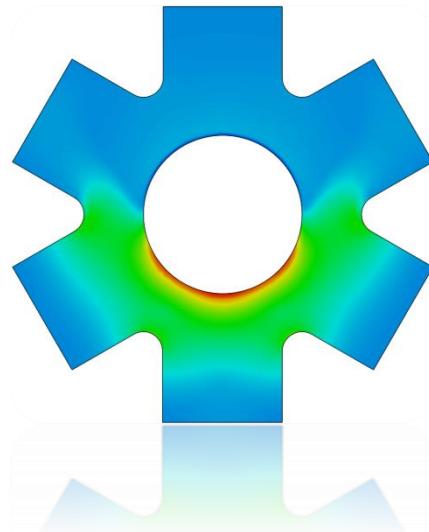


QUICK FATIGUE TOOL FOR MATLAB®

Multiaxial Fatigue Analysis Code for Finite Element Models

User Guide



© Louis Vallance 2018

Version Information

Documentation revision: 64 [18/04/2018]

Concurrent code release: 6.12-00

Acknowledgements

Quick Fatigue Tool is a free, independent multiaxial fatigue analysis project. The author would like to acknowledge the following people for making this work possible:

Dr.-Ing. Anders Winkler, SPE

Senior Technical Specialist
SIMULIA Nordics
Sweden

- Technical Advice and collaboration
- Fatigue materials data

Giovanni Morais Teixeira

Durability Technology Senior Manager
SIMULIA UK
United Kingdom

- Technical advice and collaboration

Eli Billauer

Freelance Electrical Engineer
Isreal

- Providing the code for the peak-valley detection algorithm

Adam Nieslony

Professor of Mechanical Engineering
Opole University of Technology
Poland

- Providing the code for the alternative peak-picking method

Joni Keski-Rahkonen

Senior R&D Engineer
Rolls-Royce Oy Ab
Finland

- Providing assistance with the critical plane code

Bruno Luong

- Providing the code for Cardan's formula which computes Eigenvalues for multidimensional tensor arrays

Contents

1.	Introduction	7
1.1	Overview	7
1.2	The stress-life methodology	8
1.3	The strain-life methodology.....	8
1.4	Why fatigue from FEA?	9
1.5	Syntax and conventions	12
1.6	Required toolboxes.....	15
1.7	Limitations.....	16
1.8	Additional notes.....	18
2.	Getting started.....	19
2.1	Preparing the application.....	19
2.2	How the application handles variables	19
2.3	File structure	20
2.4	Configuring and running an analysis	21
2.5	The analysis method	27
3.	Defining fatigue loadings	28
3.1	Loading methods.....	28
3.2	Creating a stress dataset file.....	32
3.3	Creating a load history	36
3.4	Load modulation	38
3.5	High frequency loadings	39
3.6	The dataset processor.....	43
4.	Analysis techniques.....	49
4.1	Background	49
4.2	Treatment of nonlinearity.....	49
4.3	Surface finish and notch sensitivity	51
4.4	In-plane residual stress	62
4.5	Controlling analysis performance	63
4.6	Analysis groups	76
4.7	S-N knock-down factors	85
4.8	Analysis continuation techniques	89
4.9	Virtual strain gauges	93
5.	Materials	97

5.1	Background	97
5.2	Material databases	98
5.3	Using material data for analysis.....	99
5.4	Creating materials using the Material Manager.....	100
5.5	Creating materials from a text file	101
5.6	General material properties	104
5.7	Mechanical material properties.....	105
5.8	Fatigue material properties	106
5.9	Composite material properties.....	111
5.10	Estimation techniques	115
6.	Analysis algorithms	117
6.1	Background	117
6.2	Stress-based Brown-Miller.....	119
6.3	Normal Stress.....	123
6.4	Findley's Method	125
6.5	Stress Invariant Parameter	132
6.6	BS 7608 Fatigue of Welded Steel Joints.....	136
6.7	NASALIFE	143
6.8	Uniaxial Stress-Life	149
6.9	Uniaxial Strain-Life	150
6.10	User-defined algorithms	151
7.	Mean stress corrections.....	154
7.1	Background	154
7.2	Goodman	157
7.3	Soderberg.....	160
7.4	Gerber	161
7.5	Morrow	162
7.6	Smith-Watson-Topper.....	163
7.7	Walker.....	164
7.8	R-ratio S-N curves.....	167
7.9	User-defined mean stress corrections.....	169
8.	Safety factor analysis	171
8.1	Background	171
8.2	Fatigue Reserve Factor.....	171

8.3	Factor of Strength	181
9.	Job and environment files.....	189
10.	Output.....	190
10.1	Background	190
10.2	Output files	192
10.3	Output variables.....	194
10.4	Viewing output.....	202
10.5	The ODB Interface.....	204
11.	FEA Modelling techniques	216
11.1	Background	216
11.2	Preparing an FE model for fatigue analysis.....	216
12.	Supplementary analysis procedures	221
12.1	Background	221
12.2	Yield criteria	222
12.3	Failure and damage initiation criteria for fibre-reinforced composites	228
12.4	Compatibility with other features.....	243
13.	Tutorial A: Analysis of a welded plate with Abaqus.....	244
13.1	Background	244
13.2	Preparing the RPT file	245
13.3	Running the analysis	245
13.4	Post processing the results	246
14.	Tutorial B: Complex loading of an exhaust manifold	249
14.1	Background	249
14.2	Preparation	251
14.3	Defining the material	252
14.4	Running the first analysis	253
14.5	Viewing the results with Abaqus/Viewer.....	255
14.6	Running the second analysis.....	256
14.7	Post processing the results	258
Appendix I.	Fatigue analysis techniques	262
Appendix II.	Materials data generation	263
Appendix III.	Gauge fatigue toolbox.....	264
References		265

1. Introduction

1.1 Overview

Quick Fatigue Tool for MATLAB is an experimental fatigue analysis code. The application includes:

- a general stress-life and strain-life fatigue analysis framework configured via a text-based interface;
- Material Manager, a material database and MATLAB application which allows the user to create and store materials for fatigue analysis (Section 5);
- Multiaxial Gauge Fatigue, a strain-life code and MATLAB application which allows the user to perform fatigue analysis from measured strain gauge histories (document *Quick Fatigue Tool Appendices: A3*);
- Export Tool, an ODB interface which allows the user to export fatigue results to an Abaqus Output Database (.odb) file for visualization in SIMULIA Abaqus/Viewer (Section 10.5); and
- supplementary analysis tools for static failure assessment (Section 12).

Quick Fatigue Tool is a family of scripts which run alongside the standard MATLAB user interface. The program does not require any executables, custom UIs or external dependencies—all of the code is available to view and customise according to the user's desires.

The general analysis framework allows the user to analyse elastic stresses from Finite Element Analysis (FEA) results. One of the main advantages of calculating fatigue lives from FEA is that it eliminates the requirement to manually compute stress concentration and notch sensitivity factors. The program is optimised for reading field output from Abaqus field report (.rpt) files. However, field output can be specified in any ASCII format provided the data structure in Section 3 is observed.

A Quick Fatigue Tool analysis requires the following inputs from the user:

1. A material definition
2. A loading definition consisting of:
 - a. Stress datasets
 - b. Load histories

The above input is specified by means of a *job* file. This is an M-file or text file containing a list of options which completely define the analysis. Analyses are performed by running the job file. Basic fatigue result output is written to the command window, and extensive output is written to a set of individual data files.

1.2 The stress-life methodology

The stress-life methodology is used for calculating fatigue damage where the expected lives are long and the stresses are elastic. The method is also well-suited to infinite life design where a pass/fail criterion based on the fatigue limit is sufficient. The stress-life approach ignores local plasticity and provides a “total life” estimate of fatigue life [1] [2] [3]. This is illustrated by Figure 1.1. If the analyst wishes to gain insights into the life up to crack initiation (N_i), or wishes to find the number of cycles required to cause crack growth ($N_i + N_p$), strain and fracture mechanics-based methods should be explored instead [4].

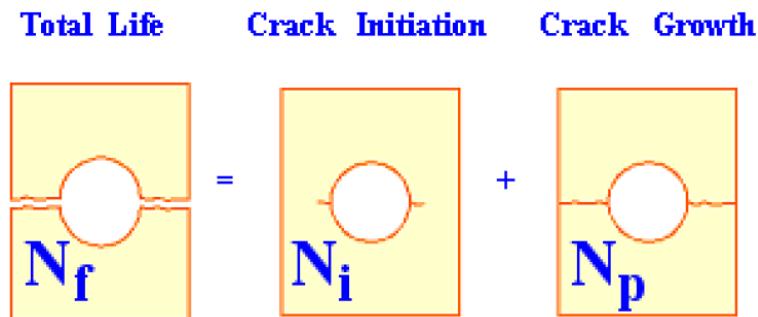


Figure 1.1: Illustration of the stress-life method where the total life, N_f , is the sum of the life to crack initiation, N_i , and life to final crack propagation, N_p .

1.3 The strain-life methodology

The strain-life methodology is used for calculating fatigue damage where the cycles are dominated by local plasticity. Although the majority of engineering structures are designed such that the operational stresses do not exceed the elastic limit, unavoidable design features such as notches can result in local plastic strains.

The strain-life methodology correlates the local plastic deformation in the vicinity of a stress concentration to the far-field elastic stresses and strains using the constitutive response determined from displacement-controlled fatigue tests on simple (smooth) laboratory specimens.

Fatigue analysis using the strain-life methodology is capable of accurate predictions of crack initiation down to a few hundred cycles. Depending on the strain-life data, failure is usually assumed as a surface crack with a length of approximately 2mm.

1.4 Why fatigue from FEA?

Modern design workflows demand a complex and multidisciplinary mind set from the analyst [5]. The combination of complex geometry and service loading can make the determination of the most important stresses an insurmountable task in the absence of powerful computer software.

The finite element method is a popular tool which allows the analyst to determine the stresses acting on a component with a high degree of accuracy. However, selection of the correct stress is often still not obvious. Take Figure 1.2 as an example.

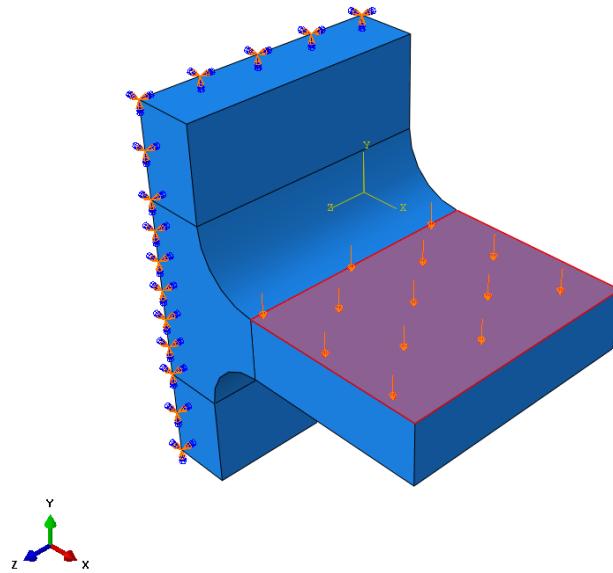


Figure 1.2: Uniaxial load applied to a fillet joint

A simple fillet joint is loaded in bending by a unidirectional pressure force. The load is applied to 18 MPa and then removed, resulting in a single pulsating loading event. Figure 1.3 shows the result of the finite element analysis. The simplest way to relate the stress to fatigue life is by the Wöhler stress-life curve [6]:

$$S = \sigma_f' N_f^b \quad [1.1]$$

The damage parameter, S , is related to the fatigue life in repeats, N_f by the material constants σ_f' and b . Considering the results from Figure 1.3, it is not obvious which stress should be chosen to take the place of the parameter S . There are several approaches for the evaluation of the fatigue life.

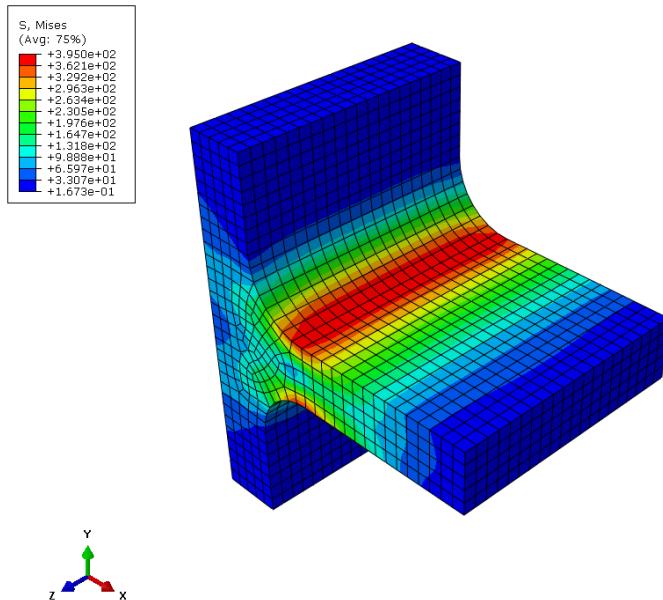


Figure 1.3: FEA stresses on the fillet joint due to bending load

A common approach is to take the node with the maximum principal stress and substitute this value into the stress-life equation. Alternatively, the model can be viewed in terms of effective stress (for example, von Mises), and using this parameter for the fatigue calculation. Both of these approaches have serious drawbacks in that they do not correctly account for the presence of multiaxiality and non-proportionality which commonly arises in fatigue loadings. Fatigue results obtained using these techniques can be in significant error and even miss the location of fatigue crack initiation.

The best practise is to employ multiaxial algorithms which correctly identify the stresses on the most damaging planes. Even unidirectional loads, such as those in the above example, can result in multiaxial stress fields. Therefore, multiaxial analysis algorithms are always recommended over uniaxial and effective stress methods.

The fillet joint is analysed using Quick Fatigue Tool with several fatigue criteria, the results of which are summarised in Figure 1.4 and the tabulated data. Algorithms with “(CP)” indicate that they are multiaxial (critical plane) methods.

The Uniaxial Stress-Life method underestimates the fatigue life, whereas the von Mises stress overestimates. By considering the maximum principal stress, the uniaxial method assumes that fatigue failure will occur on a plane perpendicular to the material surface where the shear stress is zero. In reality, most metals experience crack initiation on shear planes where there is no normal stress. For this reason, both the uniaxial and normal stress methods produce highly conservative life predictions.

The Stress-based Brown-Miller and Findley’s Method produce the most accurate results, since they consider the action of both the normal and shear stress acting on several planes.

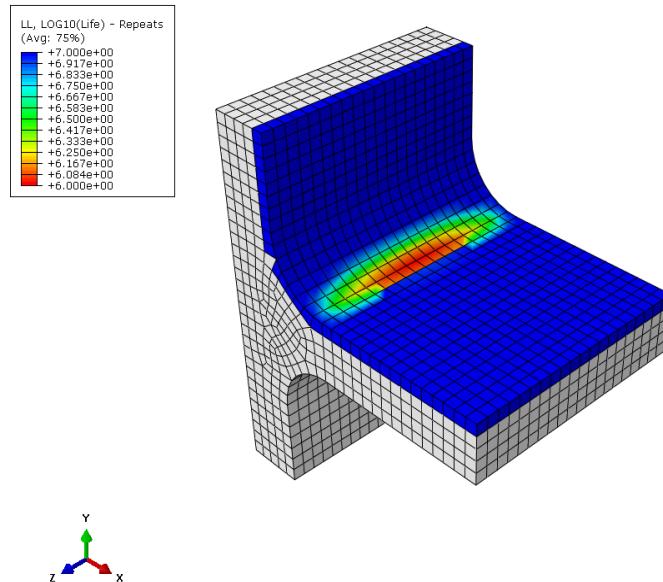


Figure 1.4: Fatigue analysis results showing (logarithmic) life using the Stress-based Brown-Miller multiaxial algorithm

Analysis algorithm	Fatigue life (repeats)
Uniaxial Stress-Life	462,000
Stress Invariant Parameter (von Mises)	1,570,000
Normal Stress (CP)	263,000
Stress-based Brown-Miller (CP)	800,000
Findley's Method (CP)	1,040,000

By combining results from FEA with a multiaxial analysis technique, the most accurate life prediction can be obtained. Due to the fact that multiple planes have to be searched in order to take into account multiaxial stress states, the multiaxial algorithms are very time-consuming compared to the uniaxial and effective stress methods. Confining the analysis to the location of maximum stress is not guaranteed to be successful since the location of crack initiation is not guaranteed to coincide with the location of maximum stress.

1.5 Syntax and conventions

1.5.1 Using Quick Fatigue Tool

The Quick Fatigue Tool analysis job is created by a combination of *job file options* and *environment variables*. Job file options represent the fundamental aspects of the fatigue analysis, such as material definition, loading and fatigue analysis algorithm. Job file options are analysis-specific. Environment variables are used to control the general behaviour of Quick Fatigue Tool, such as load gating, critical plane search precision and results format. Environment variables apply globally to all analyses, but may be configured for specific analyses.

Detailed information on job file options and environment variables can be found in the document *Quick Fatigue Tool User Settings Reference Guide*.

1.5.2 Job file options

All of the available job file options can be found in *Project\job\template_job.m*. These are standard MATLAB variables which are passed into the main analysis function when the job file is run.

Job file options can be defined as character arrays or cells, as numeric arrays, or simply left empty, depending on how the variable is defined.

1.5.3 Environment variables

All of the available environment variables can be found in *Application_Files\default\environment.m*. These are MATLAB %APPDATA% variables which are read into the program application data at the beginning of the analysis.

Environment variables can be defined as character arrays or cells, as numeric arrays, or simply left empty, depending on how the variable is defined.

Environment variables are set using the *setappdata()* method. The first argument is the name of the environment variable and the second argument is the value of the variable.

1.5.4 Units

Material data must be defined in the *SI (mm)* system of units (*MPa, strain, N, mm*). Stress datasets can use any system of units, and are automatically converted into *MPa* before the analysis.

1.5.5 Nomenclature

Unless stated otherwise, symbols take their meaning from the following tables.

Stresses and strains

σ	Cauchy stress tensor
σ_{11}	XX , direct stress
σ_{22}	YY , direct stress
σ_{33}	ZZ , direct stress
σ_{12}	XY , shear stress
σ_{13}	XZ , shear stress
σ_{23}	YZ , shear stress
ε_{11}	XX , direct strain
ε_{22}	YY , direct strain
ε_{12}	XY , shear strain
σ_{xm}	XX , mean stress
σ_{ym}	YY , mean stress
σ_{zm}	ZZ , mean stress
σ_{xym}	XY , mean stress
σ_{xzm}	XZ , mean stress
σ_{yzm}	YZ , mean stress
σ_{xa}	XX , stress amplitude
σ_{xy}	YY , stress amplitude
σ_{za}	ZZ , stress amplitude
σ_{xya}	XY , stress amplitude
σ_{xza}	XZ , stress amplitude
σ_{yza}	YZ , stress amplitude
ε_A	Gauge strain, Rosette arm A
ε_B	Gauge strain, Rosette arm B
ε_C	Gauge strain, Rosette arm C
ε^{pl}	Total strain
σ^{pl}	Total stress
σ_1	First (maximum) principal stress
σ_2	Second (middle) principal stress
σ_3	Third (minimum) principal stress
σ_{eff}	Effective stress damage parameter
σ_m	Effective mean stress
σ_a/ε_a	Effective stress/strain amplitude
σ_{max}	Maximum stress in cycle
σ_{min}	Minimum stress in cycle
σ_{aw}	Equivalent stress amplitude (Walker)
τ_{oct}	Octahedral shear stress

Material properties

E	Young's Modulus
K/K'	Monotonic/cyclic strain-hardening coefficient
n/n'	Monotonic/cyclic strain-hardening exponent
σ_u	Ultimate tensile strength
σ_y	Yield strength
ν	Poisson's ratio
σ_f'	Fatigue strength coefficient
b	Fatigue strength exponent
b_2	Second fatigue strength exponent (after knee)
$b_{2,NF}$	Fatigue life at knee
ε_f'	Fatigue ductility coefficient
c	Fatigue ductility exponent
γ	Walker gamma parameter

Fatigue

N_f	Fatigue life (cycles)
$2N_f$	Fatigue life (reversals)
α	Biaxiality ratio
σ_0	Fatigue limit
N_∞	Endurance limit
N_T	Target life
R	Load ratio

1.5.6 Documentation conventions

The following conventions are used to signify job file options and environment variables throughout the Quick Fatigue Tool documentation:

- job file options defined in MATLAB are presented in **BOLD_FACE**;
- job file options defined in text files are presented in UPPER CASE and are always preceded by an asterisk (*);
- environment variables are presented in **magenta**;
- file names and string parameters are presented in '**magenta**' and '**magenta**', respectively;
- command line parameters are presented in **magenta**;
- default or applicable parameters are underlined (_);
- default parameter values are presented in (parentheses) after the job file option or environment variable;
- items enclosed in **bold** square brackets ([]) are optional;
- items appearing in a list separated by bars (|) are mutually exclusive; and
- one value must be selected from a list of values enclosed by **bold** curly brackets ({ }).

Field output variable identifiers are presented in **BOLDFACE**.

1.6 Required toolboxes

Quick Fatigue Tool does not require any MATLAB toolboxes to function properly. However, certain toolboxes are used to enhance the functionality of the code if Quick Fatigue Tool detects that they are installed. Below is a summary of these toolboxes and the additional functionality they provide.

Toolbox	Functionality
Image Processing Toolbox	Strain gauge preview dialogue box for Gauge Fatigue Toolbox apps. See Appendix III of <i>Quick Fatigue Tool Appendices</i> for more information
Symbolic Math Toolbox	Iterative solver for the following features: <ul style="list-style-type: none">Derivation of the normal stress sensitivity constant (k) using the <i>General Formula</i> option in the Material Manager appDerivation of the reference strains from user-defined strain gauge orientations using the Multiaxial Gauge Fatigue and Rosette Analysis appsDerivation of the initial fibre misalignment angle (φ^0) for the LaRC05 composite damage initiation criteria
Statistics and Machine Learning Toolbox	RHIST and RC output variables
Signal Processing Toolbox	High frequency loadings with HF_DATASET and HF_HISTORY

1.7 Limitations

1.7.1 Fatigue from FEA

Overview

Quick Fatigue Tool has been optimised to use stress data from an Abaqus output database (*.odb*) file using the built-in field output report tool in Abaqus/CAE. Stress dataset files generated from other third party FEA processors must adhere to the rules and conventions detailed in Section 3.2.

Datasets

Quick Fatigue Tool assumes that the FEA stress datasets are elastic for both stress and strain-based fatigue analyses. In the latter case, the stresses are automatically corrected for the effect of plasticity.

Elements

Special-purpose elements and connector elements are not supported. Structural elements such as beams, pipes and wires may be used, although they have not been thoroughly tested. Using any of these elements may cause the analysis to produce error messages or crash.

Stress tensors read from FE models must use a Cartesian coordinate system.

When exporting stress datasets from Abaqus, Quick Fatigue Tool will fail to process the data if the element stresses are written to more than two locations on the element. For example, Abaqus usually writes stresses to two section points for shell elements (top and bottom faces). If the user requested field output at more than two section points, or the element is defined from a composite section, Quick Fatigue Tool will not be able to interpret the stress dataset file.

If the user wishes to analyse the surface of a composite structure, the best practise is to define a skin on the surface of the structure and export the stresses from the skin elements only.

Part instances

Quick Fatigue Tool supports stress datasets which span multiple Abaqus part instances provided that the element-node numbers are unique between instances. If the stress dataset contains duplicate element-node numbers, Quick Fatigue Tool is unable to distinguish between individual part instances and results may be reported at incorrect locations.

