pattern *<name>.<variable_name>*. In order to distinguish between variables belonging to different physics interfaces, the name string must be unique. Only letters, numbers and underscores (_) are permitted in the **Name** field. The first character must be a letter.

The default **Name** (for the first physics interface in the model) is ftg.

Q
- Modeling with the Fatigue Module
- Theory for the Fatigue Interface

In the *Structural Mechanics Module User's Guide:*

- Solid Mechanics
- Shell and Plate
- Membrane

In the *Multibody Dynamics Module User's Guide:*

- Multibody Dynamics

Domain, Boundary, Edge, Point, and Pair Nodes for the Fatigue Interface

The Fatigue Interface has these domain, boundary, edge, point, and pair nodes available from the **Physics** ribbon toolbar (Windows users), **Physics** context menu (Mac or Linux users), or right-click to access the context menu (all users).

> In general, to add a node, go to the **Physics** toolbar, no matter what operating system you are using. Subnodes are available by clicking the parent node and selecting it from the **Attributes** menu.

These nodes contain fatigue models of the following types:

- Stress-Life
- Strain-Life
- Stress-Based
- Strain-Based
- Energy-Based
- Cumulative Damage
- Harmonic Vibration
- Random Vibration

> In the *COMSOL Multiphysics Reference Manual* see Table 2-4 for links to common sections and Table 2-5 to common feature nodes. You can also search for information: press F1 to open the **Help** window or Ctrl+F1 to open the **Documentation** window.

Stress-Life

Use the **Stress-Life** node to calculate fatigue life based on a fatigue life curve that relates load cycle stress amplitude to life. The **Stress-Life** feature can be applied to domains, boundaries, edges, or points.

> See Stress-Life Models for theory information.

MODEL INPUT

See Common Physics Interface and Feature Settings and Nodes for information about the Model Input.

SOLUTION FIELD

From the **Physics interface** list select the physics interface where the load cycle was simulated.

If you select a Shell interface, also select a **Through-thickness location** — **From physics interface**, **Top**, or **Bottom**. This determines from which position in the shell stresses or strains are taken for fatigue evaluation. When you select **From physics interface**, the local z-coordinate given in the **Default Through-thickness Result Location** section in the settings for the Shell interface is used.

When you select the physics interface where the load cycle was simulated, the integration order of this interface will be automatically used to define shape functions used by the fatigue features.

FATIGUE MODEL SELECTION

Select a **Fatigue limit** — **S-N curve**, **Basquin**, or **Approximate S-N curve**. This specifies the fatigue model to be evaluated. See S-N Curve Model, Basquin Model, and Approximate S-N Curve Model for details about these models.

Select how the stress is evaluated from the **Stress** σ list — **Principal stress**, **Signed von Mises (principal)**, or **Signed von Mises (hydrostatic)**.

When using **S-N curve**, also select a **Modification** — **None** or **Stress factor**. This determines whether a modification should be applied to the S-N curve or not.

FATIGUE MODEL PARAMETERS

S-N Curve

When **S-N curve** is selected as the **Fatigue limit** select a function from the list for the **S-N curve** $f_{\text{SN}}(N)$. The list contains all functions defined under **Global Definitions**. The **Interpolation** and **Analytic** functions are recommended for use with this model. If **Stress factor** is also chosen as the **Modification** then enter a value for k. The default is 1. The stress factor can be used for shifting the S-N curve. Typically, a value $k<1$ is used to compensate for effects like surface roughness.

Basquin

When **Basquin** is selected as the **Fatigue limit**, the material parameters **Fatigue strength coefficient** σ_f' and **Fatigue strength exponent** b by default take values **From material**. For **User defined** specify other values in the fields as needed.

Approximate S-N Curve

When **Approximate S-N curve** is selected as the **Fatigue limit**, the material parameters **Transition stress** S_t, **Transition life** N_t, **Endurance limit** S_e and **Endurance life** N_e by default take values **From material**. For **User defined** specify other values in the fields as needed.

MEAN STRESS CORRECTION

Fatigue life is generally affected by the mean stress. This mean stress is defined as the average of the minimum and maximum values of stress during a load cycle.

Select a **Method** — **None**, **Gerber**, **Goodman**, or **Soderberg**. With **None**, the mean stress does not influence the fatigue life prediction. When you select **Gerber** or **Goodman** as mean stress correction method, you should also specify an ultimate tensile strength σ_u. When you select the **Soderberg** method, you should also specify a yield stress σ_{ys}.

EVALUATION SETTINGS

When **S-N curve** or **Basquin** is selected as the **Fatigue limit**, enter a value or expression for the **Cycle cutoff** N_{cut}. The default is 1e10. This is the upper limit to the model prediction of the number of cycles to failure and is assigned as solution when the stress range is very small.

Strain-Life

Use the **Strain-Life** node to calculate fatigue life based on a fatigue life curve that relates load cycle strain amplitude to life. The **Strain-Life** feature can be applied to domains, boundaries, edges, or points.

MODEL INPUT

See Common Physics Interface and Feature Settings and Nodes for information about the Model Input.

SOLUTION FIELD

From the **Physics interface** list select the physics interface where the strain-life response to the fatigue load is simulated.

If you select a Shell interface, also select a **Through-thickness location** — **From physics interface**, **Top**, or **Bottom**. This determines from which position in the shell stresses or strains are taken for fatigue evaluation. When you select **From physics interface**, the local z-coordinate given in the **Default Through-thickness Result Location** section in the settings for the Shell interface is used.

FATIGUE MODEL SELECTION

Select a **Fatigue limit** — **E-N curve**, **Coffin-Manson**, or **Combined Basquin and Coffin-Manson**. This specifies the fatigue model to be evaluated. See E-N curve Model, Coffin-Manson Model, and Combined Basquin and Coffin Manson Model for details about these models.

If **Combined Basquin and Coffin-Manson** model is selected, also specify **Mean stress correction** — **None** or **Morrow**, which controls how the effect of the mean stress should be evaluated.

If **Coffin-Manson** model is selected, also specify a **Strain type** — **None** or **User Defined**, which is used to specify strain type that controls fatigue. The list of the **Strain type** is populated with additional options when nonlinear materials are used. If the material has been modeled with **Plasticity** then **Equivalent plastic strain** is selectable. If the material has been modeled with **Creep** option also **Equivalent creep strain** is selectable. If the material has been modeled with **Viscoplasticity** option also **Equivalent viscoplastic strain** is selectable.

> ⚠️ The **Coffin-Manson** model is based on inelastic strain which is occurring in inelastic materials and can be computed with the Nonlinear Structural Materials Module.

If **Coffin-Manson** model is selected with the **User Defined** option as **Strain type**, two additional input are required. First, you need to specify the **Inelastic strain** ε_i using the pattern `<name>.<variable_name>`, and second, you need to provide the integration order that was used to compute this strain.

FATIGUE MODEL PARAMETERS

E-N Curve
When **E-N curve** is selected as the **Fatigue limit**, select a function from the list for the **E-N curve** $f_{EN}(N)$. The list contains all functions defined under **Global Definitions**. The **Interpolation** and **Analytic** functions are recommended for use with the model.

Coffin-Manson

When **Coffin-Manson** is selected as the **Fatigue limit**, the material parameters **Fatigue ductility coefficient** ε_f' and **Fatigue ductility exponent** c by default take values **From material**. For **User defined** specify other values in the fields as needed.

Combined Basquin and Coffin-Manson

When **Combined Basquin and Coffin-Manson** is selected as the **Fatigue limit**, the material parameters **Fatigue strength coefficient** σ_f', **Fatigue strength exponent** b, **Fatigue ductility coefficient** ε_f' and **Fatigue ductility exponent** c by default take values **From material**. For **User defined** specify other values in the fields as needed.

EVALUATION SETTINGS

Enter a value for the **Cycle cutoff** N_{cut}. The default is 1e10. This is the upper limit to the model prediction of the number of cycles to failure and is assigned as solution when the strain range is very small.

| | See Strain-Life Models for theory information. |

Strain-Based

Use the **Strain-Based** node to define the fatigue model selection and parameters for strain-based fatigue evaluation. The **Strain-Based** feature can be applied to domains, boundaries, edges, or points.

MODEL INPUT

See Common Physics Interface and Feature Settings and Nodes for information about the Model Input.

SOLUTION FIELD

From the **Physics interface** list select the physics interface where the load cycle is simulated.

If you select a Shell interface, also select a **Through-thickness location** — **From physics interface**, **Top**, or **Bottom**. This determines from which position in the shell stresses or strains are taken for fatigue evaluation. When you select **From physics interface**, the local z-coordinate given in the **Default Through-thickness Result Location** section in the settings for the Shell interface is used.

Select a **Criterion** — **Smith-Watson-Topper (SWT)**, **Wang-Brown**, or **Fatemi-Socie**. This specifies the fatigue model to be evaluated. See Smith–Watson–Topper (SWT) Model, Wang–Brown Model, and Fatemi–Socie Model for details about these models. Also see Elastic Notch Approximation.

Select a **Solution type** — **Full elastoplastic solution** or **Elastic solution with notch assumption**. This defines how the load cycle is processed.

If **Wang-Brown** model is selected, also specify **Mean stress correction** — **None** or **Morrow**, which controls how the effect of the mean stress should be evaluated.

FATIGUE MODEL PARAMETERS

There are several sets of fatigue model parameters, which depend on the selected criterion and options. The combinations are listed in Table 3-1 and Table 3-2. As a default, the parameters take values **From material**. For **User defined** enter other values or expressions in the fields as needed.

If the **Criterion** selected is **Fatemi-Socie**, specify how the fatigue model parameters are provided. Select from the **Shear fatigue data** list — **Implied from tensile data** or **Explicitly given**. For **Explicitly given** enter the actual torsional data.

TABLE 3-1: FATIGUE DATA

PARAMETER AND DEFAULT UNIT	APPLICABILITY
Fatigue strength coefficient σ_f (SI unit: Pa)	SWT, Wang-Brown, Fatemie-Socie (implied)
Fatigue strength exponent b (dimensionless)	SWT, Wang-Brown, Fatemie-Socie (implied)
Fatigue ductility coefficient ε_f (dimensionless)	SWT, Wang-Brown, Fatemie-Socie (implied)
Fatigue ductility exponent c (dimensionless)	SWT, Wang-Brown, Fatemie-Socie (implied)
Shear fatigue strength coefficient τ_f (SI unit: Pa)	Fatemie-Socie (explicit)
Shear fatigue strength exponent b_γ (dimensionless)	Fatemie-Socie (explicit)
Shear fatigue ductility coefficient γ_f (dimensionless)	Fatemie-Socie (explicit)
Shear fatigue ductility exponent c_γ (dimensionless)	Fatemie-Socie (explicit)
Normal strain sensitivity coefficient S (dimensionless)	Wang-Brown
Normal stress sensitivity coefficient k (dimensionless)	Fatemie-Socie

TABLE 3-2: ELASTIC AND PLASTIC CONSTITUTIVE DATA

PARAMETER AND DEFAULT UNIT	APPLICABILITY
Young's modulus E (SI unit: Pa)	SWT, Wang-Brown
Shear modulus G (SI unit: Pa)	Fatemie-Socie
Initial yield stress σ_{ys0} (SI unit: Pa)	Full elastoplastic solution
Poisson's ratio ν (dimensionless)	Elastic with notch assumption
Cyclic hardening coefficient K' (SI unit: Pa)	Elastic with notch assumption
Cyclic hardening exponent n' (dimensionless)	Elastic with notch assumption

EVALUATION SETTINGS

- Enter a **Cycle cutoff** N_{cut}. The default is 10^8. This is the upper limit to the model prediction of the number of cycles to failure and is assigned as solution when the strain range is very small.

- Enter a **Search resolution** Q. The default is 11. This parameter determines the number of evaluation points used in the search for the critical plane and thus it controls the computational time.

- For **Wang-Brown** or **Fatemi-Socie**, also select a method for the shear range evaluation from the **Shear range search method** list — **Circumscribing circle** or **Maximum distance**.

- Strain-Based Fatigue Models
- Selection of Search Directions
- Finding Maximum Shear Range
- Theory for the Fatigue Interface

Stress-Based

Use the **Stress-Based** node to define the fatigue model selection and parameters for stress-based fatigue evaluation. The **Stress-Based** feature can be applied to domains, boundaries, edges, or points.

MODEL INPUT

See Common Physics Interface and Feature Settings and Nodes for information about the Model Input.

SOLUTION FIELD

From the **Physics interface** list select the physics interface where the load cycle is simulated.

If you select a Shell interface, also select a **Through-thickness location** — **From physics interface**, **Top**, or **Bottom**. This determines from which position in the shell stresses or strains are taken for fatigue evaluation. When you select **From physics interface**, the local z-coordinate given in the **Default Through-thickness Result Location** section in the settings for the Shell interface is used.

FATIGUE MODEL SELECTION

Select a **Criterion** — **Findley**, **Matake**, **Normal Stress**, or **Dang Van**. This specifies the fatigue model to be evaluated. See Findley Criterion, Matake Criterion, Maximum Normal Stress Criterion, and Dang Van for details about these models.

FATIGUE MODEL PARAMETERS

If **Findley** or **Matake** is selected as the **Criterion**, then under **Fatigue parameters** the **Normal stress sensitivity coefficient** k as a default takes values **From material**. For **User defined** enter other values or expressions in the field as needed.

For **Criterion** options **Findley** or **Matake** and **Normal stress**, the **Limit factor** f also takes values **From material**. For **User defined** enter other values or expressions in the field as needed.

If **Dang Van** is selected as the **Criterion**, the **Hydrostatic stress sensitivity coefficient** α_{DV} and the **Limit factor** β_{DV} takes values **From material**. For **User defined** enter other values or expressions in the field as needed.

EVALUATION SETTINGS

Under **Critical plane settings**, enter a **Search resolution** Q. The default is 11. This parameter determines the number of evaluation points used in the search for the critical plane and it controls the computational time.

If **Findley** or **Matake** is selected as the **Criterion**, also select a method for the **Shear range search method** — **Circumscribing circle** or **Maximum distance**.

🔍
- Stress-Based Fatigue Models
- Selection of Search Directions
- Finding Maximum Shear Range
- Theory for the Fatigue Interface

Energy-Based

Use the **Energy-Based** node to define the fatigue model selection and parameters for energy-based fatigue evaluation. The **Energy-Based** feature can be applied to domains, boundaries, edges, or points.

MODEL INPUT

See Common Physics Interface and Feature Settings and Nodes for information about the Model Input.

SOLUTION FIELD

From the **Physics interface** list select the physics interface where the load cycle is simulated.

FATIGUE MODEL SELECTION

Select a **Criterion** — **Morrow** or **Darveaux**. This specifies the fatigue model to be evaluated. See Morrow Model and Darveaux Model for details about these models.

📝
The **Darveaux** criterion is only available at the domain level in 3D, 2D axisymmetric, and 2D components.

Select an **Energy type** — **None** or **User defined**. This specifies the energy quantity that controls the fatigue life. The list of the **Energy type** is populated with additional options when nonlinear materials are used. If the material has been modeled with **Plasticity**, also **Plastic dissipation density** is selectable. If the material has been modeled with **Creep**, also **Creep dissipation density** is selectable. If the material has been modeled with **Viscoplasticity**, also **Viscoplastic dissipation density** is selectable. If the material has been

modeled with two of the above material types than also **Total dissipation density** is selectable.

> ⚠️ The results cannot be obtained unless **Calculate dissipated energy** is selected under **Energy Dissipation** in the Linear Elastic Material, Nonlinear Elastic Material, or Hyperelastic Material node. This is because the dissipated energy depends on the load and strain history and must be calculated at the same time as the simulation of the load cycle. This adds additional degrees of freedom to the model. See Energy Dissipation in the *Structural Mechanics Module User's Guide*.

> ❗ The energy-based models depend on the dissipated energy. Energy dissipation means that the energy is consumed by the material and cannot be restored. This behavior is exhibited by inelastic materials and can be modeled with the Nonlinear Structural Materials Module or Geomechanics Module.

> 🔍 • Energy Dissipation
> • Energy-Based Fatigue Models

If the **User defined** option has been selected as the **Energy type**, two additional inputs are required. First, you need to specify the **Energy dissipation density** W_d. You can use any combination of energy quantities from the physics interface used for the stress analysis. Use the pattern `<interface>.<variable_name>`. Second, you need to provide the integration order that was used to compute this energy dissipation density.

EVALUATION SETTINGS

This section is available if **Darveaux** is selected as the **Criterion**. Select a **Volume average method** — **Individual domains** or **Entire selection**. This parameter determines how the volume average of the dissipated energy is calculated in the selected volume.

- For **Individual domains** one value is calculated for each geometrical domain.
- For **Entire selection** the average is calculated over the entire selection.

If the selection consists of only one geometrical domain, then both options give the same result. See Individual Domains versus Entire Selection for details.

FATIGUE MODEL PARAMETERS

If **Morrow** is selected as the **Criterion**, then the **Fatigue energy coefficient** W_f and **Fatigue energy exponent** m by default take values **From material**. For **User defined** enter other values or expressions in the fields as needed.

If **Darveaux** is selected as the **Criterion**, enter a value for the **Crack size** a. By default, it is set to 0. By default, the following parameters, **Crack initiation energy coefficient** K_1, **Crack initiation energy exponent** k_2, **Crack propagation energy coefficient** K_3, **Crack propagation energy exponent** k_4, and **Reference energy density** W_{ref}, take values **From material**. For **User defined** enter other values or expressions in the fields as needed for these parameters.

Cumulative Damage

Use the **Cumulative Damage** node to define the analysis type, solution field, cycle counting parameters, damage model parameters, and generalized load. The **Cumulative Damage** feature can be applied to domains, boundaries, edges, or points. If points are selected, results for Counted stress cycles and Relative usage factor are created for each point.

⊕ | Cumulative Damage Model

SOLUTION FIELD

From the **Physics interface** list select the physics interface where the stress-strain response to the fatigue load is simulated.

If you select a Shell interface, also select a **Through-thickness location** — **From physics interface**, **Top**, or **Bottom**. This determines from which position in the shell stresses or strains are taken for fatigue evaluation. When you select **From physics interface**, the local z-coordinate given in the **Default Through-thickness Result Location** section in the settings for the Shell interface is used.

ANALYSIS

To define how the load cycle is evaluated, select the analysis **Type** — **Solution history** or **Generalized loads**.

The **Solution history** processes the stress cycle, load step by load step. The **Generalized loads** processes a few basic load cases, and a list of weighting coefficients, and can thus handle very long load cycles.

⑦	Generalized Load

CYCLE COUNTING PARAMETERS

Select how the stress is evaluated from the **Stress** σ list — **Principal stress**, **Signed von Mises (principal)**, or **Signed von Mises (hydrostatic)**. See Cycle Counting Options for details.

Under **Discretization**, enter values or expressions for the **Number of mean value bins** N_{m} and **Number of range bins** N_{r}. Both are dimensionless numbers with the default value 10. These parameters define the number of discrete points used to define the load cycle distribution and are thus strongly connected to the computational time.

DAMAGE MODEL PARAMETERS

Select an **S-N curve type** — **S-N curve with R-value dependence** $\sigma_{\mathrm{SN}}(R, N)$, **S-N curve with mean stress dependence** $\sigma_{\mathrm{SN}}(\sigma_{\mathrm{m}}, N)$, or **S-N curve for amplitude stress** $\sigma_{\mathrm{SN}}(N)$. Select the S-N curve from the list, which contains all functions defined under **Global Definitions**. See Damage Estimation for details. For **S-N curve for amplitude stress**, the S-N curve can be amended with a mean stress correction. Under **Mean Stress Correction**, specify the mean stress correction **Method** — **None**, **Gerber**, **Goodman**, or **Soderberg**.,

⑦	Global Definitions, Geometry, Mesh, and Materials and Functions in the *COMSOL Multiphysics Reference Manual*

Enter the **Number of block repetitions** m. The default is 1. A block is a sequence of load events that define a representative fatigue load cycle.

Under **Evaluation settings** enter a value for the **Cycle cutoff** N_{cut}. The default is 10^{10}. Based on the **S-N curve**, the **Cycle cutoff** implicitly provides the highest stress value that gives no contribution to the fatigue usage factor. From the evaluation point of view this stress can be seen as the endurance limit.

This section is available when **Generalized loads** is selected under **Analysis**.

For the load history, enter values or expressions for the **Initial step** s_i and the **Final step** s_f. The default is 1 for both. Under **Generalized load history**, specify load history functions for all generalized loads. The number of generalized load functions must equal the number of load cases in the study specified as the **Physics interface** under **Solution Field**.

> ⚠ The stress response to the generalized unit load is calculated in the load cases of the **Solution Field**. The order of load histories in the parameter **Generalized load history** must be given in the same order as load cases specified in the calculation of the structural response.

Harmonic Vibration

Use the **Harmonic Vibration** node to define the solution field, load history definition and fatigue evaluation parameters for harmonic vibration fatigue evaluation. The **Harmonic Vibration** feature can be applied to domains, boundaries, edges, or points.

MODEL INPUT

See Common Physics Interface and Feature Settings and Nodes for information about the Model Input.

SOLUTION FIELD

From the **Physics interface** list select the physics interface where the stress response to the forced vibration is simulated.

If you select a Shell interface, also select a **Through-thickness location** — **From physics interface**, **Top**, or **Bottom**. This determines from which position in the shell stresses or strains are taken for fatigue evaluation. When you select **From physics interface**, the local z-coordinate given in the **Default Through-thickness Result Location** section in the settings for the Shell interface is used.

LOAD HISTORY DEFINITION

To define how the frequency history is prescribed, select the **Frequency history** — **Time duration**, **Number of cycles**, **Linear frequency sweep** or **Logarithmic frequency sweep**.

If **Time duration** is selected, also specify **Time history**. The **Time history** entries define the duration of the excitation at each excitation frequency. The number of excitation times

in the table must correspond to the number of excitation frequencies selected in the settings for the **Fatigue** or **Stationary** study step.

If **Number of cycles** is selected, also specify **Cycle history**. The **Cycle history** entries define number of cycles experienced at each excitation frequency. The number of cycle entries in the table must correspond to the number of excitation frequencies selected in the settings for the **Fatigue** or **Stationary** study step.

If **Linear frequency sweep** is selected, also specify the **Rate of change**, C, which is the rate of frequency change per unit time

$$C = \frac{df}{dt}$$

If **Logarithmic frequency sweep** is selected, also specify the **Logarithmic rate of change**, C_l, which is the frequency doubling rate

$$\frac{f + \Delta f}{f} = 2^{C \Delta t}$$

⚠️ It is assumed that the rate of change is slow enough so that transient effects are not important. At each frequency, the structure is considered as being in a steady state.

FATIGUE EVALUATION PARAMETERS

Under **Direction** select how the stress is evaluated from the **Stress** σ list — **Directional stress**, or **Signed von Mises**.

If **Directional stress** is used, also select how the direction is evaluated from the **Direction definition** list — **Direction vector**, or a predefined coordinate system. The list contains all coordinates systems that have been defined under the **Definitions** node in the model tree, except those of the **Scaling System** type.

If **Direction vector** is selected, also define the **Direction vector**, $\mathbf{n}_\sigma$. The default direction is along the X-axis. The normal stress in this direction is used for the fatigue evaluation.

If a predefined coordinate system is selected, also specify its **Direction axis** — **First**, **Second**, or **Third**. The normal stress along this axis is used for the fatigue evaluation.

Under **Material** select the **S-N curve** $\sigma_a(R,N)$ from the list, which contains all functions defined under **Global Definitions**. Also enter a value for the **Cycle cutoff** N_{cut}. The default is 10^{10}. Based on the **S-N curve**, the **Cycle cutoff** implicitly provides the highest stress

value that gives no contribution to the fatigue usage factor. From the evaluation point of view this stress can be seen as the endurance limit.

🔍	Harmonic Vibration Model

Random Vibration

Use the **Random Vibration** node to define the random vibration model, the solution field, the fatigue stress measure, and fatigue evaluation parameters for random vibration fatigue evaluation. The **Random Vibration** feature can be applied to domains, boundaries, edges, or points.

MODEL INPUT

See Common Physics Interface and Feature Settings and Nodes for information about the Model Input.

MOMENT COMPUTATION

From the **Random vibration model** list, select the random vibration model containing the reduced-order model vibration results. From the **Stress method** list, select the evaluation method for stresses. For **Equivalent von Mises stress (EVMS)**, select the physics interface that was used to compute stresses, from the **Physics interface for stresses** list. If you select a Shell interface, also select a **Through-thickness location** — **From physics interface**, **Top**, or **Bottom**. This determines from which position in the shell stresses or strains are taken for fatigue evaluation. When you select **From physics interface**, the local z-coordinate given in the **Default Through-thickness Result Location** section in the settings for the Shell interface is used.

⚠	Note that if **From physics interface** is selected from the **Through-thickness location list**, and the z-coordinate is changed, the Model Reduction study step must be recomputed.

If **User defined** is selected from the **Stress method** list, a user defined stress expression must be entered. From the **Shape function integration order** list, select the Gauss point order matching the user defined stress expression.

> ⚠️ The user defined stress expression must be linear with respect to displacements. Therefore, expressions involving principal stresses, and so on, are not allowed.

Select the **Lower frequency** f_L, the **Upper frequency** f_U, and the **Number of frequencies** n_f to define the integral limits and discretization in the moment computations.

CYCLE COUNTING MODEL

Select a cycle counting model — **Dirlik** or **Bendat**. From the **Upper integration limit** list, select **Automatic** or **User defined**. For **User defined**, enter a finite value for the upper integration limit S_u of the integral for the fatigue usage factor. For **Duration**, enter the time t_d that represents the duration of the random vibration.

FATIGUE MODEL SELECTION

From the **Criterion** list, select **S-N Curve**, **Basquin**, or **Approximate S-N Curve**.

FATIGUE MODEL PARAMETERS

Depending on the selected fatigue model, you have to enter the corresponding fatigue model parameters. For **S-N Curve**, select an S-N curve $f_\text{SN}(N)$ from the **S-N curve** list, which contains all functions defined under **Global Definitions**. Enter a **Cycle cutoff** N_cut. Select the **Spatially varying S-N data** check box if the S-N curve varies with position in the structure. Specify the **Life cutoff** parameter, L_cut

> ℹ️ In many situations, it is reasonable to model the S-N curve data as constant with respect to position in the structure. This will significantly reduce the computation time of the random vibration fatigue evaluation.

For **Basquin**, enter the **Fatigue strength coefficient** σ'_f, the **Fatigue strength exponent** b, the **Cycle cutoff** N_cut, and the **Life cutoff** parameter, L_cut. For **Approximate S-N curve**, enter the **Transition stress** S_t, the **Transition life** N_t, the **Endurance limit** S_e, the **Endurance life** N_e, and the **Life cutoff** parameter, L_cut.

Theory for the Fatigue Interface

In this section:

Defining Fatigue

The term *fatigue* is used to describe the phenomenon where a component fails after repeated loadings and unloadings, even though the magnitude of each individual load may be much smaller than the ultimate stress of the material. The term was coined in the middle of the nineteenth century when several railroad accidents drew attention to the subject. Even today, the vast majority of all structural failures are attributed to fatigue, which means that designing against fatigue is of utmost importance.

When a fatigue failure occurs, the process can be divided into three distinct stages:

Stage I During a large number of *load cycles* (repeated loadings and unloadings), damage is accumulated on the micromechanical scale and after some time a crack of macroscopic size is formed.

Stage 2 The macroscopic crack grows for each new load cycle.

Stage 3 When the crack has reached a certain size, the remaining material can no longer sustain the peak load and the component fails.

Usually, the details of the last two stages are considered within the topic of *fracture mechanics*; the term fatigue applies mainly to Stage 1. Because the largest part of the life of the component is spent before it is possible to observe a macroscopic crack, most designs aim to avoid ever getting such a crack. There is, however, some overlap between the disciplines and the measured number of cycles to fatigue often includes Stages 2 and 3. In the Fatigue Module the Darveaux model explicitly addresses stages 1 and 2.

Phenomenology and Testing

The underlying reason for fatigue must be investigated on the micromechanical scale where materials are not homogeneous. In an alloy there are grains and the grain boundaries cause stress concentrations. In a casting there might even be pores that are formed during the solidification. Therefore, on a local scale, the strains might be much larger than the macroscopic average values and dislocations within the crystals are activated.

STATISTICAL EFFECTS

Because the locations of these micromechanical irregularities are more or less randomly distributed, there is a large scatter in the number of load cycles that a certain type of component can be subjected to, even if the external load is well-defined. This scatter makes it necessary to test many specimens when collecting fatigue data. Two examples of these statistical effects are:

- If two sets of bars with different diameters are tested in tension with the same nominal stress, the larger one appears to have a shorter lifetime. The reason is that, within a larger volume of material, the risk of finding a microscopic defect of a certain size is larger.

- If the same type of bar is tested in both tension and bending giving the same peak stress, the one tested in bending appears to have a longer lifetime. During bending, only a small volume of the material is subjected to the highest stress.

These types of effects are not included in the Fatigue Module, so you should choose your material data with this in mind.

A pioneer in the field of fatigue was the German engineer August Wöhler who presented a classical work in 1870. His name is used in diagrams showing *stress amplitude* versus number of cycles to fatigue. These are called *Wöhler curves* or *S-N curves* (Figure 3-2).
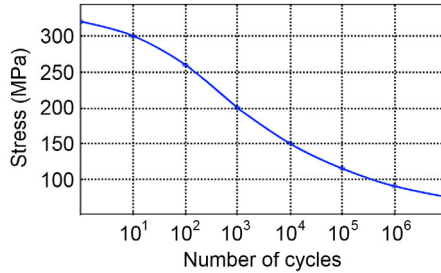

Figure 3-2: Example of an S-N curve showing the material data for brittle aluminum with ultimate tensile strength of 320 MPa.

An S-N curve is usually obtained by testing different stress levels and recording the number of cycles to failure, N. Several specimens are tested at each level so that average and scatter can be computed, giving one point on the curve.

Because the S-N curve gives the level at which a certain percentage (often 50%) of a population can be expected to fail, that value cannot always be directly used for design.

Knowledge of the scatter in terms of the standard deviation is necessary to transform the given data to another, acceptable, level. For a certain number of cycles, a certain stress level is then connected to a probability of failure. The acceptable probability (and thus stress level) in a design can differ between, for example, a passenger aircraft and a lawn mower.

There are also different factors that can affect the S-N curves:

Environmental effects: A corrosive environment is negative for the fatigue life. A material usually having a fatigue limit might not have one in a corrosive environment.

Surface finish: Most data are obtained from polished specimens. Because microscopic irregularities are involved in the formation of fatigue cracks, a rough surface decreases fatigue life.

Statistical size effects: As described in Statistical Effects.

Residual stresses from manufacturing: This is sometimes used intentionally, for example by shot peening, which results in beneficial compressive stresses in the surface of the component.

LOW-CYCLE AND HIGH-CYCLE FATIGUE

There are two different regimes of the fatigue phenomena: low-cycle fatigue (LCF) and high-cycle fatigue (HCF). The border between the two is not exact, but usually a cycle count to fatigue larger than 10^4 is considered a "high cycle". A characteristic of LCF is that significant plastic strains occur on the macroscopic scale during each load cycle.

For some materials (for example, many steel and titanium alloys), a lowest stress level exists, below which fatigue does not occur irrespective of the number of load cycles. This level is called the *fatigue limit* or *endurance limit*. Those limits are often of the order of half the ultimate tensile strength. Many materials such as aluminum and copper do not appear to have a fatigue limit.

Even for a material without a fatigue limit, such values are sometimes given. These actually represent the value of the S-N curve at a certain large number of cycles, for example, 10^8.

	Low-cycle fatigue is sometimes referred to as "strain-based" because the relevant parameter for describing LCF is strain rather than stress.

	Physically it is the strain that does the damage in HCF as well. Because HCF occurs in the elastic regime, it is possible to use either stress or strain as the parameter. The use of stress has historical and practical reasons and therefore high-cycle fatigue is often referred to as "stress-based."