If the model is defined as assemblies of part instances and the fatigue analysis spans more than one instance, the user can ensure unique element-node numbering as follows:

1. Enter the Mesh Module in Abaqus/CAE and select **Part** as the object
2. From the main menu, select **Mesh → Global Numbering Control...**
3. Specify a start label for the elements and nodes of the current part instance, such that they do not clash with other part instances which will be included for fatigue analysis

It is strongly recommended that the user works with flat input files. This generates an output database containing a single part instance, which guarantees unique element-node numbering:

1. (any module in Abaqus/CAE): From the main menu select **Model** → **Edit Attributes** → **Model name**
2. Select *Do not use parts and assemblies in input files*

1.7.2 Loading

Quick Fatigue Tool does not directly support multiple block loading. A workaround involves splitting the load spectrum into several jobs using the **CONTINUE_FROM** option, which allows Quick Fatigue Tool to automatically overlay the fatigue damage onto the previous results to give the total damage due to all blocks. Analysis continuation is discussed in Section 4.8.

1.7.3 Materials

It is assumed that stress relaxation does not occur during the loading and that the material is cyclically stable. This expedites the fatigue calculation by allowing analysis of each node without considering the effects of neighbouring nodes, but precludes the effect of global plasticity being accurately taken into account. However, for the majority of cases this should not be an issue since the stress-life method is elastic, and the strain-life method is intended for components experiencing relatively small amounts of local plasticity.

Quick Fatigue Tool is applicable to metals and some engineering plastics where the stresses and temperatures are sufficiently low that viscoelastic effects are negligible.

Treatment of local notch plasticity requires the use of strain-based fatigue methods. Treatment of crack propagation requires the use of crack growth methods such as *VCCT*, *CTOD* and *Paris Law LCF*.

Analysis of viscoelastic, hyperelastic, anisotropic and quasi-brittle materials is not supported. Materials whose fatigue behaviour cannot reasonably be modelled by linear elastic stresses and stress-life curves are not supported.

1.7.4 Performance

The MATLAB programming language is very convenient in terms of the ease and speed of development it offers. However, in runtime the code is slow in comparison to other languages. Therefore, stress datasets from even a modest finite element model can result in cumbersome analyses. The user is advised to consult Section 11 for assistance on how to minimise analysis time without compromising on the accuracy of the solution.

1.7.5 GUI appearance

It is recommended that you set your monitor DPI scaling to 125% and the resolution to 1920x1080. On Windows 7, the DPI settings are found at *Control Panel\Appearance and Personalization\Display*. On Windows 10 the settings are at the same location, but you must select *set a custom scaling level* under the “Change size of items” section.

If the above settings are not used, Material Manager, Export Tool and the Gauge Fatigue Toolbox apps may display incorrectly.

An alternative to using the Material Manager app is to import material data from a text file. For instructions on creating material text files, consult Section 5.5.

1.7.6 Validation

Quick Fatigue Tool has not been validated against any official standard. The author does not take any responsibility for the accuracy or reliability of the code. Fatigue analysis results should be treated as supplementary and further investigation is strongly recommended.

1.8 Additional notes

- it is recommended that you consult the file *README.txt* in the *Documentation* directory before proceeding to the next section of this guide;
- modifying the file structure (e.g. renaming folders) will prevent the program from functioning;
- the change log for the latest version can be found [here](#); and
- Quick Fatigue Tool is free for distribution without license, provided that the author information is retained in each source file.

To report a bug or to request an enhancement, please contact the author:

Louis VALLANCE

louisvallance@hotmail.co.uk



Vienna, March 2018

2. Getting started

2.1 Preparing the application

Preparing Quick Fatigue Tool requires minimal intervention from the user, although it is important to follow a few simple steps before running an analysis.

Make sure the working directory is set to the root of the Quick Fatigue Tool directory, e.g. `\..\\Quick Fatigue Tool\\6.x-yy`. The directory structure is shown in Figure 2.1. All folders and sub folders should be added to the MATLAB search path using the function `addpath()`, or by selecting the folders *Application_Files* and *Project*, right-clicking and selecting **Add to Path → Selected Folders and Subfolders**.

If the MATLAB working directory is not configured exactly as described above (e.g. the user enters the job directory before running a job file), the application will not run.

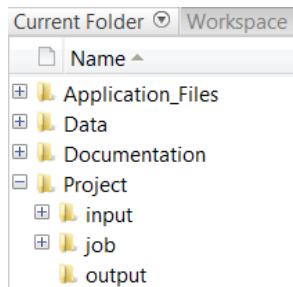


Figure 2.1: Quick Fatigue Tool file structure

Before running a fatigue analysis, it is recommended that you first run the job *tutorial_intro*. This is because the initial run of a MATLAB function requires some additional computational overhead which slows down the first analysis.

2.2 How the application handles variables

Quick Fatigue Tool does not store variables in the base workspace, nor does it modify or delete existing workspace variables. During analysis, variables are stored either in the function workspaces or as application-defined data using the `setappdata()` and `getappdata()` methods.

The application data is utilised for convenience, since variables can easily be accessed throughout the code without having to pass variables between many functions. In order to prevent unwarranted data loss, Quick Fatigue Tool does not modify existing application data by default. However, this means that variables from previous analyses will remain in the application data and could cause unexpected behaviour in subsequent analyses, such as incorrect fatigue results and spurious crashes.

In order to eliminate the possibility of such conflicts, the user is strongly advised to restart MATLAB between each analysis so that the application data is cleared. If the user does not wish to restart MATLAB for each analysis and is not concerned about Quick Fatigue Tool modifying the application data, the following environment variable may be set with a value of 3.0:

Environment File Usage: `cleanAppData = {1.0 | 2.0 | 3.0 | 4.0}`

1. Clean the application data before each analysis only
2. Clean the application data after each analysis only
3. Clean the application data before and after each analysis
4. Never clean the application data (default)

This ensures that the application data is completely cleared before and after each analysis. This has the same effect of restarting MATLAB.

The environment file and all of the available user settings are discussed in the document *Quick Fatigue Tool User Settings Reference Guide*.

2.3 File structure

Quick Fatigue Tool separates various components of the code into folders. Below is a brief description of what each folder contains:

<i>Application_Files</i>	Source code and application-specific settings. There is usually no need to modify the contents of this directory
<i>Data</i>	User-specific data (models, surface finish curves, material data, etc.)
<i>Documentation</i>	README file and User Guide
<i>input</i>	Required location for stress datasets and load histories
<i>job</i>	Job files defining analyses
<i>output</i>	Fatigue results directory. If this folder does not exist, it will automatically be created during the analysis

2.4 Configuring and running an analysis

2.4.1 Overview

The following section is a basic tutorial for new users. It leads you through a simple fatigue analysis job by explaining the meaning of the most fundamental analysis options and the resulting output.

Standard analyses are configured and submitted from an *.m* file. You will analyse a uniaxial stress history at a single node to obtain an estimate of the fatigue life.

2.4.2 Configuring a standard analysis

The analysis is configured by combining a stress dataset with a load history. The files can be found in the *Data\utorials\intro_tutorial* folder.

Defining a stress dataset

Open the stress dataset file *stress_uni.dat*. A typical stress definition consists of six components defining the Cauchy stress tensor. The components are defined as follows:

Node №	σ_{11}	σ_{22}	σ_{33}	σ_{12}	σ_{13}	σ_{23}
1	200	0	0	0	0	0

Defining a load history

Open the load history file *history_fully_reversed.dat*. A simple load history consists of two loading points. Below is a list of common load definitions:

Load type	Definition
Fully-reversed (push-pull)	[1, -1]
Pure tension (pulsating)	[0, 1]
Pure compression (pulsating)	[0, -1]

The file load history file contains the fully-reversed event, [1, -1]. This will cycle the stress component σ_{11} between 200MPa and -200MPa to create a single fatigue cycle.

Configuring the job file

Open the job file *tutorial_intro.m*. The file contains a list of options specifying all of the information necessary for fatigue analysis. Options can be strings or numeric depending on the meaning of the option. Not all options require a user setting.

The user need not worry about the number of definitions—all of the job file options are explained in the document *Quick Fatigue Tool User Settings Reference Guide* and through the chapters in this guide.

Material properties

This analysis uses SAE-950C Manten steel as the material. The materials available for analysis are located in the *Data\material\local* folder. The material properties of SAE-950C are as follows:

$$E = 203GPa$$

$$\nu = 0.33$$

$$\sigma_u = 400MPa$$

$$\sigma_c = 400MPa$$

$$\sigma_f' = 930MPa$$

$$b = 0.095$$

The material data can be viewed by loading the file *Data\material\local\SAE-950C.mat* into the MATLAB workspace.

The user specifies the material data in the job file.

Job File Usage: `MATERIAL = 'sae-950c.mat'`

Analysis algorithm

The analysis uses the Normal Stress algorithm. Since the mean stress of the cycle is zero, no mean stress correction is specified.

The user specifies the material data in the job file.

Job File Usage: `ALGORITHM = {5.0 | 'NORMAL'}`

Submitting the job for analysis

To execute the analysis, right-click on *tutorial_intro.m* and click *run* (Figure 2.2). The progress of the analysis is printed to the command window.

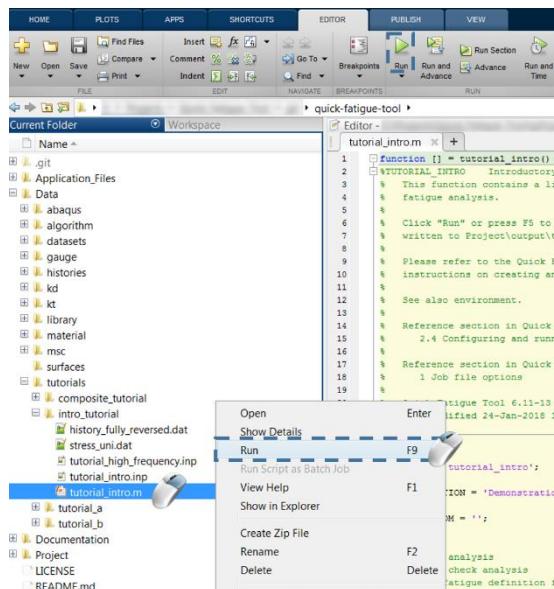


Figure 2.2: Submitting a job for analysis

Viewing the results

A summary of the fatigue results is printed to the command window, which reports a life of 5.3 million cycles to failure at node 1.

In the *Project\output* directory, a folder with the name of the job is created containing the requested output. Information regarding the progress of the analysis is written to three files:

- | | |
|---------------------------|--|
| <i>tutorial_intro.log</i> | <ul style="list-style-type: none">• input summary;• analysis groups;• critical plane summary;• factor of Strength diagnostics; and• fatigue results summary. |
| <i>tutorial_intro.msg</i> | <ul style="list-style-type: none">• pre-processor and post-processor summaries;• analysis-specific notes offering useful information to the user; and• analysis-specific warnings explaining potential issues with the analysis. |
| <i>tutorial_intro.sta</i> | <ul style="list-style-type: none">• an item-by-item summary of the analysis progress. |

The job file requested field output. This data has been written under *Project\output\tutorial_intro\Data Files\f-output-all.dat*. The field data file contains spatially-varying output variables at each item in the model. Since only one node was analysed, a single row of data is written.

The fatigue life is reported by the **L** identifier. The logarithmic life, $\text{Log}_{10}(L)$, is given by **LL**. Other pertinent variables such as the radial fatigue reserve factor (**FRFR**), the maximum stress (**SMAX**) and the worst cycle stress amplitude (**WCA**) are also reported.

2.4.2 Configuring a data check analysis

Overview

A data check runs the job file through the analysis pre-processor, without performing the fatigue analysis.

Job File Usage: `DATA_CHECK = {0.0 | 1.0 | 2.0 | 'fatigue-definition-file-name.mat'}`

- 0: Perform a full analysis (default)
- 1: Perform a data check analysis
- 2: Perform an analysis using the previous fatigue definition for the job
- '*fatigue-definition-file-name.mat*': Perform an analysis using the specified fatigue definition file

Data checks are useful for ensuring that the analysis definitions are valid, allowing the user to correct errors which may otherwise only become apparent after a long analysis run. The data check feature checks the following for consistency:

- Surface detection and ODB interface settings;
- results directory initialization;
- analysis continuation settings;
- material definitions and S–N interpolation;
- algorithm and mean stress correction settings;
- dataset and history definitions;
- principal stress histories;
- custom mean stress, FRF and surface finish data;
- yield and composite criteria analysis;
- nodal elimination;
- load proportionality; and
- virtual strain gauge definition.

Specifying a data check analysis

The user specifies a data check analysis in the job file.

Job File Usage: `DATA_CHECK = 1.0`

Output

Pertinent information regarding the data check run can be found in the message file in *Project\output\<jobName>*.

If the user requested field output from the job file, the worst tensor and principal stress per node for the whole model are written to the files *datacheck_tensor.dat* and *datacheck_principal.dat*, respectively.

The `DATA_CHECK` option can be used to save time by reusing a fatigue definition which was saved from a previous analysis. Reusing analysis data is discussed in Section 4.5.7.

2.4.3 Configuring an analysis from a text file

Overview

Quick Fatigue Tool includes a text file processor, which allows the user to submit a job from an ASCII text file containing only the options which are required to define the analysis. This results in job files which are less cumbersome than the standard .m file which must contain every option regardless of whether or not it is required.

To define a job from a text file, options are specified as keywords.

Text File Usage: *KEYWORD = value

Keywords are exactly the same as job file options, but they always begin with an asterisk (*). For example, the job file option **DATASET** is declared in the text file as *DATASET. Entries which do not begin with an asterisk, or are not proceeded with an equal sign (=) followed by a value, are ignored by the input file processor.

Job file options containing underscores are specified in the text file with spaces. For example, the option **JOB_NAME** is specified in the text file as *JOB NAME.

The following should be noted when defining jobs from a text file:

- it is not necessary to end the definition with a semi-colon;
- strings should be enclosed by apostrophes in order to distinguish them from numeric values;
- mathematical expressions must not be enclosed by brackets; and
- white spaces are ignored.

Syntax

The user must adhere to the following syntax when defining cells in the text file.

Cell type	Text input
Strings	{<string ₁ >, <string ₂ >, ..., <string _n >}
Numeric arrays	{[a ₁₁ , a ₁₂ , ..., a _{1n}], [a ₂₁ , a ₂₂ , ..., a _{2n}], ..., [a _{n1} , a _{n2} , ..., a _{nn}]}
Mixture of strings and numeric arrays	{<string ₁ >, [a ₁₁ , a ₁₂ , ..., a _{1n}]}

Any combination of strings and numerical inputs are supported, provided each element is separated by a comma.

Examples of mathematical expressions are given as

Expression type	Text input
Mathematical	a/b, a * b, etc.
Mathematical (with MATLAB functions)	sqrt(a/b), log10(a), etc.

Job submission

Jobs defined as text files are submitted from the command line.

Command Line Usage: `>> job <job-file-name>`
 `>> job <job-file-name> option`

The parameter **option** has the following definitions:

[interactive]

[datacheck | library]

interactive—print an echo of the message (.msg) file to the MATLAB command window.

datacheck—submit the analysis job as a data check analysis.

library—use the fatigue definition from the job `<job-file-name>` (same as **DATA_CHECK=2.0**).

The **datacheck** and **library** parameters are mutually exclusive.

Any file extension is accepted provided the contents is ASCII text. Job files with the extension `.inp` can be specified without appending `.inp` on the command line. For all other file types, the extension must be specified. Apostrophes are not required when specifying the input file name.

Example usage

An example of a text-based job file is given by the file `tutorial_intro.inp` in the `Data\tutorials\intro_tutorial` folder. Open the file and study its contents.

There is a text header at the beginning of the file, which is distinguished from the rest of the contents by double asterisks (**) at the beginning of each line. These lines are ignored by Quick Fatigue Tool.

The first keyword is *USER MATERIAL, which is used to define material data—the material definition is ended with *END MATERIAL. Guidance on creating material data in a text file is found in Section 5.5.

Subsequent keywords in `tutorial_intro.inp` specify analysis definitions for a uniaxial stress-life analysis. To submit the file for analysis, execute the following command:

Command Line Usage: `>> job tutorial_intro`

The material data is first read into the material database. If the material already exists in the `Data\material\local` directory, the user is prompted to overwrite the existing material data. The analysis keywords are then processed and the job is submitted for analysis.

Results of the fatigue analysis are written to `Project\output\tutorial_intro`.

2.5 The analysis method

1. A loading definition is created by combining elastic stress datasets with load histories to produce a scaled history of stresses for each item in the model
2. If a high frequency loading is provided, it is interpolated and superimposed onto the original load history
3. If requested, the load histories are pre-gated before the analysis which aims to remove small cycles from the loading
4. The principal stress history is calculated for each analysis item
5. The load history at each point in the model is assessed for proportionality. The critical plane step size may automatically be increased if the load is considered to be proportional
6. If requested, nodal elimination is performed which removes analysis items whose maximum stress range is less than the fatigue limit of the material
7. User stress-life data is interpolated to find the endurance curve for a fully-reversed cycle
8. Stresses are resolved onto planes in a spherical coordinate space to find the plane on which the most damaging stresses occur¹
9. Stresses on this plane are counted using the rainflow cycle counting method². If requested, the stress tensors on this plane are gated prior to cycle counting
10. If requested, the stress cycles are corrected for material non-linearity
11. The stress cycles are corrected for the effect of mean stress
12. A damage calculation is performed for each cycle using Miner's Rule of linear damage accumulation [7]. The endurance limit may be reduced to 25% of its original value if the cycle stress amplitudes are damaging
13. Steps 8-12 are repeated for each analysis item
14. If requested, the item with the worst life is analysed once more to calculate extensive output
15. If requested, Factor of Strength (FOS) iterations are performed. The damage is recalculated for each analysis item to obtain the linear loading scale factor which, if applied to the original loading, would result in the user-defined design life

¹ Only if the fatigue analysis algorithm is multiaxial

² Only if the load history contains more than two data points

3. Defining fatigue loadings

3.1 Loading methods

3.1.1 Overview

The loading definition forms the basis of the analysis, and describes the stress history at each point in the model. Loadings usually consist of stress datasets (user-defined or from FEA) and load histories. The stress datasets contain the static stress state at each location in the model. This can be at a node, integration point, centroid or otherwise. The load history defines the variation of the stresses through time. However, Quick Fatigue Tool does not distinguish elapsed time between loading points and hence the load history is treated as being rate-independent.

Quick Fatigue Tool includes the following approaches for defining fatigue loadings:

- uniaxial history;
- simple loading;
- multiple load history (scale and combine) loading;
- dataset sequences; and
- complex (block sequence) loading.

3.1.2 Syntax

Stress datasets are specified as ASCII text files:

```
'dataset-file-name.*' | {'dataset-file-name-1.*', 'dataset-file-name-2.*', ..., 'dataset-file-name-n.*'}
```

Load histories are specified as ASCII text files, or directly as one or more vectors:

```
'load-history-file-name.*' | {'history-file-name-1.*', 'history-file-name-2.*', ..., 'history-file-name-n.*'}
```

```
[h1, h2, ..., hn] | {[h11, h12, ..., h1n]1, [h21, h22, ..., h2n]2, ..., [hm1, hm2, ..., hmn]m}
```

3.1.3 Uniaxial history

A single load history is supplied without a stress dataset.

Job File Usage: **DATASET** = ''
 HISTORY = {*'history-file-name.*'* | [h_1, h_2, \dots, h_n] }

The load history is analysed without respect to a particular model, and is only valid for uniaxial states of stress. Uniaxial histories can only be used with the Uniaxial Stress-Life and Uniaxial Strain-Life algorithms.

Job File Usage: **ALGORITHM** = {3.0 | 'UNIAXIAL STRAIN'}
 ALGORITHM = {10.0 | 'UNIAXIAL STRESS'}

3.1.4 Simple Loading

A simple loading consists of a single stress dataset multiplied by a load history.

Job File Usage: **DATASET** = '*dataset-file-name.*'*
 HISTORY = {*'history-file-name.*'* | [h_1, h_2, \dots, h_n] }

An example of a simple fatigue loading is given by Figure 3.1.

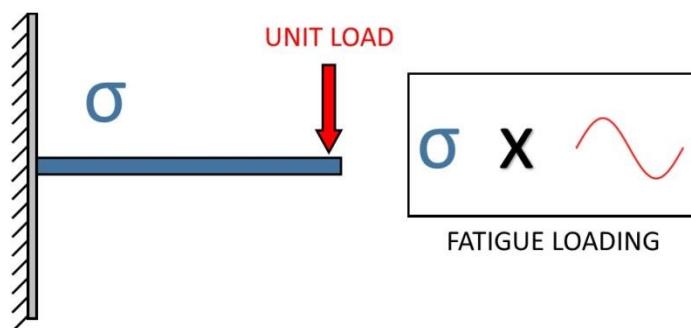


Figure 3.1: Demonstration of a simple loading. The stresses, σ , due to a unit load are multiplied by a load history

Constant amplitude loading

The load history is defined as a minimum and maximum value.

Job File Usage: **HISTORY** = [h_{min}, h_{max}]

The definition can be a single cycle, or k repetitions of the same cycle.

Job File Usage: **REPEATS** = k

Single load history

The load history is defined as a time history of the loading, scaled with the stress dataset. The dataset can be defined in two ways:

1. FEA stress from unit load, scaled by a time history of load values
2. FEA stress from maximum load, scaled by a time history of load scale factors

Both methods are valid provided that the FEA is linear and elastic.

3.1.5 Multiple load history (scale and combine) loading

A multiple load history consists of several stress datasets multiplied by the same number of histories. At each analysis item, the stress tensor from each dataset is scaled with its respective load history and combined into a single stress history. The number of stress datasets and load histories must be the same, although the number of history points in each load history need not be the same.

Job File Usage:

```

DATASET = {'dataset-file-name-1.*', 'dataset-file-name-2.*', ...,
           'dataset-file-name-n.*'}
HISTORY = {'history-file-name-1.*', 'history-file-name-2.*', ...,
           'history-file-name-n.*'}

```

An example of a scale and combine loading is given by Figure 3.2.

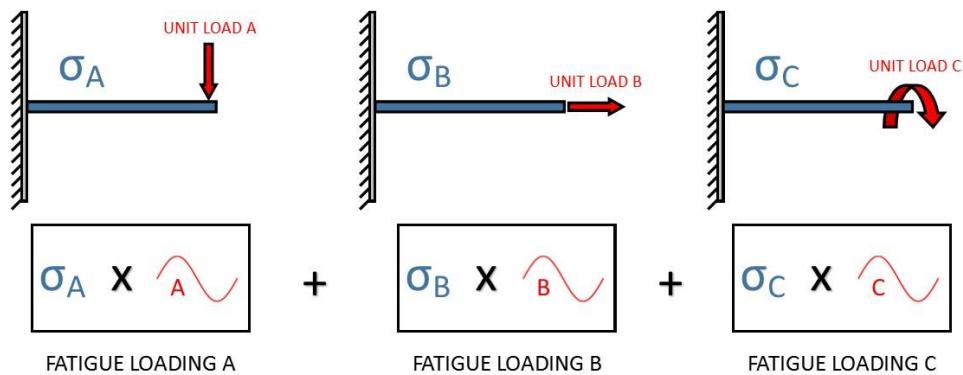


Figure 3.2: Demonstration of a multiple load history (scale and combine). The stresses, σ_A , σ_B and σ_C due to unit loads A, B and C are multiplied by their respective load histories and summed to produce the resultant fatigue loading

Scale and combine loading is only physically meaningful for elastic stresses. Each channel can be scaled by its respective load history since the load is directly proportional to the elastic FEA stress. The scale and combine method assumes that each loading channel is occurring simultaneously.

The load history may be defined by any combination of load history files and vectors.

Job File Usage:

HISTORY = {'history-file-name.*', [h₁, h₂, ..., h_n]}

3.1.6 Dataset sequences

A dataset sequence loading consists of several stress datasets.

Job File Usage:

```
DATASET = {'dataset-file-name-1.*', 'dataset-file-name-2.*',...,
'dataset-file-name-n.*'}
HISTORY = [ ]
```

An example of a dataset sequence loading is given by Figure 3.3.

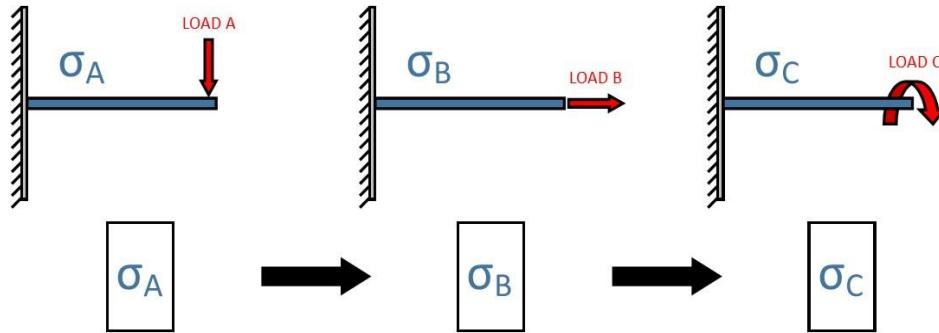


Figure 3.3: Demonstration of a stress dataset sequence loading. The fatigue loading is formed by the sequence of stress solutions σ_A , σ_B and σ_C , due to the applied loads A, B and C, respectively

Since the fatigue loading is completely described by the variation of stresses between each dataset, specification of load histories is not required.

3.1.7 Complex (block sequence) loading

Complex loadings consist of multiple loading blocks, each representing a stage of the component's duty. Block sequences are useful in proving ground tests where the component is subjected to a sequence of distinct loading events, and it is desirable to compute the individual damage contributions from each event.

Currently, sequences of loading blocks cannot be defined in a single job. However, Quick Fatigue Tool allows the user to run multiple jobs in series, wherein each job is treated as a separate loading block.

Job File Usage:

```
JOB_NAME = 'current-job-name'
CONTINUE_FROM = 'previous-job-name'
```

When a job is run as a continuation of a previous job, Quick Fatigue Tool calculates the individual damage contribution of the current job, then superimposes the result onto the previous job to give the cumulative damage of the block sequence.

Analysis continuation is discussed further in Section 4.8.

3.2 Creating a stress dataset file

3.2.1 Dataset structure

Stress datasets are text files containing a list of stress tensors. The simplest way to create a stress dataset file is to specify the tensor components as follows:

Line 1:	$\sigma_{11}, \sigma_{122}, \sigma_{133}, \sigma_{12}, \sigma_{13}, \sigma_{23}$
Line 2:	$\sigma_{211}, \sigma_{222}, \sigma_{233}, \sigma_{212}, \sigma_{213}, \sigma_{223}$
.	.
.	.
.	.
Line n	$\sigma_{n11}, \sigma_{n22}, \sigma_{n33}, \sigma_{n12}, \sigma_{n13}, \sigma_{n23}$

Each line defines the stress tensor at each location in the model.

3.2.2 Creating a dataset from Abaqus/Viewer

Stress datasets may be generated from finite element analysis (FEA). To create a stress dataset file from Abaqus/Viewer, complete the following steps:

1. In the Visualization module, from the main menu, select **Result → Options...**
2. Under “Averaging”, uncheck “Average element output at nodes”
3. From the main menu, select **Report → Field Output...**
4. Under “Step/Frame”, select the step and the frame in the analysis from which the stresses will be written
5. In the “Variable” tab, under “Output Variables”, select the position of the output (Integration Point, Centroid, Element Nodal or Unique Nodal)
6. Expand the variable “S: Stress components” and select all the available Cauchy tensor variables (S11, S22, S33, S12, S13 and S23); if plane stress elements are used, select (S11, S22, S33 and S12) and set **ELEMENT_TYPE**=1.0 in the job file; if one-dimensional elements are used, select (S11)
7. If the model contains results at multiple section points or plies, select **Settings...** from the section point method and specify the section point or ply of interest
8. In the “Setup” tab, set the file path (e.g. `\6.x-xx\Project\input\<filename>.rpt`)
9. Under “Data”, uncheck “Column totals” and “Column min/max”. Make sure “Field output” is checked
10. Click **OK**

3.2.3 Creating datasets from other FEA packages

If the user wishes to create a stress dataset from an FEA package other than Abaqus, the following standard data format must be observed for 3D stress elements:

MAIN POSITION ID (OPTIONAL)	SUB POSITION ID (OPTIONAL)	σ_{11}	σ_{22}	σ_{33}	σ_{12}	σ_{13}	σ_{23}
--------------------------------	-------------------------------	---------------	---------------	---------------	---------------	---------------	---------------

For example, a particular stress dataset may look like the following:

2	2	113.924E-03	-3.09283	77.0076E-03	-948.772E-03	-34.9769E-03	-1.08771
2	3	117.622E-03	-4.64155	-41.7606E-03	-390.323E-03	-31.3847E-03	-1.95848
2	89	142.139E-03	-2.54017	-57.3497E-03	-122.545E-03	179.476E-03	-1.97810
2	121	193.577E-03	800.952E-03	63.2928E-03	-889.262E-03	82.9093E-03	-1.12254
2	817	222.935E-03	-5.01846	9.43557E-03	-207.232E-03	-59.9510E-03	-1.93998
2	841	293.969E-03	-4.13284	50.9750E-03	-400.696E-03	-87.5143E-03	-1.40172
2	3029	200.802E-03	-3.41846	19.5431E-06	143.706E-03	178.775E-03	-2.00069
2	3053	275.228E-03	-1.49917	56.2520E-03	-145.282E-03	108.319E-03	-1.49560
3	57	12.4055	-6.48471	59.9500E-03	-2.88770	-1.17872	-1.87426
3	58	16.5065	-4.70019	-165.896E-03	-4.86937	-1.48686	-2.28258
3	820	13.1704	-5.57775	-208.927E-03	-3.59939	-2.71752	-3.96404
3	827	9.55963	-6.77472	-24.9396E-03	-1.75746	-2.11207	-3.14150
3	1017	11.4619	-13.3344	46.7205E-03	-2.41033	-774.306E-03	-2.23137
3	1018	15.2980	-12.7558	-121.544E-03	-4.58736	-1.05162	-2.75391
3	3032	12.2131	-11.4543	-144.094E-03	-3.12038	-1.93731	-4.75567
3	3039	8.75289	-11.2319	-34.1504E-03	-1.11163	-1.44957	-3.73494

The position labels can be arbitrary, but usually represent the location on the finite element model (e.g. *element.node*). Quick Fatigue Tool will quote the position of the shortest fatigue life. Position labels are not compulsory: The stress dataset can be specified with tensor information only. Furthermore, it is not compulsory to include both *main* and *sub* IDs. For example, if the stress data is unique nodal (nodal averaged), there is only one position ID which is the node number.

3.2.4 Creating datasets with different element types

Quick Fatigue Tool automatically recognises stress datasets from Abaqus containing multiple element types. Such files are split into regions, each of which defines the stress tensors for a specific element type. If the stress dataset file is user-defined, the following conventions must be observed:

3D stress elements:

MAIN POSITION ID (OPTIONAL)	SUB POSITION ID (OPTIONAL)	σ_{11}	σ_{22}	σ_{33}	σ_{12}	σ_{13}	σ_{23}
--------------------------------	-------------------------------	---------------	---------------	---------------	---------------	---------------	---------------

Plane stress elements without shell face information:

MAIN POSITION ID (OPTIONAL)	SUB POSITION ID (OPTIONAL)	σ_{11}	σ_{22}	σ_{33}	σ_{12}
--------------------------------	-------------------------------	---------------	---------------	---------------	---------------

Plane stress elements with shell face information:

MAIN POSITION ID (OPTIONAL)	SUB POSITION ID (OPTIONAL)	σ_{11} (+ve face)	σ_{11} (-ve face)	σ_{22} (+ve face)	σ_{22} (-ve face)	σ_{33} (+ve face)	σ_{33} (-ve face)	σ_{12} (+ve face)	σ_{12} (-ve face)
--------------------------------	-------------------------------	-----------------------------	-----------------------------	-----------------------------	-----------------------------	-----------------------------	-----------------------------	-----------------------------	-----------------------------

One-dimensional elements:

MAIN POSITION ID (OPTIONAL)	SUB POSITION ID (OPTIONAL)	σ_{11}
--------------------------------	-------------------------------	---------------

Below is an example of a user-defined stress dataset file containing two elements.

```
#REGION_1:
    1   1   297.850   31.6377   70.1334   -4.77388E-09   -921.773   -255.103
    1   2   70.1306   31.6377   297.843   255.077E-06   -1.55842   2.28914
    1   3   297.849   31.6377   70.1337   -4.06311E-09   -1.73031   255.097

#REGION_2:
    2   1   329.355   338.458   31.6551   31.6885   0.   0.   3.86495   4.83387
    2   2   391.347   400.629   31.5931   31.6263   0.   0.   -2.39578   -2.17509
    2   3   329.352   338.455   31.6551   31.6886   0.   0.   2.87967   3.30657
```

Each element region must be declared by a text header in order to be recognised, and the header must start with a non-numeric character. In the above example, *REGION_1* defines a 3D element and *REGION_2* defines a 2D element with results at both shell faces. Both elements are defined with element-nodal (nodal un-averaged) position labels. All the datasets in the loading must be defined with the same position labels otherwise the analysis will not run. To check whether or not the dataset definition was processed correctly, Quick Fatigue Tool prints the number of detected regions to the message file. This can be found in *Project\output\<jobName>\<jobName>.msg*.

Resolving element type ambiguities

Quick Fatigue Tool will automatically detect the element type based on the number of columns in the dataset file. For example, if there are five columns, this will be interpreted as plane stress (four columns define the tensor) with one column defining the element position (either unique nodal or centroidal). If the dataset file contains six columns, this could either be interpreted as 3D stress (all six columns define the stress tensor) with no position labels, or plane stress (four columns define the tensor) with two columns defining the element position (either element-nodal or integration point).

Job File Usage: **ELEMENT_TYPE = {0.0 | 1.0}**

- 0. Do not allow plane stress elements (default)
- 1. Allow plane stress element assumption

A complete description of how dataset files are interpreted is provided in Section 3.6.

3.2.5 Specifying stress dataset units

Since fatigue analysis uses the *SI (mm)* system of units, stress datasets are assumed to have units of *Megapascals (MPa)* by default. If the datasets use a different system of units, these must be specified in the job file so that Quick Fatigue Tool can convert them into the *SI (mm)* system.

Job File Usage: **UNITS** = {'Pa' | 'kPa' | 'MPa' | 'psi' | 'ksi' | 'Msi'}

If the stress dataset units are not listed, they can be user-defined.

Job File Usage: **UNITS** = 0
CONV = k_{conv}

The parameter k_{conv} is a constant which converts the stress dataset units into *Pascals (Pa)*.

$$[Pa] = k_{conv} \cdot [\text{dataset units}] \quad [3.1]$$

The stress data S is converted into the *SI (mm)* system using Equation 3.2.

$$S [MPa] = 10^{-6} \cdot k_{conv} \cdot S [\text{dataset units}] \quad [3.2]$$

3.3 Creating a load history

3.1.1 Overview

Load histories can be defined in three ways:

1. from a text file;
2. as a direct definition; and
3. as a workspace variable.

3.1.2 Specifying load histories from a text file

If the load history is defined from a text file, it must contain a single $1 \times n$ or $n \times 1$ vector of loading points, as follows:

$$\begin{array}{ll} P_1 & \leftarrow \text{First loading point} \\ P_2 & \\ \cdot & \\ \cdot & \\ P_n & \leftarrow \text{Last loading point} \end{array}$$

For example, a fully-reversed load history would look like that of Figure 3.4.

Job File Usage: `HISTORY = 'history-file-name.*'`

Load history files must be stored in the *Project\input* folder in order for Quick Fatigue Tool to locate the data.

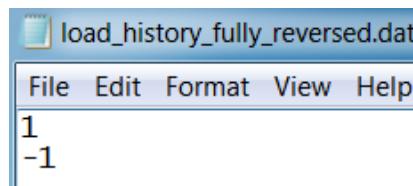


Figure 3.4: Fully-reversed load history

3.1.3 Creating a load history as a direct definition

Load histories can be specified in the job file as a $1 \times n$ vector of scale factors.

Job File Usage: **HISTORY** = $[h_1, h_2, \dots, h_n]$

Alternatively, the load history can be defined as a function.

Job File Usage: **HISTORY** = $A * sind(1:n)$

where A is a stress amplitude scale factor.

3.1.4 Creating a load history as a workspace variable

If the load history is defined as a workspace variable, it must be a $1 \times n$ or $n \times 1$ numerical array.

Job File Usage: **HISTORY** = $\{var_1, var_2, \dots, var_n\}$

In addition, the variables declared in **HISTORY** must also be specified as inputs to the function declaration and the function call.

Job File Usage: **function** [] = <jobName>(var₁, var₂, ..., var_n)

The job is then submitted by executing the job file from the command line.

Command Line Usage: >> <jobName>(var₁, var₂, ..., var_n)

For scale and combine loadings, it is possible to define a load history using a combination of text files, direct definitions and workspace variables.

Job File Usage: **HISTORY** = {'*history-file-name.**', [x₁, x₂], var}

3.1.5 Treatment of multiple load histories

The load histories do not have to be the same length. Before the analysis, all the load histories will be modified to have the same length by appending zeroes to the shorter histories. However, in order to maximise the reliability and performance of the cycle counting algorithm, it is strongly recommended that the loadings have a similar length.

Note that multiple load histories are not supported for uniaxial analysis.

3.4 Load modulation

3.4.1 Overview

The fatigue loading can be scaled and offset using the **SCALE** and **OFFSET** options, respectively. The scaled and offset fatigue load, S^* , is given by Equation 3.3.

$$S^* = S \times [(scale \times H) + offset] \quad [3.3]$$

3.4.2 Defining load scale factors

The user specifies load scale factors in the job file.

Job File Usage: **SCALE** = $[F_{s1}, F_{s2}, \dots, F_{sn}]$

If the analysis is a scale and combine loading, n is the number of dataset-history pairs; each load scale factor is multiplied by its respective dataset-history pair. If the analysis is a dataset sequence, n is the number of datasets; each load scale factor is multiplied by its respective dataset in the sequence.

If the user specifies the Uniaxial Stress-Life algorithm, a single scale factor may be specified. Load history scales can be used with any loading methods.

3.4.3 Defining load offset values

The user specifies load offset values in the job file.

Job File Usage: **OFFSET** = $[F_{o1}, F_{o2}, \dots, F_{on}]$

where n is the number of dataset-history pairs; each load offset value is summed with its respective dataset-history pair.

Since load offset values are applied to the load history points only, they may not be used with dataset sequence loadings. Load offsets may be used with all other loading methods.

3.5 High frequency loadings

3.5.1 Overview

The scale and combine method outlined in Section 3.1 does not distinguish between elapsed time and can produce physically incorrect load histories if two load signals with very different period are analysed together. The solution is to superimpose the higher frequency load data onto the lower frequency data.

3.5.2 Defining high frequency loadings

Take the example of a piston which experiences combined thermal and mechanical stresses shown in Figure 3.5.

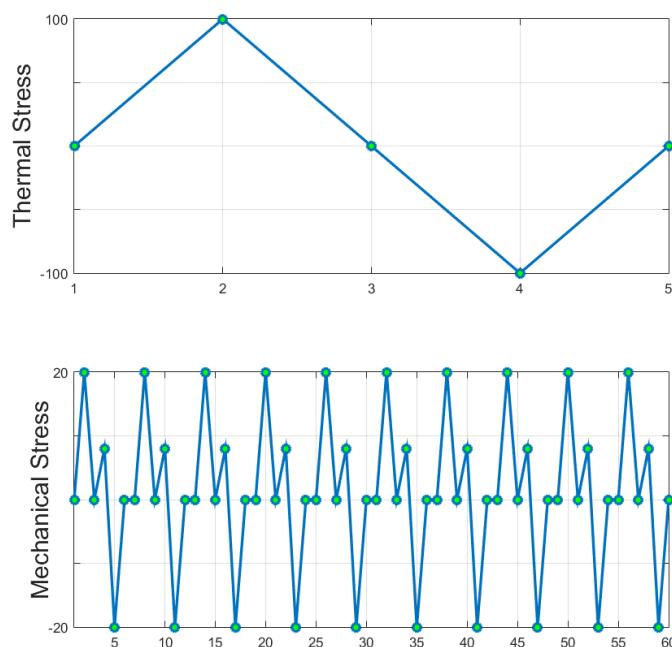


Figure 3.5: Thermal and mechanical load signals occurring over the same period

For a scale and combine loading, the two signals are defined by the load histories in Figure 3.5.

Normalised thermal load [0, 1, 0, -1, 0]

[0,1,0,0.4,-1,0,0,1,0,0.4,-1,0,0,1,0,0.4,-

Normalised mechanical load

A problem arises if the two loads occur over the same time interval. Since the mechanical load has many more time points than the thermal load, Quick Fatigue Tool will append most of the mechanical load onto the end of the load history. Using a standard scale and combine, the resulting load history would be that of Figure 3.6.

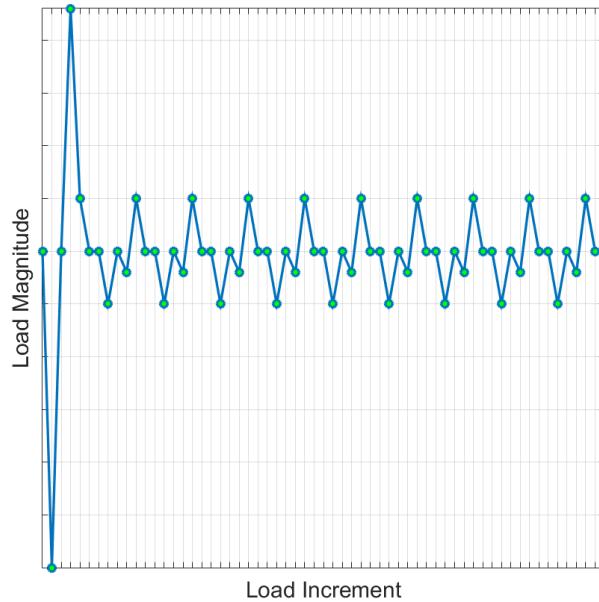


Figure 3.6: Result of using the scale and combine technique for the thermal-mechanical load

This loading definition is physically incorrect because it does not allow for the fact that the two loads occur simultaneously over the same period. The solution is to define the mechanical load as a high frequency dataset. The modified load histories are as follows:

Normalised thermal load [0, 1, 0, -1, 0]

Normalised mechanical load [0, 1, 0, 0.4, -1, 0]

In this case, the high frequency data is specified as a single repeat of the mechanical load.

Job File Usage:

<code>HF_DATASET = 'mechanical-dataset-file-name.*'</code> <code>HF_HISTORY = 'mechanical-history-file-name.*'</code> <code>HF_TIME = {100.0, 10.0}</code>
--

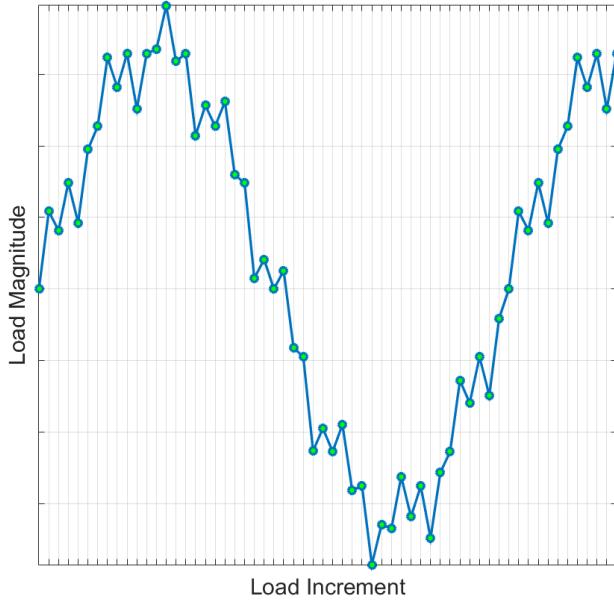


Figure 3.7: Result of using high frequency loading for the thermal-mechanical load

High frequency loading and load histories are specified in the same manner as standard datasets and load histories. In order for Quick Fatigue Tool to correctly superimpose the high frequency dataset(s), it must know the period for both loadings. In this example, the period of the low frequency data is 100 seconds and the period of a single repeat of the high frequency data is 10 seconds. This means that the high frequency dataset will be superimposed $100/10 = 10$ times into the low frequency data.

The resulting load history is shown in Figure 3.7.

Quick Fatigue Tool interpolates the thermal load so that it contains the correct number of data points for the mechanical load to be superimposed, without resulting in trailing data. In using this technique, the mechanical data is correctly represented as occurring over the same period as the thermal data.

The same loading methods apply as those outlined in Section 3.1.

3.5.3 Example usage

An example input file containing a high frequency loading definition can be found in *Data\tutorials\intro_tutorial\tutorial_high_frequency.inp*.

The job is submitted by executing the following command:

```
>> job tutorial_high_frequency
```

Results are written to *Project\output\tutorial_high_frequency*.

3.5.4 Additional guidance

The user should take into account the following points when using high frequency loading:

- high frequency loading requires the Signal Processing Toolbox to work;
- high frequency loading should be used if two or more load histories occur over the same time interval, where one or more of the load histories is a repetitive load at a much higher frequency;
- the number of analysis items in the high frequency loading must be the same as the number of items in all other stress datasets. If specific items are listed in the job file, those same items will be used in the high frequency loading;
- if the original datasets contain stresses at shell faces, the high frequency data must also contain shell face data;
- the main load history should contain at least three data points, otherwise the high frequency loading may not be interpolated properly. If the original load history contains only two data points, a zero value will be appended to the end of the history;
- when defining the high frequency load history, only a single cycle needs to be defined, along with the period for that cycle. If the entire load history is provided, the resulting load history will be incorrect;
- if load history pre-gating is enabled, the original datasets may be modified prior to the high frequency loading being added. This may result in an unexpected load history;
- if the high frequency data is not in the form of a peak-valley sequence (the loading contains intermediate data between turning points), this data will not be considered by the selected gating criterion;
- the units of the high and low frequency data must be the same; and
- using high frequency loading can increase the analysis time dramatically.

3.6 The dataset processor

3.6.1 Determining the dataset type

Quick Fatigue Tool determines the type of dataset based on the number of columns in the dataset file. The following table describes how datasets are processed.