Definitions of Fatigue Quantities

In addition to the stress amplitude, the mean stress is also important when fatigue cracks appear. A tensile mean stress decreases the fatigue life, while a compressive stress increases it. If σ_{max} is the maximum stress over the cycle and σ_{min} is the minimum stress, the following definitions are used:

Stress amplitude:

$$\sigma_a = \frac{\sigma_{\max} - \sigma_{\min}}{2}$$

Stress range:

$$\Delta\sigma = \sigma_{\max} - \sigma_{\min}$$

Mean stress:

$$\sigma_m = \frac{\sigma_{\max} + \sigma_{\min}}{2}$$

R-value:

$$R = \frac{\sigma_{\min}}{\sigma_{\max}}$$

The R-value is the most commonly used parameter to describe the mean stress level.

The most common fatigue test is where the loading is fully reversed. The mean stress is then zero and $R = -1$. The second fundamental test is the pulsating test, where the load varies between zero and a maximum value, giving $R = 0$.

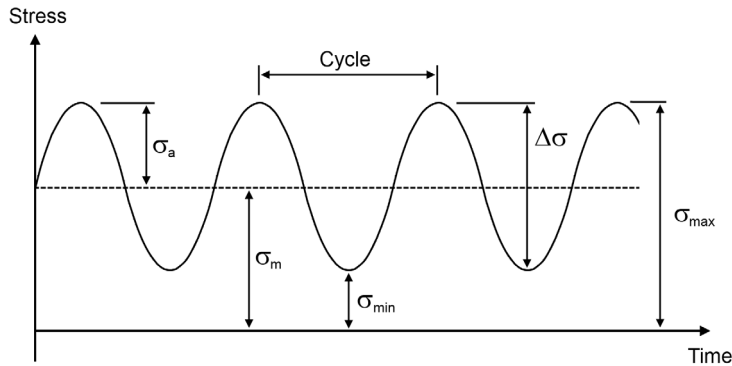
Figure 3-3: Example of cyclic loading.

	Fatigue data, like endurance limits and S-N curves can be defined either in terms of the stress amplitude or the maximum stress. Within COMSOL, it is always the amplitude which is used.

One of the simplest methods for fatigue evaluation is a relation between a fatigue controlling quantity, ξ, and the fatigue life, N, see Figure 3-4. This relation is the fatigue life curve. In most applications the fatigue controlling quantity is a stress amplitude or a strain amplitude.
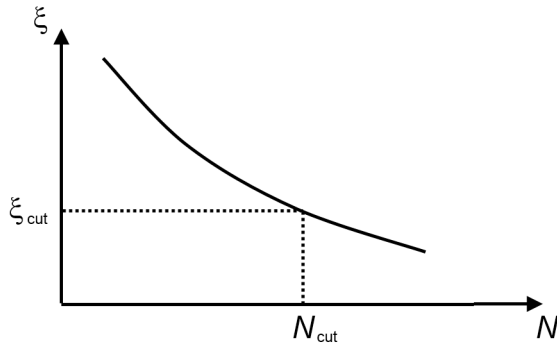
Figure 3-4: Fatigue life curve

In COMSOL the fatigue life curve is used in the following models: Stress-Life Models, Strain-Life Models, Strain-Based Fatigue Models, Cumulative Damage Model, and Harmonic Vibration Model. The fatigue life curve goes asymptotically toward infinite life for low values of the fatigue controlling quantity. The objective of a fatigue analysis is to find the critical life that is found for the largest value of the fatigue controlling quantity. Since the spread in the results can be very large, a cycle cutoff, N_{cut}, controls how far on the life curve results are evaluated. When the fatigue controlling quantity is low, $\xi < \xi_{cut}$, the fatigue life is set to the cycle cutoff, $N = N_{cut}$.

Design Strategies

There are different strategies that can be used when designing components subjected to alternating loads. For example:

- Design for an infinite lifetime by keeping the stresses sufficiently low. "Infinite" should be interpreted as much longer than the service life of the component. The method can also be used for a material without a fatigue limit. This is the most common approach and it is preferred as long as it is not unfeasible.

- Design for a fixed life (with sufficiently large factor of safety), after which the component is replaced.
- Create a damage tolerant design where the structure is inspected for cracks at regular intervals. In this case, it is the growth rate of macroscopic cracks that sets the limit of the inspection interval, and fracture mechanics methods must be used in the analysis.

Fatigue Model Selection

A fatigue model can be selected in different ways. Expert knowledge is a good starting point. It may so be that, within your organization, there is prior knowledge on the topic if a similar application has been analyzed already. Alternatively, you may also find expert knowledge through a literature search. Since about 90% of all structural failures are caused by fatigue, there is a great chance that another engineering team has already analyzed a similar application to yours.

When there is no prior knowledge on the fatigue case, a suitable fatigue model can be proposed based on a few questions regarding loading conditions and expected fatigue

failure. Figure 3-5 summarizes the key questions you should ask when evaluating fatigue using the Fatigue Module.
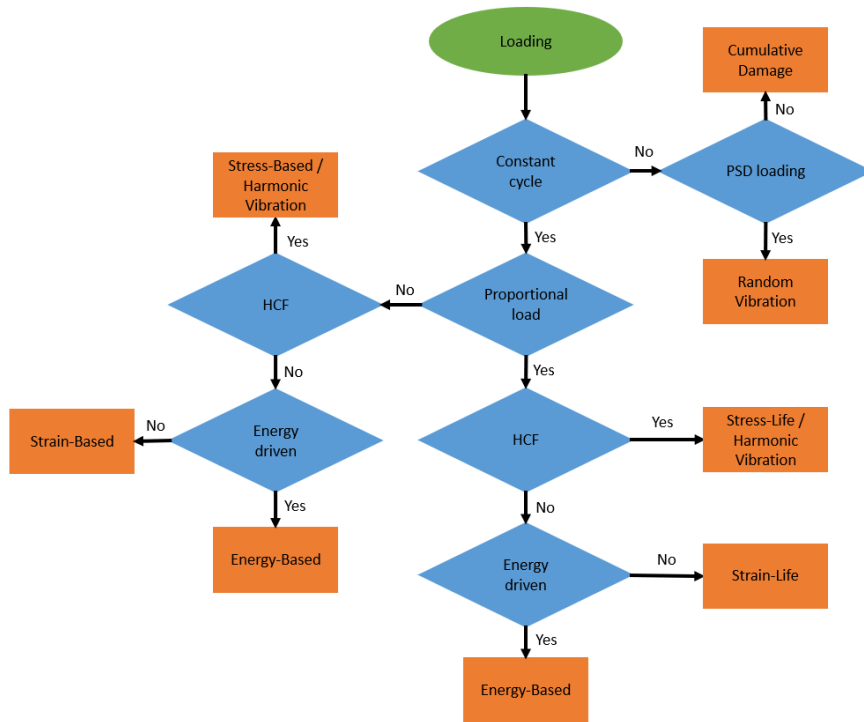

Figure 3-5: Key questions in fatigue model selection.

RANDOM LOAD FATIGUE

First, you need to determine whether the external load is random or if your application is subjected to a constant cycle. A load that is not truly random, but has sequences of nonconstant load cycles, can also fall into this category.

Random loading can further be divided into two categories, depending on the underlying stress analysis. In time-dependent and stationary analyses, random loading is described by a complete stress history. If your application is subjected to this kind of random loading, you can evaluate fatigue using the Cumulative Damage Model, where the random load is converted into a stress range distribution, rather than the single constant stress cycle — which is assumed for the other evaluation techniques. On the other hand, if the random loading is based on a random vibration (PSD based) analysis, you can evaluate fatigue using the Random Vibration Model.

At constant load cycles, the structure is affected by a repeatable load sequence. In this case, you need to determine whether the loading is proportional or nonproportional.

In proportional loading, the orientation of the principal stresses and strains does not change during the load cycle. Another way to discriminate between these two cases is to consider the characteristics of the external load. With one source of the external load, the structural response is defined by a stress tensor where all components change in phase. When the external load is applied in multiple points or if you have a traveling load, the components of the stress tensor can change out of phase. These two types of load cycles require different techniques for fatigue evaluation.

PROPORTIONAL LOADING

In proportional loading, the direction of the largest stress or strain that controls fatigue is clear. This was probably the type of application you worked with when you took your first class in fatigue. Back then, the load was always sinusoidal and classical methods such as the S-N curve, also called the Wöhler curve, were used.

In the Fatigue Module, the Stress-Life Models and Strain-Life Models models can evaluate fatigue at proportional loading. These models are based on a fatigue-life curve, which provides a direct relation between the fatigue life and the applied stress or strain amplitude.

One model in the Stress-Life family requires extra attention: Approximate S-N Curve Model. In the model, you specify two points on the S-N curve. The first one is the transition between the high- and low-cycle fatigue, while the second defines the endurance limit. The advantage of this model is that it does not require any substantial knowledge of the fatigue material data, since the two required points can be related to the ultimate tensile strength. Although it is a rough approximation, it is a good starting point when you lack material data.

The Stress-Life models are suitable for simulating high-cycle fatigue, while the Strain-Life models are frequently used in the low-fatigue regime. The transition between the low- and high-cycle fatigue varies, but is usually somewhere in the span of 1,000 to 10,000 cycles.

NONPROPORTIONAL LOADING

The challenge for nonproportional loading is to determine the range of the fatigue-controlling parameter. Since the direction of principal stresses and strains changes, so does the direction of the parameter that gives the highest impact on fatigue life.

In the Fatigue Module, this type of application can be assessed with the Strain-Based Fatigue Models and Stress-Based Fatigue Models. These are Critical Plane Methods because they evaluate many orientations in space in search for the critical plane where fatigue is expected to occur.

The Strain-Based Fatigue Models are suitable for fatigue prediction at low-cycle fatigue, while the Stress-Based Fatigue Models are frequently used to predict high-cycle fatigue. Most of the fatigue models predict the number of cycles until failure. The Stress-Based Fatigue Models predict a fatigue usage factor, which is the fraction between the applied stress and the stress limit. This indicates to the user whether the stress limit has been exceeded and failure is expected or if the component will hold for the expected fatigue life. You can view the fatigue usage factor as the inverse of a safety factor.

Several of the Stress-Based and the Strain-Based models incorporate normal stress fatigue sensitivity. The Dang Van model from the Stress-Based family takes into account the compressive stress instead and is therefore suitable for contact fatigue analysis.

FATIGUE BASED ON ENERGY

In some cases, the stress or strain alone is not sufficient to characterize the fatigue properties. You can then use the Energy-Based Fatigue Models. These combine the effect of stress and strain into energy, which is released or dissipated during a load cycle.

The Energy-Based Fatigue Models are frequently used in nonlinear materials in the low-cycle fatigue regime. Since the energy can be calculated in different ways, the Energy-Based Fatigue Models can be used in both proportionally and nonproportionally loaded applications.

    🔍 | • Energy Dissipation

Critical Plane Methods

For multiaxial cases many of the most popular fatigue criteria are based on the concept of a *critical plane*. A critical plane is a plane with a certain orientation in a point in the loaded structure that maximizes some stress or strain expression. Different models use different criteria to determine the critical plane. A successful model must be able to predict both the fatigue life and the dominant failure plane. For nonproportional

loading the orientation of the critical planes is nontrivial and the loading history needs to be examined for all possible directions in order to find the critical plane.

SELECTION OF SEARCH DIRECTIONS

The search for a critical plane is generally a computationally heavy task. In each point of the structure for which the fatigue evaluation is to be performed, the worst orientation of a plane must be found. This means that the stress/strain results have to be transformed to a large number of directions.

Consider the upper half of the unit sphere. The meridian from the north pole to the equator is subdivided into N equally spaced segments, giving $N + 1$ points. The length of such a segment is $\pi/2N$. From each point a circle of latitude is constructed. This circle is subdivided into M equally sized segments, having approximately the length $\pi/2N$. Thus, the number of direction points on each circle of latitude will differ, see Figure 3-6. The total number of normals generated in this manner is approximately $3N^2$.
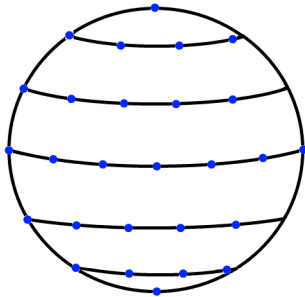

Figure 3-6: Search directions for $Q = 3$.

The accuracy of the algorithm is determined by the spacing of the points, which can be selected by the user as the **Search resolution** setting $Q\ (=N+1)$ in the **Evaluation Settings** section for the Strain-Based and Stress-Based nodes.

FINDING MAXIMUM SHEAR RANGE

On a given plane the normal stress is a scalar, but the shear stress is a two-dimensional vector. This requires an interpretation of shear stress range, $\Delta\tau$. For a multiaxial case there is no unique definition of what the maximum shear stress range is. The most strict interpretation is that $\Delta\tau$ is the diameter of the smallest circle that circumscribing the path that the τ vector creates during a load cycle, $\Delta\tau_{CC}$. This calculation is nontrivial, however, and takes significant computer resources. An alternative is to use

the maximum distance between any two points on the path instead, $\Delta\tau_{MD}$. This simplified procedure could, in extreme cases, underestimate $\Delta\tau$ by 13%. Evaluation of shear range according to both methods is shown in Figure 3-7.
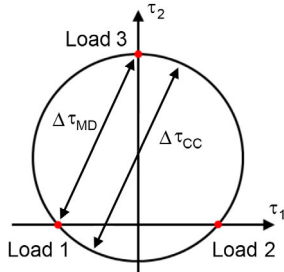
Figure 3-7: Schematic description of methods for evaluation of shear range.

The default method used in the Fatigue Module is to find the circle that circumscribe all points in the shear stress plane. Note that the smallest circumscribing circle will intersect at least two points. The choice for the simplified method is made using the **Shear range search method** setting in the **Evaluation Settings** section for the Strain-Based and Stress-Based nodes.

The discussion above, which is in terms of stresses, applies also to the corresponding strain components for a strain based evaluation.

Energy Dissipation

Elastic materials store elastic energy upon loading. This energy is restored once the structure is unloaded. Inelastic materials dissipate some energy through a nonreversible process. This dissipated energy has been used to define fatigue criteria in several materials, since it can capture the stress-strain hysteresis effect. Depending on the application, the dissipated energy is defined in different ways (Ref. 4), such as dissipated creep energy, dissipated viscoplastic energy, dissipated energy of a specific creep process, or combined dissipations from different sources. In some fatigue models, part of the elastic energy is also used in addition to the dissipated energy. It is therefore of crucial importance to know which types of energy quantities that affect fatigue and which do not. This becomes even more challenging when, for example, a material has several creep contributions (primary, secondary, and tertiary), since each creep mode has an individual contribution to the energy dissipation.

Many of the Energy-Based Fatigue Models are expressed as an exponential relation between the number of cycles to failure and the dissipated energy. These models

originate in the work by Morrow (Ref. 5 and Ref. 6), who introduced a fatigue model for ductile metals (see Morrow Model). A proposed relation has been applied with success in thermal fatigue modeling in the presence of creep or plasticity.

The definition of fatigue failure is not unique in the engineering and scientific communities. Commonly it is defined it as the crack initiation, but for some application all the history up to the final fracture is included. This ambiguity exists because certain structures are built to maintain operational integrity with cracks present, while some are considered as defective once a crack is initiated. The Morrow Model is frequently used to model failure defined as the crack initiation. Based on the numerical and experimental work on solder joint fatigue, Darveaux (Ref. 7) concluded that crack initiation only accounts for 10% of the number of cycles that are required to drive a crack through a solder joint. He proposed the Darveaux Model, which separates fatigue life into crack initiation and crack propagation.

Stress-Life Models

The relation between stress amplitude, σ_a, and life, N, is one of the oldest methods for fatigue evaluation. It is a very popular method since the relation can be easily obtained from fatigue tests. The relation takes two characteristic shapes, shown in Figure 3-8. The upper shape, which decreases until a point where it becomes a horizontal line, represents materials that have an endurance limit. If the load amplitude in a load cycle is below the endurance limit, the component does not fail in fatigue. If the endurance limit is exceeded, the component does fail in fatigue if the number of load cycles is large enough. The number of cycles where the S-N curve becomes horizontal varies for different materials but is usually in the range of 10^6 to 10^8 cycles. Several materials

do not have an endurance limit and experience fatigue even at low stress amplitudes. This fatigue behavior is demonstrated by the dashed line in Figure 3-8.
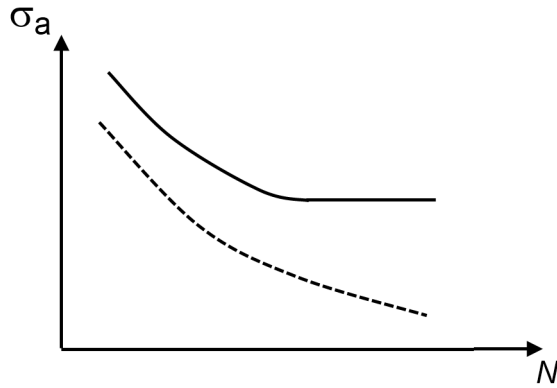
Figure 3-8: An example of a stress-life relation.