Number of columns in dataset	Assumption
1	Element type: 1D stress Position: Unknown
2	<i>IF ELEMENT_TYPE=0.0:</i> Element type: 1D stress Position: Unique-nodal or centroid <i>ELSEIF ELEMENT_TYPE=1.0:</i> Element type: 2D stress Position: Unknown
3	<i>IF ELEMENT_TYPE=0.0:</i> Element type: 1D stress Position: Element-nodal or integration point <i>ELSEIF ELEMENT_TYPE=1.0:</i> Element type: 2D stress Position: Unique-nodal or centroid
4	<i>IF ELEMENT_TYPE=0.0:</i> Element type: 2D stress Position: Element-nodal or integration point <i>ELSEIF ELEMENT_TYPE=1.0:</i> Element type: Plane stress Position: Unknown
5	Element type: Plane stress Position: Unique-nodal or centroid
6	<i>IF ELEMENT_TYPE=0.0:</i> Element type: 3D stress Position: Unknown <i>ELSEIF ELEMENT_TYPE=1.0:</i> Element type: Plane stress Position: Element-nodal or integration point
7	Element type: 3D stress Position: Unique-nodal or centroid
8	Element type: 3D stress Position: Element-nodal or integration point

9	Element type: Plane stress with shell face data Position: Unique-nodal or centroid
10	Element type: Plane stress with shell face data Position: Element-nodal or integration point
> 10	Not applicable. Quick Fatigue Tool will exit with an error.

3.6.2 1D stress element definition

One-dimensional stress elements are links which are used as beams or trusses. An example of linear and quadratic one-dimensional elements is shown in Figure 3.8.

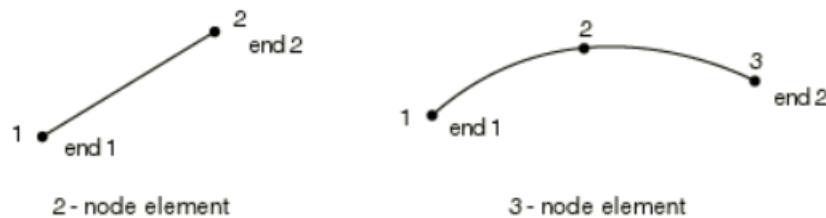


Figure 3.8: One-dimensional solid elements (courtesy of Dassault Systemes S.E.)

One-dimensional elements are defined in the stress dataset file as follows:

Required components of stress:	σ_{11}
--------------------------------	---------------

Available element result positions	Total number of columns in dataset
None	1
Unique-nodal and centroid	2
Element-nodal and integration point	3

In order for the dataset processor to recognise one-dimensional elements, plane stress element recognition must be disabled.

Job File Usage: **ELEMENT_TYPE** = 0.0

3.6.3 2D stress element definition

Two-dimensional stress elements are triangles and quadrilaterals which are used as membranes, surfaces and axisymmetric sections. An example of linear and quadratic two-dimensional elements is shown in Figure 3.9.

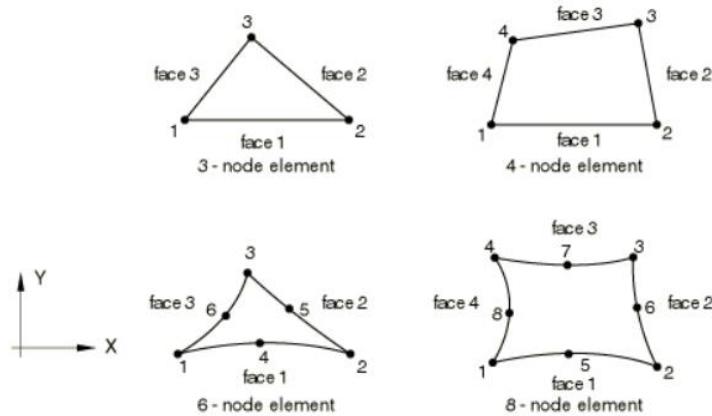


Figure 3.9: Two-dimensional and plane stress solid elements (courtesy of Dassault Systemes S.E.)

One-dimensional elements are defined in the stress dataset file as follows:

Required components of stress:	σ_{11}, σ_{12}
--------------------------------	----------------------------

Available element result position	Total number of columns in dataset	ELEMENT_TYPE
None	2	1
Unique-nodal and centroid	3	1
Element-nodal and integration point	4	0

In order for the dataset processor to recognise two-dimensional stress elements, the option **ELEMENT_TYPE** must be set according to the element result position in the dataset file.

3.6.4 Plane stress element definition

Plane stress elements are triangles and quadrilaterals which are used as shells, membranes and surfaces. An example of linear and quadratic plane stress elements is shown in Figure 3.9.

Plane stress elements are defined in the stress dataset file as follows:

Required components of stress:	$\sigma_{11}, \sigma_{12}, \sigma_{33}, \sigma_{12}$
---------------------------------------	--

Available element result positions	Total number of columns in dataset
None	4
Unique-nodal and centroid	5
Element-nodal and integration point	6
Unique-nodal and centroid (shell face data)	9
Element-nodal and integration point (shell face data)	10

In order for the dataset processor to recognise plane stress elements, plane stress element recognition must be enabled.

Job File Usage: `ELEMENT_TYPE = 1.0`

If the stress datasets originate from an FE model consisting of plane stress elements, the `.rpt` file can contain results on both faces, as shown by Figure 3.10.

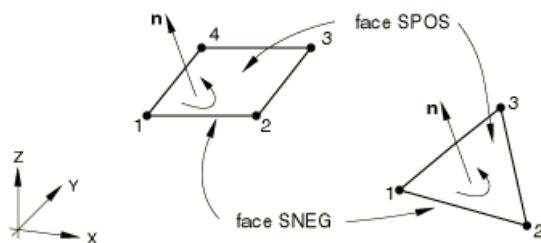


Figure 3.10: Positive normals for three-dimensional conventional shell elements (courtesy of Dassault Systemes S.E.)

Quick Fatigue Tool can read the stresses from either the positive or negative face. The default face is defined as a variable in the environment file.

Environment File Usage: `shellLocation = {1.0 | 2.0}`

1. Negative (SNEG) element face (default)
2. Positive (SPOS) element face

3.6.5 3D stress element definition

Three-dimensional stress elements are tetrahedra, pyramids, wedges and hexahedra which are used for solid structures. An example of linear and quadratic three-dimensional elements is shown in Figure 3.11.

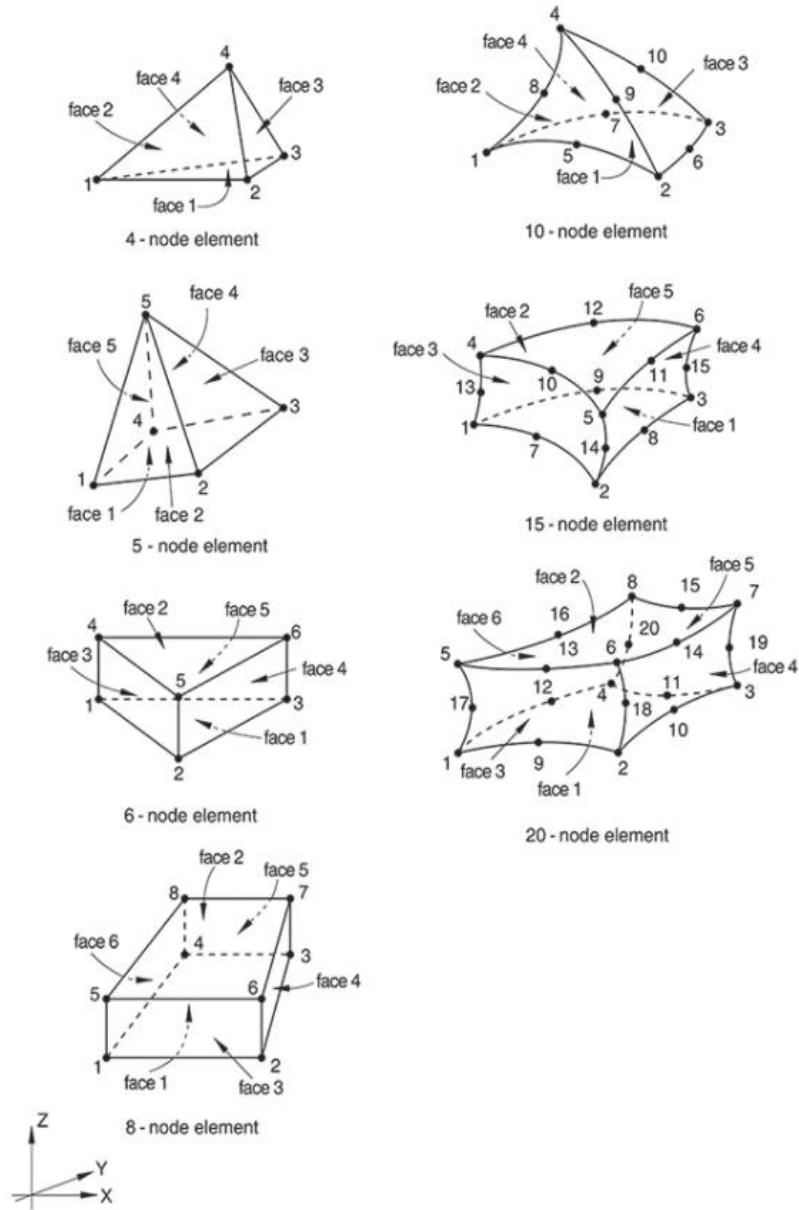


Figure 3.11: Three-dimensional solid elements (courtesy of Dassault Systemes S.E.)

Three-dimensional elements are defined in the stress dataset file as follows:

Required components of stress:	$\sigma_{11}, \sigma_{22}, \sigma_{33}, \sigma_{12}, \sigma_{13}, \sigma_{23}$
---------------------------------------	--

Available element result positions	Total number of columns in dataset
None	6
Unique-nodal and centroid	7
Element-nodal and integration point	8

3.6.6 Multiple element groups

If the stress dataset file was written from an Abaqus output database, it is possible for the data to be separated into multiple regions. This can happen if there are multiple part instances in the model, or if the model contains a mixture of element types. In such cases, Quick Fatigue Tool will automatically process each region and concatenate the stress tensors after all the datasets have been read.

Common examples of when the dataset can contain multiple regions are the analysis of surfaces with in-plane residual stress and/or surface finish. The recommended practice is to create a skin on the surface of the FE model and apply the residual stress and surface finish definitions on the skin elements using the **GROUP** option in the job file (see Sections 4.3 and 4.4 for more detailed information). Because of the combination of plane stress elements forming the skin of the component and the underlying 3D elements, Abaqus (and possibly other FEA packages) may assign duplicate node numbers between the skin and the solid bulk.

Although the fatigue calculation is not affected by the presence of duplicate node numbers, Quick Fatigue Tool may report the results at incorrect locations. Problems may also arise when writing results back to an Abaqus .odb file because Quick Fatigue Tool is unable to resolve the correct location of the node on the finite element mesh. Thus, the visualization in Abaqus/Viewer could be incorrect.

The workaround in such cases is to use stresses at the element nodes. This ensures that each node in the model has a unique identifier even in the presence of multiple element regions.

4. Analysis techniques

4.1 Background

Quick Fatigue Tool provides a selection of analysis techniques. These techniques provide useful tools for performing your analysis more efficiently and effectively.

The techniques described in this section are specified in the job and environment files. For detailed guidance on the usage of job file options and environment variables, consult the document *Quick Fatigue Tool User Settings Reference Guide*.

4.2 Treatment of nonlinearity

4.2.1 Overview

Quick Fatigue Tool always assumes that the dataset stresses are linear elastic.

When a stress-based fatigue analysis algorithm is specified, the stresses are not corrected for the effect of plasticity. This is because the stress-life methodology has been devised for linear elastic material data.

When a strain-based fatigue analysis algorithm is specified, the stresses are corrected for the effect of plasticity using the Ramberg-Osgood nonlinear elastic strain-hardening relationship. This function approximates the local nonlinear stress from elastic stresses [8] [9].

Elastic stresses are converted to nonlinear elastic stresses using Equation 4.1.

$$\varepsilon^{pl} = \frac{\sigma^{pl}}{E} + \left(\frac{\sigma^{pl}}{K} \right)^{1/n} \quad [4.1]$$

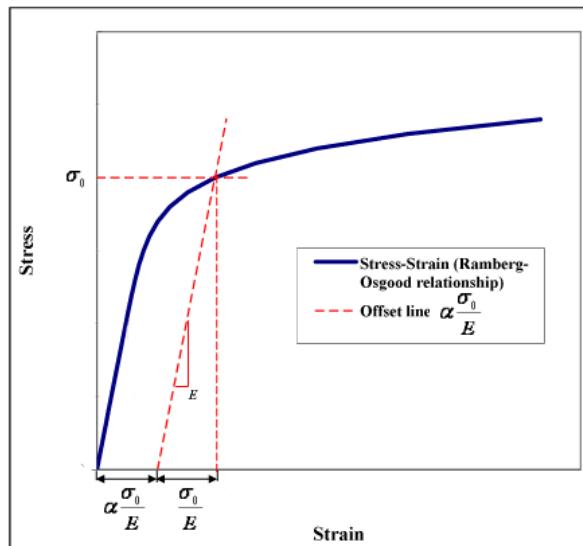


Figure 4.1: Generic representation of the stress-strain curve using the Ramberg-Osgood equation

The difference in the monotonic response between the elastic Hookean and the Ramberg-Osgood model is shown in Figure 4.1. For the offset line, the parameter α is defined as $K(\sigma_0/E)^{n-1}$, where σ_0 is the yield strength.

4.2.2 Limitations of the nonlinear model

In order to convert between linear and nonlinear quantities, Quick Fatigue Tool uses Neuber's Rule, which stipulates that the accumulated strain energy is the same at a notch as it would be in an elastic stress field far away.

The correction therefore is only valid for local notch plasticity. For smooth specimens, care must be taken. If the stress at the notch is below the yield strength, stress redistribution occurs, resulting in a larger plastic zone. Neuber's Rule does not work as well in these cases because it assumes that the peak stress is localised and that the plastic zone is surrounded by a comparatively large elastic zone, forcing the plastic zone to behave similarly to the nearby elastic stress field.

The Stress-Life methodology is based on the elastic stress at a point on the component which is not affected significantly by local stress concentrations. Traditionally, the elastic stress is used with an S-N curve which has been corrected with a notch factor to account for the presence of the stress concentration. As such, the Stress-Life methodology is not intended to be used with true stress quantities. This causes problems when performing fatigue estimates from elastic FEA, because the stresses at the notch are typically over-estimated. This can result in highly conservative life predictions compared to Strain-Life methods. For cases where the stress concentration is judged to be significant, the following is recommended in lieu of the plasticity correction:

1. Limit the fatigue analysis to the elastic stress a small distance away from the notch and apply a correction factor using the **KT_DEF**, **KT_CURVE**, **NOTCH_CONSTANT** and **NOTCH_RADIUS** options. Notch factors for specific geometries are readily available in the literature
2. Use the Strain-Life methodology

4.3 Surface finish and notch sensitivity

4.3.1 Surface finish

Surface roughness has a strong influence on the component's resistance to crack initiation [10]. While FEA is able to account for stress concentrations which arise from geometric complexity, the effect of surface finish cannot be modelled directly. Instead, a surface stress concentration factor, K_t , may be used to scale the endurance curve so that it corresponds more accurately to the surface strength of the material.

The result of applying a stress concentration factor to the endurance curve is shown in Figure 4.2.

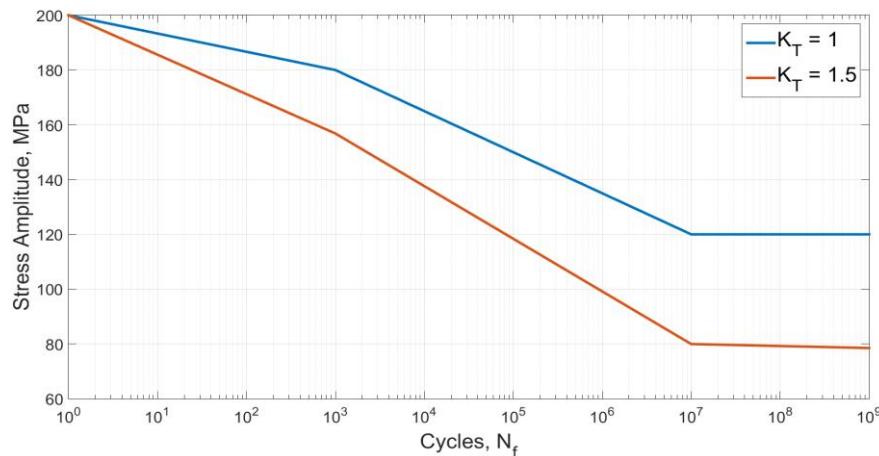


Figure 4.2: Reference endurance curve for $K_t = 1$ and after applying a stress concentration factor of $K_t = 1.5$

Quick Fatigue tool allows the surface finish to be defined in three ways:

1. as a surface stress concentration factor (K_t value);
2. from a list of surface finish types (R_a curve); and
3. as a surface roughness value (R_z value).

Defining the surface finish as a K_T value

The surface finish is specified as the stress concentration factor by leaving the value of **KT_CURVE** blank.

Job File Usage:

KT_DEF = K_T
KT_CURVE = []

Defining the surface finish as an R_a curve

The surface finish is defined as a surface type by specifying the appropriate R_a curve from a surface finish (.kt) file. Surface finish files are located in the *Data\kt* directory.

The surface finish definition files contain pre-defined curves for various surface finishes, as a function of the material's ultimate tensile strength.

The user specifies the surface finish file name '*surface-finish-file-name.kt*' and the curve number n corresponding to the surface type.

Job File Usage:

KT_DEF = ' <i>surface-finish-file-name.kt</i> '
KT_CURVE = n

The file 'default.kt' is plotted in Figure 4.3 as an example. Based on the chosen curve and the ultimate tensile strength of the material, Quick Fatigue Tool linearly interpolates to find the corresponding value of K_t . If the material's ultimate tensile strength exceeds the range specified by the K_t curve, the last value of K_t is used.

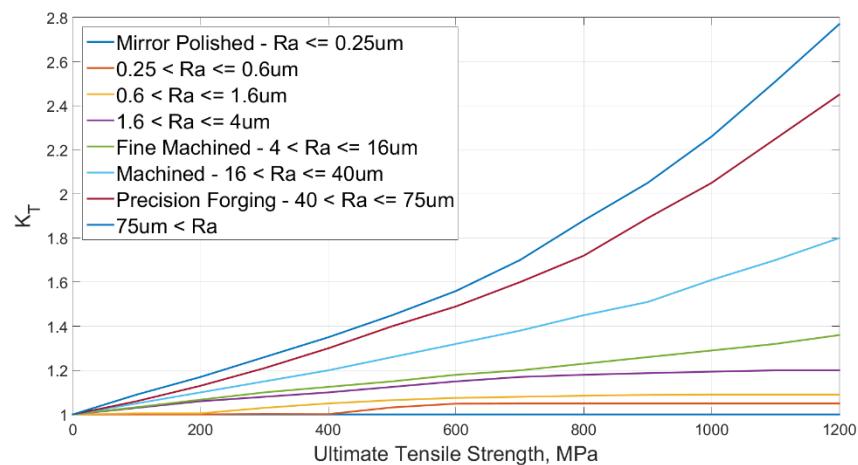


Figure 4.3: K_t curves for various surface finishes, from the file 'default.kt'

The following .kt files and the available curves are shown as

' <i>default.kt</i> '	Surface Finish
1	Mirror Polished – Ra <= 0.25um
2	0.25 < Ra <= 0.6um
3	0.6 < Ra <= 1.6um
4	1.6 < Ra <= 4um
5	Fine Machined – 4 < Ra <= 16um
6	Machined – 16 < Ra <= 40um
7	Precision Forging – 40 < Ra <= 75um
8	75um < Ra

' <i>juvinall-1967.kt</i> '	Surface finish
1	Mirror Polished
2	Fine-ground or commercially polished
3	Machined
4	Hot-rolled
5	As forged
6	Corroded in tap water
7	Corroded in salt water

'rcjohnson-1973.kt'	Surface finish
1	AA = 1uins
2	AA = 2uins
3	AA = 4uins
4	AA = 8uins
5	AA = 16uins
6	AA = 32uins
7	AA = 83uins
8	AA = 125uins
9	AA = 250uins
10	AA = 500uins
11	AA = 1000uins
12	AA = 2000uins

It is possible to specify the surface finish from a user-defined .kt file. The following file format must be obeyed:

First column: Range of UTS values over which K_t is defined

Second column: K_t values for the first curve

Third column: K_t values for the second curve

Nth column: K_t values for the (N-1)th curve

Defining the surface finish as an R_z value

The surface finish is defined as a surface roughness by specifying the appropriate R_z curve from a surface roughness (.ktx) file. Surface roughness files are located in the *Data\kt* directory.

The surface roughness definition files contain pre-defined curves for various surface roughness values, as a function of the material's ultimate tensile strength.

The user specifies the surface roughness file name '*surface-roughness-file-name.ktx*' and the curve number n corresponding to the surface roughness.

Job File Usage: *KT_DEF = 'surface-roughness-file-name.ktx'*
KT_CURVE = n

The '*Niemann-Winter-Rolled-Steel.ktx*' file is plotted in Figure 4.4 as an example. Quick fatigue tool linearly interpolates to find the K_t curve which corresponds to the user-specified R_z value. If the surface roughness value exceeds the maximum surface roughness defined in the data, the last set of K_t values are used.

Based on the ultimate tensile strength of the material, Quick Fatigue Tool linearly interpolates once more to find the corresponding value of K_t . If the material's ultimate tensile strength exceeds the range specified by the K_t curve, the last value of K_t is used.

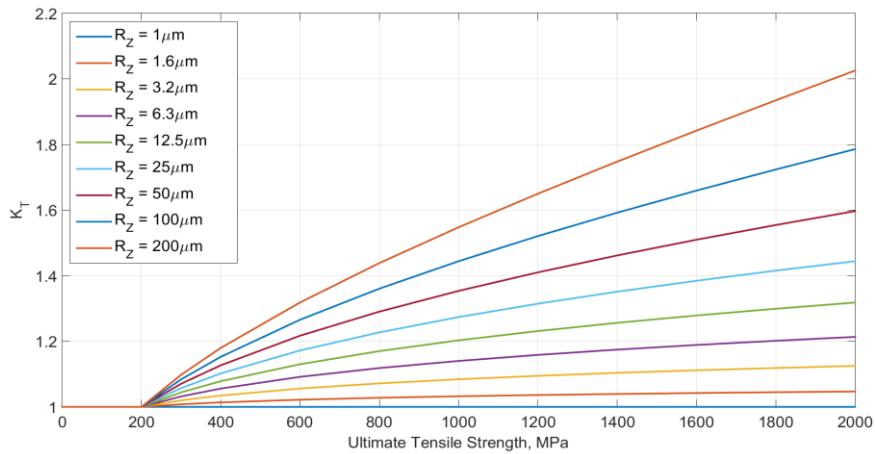


Figure 4.4: K_t curves for various surface roughness values, from the file ‘Niemann-Winter-Rolled-Steel.ktx’

The following .ktx files and surface roughness ranges are shown as

‘Niemann-Winter-Cast-Iron-Lamellar-Graphite.ktx’	
UTS Range	0 – 2000Mpa
R_z Range	1 – 200um

‘Niemann-Winter-Cast-Iron-Nodular-Graphite.ktx’	
UTS Range	0 – 2000Mpa
R_z Range	1 – 200um

‘Niemann-Winter-Cast-Steel.ktx’	
UTS Range	0 – 2000Mpa
R_z Range	1 – 200um

‘Niemann-Winter-Malleable-Cast-Iron.ktx’	
UTS Range	0 – 2000Mpa
R_z Range	1 – 200um

‘Niemann-Winter-Rolled-Steel.ktx’	
UTS Range	0 – 2000Mpa
R_z Range	1 – 200um

It is possible to specify the surface finish from a user-defined *.ktx* file. The following file format must be obeyed:

First row: Range of R_z values over which K_t values are defined

Second row: First UTS value, followed by corresponding K_t values for each R_z value

Third row: Second UTS value, followed by corresponding K_t values for each R_z value

Nth row: ($N - 1$)*th* UTS value, followed by corresponding K_t values for each R_z value

4.3.2 Effect of notch sensitivity

If the component contains a notch, the stress-life curve may require modification to account for the notch sensitivity of the material, since the stresses which contribute to fatigue of a notched component are not on the notch root surface, but a small distance into the subsurface. As such, for notch-insensitive materials, the stress concentration factor in fatigue is different to the elastic stress concentration factor, K_t . This is termed the fatigue notch factor, K_f .