The stress amplitude is calculated with

$$\sigma_a = \frac{\Delta\sigma}{2}$$

where $\Delta\sigma$ is the stress range evaluated as the difference between the largest and the smallest stresses experienced during a fatigue cycle.

- Phenomenology and Testing
- Definitions of Fatigue Quantities
- See Stress-Life for settings information.

STRESS EVALUATION IN THE STRESS-LIFE RELATION
The fatigue limit in the stress-life relation is usually obtained using uniaxial tests where the stress amplitude is measured in one direction. In reality, stress is a second-order

tensor. Therefore, the stress amplitude can be evaluated in different ways. In the Stress-Life feature the stress amplitude is calculated using one of the following options.

TABLE 3-3: STRESS AMPLITUDE

NAME	EXPRESSION
Principal stress	$\sigma = \begin{cases} \sigma_1, & \sigma_1 \geq \lvert \sigma_3 \rvert \\ \sigma_3, & \sigma_1 < \lvert \sigma_3 \rvert \end{cases}$
Signed von Mises (principal)	$\sigma = \begin{cases} \sigma_{vM}, & \sigma_1 \geq \lvert \sigma_3 \rvert \\ -\sigma_{vM}, & \sigma_1 < \lvert \sigma_3 \rvert \end{cases}$
Signed von Mises (hydrostatic)	$\sigma = \begin{cases} \sigma_{vM}, & \sigma_h \geq 0 \\ -\sigma_{vM}, & \sigma_h < 0 \end{cases}$

where σ_1 is the largest principal stress, σ_3 is the smallest principal stress, σ_{vM} is the equivalent stress according to von Mises, and σ_h is the hydrostatic (or mean) stress.

S-N CURVE MODEL

One of the oldest models for fatigue prediction is the Wöhler curve, also called the S-N curve. It relates the number of cycles, N, to the stress amplitude, σ_a, when stress cycle is constant. On the equation form it is defined as

$$\sigma_a = f_{SN}(N)$$

where f_{SN} denotes the S-N function. At low stresses the fatigue life is limited by a Cycle Cutoff. At high stresses, when the stress amplitude exceeds the highest stress as defined

by the S-N curve, the fatigue life cannot be determined. No results are computed in such regions. This is demonstrated in Figure 3-9.
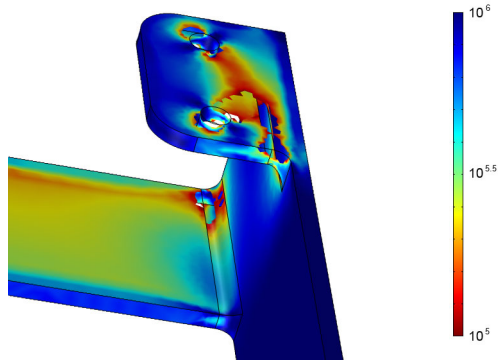
Figure 3-9: The fatigue life. Regions where the stress amplitude exceeds highest stress as defined by the S-N curve is marked with the missing color contours.

If a computed stress amplitude exceeds the highest stress amplitude in the S-N curve, a warning stating that fatigue could not be evaluated is displayed.

The Wöhler curve is mainly applied to components that operate in the high-cycle fatigue regime where the deformations are elastic.

Often an S-N curve for the actual operating condition of the component is not available. One way to handle this is to use an S-N curve obtained in the nominal condition and modify it by applying a stress modification factor, k. On the equation form the stress-life relation then becomes

$$\sigma_a = k \cdot f_{SN}(N)$$

The modification factor can contain contributions from different sources like surface finish, environmental conditions, component size, reliability level, or loading type combined with a safety factor.

BASQUIN MODEL

Basquin proposed the following exponential relation for the high-cycle-fatigue stress

$$\sigma_a = \sigma'_f (2N_f)^b$$

where σ_f and b are material constants and σ_a is the stress amplitude. N_f is the number of load cycles to failure, and $2N_f$ is the number of load reversals. This relation becomes

a linear in a log-log diagram and is often utilized in engineering applications. At low stresses the fatigue life is limited by a Cycle Cutoff.

APPROXIMATE S-N CURVE MODEL

In cases where accurate stress-life data is missing, an approximate method can be applied. Since the Basquin equation is a linear relation in the log-log space and the endurance limit is a straight line, a bilinear approximate of an S-N curve can be constructed. This requires knowledge of the endurance limit and the transition between the high-cycle fatigue (HCF) and the low-cycle fatigue. In a fatigue life (N) versus stress amplitude (σ_a) graph, the bilinear relation can be constructed as shown Figure 3-10.
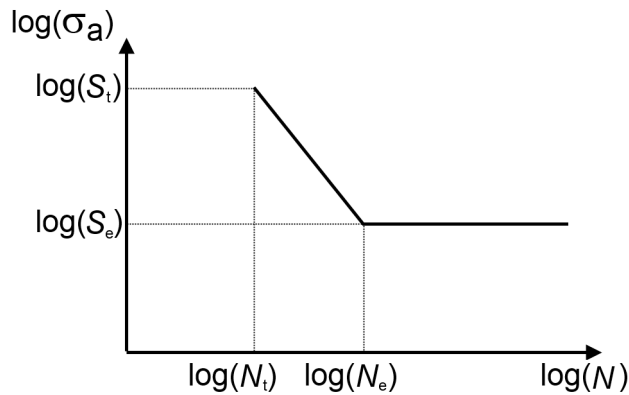

Figure 3-10: An approximate S-N curve.

The transition stress between LCF and HCF is denoted by S_t, the transition life is denoted by N_t, and the endurance limit or the endurance stress is denoted by S_e. The point that defines a knee on the S-N curve is denoted N_e and is called the endurance life. Often the key parameters of the approximate curve are expressed using the ultimate tensile stress, S_u. In steel, for example, the values are roughly $S_t = 0.9 \cdot S_u$, $N_t = 10^3$ cycles, and $S_e = 0.5 \cdot S_u$, $N_e = 10^6$ cycles.

MEAN STRESS CORRECTION

The amplitude stress that is being used in fatigue evaluation using the **S-N Curve model**, the **Basquin model**, or the **Approximate S-N Curve model** can be modified to account for the mean stress. The mean stress σ_m is defined as the average of the minimum and maximum stresses during a load cycle. The scalar stress measure used is the same as in for defining the amplitude stress.

If you select **Gerber** as **Method**, the amplitude stress is modified using the mean stress as

$$\sigma_a \leftarrow \sigma_a \left(1 - \left(\frac{\sigma_m}{\sigma_u} \right)^2 \right)$$

where σ_u is an ultimate tensile strength that you need to specify.

If you select **Goodman** as **Method**, the amplitude stress is modified using the mean stress as

$$\sigma_a \leftarrow \sigma_a \left(1 - \frac{\sigma_m}{\sigma_u} \right)$$

The Goodman and Gerber methods differ only by the shape of the interaction curve between amplitude stress and mean stress. In the Goodman case, the allowed amplitude stress decreases linearly with the mean stress, whereas in the Gerber case the interaction curve is parabolic.

If you select **Soderberg** as **Method**, the amplitude stress is modified using the mean stress as

$$\sigma_a \leftarrow \sigma_a \left(1 - \frac{\sigma_m}{\sigma_{ys}} \right)$$

where σ_{ys} is a yield stress that you need to specify. The Soderberg method is similar to the Goodman method in the sense that the interaction curve is a straight line. The Soderberg method is however more conservative, since it predicts that failure when the mean stress is at the yield limit, rather than the at the ultimate tensile stress.

Strain-Life Models

Most of the structural applications operate in the elastic regime. Local inelastic deformation can however occur at stress concentrations. Although it is concentrated to a small volume, repeated increase in inelastic strains can introduce a crack. Once a crack is formed, it is easily driven by low stresses. The fatigue is said to be strain-controlled, since the stress will be limited above the yield stress, and will thus not be good indicator of the severity of the loading. A fatigue limit based on constant strain conditions is necessary for this type of analysis. The data can be obtained from strain-controlled tests rather than from stress-controlled tests. Figure 3-11 shows a

fatigue result for such a test, where ε_a is the strain amplitude and N is the number of cycles to failure.
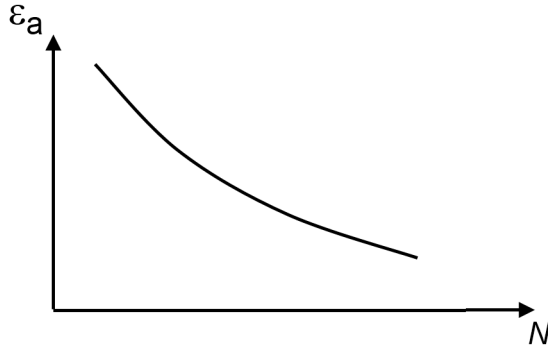

Figure 3-11: An example of a strain-controlled test.

The strain amplitude is calculated with

$$\varepsilon_a = \frac{\Delta\varepsilon}{2}$$

where $\Delta\varepsilon$ is the strain range evaluated as the difference between the largest and the smallest strain experienced during a fatigue cycle.

> See Strain-Life for settings information.

STRAIN EVALUATION IN THE STRAIN-LIFE RELATION

The fatigue limit in the strain-life relation is usually obtained using uniaxial tests, where strain amplitude is measured in one direction. In reality strain is a second order tensor. Therefore, the strain amplitude can be evaluated in different ways. In the E-N curve Model and the Combined Basquin and Coffin Manson Model the strain is evaluated according to

$$\varepsilon = \begin{cases} \varepsilon_1, & \varepsilon_1 \geq |\varepsilon_3| \\ \varepsilon_3, & \varepsilon_1 < |\varepsilon_3| \end{cases}$$

where ε_1 is the largest principal strain, ε_3 is the smallest principal strain.

E-N CURVE MODEL

The E-N curve is a relation between strain amplitude, ε_a, and fatigue life, N, that can be summarized on the form

$$\varepsilon_a = f_{EN}(N)$$

where f_{EN} is the function. At low strain the fatigue life is limited by a Cycle Cutoff. At high strain when the strain amplitude exceeds the highest strain as defined by the E-N curve the fatigue life cannot be determined. No results are computed in such regions. This is demonstrated in Figure 3-12.
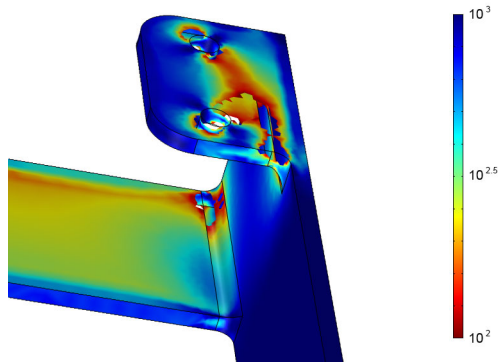
Figure 3-12: The fatigue life. Regions where the strain amplitude exceeds highest strain as defined by the E-N curve is marked with the missing color contours.

If a computed strain amplitude exceeds the highest strain amplitude in the E-N curve, a warning stating that fatigue could not be evaluated is displayed.

COFFIN-MANSON MODEL

The Coffin-Manson model is defined by

$$\frac{\Delta\varepsilon_i}{2} = \varepsilon_f'(2N_f)^c$$

where $\Delta\varepsilon_i$ is the accumulated inelastic strain in a load cycle, ε_f' and c are material constants, and $2N_f$ the number of load reversals to failure. Two reversals are equal to one cycle, so N_f is the number of cycles to failure. The fatigue relation in the original work (Ref. 5) was used for low-cycle fatigue prediction in metals and, therefore, $\Delta\varepsilon_i$ was taken as the plastic strain range.

At low strains the fatigue life is limited by a Cycle Cutoff.

Many other fatigue models are based on the work by Coffin-Manson. These, however, express the damaging inelastic strain in various ways such as equivalent creep strain range, plastic shear strain, equivalent secondary creep, and others (Ref. 4). The original relation has been generalized in the Fatigue Module so that the accumulated inelastic strain can be evaluated in several ways depending on the selected **Strain type** in the **Fatigue Model Selection** section. The different options are shown in Table 3-4 where $\Delta\varepsilon_c$ is the equivalent creep strain range and $\Delta\varepsilon_p$ is the equivalent plastic strain range. When **User defined** is used as **Strain type**, the strain variable must be specified.

TABLE 3-4: INELASTIC STRAIN TYPES

STRAIN TYPE	EXPRESSION
Equivalent creep strain	$\Delta\varepsilon_i = \Delta\varepsilon_c$
Equivalent plastic strain	$\Delta\varepsilon_i = \Delta\varepsilon_p$
Equivalent viscoplastic strain	$\Delta\varepsilon_i = \Delta\varepsilon_{vp}$
User defined	To be specified by user

⚠ A user-defined expression must be specified with the name of the physics interface. For example, if the 12-component creep strain in a physics interface with the name `solid` is evaluated in the Coffin-Manson model, then type `solid.ec12` in the expression for the **Strain type**.

COMBINED BASQUIN AND COFFIN MANSON MODEL

The fatigue limit can be provided by a combined Basquin and Coffin-Manson relation, given by

$$\varepsilon_a = \frac{\sigma_f'}{E}(2N_f)^b + \varepsilon_f'(2N_f)^c$$

where σ_f' is the fatigue strength coefficient, b is the fatigue strength exponent, ε_f' is the fatigue ductility coefficient, c is the fatigue ductility exponent, and E is Young's modulus. $2N_f$ is the number of load reversals, and thus N_f is the number of full cycles to failure at a strain amplitude of ε_a. At low strains the fatigue life is limited by a Cycle Cutoff.

It is well known that the mean stress has a significant effect on the fatigue life. Morrow proposed a mean stress correction to the Basquin part of the combined Basquin and Coffin-Manson relation according to

$$\varepsilon_a = \frac{\sigma_f' - \sigma_m}{E}(2N_f)^b + \varepsilon_f'(2N_f)^c$$

where σ_m is the mean stress of the load cycle. Since stress is a second order tensor and the mean stress is a scalar, the evaluation of the mean stress is based on the principal stress according to

$$\sigma = \begin{cases} \sigma_1, & \sigma_1 \geq |\sigma_3| \\ \sigma_3 & \sigma_1 < |\sigma_3| \end{cases}$$

Stress-Based Fatigue Models

The Findley Criterion, Matake Criterion, and Maximum Normal Stress Criterion are discussed in this section. All models are Critical Plane Methods.

FINDLEY CRITERION

The Findley criterion can be stated as

$$\left(\frac{\Delta\tau}{2} + k \cdot \sigma_n \right)_{\max} = f$$

where k and f are material parameters, $\Delta\tau$ is the maximum shear stress range on a plane, and σ_n is the largest normal stress on the same plane. The plane that maximizes the left-hand side of the equation is considered to be critical.

The fatigue usage factor, f_{us}, is the ratio between the left-hand side of the Findley criterion and the material parameter f. A value below 1 means that the component is loaded below the fatigue limit. For high compressive stress states the contribution from the normal stress can dominate the criteria predicting a negative f_{us}. In those cases, the fatigue usage factor is set to zero.

$$f_{us} = \frac{\left(\frac{\Delta\tau}{2} + k \cdot \sigma_n \right)_{\max}}{f}$$

To find the two material parameters k and f, two fatigue tests with different loading conditions are needed. This can, for example, be pure tension and pure torsion, but there are also other possibilities. For axial loading, the following relation is valid:

$$\sqrt{\left(\frac{\sigma_{max} - \sigma_{min}}{2}\right)^2 + (k \cdot \sigma_{max})^2} + k \cdot \sigma_{max} = 2f$$

Here, σ_{max} and σ_{min} are the maximum and minimum stresses at the fatigue limit, respectively (that is, infinite life). Tests with two different R values can thus be used to determine k and f. In a pure (fully reversed) torsion test with an amplitude τ_a of the torsional shear stress, the corresponding relation is

$$\tau_a = \frac{f}{\sqrt{1 + k^2}}$$

If only uniaxial test data with a single R value is available, it is possible to estimate k from the ratio between the fatigue limits under different conditions for a similar material.