The value of K_f can be approximated in several ways, and is set in the environment file.

Environment File Usage: `notchFactorEstimation = {1.0 | 2.0 | 3.0 | 4.0 | 5.0 | 6.0}`

1. Peterson (default)
2. Peterson B
3. Neuber
4. Harris
5. Heywood
6. Notch sensitivity

Peterson (default)

Quick Fatigue Tool is optimized to use stresses from finite element analysis. Therefore, the effect of K_t is implicit in the stress solution. By default, the reduction in fatigue strength due to elastic stress concentration is calculated using results obtained by Peterson. Equation 4.4 is used to scale the value of K_t as a function of endurance [11].

$$K_t(N) = 1 + \frac{K_t - 1}{0.915 + \frac{200}{(\log_{10} N)^4}} \quad [4.4]$$

The endurance curve is scaled by $K_t(N)^{-1}$ at each value of N. The Peterson relationship is visualized by Figure 4.5, for values of $K_t = 2, 3, 4$.

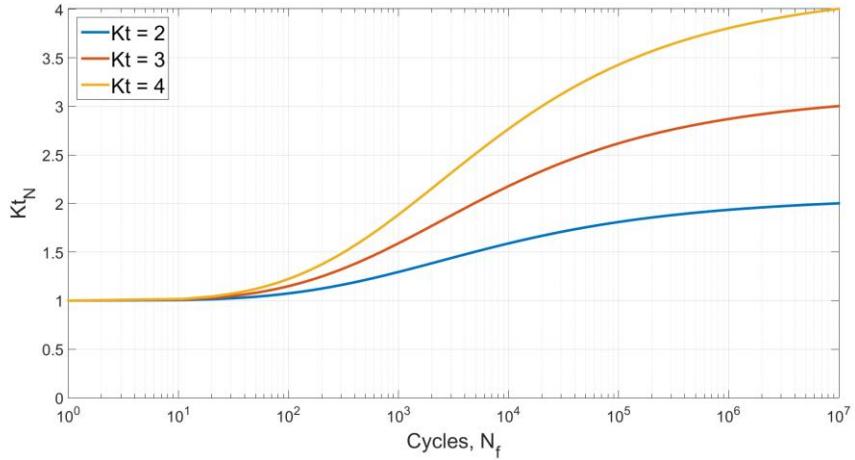


Figure 4.5: $K_t(N)$ as a function of endurance, for $K_t = 2, 3, 4$.

Peterson B

Peterson observed that, in general, good approximations for $R = -1$ loading can be obtained using Equation 4.5 [12].

$$K_f = 1 + \frac{K_t - 1}{1 + \frac{a}{r}} \quad [4.5]$$

where a is a characteristic length and r is the notch root radius. The value of a can be determined empirically as a function of the ultimate tensile strength, σ_u :

Material	a
Steel	$0.0254 \left(\frac{2070}{\sigma_u [MPa]} \right)^{1.8} mm$
Aluminium Alloy	$0.635mm$

Neuber

For parallel side grooves, Neuber developed the following approximate formula for the notch factor for $R = -1$ loading [13]:

$$K_f = 1 + \frac{K_t - 1}{1 + \sqrt{\frac{\rho}{r}}} \quad [4.6]$$

where ρ is a characteristic length.

Harris

Harris proposed the relationship in Equation 4.7 [14].

$$K_f = e^{-\frac{r}{\rho H}} + K_t \left(1 - e^{-\frac{r}{\rho H}} \right) \quad [4.7]$$

where ρH is a characteristic length. Suggested values of ρH are given below.

Material	ρH
Steel	$10^{-2} \left(\frac{84.3}{\sigma_u [\text{ksi}]} \right)^2 \text{in}$
Aluminium Alloy	0.004in

Heywood

Heywood proposed the relationship in Equation 4.8 [15].

$$K_f = \frac{K_t}{1 + 2 \sqrt{\frac{a_H}{r}}} \quad [4.8]$$

where a_H is a characteristic length. The value of a_H varies depending on the notch type. Typical values proposed for steel are shown in the table below. The values assume that the notch root radius is measured in inches.

Notch Type	a_H
Hole	$\left(\frac{5}{\sigma_u [\text{ksi}]} \right)^2$
Shoulder	$\left(\frac{4}{\sigma_u [\text{ksi}]} \right)^2$
Groove	$\left(\frac{3}{\sigma_u [\text{ksi}]} \right)^2$

Notch sensitivity

The fatigue notch factor can be defined in terms of the notch sensitivity, q .

$$K_f = 1 + q(K_t - 1) \quad [4.9]$$

Typical values of q for steels and aluminium alloys are shown in Figures 4.6-7 for bending and torsion, respectively [16].

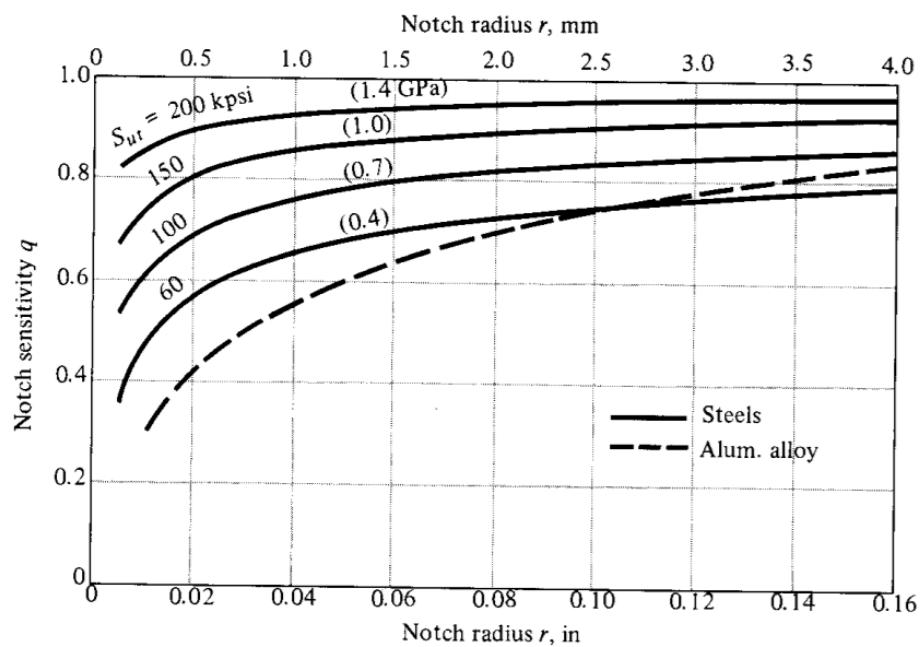


Figure 4.6: Notch sensitivity factors for steels and aluminium alloys (bending)

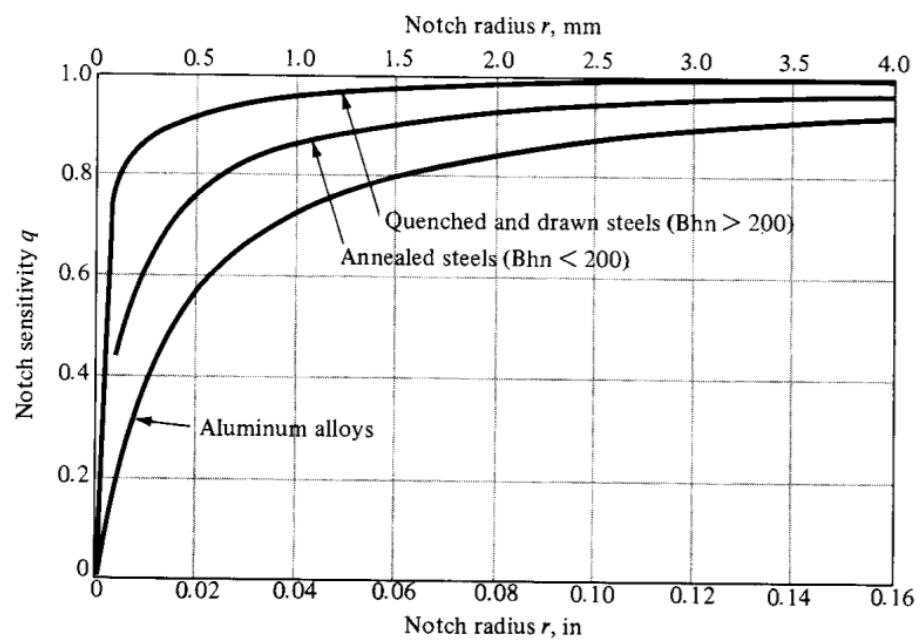


Figure 4.7: Notch sensitivity factors for steels and aluminium alloys (torsion)

Defining notch parameters

The notch characteristic length and the notch root radius are defined in the job file.

Job File Usage: **NOTCH_CONSTANT** = C_n
 NOTCH_RADIUS = C_r

The notch constant is defined according to the table below.

Notch factor estimation method	Notch constant
Peterson (default)	<i>Not applicable</i>
Peterson B	a
Neuber	ρ
Harris	ρH
Heywood	a_H
Notch sensitivity	q

The notch root radius defines the parameter r in Equations 4.5-4.8.

Specifying notch factors for FEA stresses

Since Quick Fatigue Tool assumes that the FEA stresses account for the effects of geometry, the default meaning of K_t is that of a supplementary factor which describes surface finish effects which cannot easily be modelled in finite elements. To that end, the user must be careful when considering the inclusion of the fatigue notch factor if the stresses originate from FEA.

If the S–N data is produced from smooth (un-notched) specimens then the nominal stress is equal to the local stress, as there is no stress gradient effect. In this case, the fatigue notch factor is not required since the FE solution provides the local stress at the notch surface. However, if the S–N data originated from a notched test, then the FEA stresses at the notch will produce excessively conservative fatigue life predictions. In such cases, the user must follow the procedure outlined by their chosen stress-life guideline in order to estimate the pseudo nominal stress a certain distance away from the notch tip, and use this stress on the notched S–N curve instead.

The fatigue notch factor is estimated from the value of K_t .

Job File Usage: **KT_DEF** = n

where n is not the surface finish factor, but the elastic stress concentration factor. By defining the notch sensitivity constant and the notch root radius, the corresponding notch sensitivity is estimated. The stresses (or the S–N curve) will be scaled in the same way as is done with a standard surface finish definition (Figure 4.2).

4.3.3 Modelling guidance

Specifying K_t on a material surface

Unless analysis groups are defined, the surface finish definition is applied to every analysis item in the model. This behaviour is incorrect if the model contains subsurface nodes. If subsurface nodes are being analysed, the recommended practice is to define a skin on the finite element model and apply the surface finish definition to the skin using a separate analysis group.

Job File Usage: `GROUP = {'skin-group-file-name.*', 'DEFAULT'}`

The procedure for creating analysis groups is described in Section 4.6. The surface finish is then defined in the job file.

Job File Usage: `KT_DEF = {'surface-finish/roughness-file-name.kt/ktx', []}`
`KT_CURVE = n`

Specifying K_f directly

If the fatigue test data was measured for a smooth specimen but the component contains a notch, the user can specify the fatigue notch factor directly if it is already known. By assuming that the material is fully notch-sensitive ($q = 1$), then $K_t = K_f$. This means that the surface finish factor can be used as the notch sensitivity factor.

Environment File Usage: `notchFactorEstimation = 6.0`

Job File Usage: `KT_DEF = K_f`
`KT_CURVE = []`
`NOTCH_CONSTANT = 1.0`

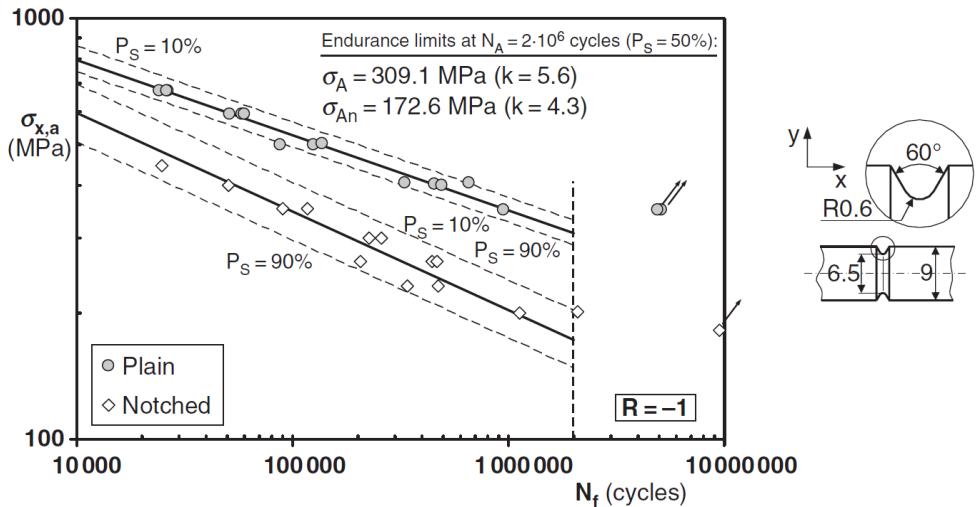


Figure 4.8: Fatigue results generated by testing plain and V-notched cylindrical specimens of S690 steel under rotating bending [17]

Take Figure 4.8 as an example. This data was obtained by Susmel [17], and shows the S-N curves for a notched and a smooth specimen. Using the data at the endurance limit (two million cycles), the value of K_f is estimated to be $\sigma_A/\sigma_{An} = 1.791$ at 50% probability of survival. For a load ratio of $R = -1$, a cycle with a stress amplitude of 309.1 MPa will produce the same life on a smooth specimen as a cycle with a stress amplitude of 172.6 MPa on the notched specimen.

This can be verified in Quick Fatigue Tool by defining the smooth specimen material data with the following S-N data points:

N – Values	S – Values
10000	800
2000000	309.1

The two definitions below should result in a fatigue life of two million cycles. The Uniaxial Stress-Life algorithm is used for both definitions. For the second definition, the notch sensitivity is used as the fatigue notch factor estimation method.

Definition A

Job File Usage: **HISTORY** = [309.1, -309.1]
KT_DEF = 1.0
NOTCH_CONSTANT = []

Definition B

Job File Usage: **HISTORY** = [172.6, -172.6]
KT_DEF = 1.791
NOTCH_CONSTANT = 1.0

4.4 In-plane residual stress

4.4.1 Overview

Many engineering components exhibit surface residual stresses due to their manufacturing process. Compressive residual stresses are often introduced by design. However, tensile residual stresses can accelerate fatigue damage accumulation.

4.4.2 Defining in-plane residual stress

An in-plane residual stress component can be specified directly in the job file.

Job File Usage: **RESIDUAL** = σ_R

The stress is assumed to act uniformly in all directions and is added to the mean stress of each cycle.

4.4.3 Modelling guidance

Unless analysis groups are defined, the residual stress definition is applied to every analysis item in the model. This behaviour is incorrect if the model contains subsurface nodes. If subsurface nodes are being analysed, the recommended practice is to define a skin on the finite element model and apply the residual stress to the skin using a separate analysis group.

Job File Usage: **GROUP** = {'*skin-group-file-name.**', 'DEFAULT'}

The procedure for creating analysis groups is described in Section 4.6. The residual stress is then defined in the job file.

Job File Usage: **RESIDUAL** = [σ_R , 0.0]

Since the residual stress is applied directly to the fatigue cycle and assumes the orientation of the critical plane, the quantity cannot be visualized with field output. For example, the variables SMAX and WCM do not include the effect of residual stress. Furthermore, the residual stress is not considered by nodal elimination.

4.4.4 Limitations

Definition of residual stress is not compatible with the BS 7608 algorithm. If residual stress is defined with Findley's Method, the stress is added to the cycle during the damage calculation instead of the mean stress.

4.5 Controlling analysis performance

4.2.1 Overview

Quick Fatigue Tool utilises a suite of algorithms which attempt to automatically reduce the analysis time without negatively impacting the accuracy of the results:

- time history peak-picking;
- load proportionality checking;
- analysis region control;
- nodal elimination;
- high performance principal stress calculation; and
- reuse of analysis data.

In this section, the above techniques are described.

4.5.2 Pre-processing time histories

Load histories often contain data which does not contribute to fatigue and can have a spurious effect on the fatigue calculation. Take the signal in Figure 4.9 as an example.

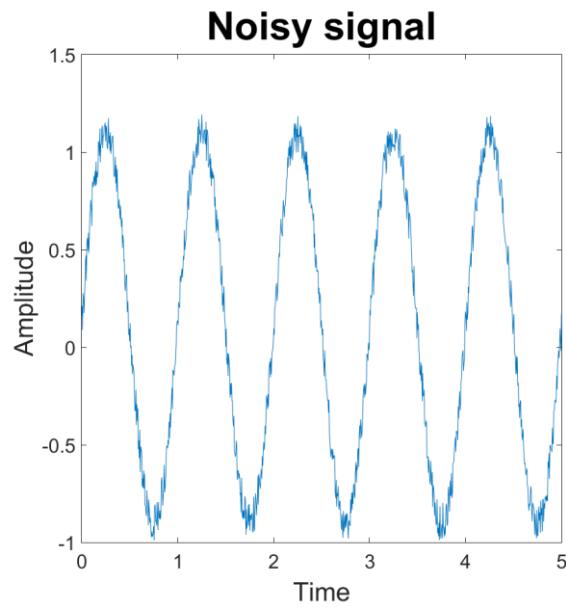


Figure 4.9: Noisy test signal

Analysing signals which contain small cycles puts additional computational load on the cycle counting algorithm and can impact the analysis time significantly, without improving the estimate of the fatigue damage. Quick Fatigue Tool offers three procedures for removing redundant cycles.

1. pre-gate load histories;
2. gate tensors; and
3. noise reduction.

Quick Fatigue Tool assumes that there is no correlation between the phase of the load histories and the damage parameter on the critical plane. Therefore, the default behaviour is to gate the stress tensors as this is the most reliable method.

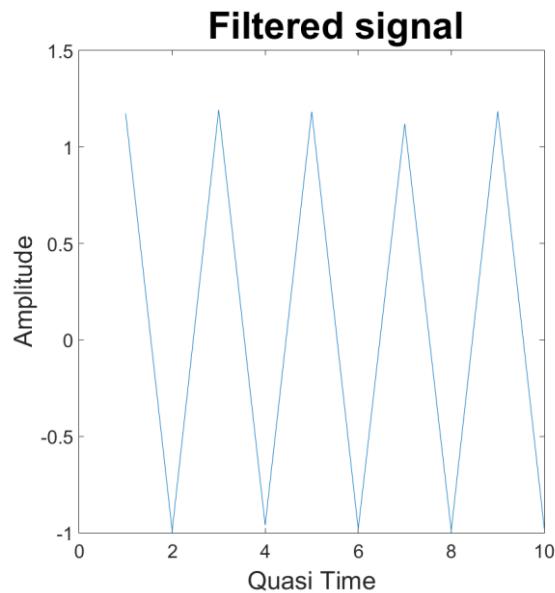


Figure 4.10: Filtered signal

Pre-gate load histories

Load history pre-gating converts each load history into a peak-valley sequence before being combined to create the fatigue loading.

Environment File Usage: `gateHistories = {0.0 | 1.0 | 2.0}`

Retain every point in the load histories (default)

When load history pre-gating is disabled, every point in the load history is considered for fatigue analysis.

Environment File Usage: `gateHistories = 0.0`

Pre-gate load histories as percent gate

Each load history is filtered according to a percent gating criterion. If the difference between two consecutive history points in the i th loading is less than a specified percentage g_i , then the current point is removed from the history.

Environment File Usage: `gateHistories = 1.0`
`historyGate = [g1, g2, ..., gn]`

Up-to n gating values are specified for n loading channels.

Figure 4.10 shows the result of gating the noisy signal from Figure 4.9 with a 15% gating criterion.

Pre-gate load histories using Nielsony's method

The load history is gated by identifying local extrema in the signal.

Environment File Usage: `gateHistories = 2.0`

Effectiveness of load history pre-gating

The pros and cons of load history gating are as follows:

Pros:

- load history gating is performed once for each loading channel before the start of the analysis, so it is very fast; and
- for simple loadings, load history gating has a similar accuracy to tensor gating.

Cons:

- fatigue results may be in error if the loading consists of multiple histories. The method is applied separately to each history so there is no guarantee that the phase relationship between the loading channels is maintained.

When using pre-gated load histories with multiple load channels, the user should compare the change in life values between the gated and the original signal. If the difference is significant, load history pre-gating should be disabled.

Constant amplitude loading

When load history pre-gating is enabled and Quick Fatigue Tool detects a constant amplitude loading condition, then only the first cycle is analysed. This avoids invoking the rainflow cycle counting algorithm unnecessarily. The number of repeats is automatically adjusted to account for the additional cycles, according to Equation 4.10.

$$R_T = R_{job} \times R_{load} \quad [4.10]$$

The total number of repeats, R_T , is the product of the number of repeats defined by the **REPEATS** option, R_{job} , and the number of repeats in the load history, R_{load} .

History point h_i is considered to be constant amplitude with respect to the point h_{i-2} provided that the two points lie within the user-defined tolerance specified by the **historyGate** environment variable. If multiple gating values are specified, then the first value will be used.

Gate tensors

Tensor gating converts the tensor histories into a peak-valley sequence before being rainflow cycle counted. When load history gating is disabled, the original ungated loading is used to determine the principal stress and damage parameter histories.

Environment File Usage: `gateTensors` = {0.0 | 1.0 | 2.0}

Retain every point in the load histories

When tensor gating is disabled, every point in the tensor history is considered for fatigue analysis.

Environment File Usage: `gateTensors` = 0.0

Gate tensors as percent gate (default)

Each tensor history is filtered according to a percent gating criterion. If the difference between two consecutive history points in the i th loading is less than a specified percentage g_i , then the current point is removed from the history.

Environment File Usage: `gateTensors` = 1.0
 `tensorGate` = $[g_1, g_2, \dots, g_n]$

Up-to n gating values are specified for n loading channels.

Gate tensors using Nielsony's method

The load history is gated by identifying local extrema in the signal.

Environment File Usage: `gateTensors` = 2.0

Effectiveness of tensor gating

The pros and cons of tensor gating are listed below.

Pros:

- The most accurate method of gating. Since the damage parameter accounts for the combination of the fatigue loading, the phase relationship between the loading channels is always maintained

Cons:

- The damage parameter is gated per plane, per node, thus tensor gating is much slower than pre-gated load histories

Zero derivative gating

If it is necessary to remove intermediate data (points which lie between peaks and valleys), but additional gating is not required, the gate value can be set to zero. Quick Fatigue Tool will use a zero derivative method and all peak-valley pairs will be retained.

Environment File Usage: `tensorGate` = 0.0
 `historyGate` = 0.0

Noise reduction

A low pass filter is applied to each load history in order to remove signal noise from the load data. The user specifies the number of averaging segments n .

Environment File Usage: `noiseReduction` = 1.0
 `numberOfWindows` = n

Care should be taken when using the low-pass filter, since the stress amplitude is always reduced. This may result in excessively optimistic fatigue life results. Noise reduction should not be used as an alternative to gating, and its use is not recommended in general unless the load signal is highly affected by measurement noise.

4.5.3 Determining load proportionality

If the direction of the principal stress does not change during the loading, then the stresses are proportional and critical plane searching is not necessarily required.

Quick Fatigue Tool automatically checks the model for proportional loading before the start of the analysis and skips critical plane searching at these regions for certain algorithms.

The user specifies load proportionality checking in the environment file.

Environment File Usage: `checkLoadProportionality = {0.0 | 1.0}`

0. Do not perform load proportionality checks
1. Perform load proportionality checks (default)

Specifying the proportionality tolerance

The principal stress history of the loading is calculated at each location in the model, along with the orientation of the first principal stress. If the largest change of the angle of the first principal stress does not exceed the specified tolerance, the loading is assumed to be proportional.

The user specifies the proportionality tolerance, k_{tol} .

Environment File Usage: `proportionalityTolerance = k_{tol} (1)`

The purpose of defining a tolerance is to account for solution noise which may cause the direction of the principal stresses to “wobble”, even if the loading is theoretically proportional.

Load proportionality checking is compatible with the following fatigue analysis algorithms:

- Stress-based Brown-Miller;
- Normal Stress; and
- BS 7608.

4.5.4 Specifying the analysis region

It is possible to restrict the analysis to a specific region of the model. For very large models, it may be economical to perform the analysis only at the locations where fatigue failure is likely to occur. Alternatively, if the location of fatigue failure has already been determined by a previous analysis, an additional analysis may be performed at this location with additional output and/or more rigorous critical plane searching.

Quick Fatigue Tool includes the following approaches for specifying the analysis region:

- whole model;
- ODB element surface (default);
- maximum principal stress range;
- hotspot; and
- user-defined.

Job File Usage: `ITEMS = {'ALL' | 'SURFACE' | 'MAXPS' | 'file-name.*' | [ID1,..., IDn]}`

Analysing the whole model

All items in the model are analysed unless they are removed by nodal elimination. If the **DATASET** option is not used for analysis, the **ITEMS** option is ignored.

Job File Usage: `ITEMS = 'ALL'`

Analysing an ODB element surface

The analysis region is restricted to items on the element free surface of the model Abaqus .odb file.

Job File Usage: `ITEMS = 'SURFACE'`

An element is considered to be on the surface if it has at least one node on the surface. This option is useful in the majority of cases, where fatigue cracks initiate on the component surface. If subsurface cracks are likely to initiate, `ITEMS='ALL'` should be used instead.

If there is no .odb file specified, the whole model is used as the analysis region. ODB element surface detection is not supported for uniaxial analysis methods.

Specifying the model output database

Surface detection requires the Abaqus model output database (.odb) file and part instance name to be specified.

Job File Usage: `OUTPUT_DATABASE = 'model-odb-file-name.odb'`
`PART_INSTANCE = 'part-instance-name'`

The specified part instances should match those defined in the stress dataset files. If the user specifies part instances which do not exist in the dataset then surface detection will not take effect. The element surface is read according to the user-specified result position.

Job File Usage: **RESULT_POSITION** = {'ELEMENT_NODAL' | 'UNIQUE_NODAL' | 'CENTROID'}

The user should ensure that the specified result position matches the position of the stress datasets. Surface detection is not supported for integration point results.

Shell elements

If the model output database contains shell elements, Quick Fatigue Tool treats the element surface as the entire shell. Alternatively, the user can specify to treat the element surface as free shell faces.

Environment File Usage: **shellFaces** = {0.0 | 1.0}

0. Treat surface as whole shell element (default)
1. Treat surface as free shell element faces

The difference between these two surface definitions is shown in Figure 4.11.

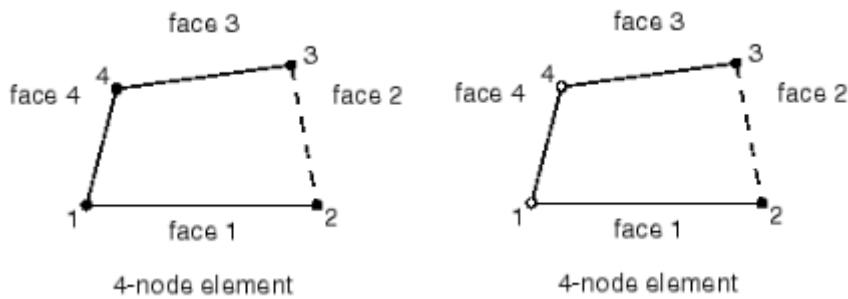


Figure 4.11: 4-node shell elements whose free faces are shown by dashed faces. Surface nodes are indicated by solid black circles. (L) Treat shell surface as whole shell. (R) Treat shell surface as free shell faces.

Supported element types

The surface detection algorithm supports most Abaqus elements with displacement degrees of freedom. A list of compatible elements is found in Appendix IV “List of supported elements for surface detection” of the document *Quick Fatigue Tool Appendices*.

Limiting the search region to elements defined in the stress dataset (default)

If the stress dataset contains element-nodal or centroidal stress data, surface detection can be limited to only the elements in the output database file which are defined in the dataset.

Environment File Usage: **searchRegion** = 0.0

When the search region is limited to dataset elements, Quick Fatigue Tool treats the dataset as the entire model. This is the preferred method because it is much faster in cases where the part instance contains a very large number of elements compared to the stress dataset. However, element boundaries which exist in the dataset may, in reality, be attached to elements in the .odb file which do not constitute a free surface. Therefore, when **searchRegion**=0.0, the surface detection algorithm may overestimate the number of elements on the surface.

Searching the whole ODB part instance

The search region can be defined as every element in the output database part instance.

Environment File Usage: `searchRegion = 1.0`

If the dataset contains unique nodal stress data, the entire part instance is always included by the surface detection algorithm.

If more than one part instance is specified with `PART_INSTANCE`, then all elements in each part instance are always included by the surface detection algorithm.

Reusing surfaces

Whenever Quick Fatigue Tool reads the surface of an `.odb` file, the surface items for the current model and specified part instances are written to the surface definition file '`Data\surfaces\<model-ID>_surface.mat`'. At the start of each job, if `ITEMS='SURFACE'`, the surface definition file is used to extract the model surface instead of the `.odb` file; this is much more time-efficient when running the same job in multiple configurations.

Environment File Usage: `surfaceMode = {0.0 | 1.0}`

0. Always read model surface from the ODB
1. Read model surface from the surface definition file (default—if available)

Alternatively, the user specifies the surface definition file directly in the job file.

Job File Usage: `ITEMS = '<model-ID>_surface.mat'`

Analysing the item with the maximum principal stress range

The analysis region is restricted to the item with the largest principal stress range.

Job File Usage: `ITEMS = 'MAXPS'`

This method allows the user to quickly identify an analysis item that is likely to reside in one of the damage hotspots in the model.

This setting can be useful for very large models where even a simple fatigue analysis (such as Stress Invariant Parameter) could take a long time to complete. Using the '`MAXPS`' option allows the user to select a more rigorous analysis algorithm on a single analysis item without first having to run a whole model analysis. This option is not intended to replace a complete analysis, however; the location of maximum stress range does not necessarily coincide with the location of maximum fatigue damage. Therefore, the '`MAXPS`' option should be used with great care if the loading is non-proportional.

Even if the job contains non-default group definitions, Quick Fatigue Tool will still search the whole model as defined by the `DATASET` option. If the item with the largest principal stress range exists outside any of the groups, the analysis will be aborted. This is because the code cannot determine valid properties for an item which does not have any material or analysis data associated with it.

Analysing items listed in a text file

The analysis region is restricted to the items defined in a text file.

Job File Usage: **ITEMS** = '*file-name.**'

Defining the analysis region from hotspots

Analysis items with lives below the design life N_d are written to the text file *Project\output\<jobName>\Data Files\hotspots.dat*. This text file can be used to specify the analysis region in a subsequent job.

First job:

Job File Usage: **HOTSPOT** = 1.0
DESIGN_LIFE = N_d

Second job:

Job File Usage: **ITEMS** = '*hotspots.dat*'

Defining the analysis region from an automatically-generated items file

Any of the following files generated by Quick Fatigue Tool may be specified as the analysis region: '*<model-ID>_surface.mat*', '*warn_lcf_items.dat*' or '*warn_overflow_items.dat*'.

Defining the analysis region as a user-defined item ID list

The analysis region is restricted to the item IDs ID_1 to ID_n . The item numbers correspond to rows in the stress dataset file.

Job File Usage: **ITEMS** = [ID_1, \dots, ID_n],

Consider the FEA definition in Figure 4.12. If **ITEMS**=16, the analysis will only consider the 16th item in the definition. Note that if the definition file contains a header, the value of **ITEMS** will not correspond to the line number in the file.

20	Node	Label	S.S11	S.S22	S.S33	S.S12	S.S13	S.S23
21			@Loc 1	@Loc 1	@Loc 1	@Loc 1	@Loc 1	@Loc 1
22								
23	1	-795.763E-03	-4.48699	-2.95655	6.92934	-3.93393	1.82896	
24	2	4.90945	1.50557	-4.52454	5.76533	-3.86309	3.06441	
25	3	415.521E-03	726.238E-03	640.271E-03	592.421E-03	1.13344	1.97704	
26	4	9.63988	-8.78137	49.1831E-03	5.92665	834.660E-03	1.07187	
27	5	2.33473	3.17949	514.433E-03	2.61164	1.37854	1.49722	
28	6	-465.532E-03	-14.7080	-423.602E-03	954.029E-03	484.061E-03	3.20978	
29	7	-6.34151	2.17191	-5.48330	-4.71242	6.09594	5.78890	
30	8	-1.61290	3.09481	-1.66385	985.416E-03	1.58106	-842.751E-03	
31	9	-20.6121	-12.0342	-12.6017E-03	-15.7107	118.645E-03	40.7449E-03	
32	10	-4.48167	-4.84691	6.92384E-03	-4.60756	29.1547E-03	33.7680E-03	
33	11	4.41018	4.17798	342.322E-03	4.38294	1.23794	1.21576	
34	12	4.38159	4.72635	348.698E-03	4.66588	1.23816	1.30647	
35	13	49.8089	-12.6640	-1.03326	6.16651	-2.52934	-1.17407	
36	14	19.4927	7.00084	3.45965	7.36442	3.71231	176.971E-03	
37	15	17.9602	9.06355	4.16721	6.33369	2.33977	-593.006E-03	
38	16	308.628E-03	-3.64435	653.559E-03	548.751E-03	-481.142E-03	-1.08482	
39	17	9.56567	-838.844E-03	1.00148	865.819E-03	2.04524	-170.132E-03	
40	18	6.68310	-6.61508	-13.8474E-03	-1.64620	-735.383E-03	-395.798E-03	
41	19	7.48860	-3.53067	356.557E-03	-1.50177	-1.02732	-202.813E-03	
42	20	3.53093	-858.371E-03	176.520E-03	2.74948	-934.752E-03	-789.951E-03	

Figure 4.12: Example FEA definition file

After each analysis, Quick Fatigue Tool writes to the message file the ID(s) of the item(s) in the stress dataset(s) with the worst life. This value can be used in conjunction with the **ITEMS** option to re-run the analysis at the worst location in the model.

4.5.5 Nodal elimination

Overview

The analysis time can be reduced by ignoring analysis items whose maximum stress is unlikely to cause damage. When the nodal elimination algorithm is enabled, Quick Fatigue Tool checks the maximum principal stress range at each item before beginning the analysis. If the stress is below a certain percentage of the conditional stress, σ_{COND} , then the item is not included for analysis. The value of σ_{COND} is calculated as a function of the target life, N_T .

Enabling nodal elimination

Nodal elimination is enabled in the environment file.

Environment File Usage: **nodalElimination** = {0.0 | 1.0 | 2.0}

Retaining all nodes for analysis

The nodal elimination algorithm is disabled and all items in the model are analysed even if they do not experience fatigue damage.

Environment File Usage: **nodalElimination** = 0.0

Nodal elimination based on the material endurance limit (default)

The target life N_T is determined by the material constant amplitude endurance limit (CAEL).

Environment File Usage: **nodalElimination** = 1.0

The value of σ_{COND} is the stress amplitude which will result in a life of N_T cycles.

The value of σ_{COND} may be specified directly if a user-defined endurance limit is specified in the environment file.

Environment File Usage: **fatigueLimitSource** = 3.0
 userFatigueLimit = σ_{COND}

Nodal elimination based on the component design life

The target life N_T is determined by the component design life.

Environment File Usage: **nodalElimination** = 2.0

Job File Usage: **DESIGN_LIFE** = N_T

Calculation of the condition stress, σ_{COND}

If an analysis is run with the Stress-based Brown-Miller algorithm, then the conditional stress is obtained from Equation 4.11:

$$\sigma_{COND} = E \left(1.65 \frac{\sigma_f'}{E} (N_T)^b + 1.75 \varepsilon_f' (N_T)^c \right) \quad [4.11]$$

where σ_{COND} is the conditional stress at which the life is equal to the target life, N_T . The analysis item is then removed if the following inequality is satisfied:

$$f_{thr} \cdot \sigma_{COND} > 0.5 \cdot (\sigma_1 - \sigma_3)|_{max} \quad [4.12]$$

where $(\sigma_1 - \sigma_3)|_{max}$ is the maximum difference between the first and third principal stresses in the loading and f_{thr} is the elimination threshold scale factor.

Scaling the conditional stress with f_{thr}

If the fatigue loading contains only one cycle, then the conditional stress can be determined directly from the fatigue limit. However, if the loading contains multiple cycles then it is possible for finite life even if the majority of the cycles are below the fatigue limit.

Consequently, it is necessary to scale the conditional stress by a factor f_{thr} , such that damaging cycles are not neglected.

Environment File Usage: `thresholdScalingFactor` = f_{thr} (0.8)

The default value of f_{thr} is 80%.

4.5.6 Principal stress calculation

Overview

Quick Fatigue Tool determines the principal stress history of the loading before the fatigue analysis. The built-in `eig()` MATLAB function only accepts two-dimensional data, which can make the calculation very time consuming for large models with long histories. This is because the principal stresses are calculated separately for each point in the load history.

For very large models, the three-dimensional Eigensolver written by Bruno Luong is recommended. This method calculates the principal stresses for the entire load history in a single calculation for each analysis item, resulting in potentially shorter pre-processor times.

Environment File Usage: `eigensolver` = {1.0 | 2.0}

Using the built-in Eigensolver

The built-in Eigensolver uses the MATLAB function `eig()` to determine the principal stress history.

Environment File Usage: `eigensolver` = 1.0

Using the high performance Eigensolver (default)

The high performance Eigensolver uses the Luong's method to determine the principal stress history.

Environment File Usage: **eigensolver** = 2.0

Luong's method is used by default. Small numerical round-off errors in the calculation may cause Quick Fatigue Tool to report a very small mean stress in the results, even if the loading has no mean stress—this does not affect the fatigue life result.

If the load history contains less than three points, Quick Fatigue Tool always uses the built-in Eigensolver.

4.5.7 Reusing analysis data

Overview

Quick Fatigue Tool saves the fatigue definition of each analysis job to a MATLAB binary file located in *Data\library*. The definition file contains the following information about the analysis job:

- main ID (element/centroid) numbers;
- sub ID (node/integration point) numbers;
- stress tensor histories at each location in the model;
- stress invariant histories at each location in the model;
- items which were removed by the nodal elimination algorithm; and
- un-gated uniaxial stress history (if applicable).

If a particular analysis job is re-run with different job settings, and the above information is unchanged, then it is not necessary to repeat the tasks which generate the fatigue definition. In such cases, the definition can be recalled from the library.

Job File Usage: **DATA_CHECK** = {2.0 | '*fatigue-definition-file-name.mat*'}

Recalling a fatigue definition using the job name

The user specifies the job name matching that of the fatigue definition being recalled.

Job File Usage: **JOB_NAME** = '*job-name*'
 DATA_CHECK = 2.0

Quick Fatigue Tool searches for the fatigue definition file named '*Data\library\[J]<job-name>_fd.mat*'.

Recalling a fatigue definition from a fatigue definition file

The user specifies the fatigue definition file directly.

Job File Usage: **DATA_CHECK** = '*fatigue-definition-file-name.mat*'

Additional guidance

It is important that the fatigue definition of the current job matches that of the fatigue definition file; any changes to the definition of the following options may result in undesirable behaviour:

- **DATASET; HISTORY; SCALE; OFFSET; HF_DATASET; HF_HISTORY; HF_TIME; HF_SCALE; ELEMENT_TYPE; GROUP; ALGORITHM; ITEMS.**

4.6 Analysis groups

4.6.1 Overview

Analysis groups are used to define regions in the model having distinct properties. Analysis groups can have their own definitions for the following job file options:

Property	Job file option
Material properties	MATERIAL
S-N data scale factor	SN_SCALE
S-N knock-down curves	SN_KNOCK_DOWN
Fatigue Reserve Factor envelope definition	FATIGUE_RESERVE_FACTOR
Surface finish definition	KT_DEF
Surface finish curve	KT_CURVE
Residual stress	RESIDUAL
Notch sensitivity constant	NOTCH_CONSTANT
Notch root radius	NOTCH_RADIUS

Analysis groups can have their own definitions for the following environment variables:

Property	Environment variable
Goodman envelope definition	modifiedGoodman
Goodman mean stress limit	goodmanMeanStressLimit
User-defined Walker γ parameter	userWalkerGamma
User-defined fatigue limit	userFatigueLimit
User FRF tensile mean stress normalization parameter	frfNormParamMeanT
User FRF compressive mean stress normalization parameter	frfNormParamMeanC
User FRF stress amplitude normalization parameter	frfNormParamAmp

Analysis groups can be defined in two ways:

1. An item ID list
2. An FEA subset

An item ID list is a list of indexes corresponding to the row numbers in the stress dataset(s) as they are defined in the file. An FEA subset is a list of position IDs referencing the location of the elements, nodes, centroids or integration points in the model.

Job File Usage:

GROUP = {*'group-file-name-1.*'*, ..., *'group-file-name-n.*'*}

Groups are defined as ASCII text files.

4.6.2 Defining analysis groups as an item ID list

Consider the stress dataset file in Figure 4.13.

Element Label	Node Label	S.S11 @Loc 1	S.S22 @Loc 1	S.S33 @Loc 1	S.S12 @Loc 1	S.S13 @Loc 1	S.S23 @Loc 1
	1	43	89.1674	12.8745	429.077E-03	40.4575	20.3763
	1	44	89.1674	12.8745	429.077E-03	40.4575	20.3763
	1	537	89.1674	12.8745	429.077E-03	40.4575	20.3763
	1	538	89.1674	12.8745	429.077E-03	40.4575	20.3763
	1	591	89.1674	12.8745	429.077E-03	40.4575	20.3763
	1	592	89.1674	12.8745	429.077E-03	40.4575	20.3763
	1	1996	89.1674	12.8745	429.077E-03	40.4575	20.3763
	1	2014	89.1674	12.8745	429.077E-03	40.4575	20.3763
2	44	96.7538	7.40922	-1.47880	28.9776	18.0028	674.289E-03
2	45	96.7538	7.40922	-1.47880	28.9776	18.0028	674.289E-03
2	538	96.7538	7.40922	-1.47880	28.9776	18.0028	674.289E-03
2	539	96.7538	7.40922	-1.47880	28.9776	18.0028	674.289E-03
2	592	96.7538	7.40922	-1.47880	28.9776	18.0028	674.289E-03
2	593	96.7538	7.40922	-1.47880	28.9776	18.0028	674.289E-03
2	2014	96.7538	7.40922	-1.47880	28.9776	18.0028	674.289E-03
2	2015	96.7538	7.40922	-1.47880	28.9776	18.0028	674.289E-03

Figure 4.13: Example stress dataset

The dataset consists of two quadratic elements (eight-node hexahedrons). The dataset can be split between two analysis groups, each defining one of the elements in the dataset. This can be achieved by creating an item ID list defining each element. For example, the ID list for the first element is the integer series from 1 to 8, while the second element is defined as the integer series from 9 to 16. These are simply the row numbers corresponding to the nodes of the two elements. Example text file contents for each group are given below.

‘element 1.txt’	‘element 2.txt’
1	9
2	10
3	11
4	12
5	13
6	14
7	15
8	16

4.6.3 Defining analysis groups as an FEA subset

Item ID lists are a direct and relatively simple way of defining analysis groups for small models. However, consider the model of an excavator arm shown by Figure 4.14.

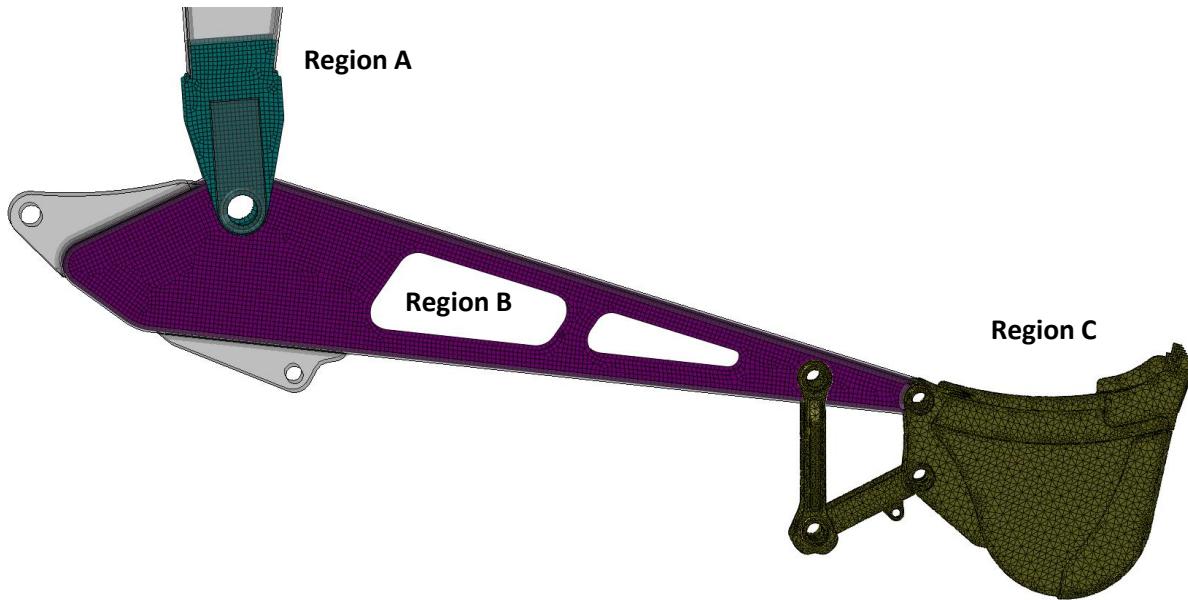


Figure 4.14: Finite element model of an excavator arm with two regions of interest

In this instance, creating groups for regions A, B and C with an item ID list would be a very cumbersome task. Instead, FEA subsets can be used which define the groups by their element labels. In Abaqus, this is easily achieved by creating a display group of the region of interest, then exporting the element labels as an *.rpt* file. The process of generating such a file is exactly the same as for generating FEA stress datasets, and is discussed in detail in Section 3.2.