MATAKE CRITERION

The Matake criterion is very similar to the Findley criterion. The difference is that it is the plane with maximum shear stress range that is taken as the critical plane, and the maximum normal stress is evaluated on that plane. The expression is:

$$\left(\frac{\Delta\tau}{2}\right)_{max} + k \cdot \sigma_n = f$$

At high compressive stress states when the Matake relation becomes negative, just as for Findley criteria, the fatigue usage factor is set to zero.

MAXIMUM NORMAL STRESS CRITERION

In the maximum normal stress criterion, the contribution from the shear stress is ignored and the plane which experiences the maximum normal stress range, $\Delta\sigma_n$, is considered as the critical plane. The criterion is defined with following expression

$$(\Delta\sigma_n)_{max} = f$$

This is a one parameter criterion where only the fatigue limit in tension is needed. In order to match a simple uniaxial test, the fatigue limit should include any mean stress effects.

!	This criterion is not suitable for materials or loadings where the mean stress is important.

DANG VAN

The Dang Van model combines the observations of the importance of the shear stress on the crack nucleation and the influence of the hydrostatic stress on the crack propagation via relation

$$\tau_{\text{max}} + \alpha_{\text{DV}}\sigma_{\text{H}} = b_{\text{DV}}$$

where τ_{max} is the maximum shear stress, σ_{H} is the hydrostatic stress, while a_{DV} and b_{DV} are material constants.

The stress state at the macroscopic level, Σ_{ij}, differs from the stress state at the microscopic level, σ_{ij}. At the macroscopic level the material is homogeneous, while at the microscopic level it is inhomogeneous. According to the Dang Van theory, the two stress tensors are related to each other via a plastic shakedown that prevents crack growth from a critical grain to the neighboring one. Affected by the multiaxial loading, the material undergoes combined kinematic and isotropic hardening and as a result the center and the radius of the yield surface changes. The center of the smallest hypersphere in stress space which encloses the entire load history determines the residual stress tensor of the shakedown. It is a 6 dimensional hypersphere with the axes $s_{\text{xx}}/(\sqrt{2})$, $s_{\text{yy}}/(\sqrt{2})$, $s_{\text{zz}}/(\sqrt{2})$, s_{xy}, s_{yz}, and s_{xz}, where s_{ij} is the deviatoric stress tensor. The resulting microscopic stress is given by

$$\sigma_{ij} = \Sigma_{ij} + \text{dev}(\rho)$$

where ρ is the residual stress tensor of the shakedown. A schematic description of the residual stress tensor for a biaxial case is shown in Figure 3-13.
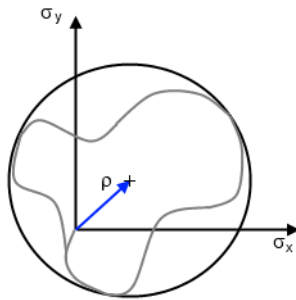
Figure 3-13: Elastic shakedown in a biaxial load case. The gray line denotes the load history.

The maximum shear stress is evaluated as

$$\tau_{max}(t) = \frac{\sigma_1(t) - \sigma_3(t)}{2}$$

and the hydrostatic stress is evaluated as

$$\sigma_H(t) = \frac{\sigma_1(t) + \sigma_2(t) + \sigma_3(t)}{3}$$

where σ_1, σ_2, and σ_3 are the principal stresses. The function argument t denotes the time dependence of the stress variables. The fatigue usage factor is finally computed as

$$f_{us} = \frac{(\tau_{max}(t) + \alpha_{DV}\sigma_H(t))_{max}}{\beta_{DV}}$$

Ⓠ	Stress-Based

Strain-Based Fatigue Models

The Fatigue Module has three strain-based Critical Plane Methods: Smith–Watson–Topper (SWT) Model, Wang–Brown Model, and Fatemi–Socie Model. These types of models are frequently used for low-cycle fatigue modeling.

In a low-cycle fatigue analysis, the plastic strains in each cycle are significant on a macroscopic scale. These strains must then be computed, which many times is the main challenge of the analysis. There are two fundamental methods to handle this:

• a full elastoplastic analysis, and

• elastic analysis with an approximation for the plasticity.

Both methods are available for all models via the **Solution type** parameter in the **Fatigue Model Selection** section for Strain-Based. The option for plasticity approximation is only available with Smith–Watson–Topper (SWT) Model, Wang–Brown Model, and Fatemi–Socie Model.

Elastoplastic Analysis

The most straightforward method is to perform a full elastoplastic analysis and feed the results to the fatigue evaluation. However, this approach can be computationally expensive since the relevant strain input come from a *stabilized cycle*. With plasticity,

due to shake-down or ratcheting, for example, it might be so that several cycles must be computed before such a cycle is obtained.

Elastic Analysis
It is also possible to only use an elastic analysis and then approximate the plastic strains. This technique is computationally efficient. The underlying assumption is that the plastic strains are localized so that the small plastic region can be considered to be locally displacement controlled by its elastic surroundings. This is the case at a notch.

⚠ Equations describing strain-based models use number of reversals, $2N_f$, while the result displays the number of cycles to failure, N_f.

SMITH–WATSON–TOPPER (SWT) MODEL
Smith–Watson–Topper (SWT) is a type of critical plane model where the plane normal to the maximum normal strain range, $\Delta\varepsilon_n$, is considered. The model is described by

$$\sigma_{n,max} \cdot \left(\frac{\Delta\varepsilon_n}{2}\right)_{max} = \frac{\sigma_f'^2}{E}(2N_f)^{2b} + \sigma_f'\varepsilon_f'(2N_f)^{b+c}$$

The left-hand side is commonly called the SWT parameter and contains the maximum normal stress during the cycle on the critical plane, $\sigma_{n,max}$. The right-hand side contains the cycles to failure, N_f, and material parameters σ_f', E, b, ε_f', and c. At low stresses and strains the fatigue life is limited by a Cycle Cutoff.

WANG–BROWN MODEL
The Wang–Brown model is based on finding the plane with maximum shear strain range, $\Delta\gamma$. It contains one extra material parameter, S, which represents the sensitivity to the normal strain range, $\Delta\varepsilon_n$, on the critical plane. In reality, S is not a constant but has some dependence on the load level. It is usually of the order 1.0–2.4 for LCF, but it can be as low as 0.3 close to the fatigue limit. The model is described with

$$\left(\frac{\Delta\gamma}{2}\right)_{max} + S\Delta\varepsilon_n = (1 + \nu + (1 - \nu)S)\frac{\sigma_f'}{E}(2N_f)^b + \frac{(3 + S)}{2}\varepsilon_f'(2N_f)^c$$

where the right-hand side contains the cycles to failure, N_f, and material parameters σ_f', E, b, ε_f', and c.

If the model is used in the HCF regime, it might be necessary to compensate for mean stress effects. This can be done using Morrow's mean stress correction. The Basquin term of the equation is then modified so that

$$\left(\frac{\Delta\gamma}{2}\right)_{\text{max}} + S\Delta\varepsilon_n =$$

$$(1 + \nu + (1 - \nu)S)\frac{\sigma_f' - 2\sigma_{n,\,\text{mean}}}{E}(2N_f)^b + \frac{(3 + S)}{2}\varepsilon_f'(2N_f)^c$$

Here, $\sigma_{n,\,\text{mean}}$ is the mean normal stress on the maximum shear plane. It is computed as the average of the maximum and minimum normal stress on the critical plane during the load cycle. At low strains the fatigue life is limited by a Cycle Cutoff.

FATEMI–SOCIE MODEL

The Fatemi–Socie model builds on the same ideas as the Wang–Brown model and also considers the plane with the largest shear strain range $\Delta\gamma$. Then, rather than the normal strain, it uses the maximum normal stress during the cycle on the critical plane, $\sigma_{n,\text{max}}$, to model the influence of an opening of the microcrack. In its fundamental form the model is formulated using the fatigue properties for pure shear, something that could be obtained from a torsion test. The Fatemi–Socie relation can be written as

$$\left(\frac{\Delta\gamma}{2}\right)_{\text{max}} \cdot \left(1 + k\frac{\sigma_{n,\text{max}}}{\sigma_{\text{ys0}}}\right) = \frac{\tau_f'}{G}(2N_f)^{b_\gamma} + \gamma_f'(2N_f)^{c_\gamma}$$

where the right-hand side contains the load cycles to failure, N_f, and material parameters τ_f', G, b, γ_f', and c. The normal stress sensitivity constant, k, can be set to 1 as an initial approximation. In reality, k is not a constant but has some dependence on the load level. σ_{ys0} is the initial yield stress of the material. At low stresses and strains the fatigue life is limited by a Cycle Cutoff.

Often the true material constants for shear are not available. In that case it is possible to approximate them from results obtained using tensile tests. This option is available and the conversion is done using the following relations:

$$\tau_f' = \frac{\sigma_f'}{\sqrt{3}}$$

$$b_\gamma = b$$

$$\gamma_f' = \sqrt{3}\varepsilon_f'$$

$$c_\gamma = c$$

ELASTIC NOTCH APPROXIMATION

Neuber's rule states that for a notch the product of elastically computed stress and strain is equal to the product of the actual, inelastic, stress and strain. Strictly speaking,

it is defined only in terms of a uniaxial stress state. In practice, the stress states are often multiaxial, so here Neuber's rule is expressed with equivalent stresses, σ_{eq}, and strains, ε_{eq}.

$$\sigma_{eq,e} \cdot \varepsilon_{eq,e} = \sigma_{eq} \cdot \varepsilon_{eq} \tag{3-1}$$

In this equation the left-hand side has an "e" denoting the results of an elastic analysis, while the right-hand side contains the actual values.

In strain-based fatigue analysis it is common to assume a Ramberg–Osgood material law when modeling the cyclic plastic behavior

$$\varepsilon_{eq} = \frac{\sigma_{eq}}{E} + \left(\frac{\sigma_{eq}}{K'}\right)^{\frac{1}{n'}} \tag{3-2}$$

where E is the modulus of elasticity. The parameters K' and n' are material constants of the cyclic response and not the monotonic response obtained from a standard tensile test.

Initially, Equation 3-1 and Equation 3-2 are solved together to obtain the elastoplastic equivalent (in a von Mises sense) stresses and strains. Hoffmann and Seeger (Ref. 1) have developed an algorithm for approximate computation of the stress and strain amplitudes in a multiaxial case. By following the Hencky's rule and utilizing the generalized notation of the Hooke's law they obtained following expression for the total elastoplastic strains

$$\varepsilon_1 = \frac{\varepsilon_{eq}}{\sigma_{eq}}(\sigma_1 - \bar{v}\sigma_2)$$

$$\varepsilon_2 = \frac{\varepsilon_{eq}}{\sigma_{eq}}(\sigma_2 - \bar{v}\sigma_1)$$

$$\varepsilon_3 = -\bar{v}\frac{\varepsilon_{eq}}{\sigma_{eq}}(\sigma_1 + \sigma_2)$$

$$\bar{v} = \frac{1}{2} - \left(\frac{1}{2} - v\right) \cdot \frac{\sigma_{eq}}{E\varepsilon_{eq}}$$

where v is the Poisson's ratio, $\bar{v}$ is the effective Poisson's ratio and σ_1 and σ_2 are the principal stresses. The third principal stress is zero. The expression for the equivalent stress is given by

$$\sigma_{eq} = \sqrt{\frac{1}{2}(\sigma_1^2 + \sigma_2^2 + (\sigma_1 - \sigma_2)^2)}$$

This provides a set of five unknowns and four equations. As the last equation Hoffmann and Seeger proposed that ratio between the two principal strain in the elastoplastic notch region equals the ratio of corresponding principal strains in the elastic case

$$\phi = \frac{\varepsilon_2}{\varepsilon_1} = \frac{\varepsilon_{2,e}}{\varepsilon_{1,e}}$$

Their work shows that the ratio is almost constant in a notch and justifies the use of this assumption. From the equations above the solution to the first principal stress and strain is given by

$$a = \frac{\phi + \bar{v}}{1 + \bar{v}\phi}$$

$$\varepsilon_1 = \frac{1 - va}{\sqrt{1 - a + a^2}} \cdot \varepsilon_{eq}$$

$$\sigma_1 = \frac{1}{\sqrt{1 - a + a^2}} \cdot \sigma_{eq}$$

Numerically, directions of the principal notch stresses are evaluated in following way:

1 The direction of σ_1 is taken as the direction of the elastic principal stress that is largest in magnitude.

2 The direction of σ_3 is taken as the direction of the elastic principal stress that is smallest in magnitude.

3 The direction of σ_2 is taken as the direction of the remaining principal elastic stress.

Energy-Based Fatigue Models

The Fatigue Module has the Morrow Model and Darveaux Model to predict fatigue based on the Energy Dissipation.

See Energy-Based for settings information.

MORROW MODEL

Morrow (Ref. 6) proposed an exponential fatigue relation in elastoplastic materials given by

$$\Delta W_d = W_f' \, (2N_f)^m$$

where ΔW_d is the range of the dissipated energy density during one cycle, N_f is the number of load cycles until failure (one fatigue cycle consists of two reversals), and W_f' and m are material constants.

In the original work, ΔW_d was taken as the range of the plastic dissipated density. This has been extended in the Fatigue Module where the Morrow model can be used with other dissipated energies, depending on the parameter **Energy type** selected under Fatigue Model Selection for the Energy-Based node. The dissipated energy is evaluated according to Table 3-5 where ΔW_c is the range of the creep dissipated energy density during one cycle and ΔW_p is the range of the plastic dissipated energy density during one cycle.

TABLE 3-5: DISSIPATED ENERGY TYPES

ENERGY TYPE	EXPRESSION
Creep dissipation density	$\Delta W_d = \Delta W_c$
Plastic dissipation density	$\Delta W_d = \Delta W_p$
Viscoplastic dissipation density	$\Delta W_d = \Delta W_{vp}$
Total dissipation density	$\Delta W_d = \Delta W_c + \Delta W_p + \Delta W_{vp}$
User defined	To be specified by used

DARVEAUX MODEL

Darveaux (Ref. 7) relates fatigue life to a volume average of dissipated energy according to

$$N = K_1 \left(\frac{\Delta W_{ave}}{W_{ref}} \right)^{k_2} + \frac{a}{K_3 \left(\frac{\Delta W_{ave}}{W_{ref}} \right)^{k_4}}$$

$$\Delta W_{ave} = \frac{\int \Delta W_d dV}{\int dV}$$

where the first term in the equation represents the number of cycles necessary to initiate fatigue and the second term defines the number of cycles during crack growth. N is the number of cycles necessary to grow a crack until it has reached the size a. K_1,

k_2, K_3, and k_4 are material constants. The average dissipated energy density range ΔW_{ave} is based on the dissipated energy density range ΔW_d that is evaluated over the material volume V. The integration of the energy density reduces the sensitivity to meshing since singularities can be expected at sharp geometrical changes, corners, or domain interfaces. Phenomenologically this is explained by the fact that a crack propagates through a layer and the energy dissipated in that layer controls fatigue.

	In order to only consider the crack initiation, set $a = 0$. In case only the crack growth is evaluated, set $K_1 = 0$.

The dissipated energy density range can take different values specified by the **Energy type** parameter as defined for the Energy-Based node. Available options are summarized in Table 3-5.

Individual Domains versus Entire Selection

The computation of ΔW_{ave} is determined by the parameter **Volume integration**. When **Individual domains** is selected, the average value is calculated over each geometrical domain specified in the selection. When **Entire selection** is selected, a single average value is calculated based on the dissipated energies in all domains in the selection. The difference between these two options is illustrated in Figure 3-14, where two equal solder joints are shown. One of them is subdivided into three geometrical domains with volumes V_1, V_2, and V_3 while the second one consists of only one domain with volume V_4. Since both joints have the same dimensions, $V_4 = V_1 + V_2 + V_3$.
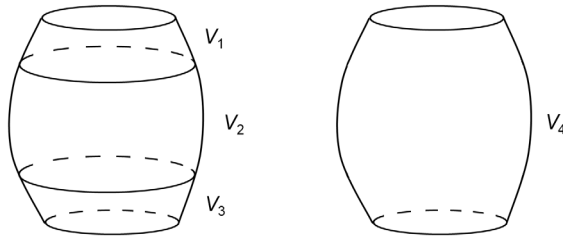
Figure 3-14: Two equal bodies consisting of three and one geometrical domains.