For example, consider again the dataset in Figure 4.13. An FEA subset may then resemble that shown in Figure 4.15.

Element Label	Node Label	S.S11 @Loc 1	S.S22 @Loc 1	S.S33 @Loc 1	S.S12 @Loc 1	S.S13 @Loc 1	S.S23 @Loc 1
1	43	89.1674	12.8745	429.077E-03	40.4575	20.3763	2.23029
1	44	89.1674	12.8745	429.077E-03	40.4575	20.3763	2.23029
1	537	89.1674	12.8745	429.077E-03	40.4575	20.3763	2.23029
1	538	89.1674	12.8745	429.077E-03	40.4575	20.3763	2.23029
1	591	89.1674	12.8745	429.077E-03	40.4575	20.3763	2.23029
1	592	89.1674	12.8745	429.077E-03	40.4575	20.3763	2.23029
1	1996	89.1674	12.8745	429.077E-03	40.4575	20.3763	2.23029
1	2014	89.1674	12.8745	429.077E-03	40.4575	20.3763	2.23029

Figure 4.15: FEA subset of the dataset in Figure 4.9

The group is literally a subset of the element labels from the original dataset. When using FEA subsets, the following guidance should be observed:

1. The results position between the FEA subset and the original dataset must agree, i.e. if the original dataset uses element-nodal position labels, then so should the FEA subset
2. Including field data with the FEA subset is not compulsory. However, since Abaqus requires at least one field to be exported, FEA subsets written by Abaqus/Viewer will always contain at least one column of field data

It is quite possible that a group defined as an FEA subset will reference more than one element with the same position label. In order for the group definition to be correct, Quick Fatigue Tool must somehow match the ID to the correct location in the FE model. In the event that duplicate IDs are found in a group, field data in the group definition can be used to check the stress tensor of the duplicate IDs against the tensors in the original stress dataset. Therefore, it is recommended that FEA subsets retain the same field information as the original datasets.

Quick Fatigue Tool automatically attempts to resolve duplicate IDs based on the availability of the field data; no additional intervention is required by the user. However, if the loading is defined as either a multiple scale and combine or as a dataset sequence (the **DATASET** option was specified with more than one argument), then the field data from the group file must agree with the last dataset in the loading.

4.6.4 Arranging groups in the job file

Both item ID lists and FEA subsets can be used together for an analysis, and the groups do not have to be mutually exclusive. Quick Fatigue Tool will process each group as they are encountered in the **GROUP** option and will exclude the items so that they will not be read a second time if they appear in subsequent groups. Therefore, the order in which groups are defined will affect which group the items are assigned to if any of the definitions are inclusive of each other. The hierarchical nature of the group reading process is illustrated by Figure 4.16.

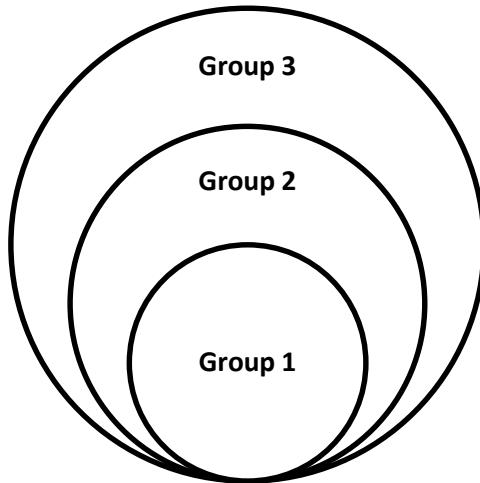


Figure 4.16: Group hierarchy

Group 1 is a subset of Group 2, which in turn is a subset of Group 3. Consider the case where the groups are defined in the job file.

Job File Usage: `GROUP = {'group-1.*', 'group-2.*', 'group-3.*'}`

Quick Fatigue Tool will first read Group 1. All items from Group 1 will be read into the group definition, and excluded from re-definition in later groups. Hence, Group 2 will include all of its items *except* those already belonging to Group 1. The same logic will apply to Group 3, which will have all items from Group 2 and Group 1 excluded from its definition.

The groups defined in the excavator model from Figure 4.10 could be defined in the job file.

Job File Usage: `GROUP = {'region-A.*', 'region-B.*'}`

Quick Fatigue Tool will analyse all of Region A and Region B. None of the items from Region C will be analysed. In this case, the order does not matter since the groups are mutually exclusive.

If all three regions are to be analysed, but only Region C requires individual properties, the following group definition may be used.

Job File Usage: `GROUP = {'region-C.*', 'DEFAULT'}`

In this instance, Quick Fatigue Tool will analyse Region C with individual properties for that group, followed by all other items in the model. In general, use of the '`DEFAULT`' parameter instructs the program to analyse all remaining items in the model (all other items in the original dataset which do not belong to any preceding groups). The '`DEFAULT`' parameter may only be used as the last argument in the `GROUP` option.

The '**DEFAULT**' parameter is used by its self to analyse the whole model, with no group definitions.

Job File Usage: **GROUP** = {'**DEFAULT**'}

4.6.5 Determining the group type

Quick Fatigue Tool determines whether a group definition is an item ID list or an FEA subset based on the contents of the file, and depending on the user setting in the environment file.

Environment File Usage: **groupDefinition** = {0.0 | 1.0}

Allow Quick Fatigue Tool to determine the group data type

Quick Fatigue Tool assumes that the group definition is an item ID list if there is only one column of data in the file, and an FEA subset if there is more than one column.

Environment File Usage: **groupDefinition** = 0.0

Always read group data as an FEA subset (default)

Quick Fatigue Tool always assumes that the group definition is an FEA subset. If the group definition is an item ID list, the group processor may exit with an error.

Environment File Usage: **groupDefinition** = 1.0

4.6.6 Assigning properties to groups

Overview

Group properties (job file options and environment variables) are specified by assigning multiple definitions according to the number of groups in the analysis.

Settings which require a definition for each analysis group

Usually, the number of property definitions should match the number of groups in the analysis. If the analysis contains n groups where $n > 1$, and the user assigns a single definition to a property, Quick Fatigue Tool automatically propagates that definition to all of the analysis groups. Otherwise, the number of property definitions must exactly match the number of group definitions.

Most properties accept n definitions for n analysis groups.

Job File Usage: **MATERIAL** = {'*material-file-1.mat*', ..., '*material-file-n.mat*'}

This requirement applies to the following job file options and environment variables:

Job file option	Environment variable
MATERIAL	frfNormParamMeanT
SN_SCALE	frfNormParamMeanC
NOTCH_CONSTANT	frfNormParamAmp
FATIGUE_RESERVE_FACTOR	userWalkerGamma
RESIDUAL	userEnduranceLimit midifiedGoodman goodmanMeanStressLimit

Settings which are optional for each analysis group

S-N knock-down factors may not be required for every analysis group.

Job File Usage: **SN_KNOCK_DOWN** = {'knock-down-file-1.kd', [], 'knock-down-file-n.kd'}

In this example, knock-down factors have been specified in groups 1 and n , and none is defined for Group 2. In cases where some groups do not require a definition, this must be indicated by an empty assignment ([]), otherwise the group definition will be processed incorrectly.

A combination of surface finish definition files and K_T values can be specified over several groups.

Job File Usage: **KT_DEF** = {'surface-finish-file-1.kt', 1.2, 'surface-finish-file-n.kt'}

In such cases, the **KT_CURVE** option needs only to be specified according to the number of surface finish files defined by **KT_DEF**.

Job File Usage: **KT_CURVE** = [C_1, C_n]

In this example, the curve numbers C_1 and C_n correspond to the surface finish files 'surface-finish-file-1.kt' and 'surface-finish-file-n.kt', respectively; a curve number for the K_T value is not required since the surface finish is defined directly.

4.6.7 Group definition syntax

A list of all properties eligible for group definitions, along with typical syntax is provided in the table below.

Job file option/Environment variable	Example definition
MATERIAL	{'material-file-1.mat',..., 'material-file-n.mat'}
SN_SCALE	[F_{s1} , ..., F_{sn}]
SN_KNOCK_DOWN	{'knock-down-file-1.kd', 'knock-down-file-n.kd'}
FATIGUE_RESERVE_FACTOR	{ E_1 ,..., 'msc-file-name-n.msc'}
KT_DEF	{ K_{T1} , 'surface-finish.file-2.kd', 'surface-finish.file-3.kd'}
KT_CURVE	[$value_2$,..., $value_3$]
RESIDUAL	[$value_1$,..., $value_n$]
NOTCH_CONSTANT	[$value_1$,..., $value_n$]
NOTCH_RADIUS	[$value_1$,..., $value_n$]
frfNormParamMeanT	{ $value_1$, '<param ₂ >',..., $value_n$ }
frfNormParamMeanC	{ $value_1$, '<param ₂ >',..., '<param _n >'}
frfNormParamAmp	{<param ₁ >, <param ₂ >,..., $value_n$ }
userWalkerGamma	[$value_1$,..., $value_n$]
userEnduranceLimit	[$value_1$,..., $value_n$]
modifiedGoodman	[$value_1$,..., $value_n$]
goodmanMeanStressLimit	{ $value_1$, '<param ₂ >',..., $value_n$ }

4.6.8 Limitations

Job and environment file usage

The following caveats should be noted for the use of analysis groups:

- materials must be defined as a cell;
- values for **SN_SCALE** are only required if **USE_SN** = 1.0;
- a combination of surface finish .kt/.ktx files and/or surface finish values can be used within the same **KT_DEF** statement;
- the number of values in **KT_CURVE** need only reflect the number of .kt/.ktx files defined in **KT_DEF**;
- the number of values in **frfNormParamMeanT**, **frfNormParamMeanC** and **frfNormParamAmp** need only reflect the number of custom FRF envelope definitions in **FATIGUE_RESERVE_FACTOR**;
- the number of values in **goodmanMeanStressLimit** need only reflect the number of zero-valued entries in **modifiedGoodman** (the Goodman limit stress can only be defined for the standard Goodman envelope);
- if the '**DEFAULT**' parameter is used as the last argument in a group definition, the default group is included in the total number of groups;
- if the default analysis algorithm or mean stress correction is specified, the algorithm and mean stress correction used for analysis is chosen from the last argument of **MATERIAL**; and
- if the **DESIGN_LIFE** option is set to the material's constant amplitude endurance limit ('**CAEL**'), the endurance value is chosen from the last argument of **MATERIAL**.

Analysis algorithms

Analysis groups in Quick Fatigue Tool currently do not support multiple definitions of the following:

- mean stress corrections/analysis algorithms; and
- BS 7608 material properties.

If a different algorithm or mean stress correction is required for each analysis group, the workaround is to split the analysis into multiple jobs and superimpose the fatigue results onto a single field output file using the option **CONTINUE_FROM**. Analysis continuation techniques are discussed in Section 4.8.

4.7 S-N knock-down factors

4.7.1 Overview

A set of S-N scale factors can be applied to the material S-N data in the form of knock-down factors, which scale the stress data points for each specified life value. Knock-down factors are defined in a separate *.kd* file and specified in the job file.

Job File Usage: **SN_KNOCK_DOWN** = {'*knock-down-file-name.kd*'}
 USE_SN = 1.0

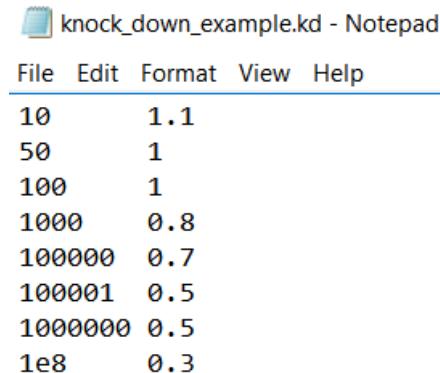
4.7.2 Defining a knock-down curve file

The knock-down curve file is defined as follows:

First column: Life values (N_f) at which knock-down factors are to be applied

Second column: Knock-down factors corresponding to each life value

An example *.kd* file is given by Figure 4.17.



File	Edit	Format	View	Help
10	1.1			
50	1			
100	1			
1000	0.8			
100000	0.7			
100001	0.5			
1000000	0.5			
1e8	0.3			

Figure 4.17: Example *.kd* file containing life values in the first column and knock-down factors in the second column

4.7.3 Applying knock-down factors to the S-N data

The knock-down factors are applied to the S-N data before the analysis to produce the modified endurance curve. The life values in the *.kd* file are treated as sample points and as such, they do not have to match the position of the S-N data points. Quick Fatigue Tool will automatically interpolate and extrapolate the S-N data before scaling each data point. An example of this process is shown by the following table. Note that bracketed values in bold have been interpolated or scaled.

Original S–N curve		Knock-down curve		Scaled S–N curve	
S	N	Factor	N	S	N
(800)	(10)	1.1	10	(880)	(10)
(1725)	(50)	1	50	(1725)	(50)
(1400)	(100)	1	100	(1400)	(100)
700	1000	0.8	1000	(560)	(1000)
350	10000	(0.75)	(10000)	(263)	(10000)
(313)	(100000)	0.7	100000	(219)	(100000)
(313)	(100001)	0.5	100001	(157)	(100001)
(280)	(1000000)	0.5	1000000	(140)	(1000000)
250	10000000	(0.4)	(10000000)	(100)	(10000000)
-	-	0.3	100000000	-	-

Knock-down data specified below the minimum life of the original S–N data is extrapolated. However, Quick Fatigue Tool assumes that the last data point in the original S–N data represents the material's endurance limit and thus knock-down data provided beyond this point is not extrapolated.

The original S–N, knock-down and modified S–N curves are illustrated by Figures 4.18-20. If the material contains S–N curves for multiple R-ratios, each curve is scaled by the knock-down curve before the beginning of the analysis.

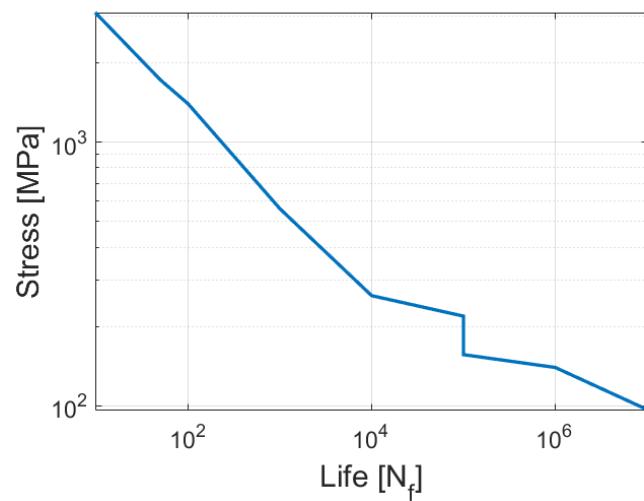
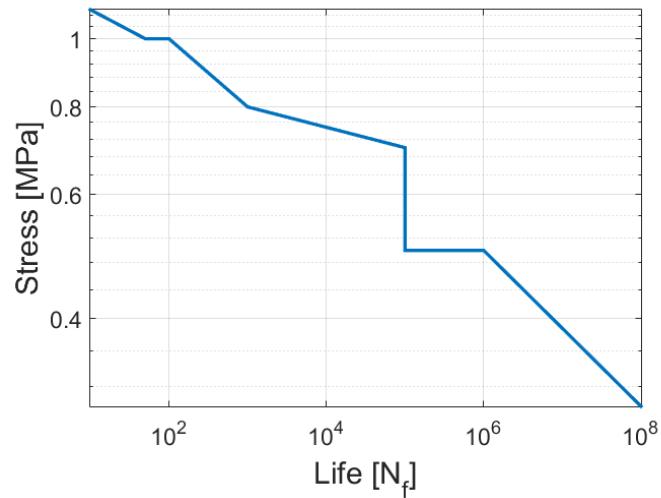
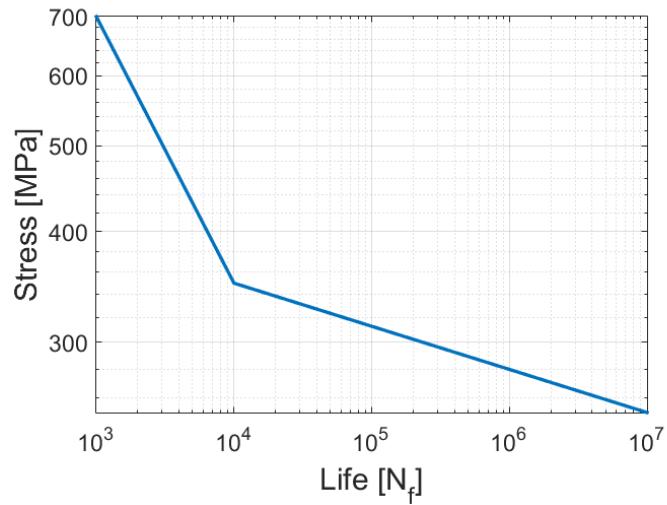


Figure 4.18-20: Original S-N curve (top), knock-down factors (middle) and modified S-N curve (bottom)

4.7.4 Example applications

Knock-down curves can be used in addition to the surface finish definition in order to account for additional manufacturing effects. Imperfections of the manufacturing process can lead to defects, which reduce the endurance of the material.

Knock-down curves may be used with cast metals where inclusions result in a local loss of fatigue performance. Another application is in the injection moulding of plastic components. During this process multiple flow regions can meet, causing a weld line at the interface of the two melts. During cooling of the newly formed part, voids may develop wherein large thermal gradients exist. Both of these phenomena can be accounted for via the use of S-N knock-down factors.

One possible approach to using knock-down factors would be to identify the nodes on the FE model which represent the manufacturing defect, and export a dataset file containing the node numbers and field data, the process of which is described in Section 3.2. This dataset can then be used to define a group in the job file with an individual knock-down curve applied. Such a configuration could be achieved by using the following options.

Job File Usage: MATERIAL = {'knock_down.mat', 'default.mat'}
 USE_SN = 1.0
 SN_KNOCK_DOWN = {'knock_down_curve.kd', []}
 GROUP = {'defect_group.rpt', 'DEFAULT'}

4.7.5 Exporting knock-down curves

The knock-down curves can be exported to MATLAB figures in the environment file.

Environment File Usage: figure_KDSN = {0.0 | 1.0}

0. Do not export knock-down curves
1. Export knock-down curves as a MATLAB figure (default—if available)

S-N knock-down curves are exported for groups which have knock-down factors specified. MATLAB figures must be requested in the job file.

Job File Usage: OUTPUT_PICTURE = 1.0

If the material contains S-N curves for multiple R-ratios, only the curve for $R = -1$ is exported. If an $R = -1$ curve is not defined then it is automatically interpolated.

4.8 Analysis continuation techniques

4.8.1 Overview

Quick Fatigue Tool provides the capability to perform an analysis as a continuation of a previous job. Field output from the current job is written onto the field output from a previous job. An analysis which uses the continuation feature:

- can be used to model block loading, where each job defines a distinct loading event;
- can be used to assign different analysis algorithms to multiple regions in a model; and
- allows the user to specify completely new definitions for any job file option for each job.

When used in conjunction with the ODB interface, analysis continuation:

- can be used to superimpose field data onto the same mesh;
- can be used to append field data onto a mesh at locations different to those which were analysed in the previous job; and
- a combination of the above.

A job which uses analysis continuation runs in the usual way. At the end of the analysis, field data is superimposed onto the field output file from a specified job. The rules for combining field data depend on the variable type, and are listed in the table below.

Variables	Rule
D	$D^* = D_1 + D_2$
L; LL; DDL	$L^* = \left(\frac{1}{L_1} + \frac{1}{L_2} \right)^{-1}$
FOS; SFA; FRFR; FFH; FRFV	$F^* = \min(F_1, F_2)$
FRFW	Derived from the values of $FRFR^*$, $FRFH^*$ and $FRFV^*$
SMAX; SMXP; SMXU; TRF; WCM; WCA; WCDP; YIELD	$F^* = \max(F_1, F_2)$
WCATAN	Derived from the values of WCM^* and WCA^*

The subscripts 1 and 2 correspond to the first and second jobs, respectively. The superscript * corresponds to the superimposed field variable.

4.8.2 Referencing the previous job

The user specifies the job name from the previous analysis in the current job file.

Job File Usage: `CONTINUE_FROM = 'previous-job-name'`

The name of the previous job is the name given by the option `JOB_NAME` in the previous job file. Field output must be requested for the first job. Field output is written automatically for the second job, provided that a valid definition of `CONTINUE_FROM` is specified.

Job File Usage: `OUTPUT_FIELD = 1.0`

4.8.3 Example applications

Multiple analysis algorithms

Analysis continuation can be used to circumvent the limitation of analysis groups, which does not support multiple definitions for the analysis algorithm or mean stress correction. Analysis continuation allows the user to define the algorithm and mean stress correction for multiple regions in the model, and analyse each region as a separate job. Field data is superimposed onto the previous field data file in a chained fashion by specifying the name of the previous job for each analysis. Finally, the cumulative results of all analyses may be written to an `.odb` file using the ODB interface, which is discussed in Section 10.5.

Multiple block loading

The definition of multiple loading blocks is not directly supported by Quick Fatigue Tool. However, analysis continuation can be used to separate the load spectrum across several job files, where each analysis represents a particular loading block, or event, in the component's operational duty.

Consider the loading profile for a given component:

Block #	Descriptor
1	Applied load in 1-direction; fully-reversed [1, -1] load history; 3 repeats
2	Applied load in 2-direction; pulsating [0, 1] load history; 10 repeats
3	Applied load in 1-direction; mixed [1, -1, 2, -1] load history; 1 repeat

Each block is defined as a separate job file, along with their own definitions for the `DATASET`, `HISTORY` and `REPEATS` options.

4.8.4 Additional guidance

Mismatching models

When superimposing results onto a previous analysis, Quick Fatigue Tool searches for matching item IDs between the two field output files. Field data at items which match with the previous file are superimposed to create the field data for the combined result. Items which do not match with the previous job are appended onto the end of the field output file. This is advantageous, since the loading in each block does not have to be applied to the same region between analyses; the definition of `DATASET` does not have to be consistent.

Such a scenario is illustrated by Figure 4.21.

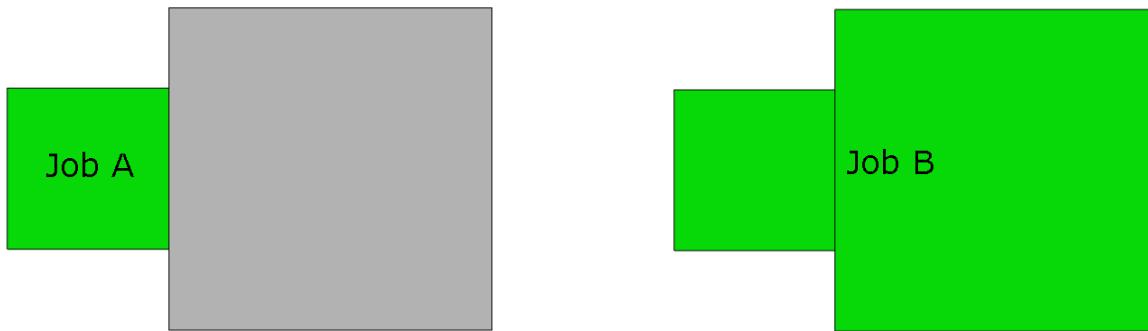


Figure 4.21: FE model split into two analysis regions (green). The first loading block is applied to the smaller region, while the second loading block is applied to the whole model.