Assume that both solder joints are subjected to the same load and boundary conditions. Then the same energy is dissipated in them. When the model with three geometrical domains is evaluated, the option **Individual domains** calculates

$$\Delta W_{ave}^1 = \frac{\displaystyle\int_{V_1} \Delta W_d dV}{\displaystyle\int_{V_1} dV}$$

$$\Delta W_{ave}^2 = \frac{\displaystyle\int_{V_2} \Delta W_d dV}{\displaystyle\int_{V_2} dV}$$

$$\Delta W_{ave}^3 = \frac{\displaystyle\int_{V_3} \Delta W_d dV}{\displaystyle\int_{V_3} dV}$$

while the option **Entire selection** calculates

$$\Delta W_{ave}^{1+2+3} = \frac{\displaystyle\int_{V_1+V_2+V_3} \Delta W_d dV}{\displaystyle\int_{V_1+V_2+V_3} dV}$$

The same result is also obtained when a solder joint that consists of one domain only and is evaluated, thus

$$\Delta W_{ave}^4 = \Delta W_{ave}^{1+2+3}$$

For selections that consist of one geometrical domain both the options of the **Volume average method** parameter calculate the same result.

⚠️   When using the **Entire selection** option, there is no restriction that the geometrical domains in the selection must be connected.


🔍   • Energy-Based

      • Energy Dissipation

Cumulative Damage Model

Fatigue damage caused by a random load history or a variable load (see Figure 3-15) is not as easily quantified as damage from a constant load cycle. Correct simulation of the fatigue process plays a key role to predict the life of the structure. The nature of the service load history needs to be determined and the accumulated damage must be defined.
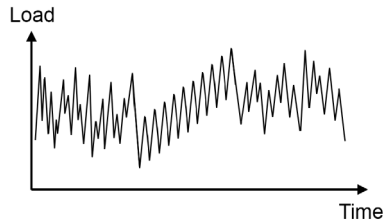
Figure 3-15: Random load history.

In the Cumulative Damage evaluation, the load is first processed with the cycle counting method, rainflow counting (Ref. 2), and followed by the damage estimation according to the Palmgren-Miner linear damage model (Ref. 3).

> See Cumulative Damage for settings information.

CYCLE COUNTING OPTIONS

The Rainflow counting method counts load cycles and transfers it into a stress distribution of the applied load history. Before this can be done, the stress tensor needs to be converted into an equivalent scalar stress. This choice is done in the specification

of the **Stress** settings in the **Cycle Counting Parameters** section. The following stress definitions can be evaluated:

TABLE 3-6: EQUIVALENT STRESS SCALAR EXPRESSION

NAME	EXPRESSION
Principal stress	$\sigma = \begin{cases} \sigma_1, & \sigma_1 \geq |\sigma_3| \\ \sigma_3, & \sigma_1 < |\sigma_3| \end{cases}$
Signed von Mises (principal)	$\sigma = \begin{cases} \sigma_{vM}, & \sigma_1 \geq |\sigma_3| \\ -\sigma_{vM}, & \sigma_1 < |\sigma_3| \end{cases}$
Signed von Mises (hydrostatic)	$\sigma = \begin{cases} \sigma_{vM}, & \sigma_h \geq 0 \\ -\sigma_{vM}, & \sigma_h < 0 \end{cases}$

In the expressions in Table 3-6, σ is the stress measure used in the Rainflow cycle counting method, σ_1 is the largest principal stress, σ_3 is the smallest principal stress, σ_{vM} is the von Mises equivalent stress, and σ_h is the hydrostatic (or mean) stress.

The Rainflow counting method reduces the stress history into a stress distribution that consists of a discrete number of *bins,* where each bin is characterized by an amplitude stress, σ_a, and a mean stress, σ_m, and holds the counted number of cycles having approximately these values. Other parameters can also be used to define a bin. These are maximum stress, minimum stress, maximum tensile stress, or R-value. In Figure 3-16 an example of a dataset is shown where the load history has been reduced to 16 bins. In the bin $\sigma_a = 70$ MPa and $\sigma_m = 30$ MPa, 67 load cycles are found. This

means that in the original load response 67 cycles are present in the range 65 MPa$<\sigma_a$
$<$75 MPa and 25 MPa$<\sigma_m<$35 MPa.

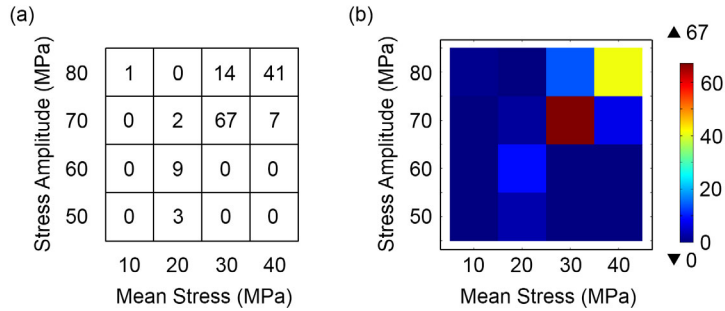*Figure 3-16: Reduced load history with the Rainflow counting method. (a) Schematic representation. (b) Results variable counted stress cycles, ftg.csc.*

The rainflow counted cycles of all bins are collected in the variable `ftg.csc` (counted stress cycles). This variable can be visualized with the Matrix Histogram, see Figure 3-16.

---

②   Definitions of Fatigue Quantities

DAMAGE ESTIMATION

Once the stress distribution is known, the Palmgren–Miner linear damage rule is used to calculate a cumulative damage. This damage in the Fatigue Module is called the fatigue usage factor and evaluated as

$$f_{us} = m \sum_{i=1}^{q} \frac{n_i}{N_i} \qquad (3\text{-}3)$$

where n_i is the number of cycles in bin i, N_i is the maximum number of cycles until fatigue occurs for bin i, q is the number of bins, and m is number of repeated cycle blocks in the load history, see Figure 3-17. Usually a fatigue usage factor of 1 or larger means that the component fails due to fatigue.
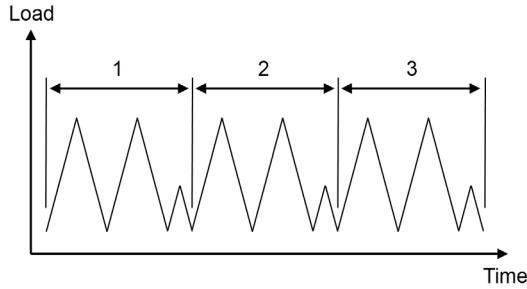
Figure 3-17: Three repeatable cycle blocks.

If the cycles in the bins describe the entire load history, the stress cycle is not repeated and thus $m = 1$.

⚠️ The Palmgren-Miner rule does not take into account the sequence effect which is well known from experiments. In the model, high loads applied in the beginning or in the end of the load cycle give the same contribution to the damage.

The limiting stress as a function of the number of cycles until failure is obtained from the S-N curve, also called the Wöhler curve, see Figure 3-18. For different R-values (R), different material responses can be expected.
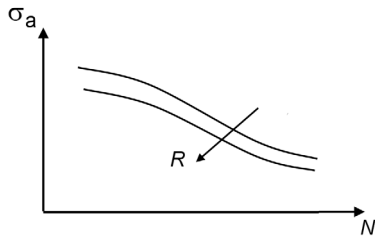
Figure 3-18: An example of the stress amplitude as a function of the number of cycles until failure.

It is recommended that the S-N curve is defined using an **Interpolation** function of **Grid** type, or with an **Analytical** function. Other function types can also be used. It is

important that the arguments of the function are ordered with R-value as the first argument and number of cycles as the second argument.

In the *COMSOL Multiphysics Reference Manual:*

- Functions
- Analytic and Interpolation

The evaluation of the S-N curve in the Cumulative Damage node is based on the following relations, for the S-N curve type using a load ratio definition:

- If $\sigma_a(R, N) > \sigma_a(R, 0.1)$ then a static failure is expected and no results are computed.
- If $\sigma_a(R, N) < \sigma_a(R, N_{cut})$ the load bin gives no contribution to the fatigue usage factor.
- If $\dfrac{\partial \sigma_a}{\partial N} > 0$ the solution to the S-N curve cannot be found and no results are computed.

The S-N curve evaluation logic above also applies to the case of an S-N curve with a mean stress dependence, and the case of an amplitude S-N curve with a mean stress correction.

For the cases when the results are not computed a message is shown to the user.

If you only have access to an amplitude S-N curve (no R-value or mean stress dependence), and you do not use any mean stress correction, the influence of the mean stress is not reflected by the results.

In the Cumulative Damage method, the stress amplitude is calculated using the S-N curve. Different material databases may provide S-N curves based on different stress measures such as the amplitude stress or the maximum stress. You may need to convert the data before using the S-N curve in the Fatigue Module.

The cumulative damage depends on the combination of stress levels and the number of counted cycles (see Equation 3-3). Each bin contributes to the fatigue usage factor with

$$r_i = \frac{n_i m}{N_i f_{us}} \qquad (3\text{-}4)$$

where index i denotes the bin counter. The contribution to f_{us} by all bins is collected in the relative usage factor ftg.rus. The sum of all individual components of the relative usage factor is 1.

COMPUTATION OF THE RESULTS

The S-N curve can be divided into three regions, shown in Figure 3-19. At high stresses, region 1, the fatigue life cannot be predicted since other mechanisms than the stress amplitude control fatigue. In this regime low-cycle fatigue and static failure can be expected. At intermediate stresses, region 2, the fatigue life is well defined and follows the S-N curve. At stresses below the endurance limit, the life is infinite and fatigue will not occur.
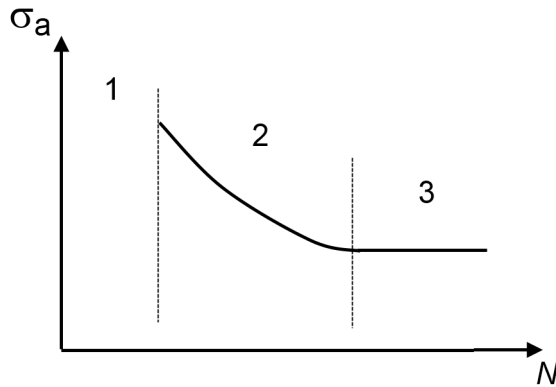

Figure 3-19: Three life regions in the S-N curve: undefined life 1, finite life 2, and infinite life 3.

The results calculated in the fatigue analysis depend on the location at the S-N curve where a stress cycle is encountered. Sometimes, it is not possible to determine a lifetime (number of allowed cycles) for a certain stress bin. Table 3-7 summarizes the results which can occur. For certain cases, there is a difference depending on whether a single point is evaluated or multiple points are evaluated. Generally, when stresses are within the finite life region all results are well defined and computed. When a single point is evaluated the counted stress cycles can always be computed but the computation of remaining results depends on the life region were stresses are encountered.

TABLE 3-7: ANALYSIS RESULTS DEPENDING ON THE S-N CURVE LIFE REGION

RESULT	UNDEFINED LIFE: SINGLE POINT	UNDEFINED LIFE: MULTIPLE POINTS	FINITE LIFE	INFINITE LIFE: SINGLE POINT	INFINITE LIFE: MULTIPLE POINTS
Fatigue usage factor	Undefined	Undefined	Defined	Zero	Zero
Counted stress cycles	Defined	Undefined	Defined	Defined	Undefined
Relative usage factor	Undefined	Undefined	Defined	Zero	Undefined

In Table 3-7, "Undefined" means that results are undefined and cannot be computed, "Defined" means that results are well defined and are computed, and "Zero" means that results are stored as zero. In an analysis where results are available in some regions, the fatigue usage factor is always displayed. In regions where undefined life is encountered, no results are computed. In case of a 2D plot, no color is shown in such areas as demonstrated in Figure 3-20.

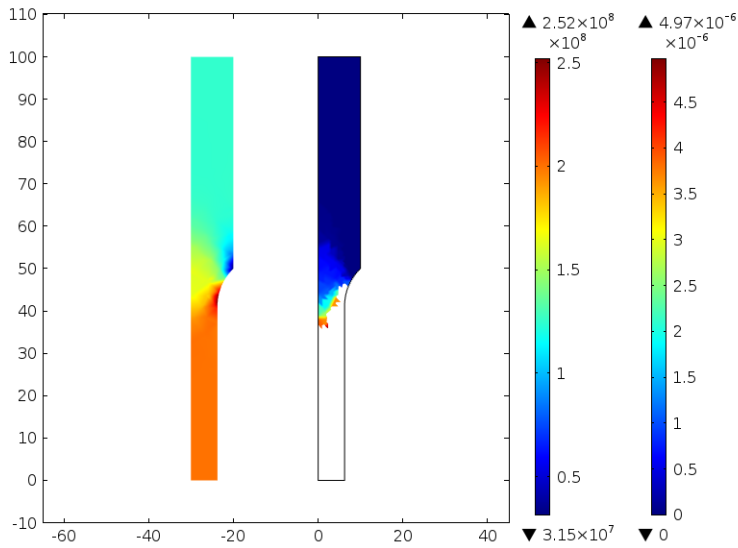*Figure 3-20: Stress amplitude to the right and fatigue usage factor to the left.*

The example shown in Figure 3-20 is based on an S-N curve with top stress of 200 MPa and endurance limit at 125 MPa. This means that when stresses are above 200 MPa, a fatigue life cannot be predicted, and when stresses are below 125 MPa an infinite life is expected. In the example, stresses below 125 MPa are experienced in the

thick top part of the specimen, and as a result a zero usage factor is calculated in this region. Stresses above 200 MPa are experienced in the thin bottom part of the specimen. As the fatigue usage factor cannot be defined in this part, no results are calculated in this region, as indicated by the missing color contours in the right picture. If a computed stress amplitude exceeds the highest stress amplitude in the S-N curve, a warning stating that fatigue could not be evaluated is displayed.

> ⚠ The stress cycle distribution of the counted stress cycles is always computed when evaluating Cumulative Damage in a single point, regardless of the outcome of the damage evaluation.

GENERALIZED LOAD

When a load cycle consists of a large number of events, the setup and simulation of the load cycle can be time consuming. This time can be greatly reduced if the influence of the external load, through the principle of superposition, is decomposed into a load scenario defined with few generalized loads. This technique can only be used when nonlinear effects can be ignored.

Based on the superposition the stress state in a point can be calculated with

$$\sigma_{ij}(t) = \sum_{k=1}^{p} \sigma_{ij}^{k} f^{k}(t) \tag{3-5}$$

where

- σ_{ij} is the resulting stress tensor
- k indicates generalized load case
- σ_{ij}^{k} is the stress response to a unit load of generalized load case k
- f^{k} is the load history of the generalized load case k
- t is the time or parameter variable.

The influence of the total load history is described in a number of modeling steps. Refer to Figure 3-23 to see an example of the steps as explained next.

Step 1 and Step 2

Start by computing $\sigma_{ij}{}^{k}$. Then define generalized unit loads as **Load Groups** (step 1) and set up the solution in the **Define load cases** option of the **Study Extensions** section in the preceding stress/strain **Study** (step 2), see Figure 3-21.

☑ Define load cases						
Load case	IgX	Weight	IgY	Weight	IgZ	Weight
Load case X	✔	1.0	✘	1.0	✘	1.0
Load case Y	✘	1.0	✔	1.0	✘	1.0
Load case Z	✘	1.0	✘	1.0	✔	1.0

↑ ↓ + ⥯

Figure 3-21: Definition of the stress response to a unit load for all generalized loads.

Step 3

When the study is computed (step 3), $\sigma_{ij}{}^{k}$ is calculated and acts as a multiplier to the time history of the generalized loads, f^{k}.

Step 4 and Step 5

Provide load histories using functions under **Global Definitions** (step 4) and further specify the functions with the name in the **Generalized load history** parameter of the **Generalized Load Definition** section in the Cumulative Damage node (step 5) (Figure 3-22).

Generalized load history
fX
fY
fZ

+ ⥯

Figure 3-22: Specification of load history functions.

The order of generalized unit loads in the stress/strain **Study** must correspond to the order of load histories specified in the **Generalized load history** parameter; compare Figure 3-21 with Figure 3-22. Finally, calculate the fatigue usage factor by solving the fatigue study (step 6). All steps are schematically shown in Figure 3-23.
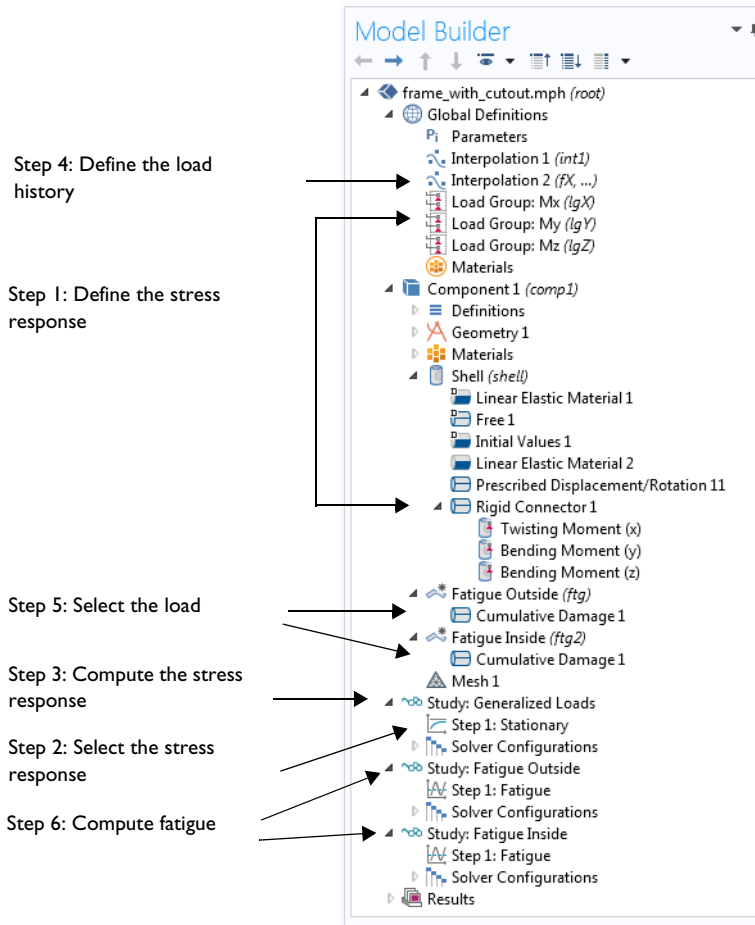
Step 4: Define the load history

Step 1: Define the stress response

Step 5: Select the load

Step 3: Compute the stress response

Step 2: Select the stress response

Step 6: Compute fatigue

Figure 3-23: Calculation steps for generalized load based Cumulative Damage. This example uses the Frame with Cutout model from the Fatigue Module Applications Libraries.

The load history evaluated by the cycle counting method is limited by the **Initial step** s_i and **Final step** s_f parameters (see Figure 3-24). This makes it possible to evaluate the

influence of parts of the load history on the cumulative damage, with no change of the load history function. See Figure 3-24.
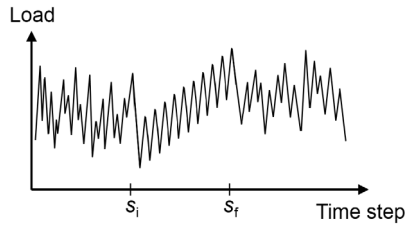
Figure 3-24: Load cycle event limits.

> ⚠️ The generalized load technique utilizes the principle of superposition and therefore excludes any nonlinear effects.

Harmonic Vibration Model

In a harmonic vibration fatigue analysis, a component is subjected to a forced harmonic excitation. In general, when the excitation frequency approaches an eigenfrequency of the object the stresses show a strong increase.

In the Harmonic Vibration evaluation, only deterministic vibration can be simulated; random, PSD based vibration analysis is outside its scope.

The analysis is done in two steps. First the stress responses at given excitation frequencies are computed using a frequency domain study step. In the subsequent fatigue study, the damage is estimated according to the Palmgren-Miner linear damage rule which uses an S-N curve to determine the fatigue life.

VIBRATION CONTROLLED LOAD

In general, when a component is excited at a constant frequency its dynamic response undergoes two phases. An initial transient phase is followed by a steady state phase. During the transient phase, which is usually short in time, the dynamic response goes from the state of the previous excitation frequency, or stand still, to the state of the

current excitation frequency. During the steady state phase, a component oscillates, so that a repetitive stress response is experienced for each consecutive cycle.
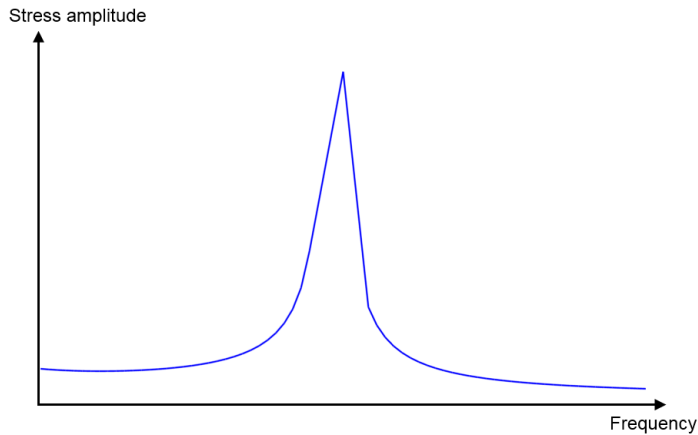
Figure 3-25: Stress response when passing an eigenfrequency.

STRESS STATE AT A CONSTANT EXCITATION FREQUENCY

When computing a stress state history at forced vibration, two cases have to be considered. The first one represents operating conditions when a component is only subjected to an excitation frequency. The second case represents conditions when a component subjected to an excitation frequency and additional static loads, such as gravity or prestress. The excitation frequency will force the component to vibrate back and forth thus causing an alternating stress that can be seen as an amplitude stress state, Σ_a. The stress response from the stationary loads causes a constant stress state in the component, that can be seen as a mean stress state, Σ_m. The stress response from both load mechanisms gives a full stress tensor denoted Σ.

Since the S-N curve is heavily influenced by the R-value, R, which in turn depends on the mean stress and the stress amplitude, the stress state from both load mechanisms must be considered.

Forced Vibration

When no static loads are present, the stress state at each excitation frequency can be computed using a single **Frequency Domain** study. For each excitation frequency the stress state is

$$\Sigma^p(t) \ = \ \Sigma_a^{\ p}(t)$$

where p is the number of the excitation frequency, while the argument t is the time. The time dependence is in this case harmonic.

Forced Vibration with Stationary Loads

In the presence of static loads, for example self-weight, a **Prestressed Analysis, Frequency Domain** study computes both the mean stress state, Σ_m, and the amplitude stress state, Σ_a, for each excitation frequency. The stress state at a given excitation frequency is given by

$$\Sigma^p(t) = \Sigma_m + \Sigma_a^{\ p}(t)$$

Note that the mean stress state is the same for all excitation frequencies while the amplitude stress state changes at each excitation frequency.

STRESS EVALUATION

The first step in a fatigue evaluation is the transformation of the stress state at a certain excitation frequency into a fatigue controlling stress, σ, which is a scalar used to evaluate fatigue life from the S-N curve.

Fatigue Mean Stress

The mean stress is computed from the stress state caused by the static loads. Since these loads are constant at all excitation frequencies, the resulting stress state, Σ_m, is also constant and so is the mean stress, σ_m. When the option **Directional stress** is used, the mean value is taken as the normal stress on the plane defined by the **Direction vector**. For example, if the stress in the x direction is examined then $\sigma_m = \sigma_{xx}$.

In case of **Signed von Mises**, the mean stress is computed using the von Mises equivalent stress from the static load case.

$$\sigma_{vM} = \sqrt{\sigma_{xx}^2 + \sigma_{yy}^2 + \sigma_{zz}^2 - \sigma_x\sigma_y - \sigma_x\sigma_z - \sigma_y\sigma_z + 3(\tau_{xy}^2 + \tau_{xz}^2 + \tau_{yz}^2)}$$

The sign of the mean stress is controlled by the hydrostatic stress, which is computed as

$$\sigma_h = \frac{\sigma_{xx} + \sigma_{yy} + \sigma_{zz}}{3}$$

The mean stress is

$$\sigma_m = \begin{cases} \sigma_{vM}, & \sigma_h \geq 0 \\ -\sigma_{vM}, & \sigma_h < 0 \end{cases}$$

Fatigue Stress Amplitude

The fatigue stress amplitude is computed from the amplitude stress state, and is in general different for each excitation frequency. Frequency domain results are complex valued. In general, the components of the stress tensor have different phase angles in the complex plane, indicating that the maximum values do not occur at the same time.

A complex-valued stress component is given by the general form

$$\sigma_{kl} = a_{kl} + i b_{kl} = r_{kl} e^{i\theta_{kl}}$$

where i is the imaginary unit. The modulus of a complex stress, r_{kl}, is the amplitude stress for the stress tensor component, while the argument of the complex number, θ_{kl}, is the phase shift of a stress component. These variables are computed using

$$r_{kl} = \sqrt{a_{kl}^2 + b_{kl}^2}$$

$$\theta_{kl} = \operatorname{atan}\left(\frac{b_{kl}}{a_{kl}}\right) + n\pi \qquad n = \begin{cases} 0 & a \geq 0 \\ 1 & a < 0 \end{cases}$$

Thus, the stress component varies between r_{kl} and $-r_{kl}$ during one cycle and the stress history is defined by

$$\sigma_{kl} = r_{kl}\cos(\omega t + \theta_{kl})$$

where ω is the angular frequency that is related to the excitation frequency through $\omega = 2\pi f$.

The **Directional stress** option considers a stress normal to a given plane. If, for example, the yz-plane is considered, the normal stress is given by

$$\sigma_{xx} = r_{xx}\cos(\omega t + \theta_{xx})$$

and the fatigue stress amplitude is taken as the highest stress during the excitation period which is the modulus of σ_{xx}. Thus,

$$\sigma_a = r_{xx}$$

The **Signed von Mises** option computes an equivalent stress which depends on all components of the complex stress tensor. The square of the von Mises stress is given by

$$\sigma_{vM}^2 = \sigma_{xx}^2 + \sigma_{yy}^2 + \sigma_{zz}^2 - \sigma_{xx}\sigma_{yy} - \sigma_{xx}\sigma_{zz} - \sigma_{yy}\sigma_{zz} + 3(\sigma_{xy}^2 + \sigma_{xz}^2 + \sigma_{yz}^2) \qquad (3\text{-}6)$$

The square of the each of the stress components is

$$\sigma_{kl}^2 = r_{kl}^2 \cos^2(\omega t + \theta_{kl}) = \frac{r_{kl}^2}{2} + \frac{r_{kl}^2}{2} \cos(2\omega t + 2\theta_{kl}) \qquad (3\text{-}7)$$

The cross product between the two normal stress components follows the pattern

$$\sigma_{xx} \cdot \sigma_{yy} = r_{xx} \cos(\omega t + \theta_{xx}) \cdot r_{yy} \cos(\omega t + \theta_{yy}) =$$

$$= r_{xx} \cdot r_{yy} \cdot \cos(\omega t + \theta_{xx}) \cdot \cos(\omega t + \theta_{yy}) =$$

$$= \frac{r_{xx} \cdot r_{yy}}{2}[\cos(\omega t + \theta_{xx} - (\omega t + \theta_{yy})) + \cos(\omega t + \theta_{xx} + \omega t + \theta_{yy})] = \qquad (3\text{-}8)$$

$$= \frac{r_{xx} \cdot r_{yy}}{2}[\cos(\theta_{xx} - \theta_{yy}) + \cos(2\omega t + \theta_{xx} + \theta_{yy})] =$$

$$= \frac{r_{xx} \cdot r_{yy}}{2} \cos(\theta_{xx} - \theta_{yy}) + \frac{r_{xx} \cdot r_{yy}}{2} \cos(2\omega t + \theta_{xx} + \theta_{yy})$$

Both the squares of the individual stress components, Equation 3-7, and the cross products between the two normal stress components, Equation 3-8, can be written on the general form

$$\sigma_{ij} \cdot \sigma_{kl} = \alpha_{ijkl} + \beta_{ijkl} \cos(2\omega t + \gamma_{ijkl})$$

where for $i = k$ and $j = l$ or for $i = j$ and $k = l$

$$\alpha_{ijkl} = \frac{r_{ij} \cdot r_{kl}}{2} \cos(\theta_{ij} - \theta_{kl})$$

$$\beta_{ijkl} = \frac{r_{ij} \cdot r_{kl}}{2}$$

$$\gamma_{ijkl} = \theta_{ij} + \theta_{kl}$$

The square of the equivalent stress, Equation 3-6, can be rewritten into following form

$$\sigma_{vM}^2 = \sigma_{xx}^2 - \sigma_x\sigma_y + 3\tau_{xy}^2 + \text{cyc.} =$$

$$= \alpha_{xxxx} + \beta_{xxxx}\cos(2\omega t + \gamma_{xxxx}) - (\alpha_{xxyy} + \beta_{xxyy}\cos(2\omega t + \gamma_{xxyy})) +$$
$$(\ + 3(\alpha_{xyxy} + \beta_{xyxy}\cos(2\omega t + \gamma_{xxxx})) + \text{cyc.} = \ )$$

$$= \alpha_{xxxx} - \alpha_{xxyy} + 3\alpha_{xyxy} + \text{cyc.} + [\beta_{xxxx}\cos(2\omega t + \gamma_{xxxx}) -$$
$$- \beta_{xxyy}\cos(2\omega t + \gamma_{xxyy}) + 3\beta_{xyxy}\cos(2\omega t + \gamma_{xxxx}) + \text{cyc.}] =$$

$$= \sum_{i=1}^{9} m_i + \sum_{i=1}^{9} a_i\cos(2\omega t + \phi_i)$$

where

$$m_1 = \alpha_{xxxx} \qquad a_1 = \beta_{xxxx} \qquad \phi_1 = \gamma_{xxxx}$$

$$m_2 = -\alpha_{xxyy} \qquad a_2 = -\beta_{xxyy} \qquad \phi_2 = \gamma_{xxyy}$$

$$m_3 = 3\alpha_{xyxy} \qquad a_3 = 3\beta_{xyxy} \qquad \phi_3 = \gamma_{xyxy}$$

$$\dots$$

Thus, the square of the equivalent stress contains constant terms and time dependent terms. Since all time-dependent terms varies harmonically with the same period the equivalent stress can be reformulated into

$$\sigma_{vM}^2 = m + a\cos(2\omega t + \phi)$$

where

$$a^2 = \sum_i \sum_j a_i a_j \cos(\phi_i - \phi_j)$$

$$\tan\phi = \frac{\sum_i a_i \sin\phi_i}{\sum_. a_i \cos\phi_i}$$

$$m = \sum_i m_i$$

It is not surprising that the equivalent stress is time dependent and that during one cycle it reaches the peak stress twice. Phenomenologically as an object vibrates, it moves back and forth. In a bending mode it means that the object will bend in one direction and then in the other direction. Since the equivalent stress does not discriminate between tension and compression it will give the same value in both cases. Interestingly enough, the equivalent stress is not harmonic, while its square is.

From the fatigue point of view only the highest equivalent stress during the cycle is of interest. This stress is considered to be the amplitude fatigue stress and is evaluated as

$$\sigma_a = \max|\sigma_{vM}| = \max\left|\sqrt{m + a\cos(2\omega t + \phi)}\right| = \sqrt{m + a}$$

FATIGUE EVALUATION

The S-N curve provides the stress amplitude as a function of the R-value and the number of cycles to failure, N. Since both the fatigue mean stress and the fatigue stress amplitude are known for each excitation frequency, the R-value is computed by

$$R^p = \frac{\sigma_m - \sigma_a^p}{\sigma_m + \sigma_a^p}$$

where the subscript p denotes the frequency dependence. Once the stress amplitude and R-value are known for each excitation frequency, f_p, the limiting number of cycles at each excitation frequency, N_p, is extracted from the S-N curve. The relation between the duration time of the excitation at a given frequency, t_p, and the number of cycles experienced is given by

$$n_p = t_p \cdot f_p$$

The partial damage from a given excitation frequency is simply the ratio n_i/N_i. Following the Palmgren-Miner linear damage rule the damage from all excitation frequencies is summarized using

$$f_{us} = \sum_{p=1}^{q} \frac{n_p}{N_p}$$

where q denotes the number of excitation frequencies. The fatigue usage factor f_{us} denotes the cumulative damage. Values below 1 means that fatigue is not expected during the analyzed frequency sweep.

FREQUENCY EVALUATION IN FREQUENCY SWEEP COMPUTATIONS

In a sweep computation the stress response is evaluated at discrete frequencies. The frequency in between may however in reality change over time. From the evaluation point of view, the history is seen as a number of constant amplitude blocks, each containing a number of cycles.

The number of frequency blocks depends on the number of frequencies, F, specified in the **Frequency Domain** study, where the stress response is computed at each frequency. A frequency block stretches between two evaluation frequencies and the stress response at the evaluation frequency that gives a higher stress value is used in the computation of the damage. As an example, in the figure below the stress response is computed at 7 frequencies in the **Frequency Domain** study. This gives 6 evaluation blocks. Note that blocks do not need to stretch over equal frequency intervals. If a computation is made around an eigenfrequency the stress response at this evaluation frequency will be used in the fatigue computation for both neighboring blocks of the evaluation frequency. The use of the higher stress for the entire block gives a conservative estimate. Due to the nonlinear dependence between stress and fatigue life, the larger stress cycles will dominate the fatigue life prediction.
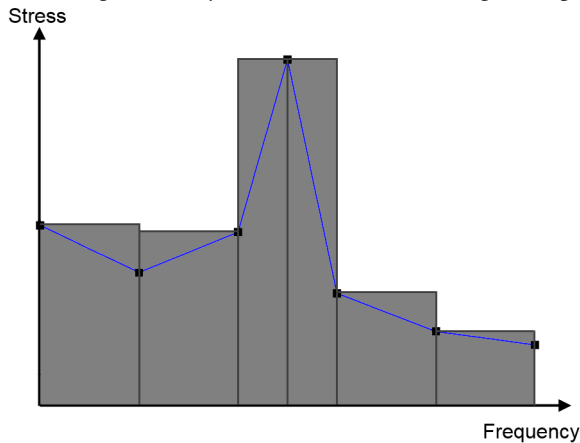
Figure 3-26: Stress evaluation in sweep computations. The gray blocks indicate the stress used, whereas the points on the blue curve are the computed stresses.

Linear Frequency Sweep

The sweep rate is the change in frequency per time unit. This can be written as

$$C = \frac{f_{p+1} - f_p}{t_b}$$

where t_b is the time duration of a block, f_{p+1} is the higher evaluation frequency, and f_p is the lower evaluation frequency. The number of cycles in each block, n_b, is evaluated using

$$n_b = \frac{(f_{p+1} - f_p) \cdot t_b}{2} = \frac{(f_{p+1} - f_p)^2}{2C}$$

Note that the subscript b is the block number while p is the frequency excitation number. The relation between these two is $b = p - 1$.

Logarithmic Frequency Sweep

In an experimental frequency sweep over a large range of frequencies, it is common that the rate of frequency change is given in octaves, which is a doubling of the frequency. The reason behind this choice is to give the same time for doubling of frequencies when evaluating frequencies of different magnitudes. The rate of change is then a logarithmic sweep rate, C_l, that is a proportionality constant between the binary logarithm of the excitation frequency and the time. The binary logarithm is the logarithm to the base 2. The relation between the time it takes to sweep between two frequencies, which in the computation is the block time, is given by

$$\frac{f_{p+1}}{f_p} = 2^{C_l \cdot t_b}$$

The block time is thus

$$t_b = \frac{1}{C_l} \cdot \log_2 \frac{f_{p+1}}{f_p}$$

The number of cycles in the evaluation block is obtained by integration of the frequency function

$$n_b = \int_0^{t_b} f \, dt = \int_0^{t_b} f_p \cdot 2^{C_l t} \, dt = \frac{f_{p+1} - f_p}{C_l \cdot \ln 2}$$

Random Vibration Model

When the loading on a structure is random, and given by a PSD spectrum, a stress life approach can be used to assess fatigue using the Palmgren-Miner linear damage model (Ref. 3). The loading PSD will effectively produce a stress response PSD at every location in the structure, and this response PSD is used to assess fatigue. As the process is random, no finite time sample of the response will be identical to the next. Instead,

assuming that any time sample is long enough, the statistics of each such sample should be the same, see Random Vibration Theory for a detailed discussion. Statistical properties that are meaningful to a fatigue analysis include the number of upward zero crossings per unit time n_0, and the number of peaks per unit time n_p. Figure 3-27 shows a schematic of a stress time sample.
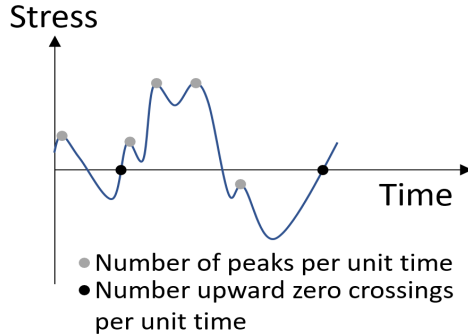
Figure 3-27: Peaks and upward zero crossings in a stress time sample.

In addition, the irregularity factor is defined as the number of upward zero crossings per unit time, divided by the number of peaks per unit time, in the time sample. The irregularity factor is always between zero and one, where a value of one suggests that the process is a narrow band process. The irregularity factor is defined as

$$\gamma = \frac{n_0}{n_p}$$

In order to extract these and other relevant statistical quantities from a stress response PSD, the moments of the spectrum are computed. The kth moment is defined as

$$m_k = \int_0^\infty f^k G(f) \mathrm{d}f$$

where $G(f)$ is the stress response PSD, and f is the frequency. In particular, the zeroth moment m_0 is related to the RMS of the stress response through

$$\sigma_{\mathrm{rms}} = \sqrt{m_0}$$

The number of upward zero crossings per unit time n_0, the number of peaks per unit time n_p, and the irregularity factor γ can be expressed in terms of the moments of the stress response spectrum as

$$n_0 = \sqrt{\frac{m_2}{m_0}}$$

$$n_\mathrm{p} = \sqrt{\frac{m_4}{m_2}}$$

$$\gamma = \frac{n_0}{n_\mathrm{p}} = \frac{m_2}{\sqrt{m_0 m_4}}$$

In a fatigue analysis in the time domain, the Palmgren–Miner damage summation rule is often used in conjunction with a rainflow stress counting algorithm, with a discrete binning of stresses into a stress histogram, see the section on the Cumulative Damage Model for an overview. In the case of fatigue during random vibrations, the discrete stress histogram is replaced by a continuous, normalized probability density distribution function $P(\sigma_\mathrm{a})$, where σ_a is a stress amplitude. A requirement for $P(\sigma_\mathrm{a})$ is that if it is integrated from zero to infinity, for the total probability, the value is equal to one. The number of stress cycles $n(\sigma_\mathrm{a})$ at a given stress level σ_a is

$$n(\sigma_\mathrm{a}) = t_\mathrm{p} n_\mathrm{p} P(\sigma_\mathrm{a})$$

where t_p is the duration of the random vibration process. Note, that the probability of exactly an amplitude stress σ_a is zero, and in order to compute fatigue damage, we must use the Palmgren–Miner damage summation in a continuous sense, and integrate over all stress amplitudes:

$$f_\mathrm{us} = t_\mathrm{p} n_\mathrm{p} \int_0^\infty \frac{P(\sigma_\mathrm{a})}{N(\sigma_\mathrm{a})} \mathrm{d}\sigma_\mathrm{a} \tag{3-9}$$

Hence, the fatigue usage factor f_us is computed by integrating the damage at each stress amplitude level, from zero to infinity. In practice, the infinite upper integration limit can be replaced by a sufficiently large, but finite value. The probability density distribution function diminishes at high stress levels, making it increasingly improbable to experience these stresses. In the expression above, $N(\sigma_\mathrm{a})$ is the expected number of cycles to failure at stress amplitude σ_a, as dictated by the specific fatigue model and the fatigue properties of the structural material in the structure. The function $N(\sigma_\mathrm{a})$ is defined by, for example, an S-N curve or the Basquin model. A fatigue usage factor exceeding one usually means that the structure fails due to fatigue.

THE PROBABILITY DENSITY FUNCTION

The probability density distribution function $P(\sigma_a)$ can be described using different models. These functions are essentially what define the counting of stress cycles, and depending on the characteristics of the stress response spectrum, a certain model may be more suitable than another. In this section, the models implemented in COMSOL Multiphysics are described.

The Bendat Model

In 1964, Bendat (Ref. 8) presented a theory for predicting fatigue damage due to random vibrations. This model is suitable for fatigue analyses of narrow band stress response spectra. In practice, this may a limitation in its applicability to more general spectra, as it tends to be overly conservative. The number of stress cycles $n(\sigma_a)$ at an amplitude stress level σ_a, using Bendat's model, is given by

$$n(\sigma_a) = t_p n_p \frac{\sigma_a}{m_0} \exp\left(-\frac{\sigma_a^2}{2m_0}\right)$$

and the fatigue usage factor is computed using Equation 3-9.

The Dirlik Model

The model by Dirlik (Ref. 9) is empirical and it was developed based on Monte Carlo simulations. The Dirlik model is suitable for more general stress response spectra, and it is not limited to narrow band spectra, as is the Bendat model. The number of stress cycles $n(\sigma_a)$ at an amplitude stress level σ_a, using the Dirlik model, is given by

$$n(\sigma_a) = t_p n_p \frac{1}{\sqrt{m_0}} \times$$
$$\left(\frac{D_1}{Q}\exp\left(-\frac{Z(\sigma_a)}{Q}\right) + \frac{D_2 Z(\sigma_a)}{R^2}\exp\left(-\frac{Z(\sigma_a)^2}{2R^2}\right) + D_3 Z(\sigma_a)\exp\left(-\frac{Z(\sigma_a)^2}{2}\right)\right)$$

with

$$Z(\sigma_a) = \frac{\sigma_a}{\sqrt{m_0}}$$

and

$$D_1 = \frac{2(x_m - \gamma^2)}{1 + \gamma^2}$$

$$D_2 = \frac{1 - \gamma - D_1 + D_1^2}{R^2}$$

$$D_3 = 1 - D_1 - D_2$$

$$R = \frac{\gamma - x_m - D_1^2}{1 - \gamma - D_1 + D_1^2}$$

$$Q = \frac{1.25(\gamma - D_3 - D_2 R)}{D_1}$$

$$x_m = \frac{m_1}{m_0} \frac{1}{n_p}$$

STRESS STATE FOR FATIGUE EVALUATION

Up to this point, we have not discussed the underlying stress measure to use for the computation of the moments of the stress response PSD. It must be noted that the random vibration fatigue analysis only considers results from pure vibrations. In other words, contributions from a superimposed mean stress are not considered. Permissible stress measures must therefore:

- Be linear with respect to the displacement field
- Have zero mean value

For instance, any linear combination of the individual stress components is permissible, while the von Mises stress or the principal stresses are not. In particular, the components the stress tensor rotated to a local coordinate system fulfill these requirements. It is not uncommon that the stress state in a critical location is uniaxial in local directions.

In practice, it is desirable to devise methods to account for the multiaxiality of the stress state. Preumont and Piéfort (Ref. 10) developed a method, in which a so-called *equivalent von Mises stress* is used. This method has been implemented in COMSOL Multiphysics, and it is described below.

The equivalent von Mises, or EVMS, is based on the definition of the von Mises stress measure. The square of this measure can be expressed as

$$\sigma_{eq}^2 = \sigma^T Q \sigma \tag{3-10}$$

where σ is the stress vector with components, using standard notation, σ_x, σ_y, σ_z, σ_{xy}, σ_{yz}, and σ_{xz}, and the matrix Q is given by

$$Q = \begin{bmatrix} 1 & -\frac{1}{2} & -\frac{1}{2} & 0 & 0 & 0 \\ -\frac{1}{2} & 1 & -\frac{1}{2} & 0 & 0 & 0 \\ -\frac{1}{2} & -\frac{1}{2} & 1 & 0 & 0 & 0 \\ 0 & 0 & 0 & 3 & 0 & 0 \\ 0 & 0 & 0 & 0 & 3 & 0 \\ 0 & 0 & 0 & 0 & 0 & 3 \end{bmatrix}$$

Note that the von Mises stress itself is not suitable for random vibration fatigue evaluations, as it is nonlinear and always nonnegative. Taking the expectancy value of Equation 3-10 gives

$$E[\sigma_{eq}^2] = Q{:}E[\sigma\sigma^T]$$

In this equation, $E[\sigma\sigma^T]$ is the covariance matrix of the stress vector σ. It is given by

$$E[\sigma\sigma^T] = \int_0^\infty \mathrm{Re}\{G_{\sigma\sigma}(f)\}df$$

where $G_{\sigma\sigma}(f)$ is the PSD matrix of the stress vector. Next, a PSD $G_{EVMS}(f)$ is expressed based on the von Mises stress quadratic form such that

$$E[\sigma_{eq}^2] = \int_0^\infty G_{EVMS}(f)df$$

Note that this PSD is *not* the PSD of the von Mises stress itself, but by construction, it shares the mean square value. Thus,

$$\int_0^\infty G_{EVMS}(f)df = Q{:}\int_0^\infty \mathrm{Re}\{G_{\sigma\sigma}(f)\}df$$

Preumont and Piéfort *define* this to be a Gaussian random process such that

$$G_{EVMS}(f) = Q{:}G_{\sigma\sigma}(f) \qquad\qquad (3\text{-}11)$$

holds. By construction, the $G_{\text{EVMS}}(f)$ PSD has a zero mean value, and it reduces to the correct, alternating stress in the case of pure uniaxial loading. The PSD function definition in Equation 3-11is used to define the corresponding spectral moments of stress. The kth moment is defined as

$$m_k \;=\; \int_0^\infty f^k G_{\text{EVMS}}(f)\mathrm{d}f \;=\; Q\!:\!\int_0^\infty f^k \text{Re}\{G_{\sigma\sigma}(f)\}df$$

These spectral moments, with the understanding that they are derived from the definition of the equivalent von Mises stress PSD, can be used by the cycle counting models outlined in The Probability Density Function.

References for the Fatigue Module

1. M. Hoffmann and T. Seeger, "A Generalized Method for Estimating Multiaxial Elastic-Plastic Notch Stresses and Strains, Part 1: Theory," *J. Engineering Materials and Technology*, vol. 107, pp. 250–254, 1985.

2. *Standard Practices for Cycle Counting in Fatigue Analysis*, ASTM E 1049-85, ASTM International, 2005.

3. A. Fatemi and L. Yang, "Cumulative Fatigue Damage and Life Prediction Theories: A Survey of the State of the Art for Homogeneous Materials," *Int. J. Fatigue*, vol. 20, no. 1, pp. 9–34, 1998.

4. W.W. Lee, L.T. Nguyen, and G.S. Selvaduray, "Solder joint fatigue models: review and applicability to chip and scale packages," *Microelectronic Reliability*, vol. 40, pp. 231–244, 2000.

5. D.F. Socie and G.B. Marquis, *Multiaxial Fatigue*, Society of Automotive Engineers, Inc., 2000.

6. J.D. Morrow, "Cyclic plastic strain energy and fatigue of metals," *ASTM STP-378*, pp.45–87, 1965.

7. R. Darveaux, "Effect of Simulation Methodology on Solder Joint Crack Growth Correlation," *Electronic Components and Technology Conference, IEEE*, pp. 1048–1058, 2000.

8. J.S. Bendat, "Probability functions for random responses: Prediction of peaks, fatigue damage, and catastrophic failures," *NASA Report on Contract NAS-5-4590*, 1964.

9. T. Dirlik, "Application of Computers in Fatigue Analysis," Ph.D. Thesis, University of Warwick, Coventry, 1985.

10. A. Preumont and V. Piéfort, "Predicting Random High-Cycle Fatigue Life With Finite Elements," *J. Vibr. Acoustics*, vol. 116, pp. 245–248, 1994.

Index