After running *Job A*, fatigue results are written to the left-hand portion of the model. After running *Job B* with **CONTINUE_FROM='Job A'**, fatigue results common to the analysis region from *Job A* are superimposed onto the previous data. Fatigue results corresponding to the right-hand portion of the model are appended to the field data without superimposition. This is illustrated by Figure 4.22. Note how the results at the left-hand side have changed to account for the second loading block from *Job B*.

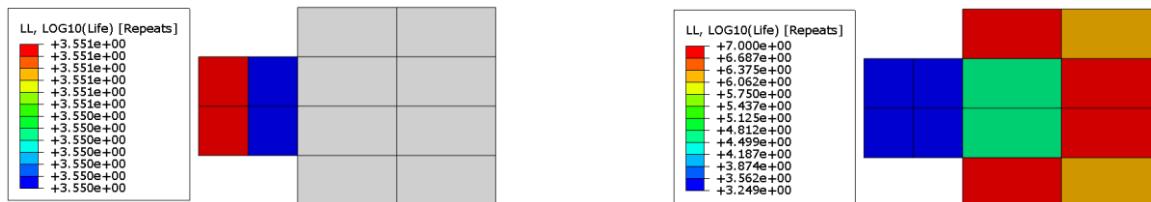


Figure 4.22: Fatigue results for the model depicted in Figure 4.21. Common nodes are superimposed; new nodes are appended as additional field data

Changing the analysis algorithm

It is possible to use a different fatigue analysis algorithm from the previous job. This is especially advantageous if the analysis items in the second job differ completely to those in the first job. For example, a model containing welded features may be analysed with BS7608 near the weld seams, and a different stress-life algorithm at other non-welded locations.

If the analysis items in both models are the same (i.e. damage values are superimposed onto previous results), the user should not move from stress-life to strain-life. Since the stress and strain histories calculated by the strain-life methodology are a function of all previously calculated inelastic strains, the correct damage parameter cannot be obtained if the preceding fatigue history is elastic.

Updating the material state

When a strain-based algorithm is selected for fatigue analysis, Quick Fatigue Tool automatically saves the final material state. If analysis continuation is used with another strain-based procedure, the load history of the second job is automatically adjusted to ensure that material hysteresis is preserved.

Environment File Usage: `importMaterialState = {0.0 | 1.0}`

0. Do not import the material state
1. Import the material state (default—if applicable)

4.8.5 Limitations

Analysis continuation is subject to the following limitations:

- the load equivalency (`LOAD_EQ`) must be the same for all jobs;
- the design life (`DESIGN_LIFE`) must be the same for all jobs;
- the results position must be the same for all jobs;
- the number of cycles quoted in the log file only applies to the most recent analysis;
- the field output file must be located in the default directory according to the job name. Renaming of files or folders will prevent Quick Fatigue Tool from locating the necessary files;
- the analysis will crash if the previous field output file is opened by an external process during the fatigue analysis;
- load transitions are not supported. The cycle counting algorithm will treat each block as a separate event, therefore the effect of large cycles in previous blocks will not be taken into effect in subsequent blocks. If the yield calculation is active over multiple blocks, the effect of hardening is not imported to the next job, rather, the stress-strain state will be reset to zero at the beginning of each block;
- the values of `L` and `LL` are capped at the constant amplitude endurance limit of the material corresponding to the second analysis;
- virtual strain gauge definitions are not carried forward to subsequent analyses;
- combining stress-based and strain-based algorithms is not permitted;
- if ODB element/node sets are written to the `.odb` file after each analysis, the set names must be unique between jobs, otherwise the ODB interface will exit with an error;
- while the ODB interface can handle collapsed elements, results from these elements cannot be superimposed onto previous field data, since Quick Fatigue Tool is unable to resolve the ambiguity caused by duplicate position labels; and
- analysis continuation is not supported for composite criteria analysis.

4.9 Virtual strain gauges

4.9.1 Overview

Virtual strain gauges are used to assess the behaviour of calculated load histories compared to measured strain data, as a means to validate the input stresses. In FEA, strain gauges can be difficult to model and usually require the definition of axial connector elements positioned at strategic locations on the mesh surface which correspond to the laboratory test. Quick Fatigue Tool offers the specification of a virtual strain gauge, which is a location on the model (integration point, node, etc.) at which the strain history is measured in a particular direction.

4.9.2 Gauge conventions

Gauge layout

The virtual strain gauge has a rectangular rosette format, an example of which is shown in Figure 4.23.

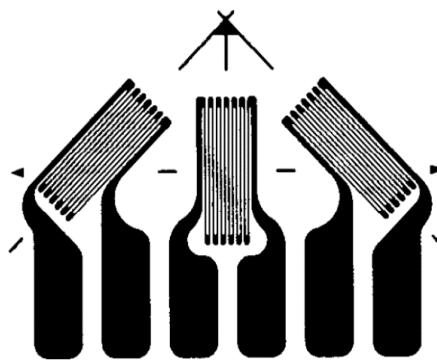


Figure 4.23: Typical layout of a rectangular rosette gauge

The gauge is represented in Cartesian space in Figure 4.24, with the rosette arms *A*, *B* and *C* orientated according to the angles α , β and γ , counter clockwise from the positive x-direction.

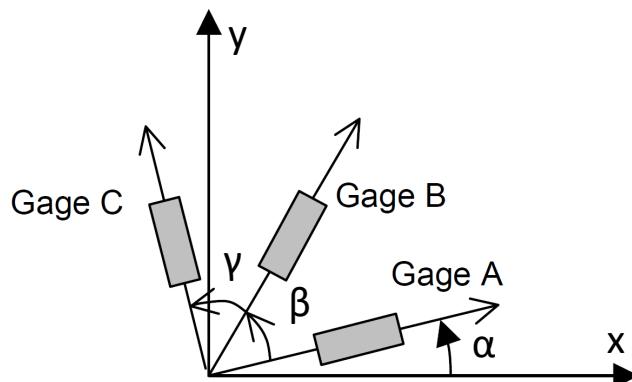


Figure 4.24: Rosette gauge orientation relative to Cartesian axes

Stress definition

The stress dataset should contain plane stress data—out-of-plane components of stress are not considered by the calculation.

The Cartesian directions of the Rosette strain gauge correspond to the stress tensor components according to the following table:

Gauge direction	Component of stress
<i>x – direction</i>	σ_{11}
<i>y – direction</i>	σ_{22}
<i>xy – direction</i>	σ_{12}

4.9.3 Technical background

The calculation of the strain histories is based on the linear elastic stresses defined in the stress data set file. If the material contains values of the cyclic strain-hardening coefficient and exponent (K' and n' , respectively) then the elastic stresses are first converted to elasto-plastic strain histories. These histories are then resolved onto the directions of the gauge arms according to Equations 4.13-15.

$$\varepsilon_A = \frac{\varepsilon_{11} + \varepsilon_{22}}{2} + \frac{\varepsilon_{11} - \varepsilon_{22}}{2} \cos 2\alpha + \varepsilon_{12} \sin 2\alpha \quad [4.13]$$

$$\varepsilon_B = \frac{\varepsilon_{11} + \varepsilon_{22}}{2} + \frac{\varepsilon_{11} - \varepsilon_{22}}{2} \cos 2(\alpha + \beta) + \varepsilon_{12} \sin 2(\alpha + \beta) \quad [4.14]$$

$$\varepsilon_C = \frac{\varepsilon_{11} + \varepsilon_{22}}{2} + \frac{\varepsilon_{11} - \varepsilon_{22}}{2} \cos 2(\alpha + \beta + \gamma) + \varepsilon_{12} \sin 2(\alpha + \beta + \gamma) \quad [4.15]$$

The plasticity correction uses the same algorithm as the Multiaxial Gauge Fatigue app and is described in the document *Quick Fatigue Tool Appendices: A3.2.4*.

If cyclic data is not provided, then the strains are calculated from the stresses elastically, according to Equations 4.16-18.

$$\varepsilon_{11} = \sigma_{11} E^{-1} \quad [4.16]$$

$$\varepsilon_{22} = \sigma_{22} E^{-1} \quad [4.17]$$

$$\varepsilon_{12} = 2\sigma_{12}(1 - \nu)E^{-1} \quad [4.18]$$

4.9.4 Specifying the position of the strain gauge

Virtual strain gauges are defined by specifying the major and minor position IDs identifying the location of the gauges 1 to n on the model.

Job File Usage: **GAUGE_LOCATION** = {'<mainID>.<subID>₁', ..., '<mainID>.<subID>_n'}

For uniaxial analyses, the gauge location is always '**1.1**'. The user may specify multiple strain gauges by listing the position IDs as separate strings.

If the stress dataset does not include position IDs then the major ID is the row number in the dataset file; the minor ID is always 1. For example, the gauge location might be specified as '**1.1**' or '**2.1**'.

4.9.5 Specifying the orientation of the strain gauge

The strain gauge orientation is defined by providing the values of α , β and γ for gauges 1 to n , according to Figure 4.23.

Job File Usage: **GAUGE_ORIENTATION** = {[α, β, γ]_{1,...,n}}

Alternatively, the user may use the flags '**RECTANGULAR**' and '**DELTA**' to indicate that the gauge has a rectangular [$0^\circ, 45^\circ, 45^\circ$] or delta [$30^\circ, 60^\circ, 60^\circ$] layout, respectively.

Job File Usage: **GAUGE_ORIENTATION** = {'**RECTANGULAR**' | '**DELTA**'}

If multiple gauges are defined with the same orientation then only a single definition of **GAUGE_ORIENTATION** is required—otherwise, an orientation for each gauge must be specified.

4.9.6 Example usage

Consider the model in Figure 4.25. A gauge is to be defined at element 657, node 7. The gauge is aligned with the global x-axis, giving an orientation of $\alpha = 0^\circ$, $\beta = 45^\circ$ and $\gamma = 45^\circ$. The virtual gauge is defined in the job file.

Job File Usage: **GAUGE_LOCATION** = {'657.7'}
GAUGE_ORIENTATION = {[0.0, 45.0, 45.0]}

Additional gauges can be added by appending position IDs and orientations as necessary.

Results for each gauge are written to a text file and stored in *Project\output\<jobName>\Data Files*.

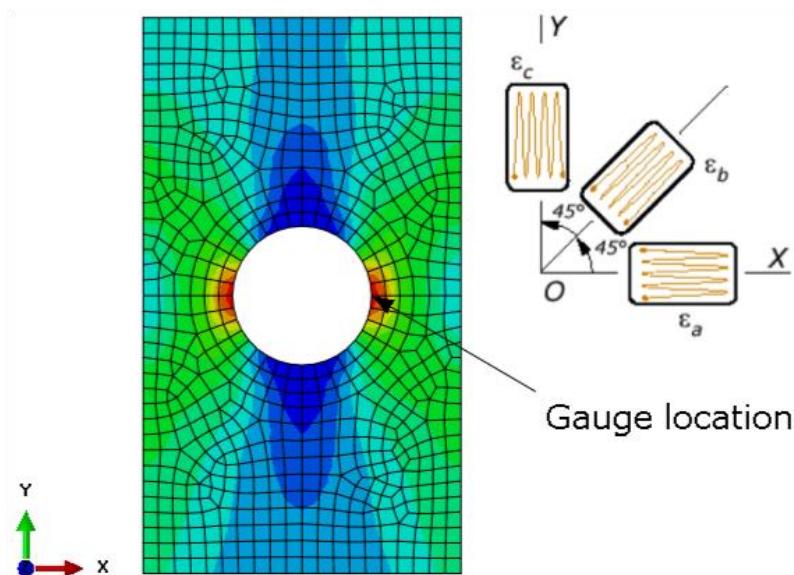


Figure 4.25: Virtual gauge location on FE model

4.9.7 Modelling guidance

Virtual strain gauges are intended for components in a state of plane stress or strain. The gauge will only detect the two-dimensional state of strain relative to the global x–y (σ_{11} – σ_{22}) plane. The strain gauge definition is facilitated by defining the gauges on plane stress elements. If the model contains solid elements, a skin (a layer of shell elements) should be applied over the surface and the gauges defined on these elements.

The position IDs used to define the gauges should be consistent with the element position used to generate the stress data set file. For example, if the stresses were exported at integration points or element nodes, both the main and sub IDs are required. For centroidal and unique nodal data, the item is defined by the main ID; the sub ID always has a value of 1.0.

Output from virtual strain gauges can be used as input to the Multiaxial Gauge Fatigue application (*Quick Fatigue Tool Appendices: A3.2*). The user must ensure that the orientations specified by **GAUGE_ORIENTATION** match those defined in the application.

If the user wishes only to extract virtual strain gauge histories, the fatigue analysis can be omitted by setting **DATA_CHECK=1** in the job file. Alternatively, the Virtual Strain Gauge application (*Quick Fatigue Tool Appendices: A3.4*) can be used to generate strain gauge histories from a strain tensor definition; both methods use the same underlying analysis technique discussed in this section.

4.9.8 Limitations

Virtual strain gauges convert the linear elastic stresses to elasto-plastic strains using a simple multilinear hardening rule. The correction is applied separately to each strain component and as such, results will be inaccurate where a large amount of plasticity is present in the material. The virtual strain gauge should be used as a rough estimate of the local nonlinear strain history only; in cases where a more thorough treatment is required, the user is advised to define the gauge directly on the finite element model.

5. Materials

5.1 Background

5.1.1 Overview

This section describes how to define and use material properties for analysis in Quick Fatigue Tool. In general, this involves:

- creating material properties interactively with the Material Manager application, or from a text file; and
- specifying the material in the job file.

Material data is stored as a MATLAB binary (.mat) file and is located by default in the directory *Data\material\local*.

5.1.2 Accessing the Material Manager

The Material Manager application is used to create and edit material data. It can be accessed either from the command line or by installing the Material Manager GUI application.

To install the Material Manager, double-click the file *Material Manager.mlappinstall* in the *Application_Files\toolbox* directory. The app will appear in the apps bar in MATLAB.

Figure 5.1 shows the Material Manager GUI.

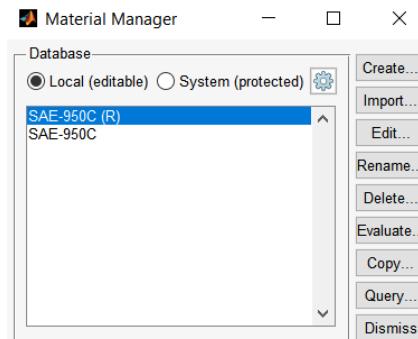


Figure 5.1: Material Manager GUI

The Material Manager GUI is launched by doing one of the following:

- select the Material Manager launch icon from the APPS ribbon; or
- execute the command *material.manage* from the MATLAB command line.

5.2 Material databases

5.2.1 Overview

Material Manager separates material data into two databases:

Local

- Local copies of materials are stored here
- Materials in this database can be modified
- Materials in this database can be used for analysis

System

- Database containing materials included with the Quick Fatigue Tool application
- Materials in this database cannot be modified
- Materials in this database must be fetched in order to be used for analysis

If the full path to the material `.mat` file is specified in the job file, Quick Fatigue Tool will search for the material in this location only. If the material is given without a path, Quick Fatigue Tool will search for the material in the following order:

1. Local material database
2. Default local database path (`<quick-fatigue-tool-root>\Data\material\local`)
3. MATLAB search path (first encounter)

The local material database is the work directory used by the Material Manager application for storing material data.

5.2.2 Specifying the local material database

When Material Manager is started for the first time, the user is prompted to specify the directory for the local material database. If the default location (`<quick-fatigue-tool-root>\Data\material\local`) is available, this is selected automatically. Once the directory is selected, Quick Fatigue Tool saves the location into the `%APPDATA%` and writes a text file in that directory as a marker, so that Quick Fatigue Tool can recall the local database after restarting MATLAB.

The local material database can be changed at any time by selecting the  button from the main Material Manager GUI, or by specifying the database on the command line.

Material Manager Usage: Database region of the *MaterialManager* dialogue: Select . In the dialogue box that appears, select the default location with the check box, or specify a user-defined path in the edit region of the GUI.

Command Line Usage: `>> material.database('<database-path>')`

5.3 Using material data for analysis

5.3.1 Specifying the material in the job file

The user specifies the material in the job file as the name of the *.mat* file containing the material data.

Job File Usage (.m): MATERIAL = '*material-file-name.mat*'

Job File Usage (.txt): *MATERIAL = '*material-file-name.mat*'

Detailed guidance on specifying job file options in a text file is provided in Section 2.4.3.

5.3.2 Fetching materials from the system database

To use a material from the system database, the material is fetched from the file *mat.mat* in *Data\material\system* and a copy is stored in the local database.

Material Manager Usage: Database region of the *MaterialManager* dialogue: Select **System (protected)**. Select a material from the list of system materials. Select **Fetch....** Verify the name of the material. Select **OK**.

Command Line Usage: >> material.fetch()
 >> <database number>
 >> <material number>

The fetched material appears in the *Local* database list in the Material Manager GUI. A list of materials in the local database can be shown in the MATLAB command window.

Command Line Usage: >> material.list()

5.4 Creating materials using the Material Manager

5.4.1 Overview

Material properties are defined by launching the material property editor.

Material Manager Usage: To create a new material, from the *MaterialManager* dialogue select **Create....** To modify an existing material, select **Edit....**

Command Line Usage: `>> material.create`
 `>> material.edit('material-name')`

The material property editor is shown in Figure 5.2.

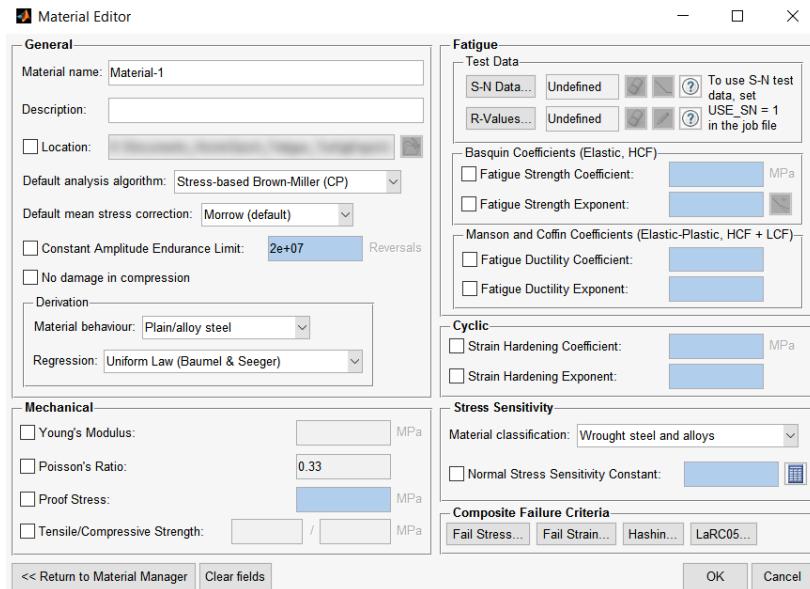


Figure 5.2: Material property editor for a user-defined material

Property values are specified by first checking the property to indicate that it is user-defined. When a property is unchecked, its input field turns blue, meaning that the property is not defined by the user and will be derived automatically if applicable.

5.4.2 Units

Properties with units of stress must be specified in *Megapascals*. The constant amplitude endurance limit is given in *repeats* ($2N_f$); this is the number of cycles, N_f , multiplied by two.

5.4.3 M-file Alternative to Material Manager

A script containing the material data structure can be found in *Application_Files\material_manager\createMaterialDirect.m*. This file is used to create materials without the Material Manager GUI. The user enters material properties into the variable *material_properties*. Upon running the script, the material is saved as a *.mat* file into the local material database (*Data\material\local*).

5.5 Creating materials from a text file

5.5.1 Overview

Material data defined in a text file can be used for

- importing materials into the local database using Material Manager; and
- defining material data directly in the job file for text-based job submission.

Material properties are declared using material keywords in combination with parameters and data lines (if applicable).

5.5.2 Material keyword syntax

Starting a material definition

Every material definition must begin with the material name assignment.

Text File Usage: *USER MATERIAL, name

This keyword declares the material definition and assigns a material name using the *name* parameter.

Keywords with no data lines

Some keywords do not have any associated data lines.

Text File Usage: *REGRESSION, {UNIFORM | UNIVERSAL | MODIFIED | 9050 | NONE}

This keyword specifies the regression algorithm used to derive unknown material properties.

An example usage of this keyword is given below:

*REGRESSION, UNIFORM

There are no data lines associated with this keyword—only the keyword and a single mutually exclusive parameter is required.

Keywords with data lines

Some keywords require a data line to complete their definition.

Text File Usage: *CAEL
First (and only) line: N_{∞}

This keyword defines the constant amplitude endurance limit. There is no associated parameter. The first (and only) data line defines the constant amplitude endurance limit value, N_{∞} .

Some keywords accept multiple optional values on their data line.

Text File Usage: *MECHANICAL
First (and only) line: $E, \nu, \sigma_u, \sigma_y$

This keyword defines the mechanical constants for the material. The data line entries E , ν , σ_U and σ_y specify the Young's Modulus, Poisson's ratio, the ultimate tensile strength and the yield strength, respectively.

It is not necessary to define all the properties on the first data line—if a value is not required, it can be skipped with a comma.

An example usage of this keyword where all properties are defined is given as

```
*MECHANICAL  
200e3, 0.3, 400, 325
```

An example usage of this keyword where only the Young's Modulus and the ultimate tensile strength are defined is given as

```
*MECHANICAL  
200e3, , 400, ,
```

Ending a material definition

Every material definition must begin with the material name assignment.

Text File Usage: *END MATERIAL

There are no parameters or data lines associated with this keyword.

5.5.3 Importing materials from a text file

Materials are imported into the local database by using the *Import* function in Material Manager. The material text file is read through a text file processor and the definitions are saved as a MATLAB binary (.mat) file in *Data\material\local*.

Material data is imported into the local database using the Material Manager GUI or via the command line.

Material Manager Usage: From the *MaterialManager* dialogue select **Import....** Change the file selection filter to **Normal text file (*.txt)**. Select the text file containing material data and select **Open**.

Command Line Usage: >> material.import('material-file-name.*')

5.5.4 Specifying material properties in a job file

Material data may be defined as part of a text-based job file. Job submission from text files is discussed in Section 2.4.3. The material definition may be placed anywhere in the job file provided that it begins and ends with the keywords *USER MATERIAL and *END MATERIAL, respectively. Failing to do so may result in an error.

Input file template

```
*JOB NAME = holePlate
*MATERIAL = steel
*DATASET = stressData.dat
*HISTORY = [1, -1]
beginning of material definitions:
*USER MATERIAL, steel
*CAEL
2e7
*MECHANICAL
200e3, , 400, ,
*FATIGUE, constants
930, -0.095, ,
1, 1, ,
*END MATERIAL
*USER MATERIAL, aluminium
*MECHANICAL
79e3, , 110, ,
*FATIGUE, test data
10000, 62.7, 51.6
1e6, 38.3, 32.7
*R RATIOS
-1, 0
*END MATERIAL
additional options to define the fatigue analysis:
*OUTPUT FIELD = 1
*FATIGUE RESERVE FACTOR = 1
```

Note that the job file contains two material definitions. Although only one of the materials is referenced by the *MATERIAL keyword, both materials are copied to the local material database as .mat files.

5.5.5 Material keyword reference

A complete list of material keywords and their associated syntax can be found in the document *Quick Fatigue Tool User Settings Reference Guide*.

If the user hovers their mouse over a property in the material property editor dialogue box (Figure 5.2), the associated text file keyword is revealed as a tooltip string.

5.6 General material properties

5.6.1 Overview

General material properties are non-physical attributes of the material. It is not compulsory to define general properties and Quick Fatigue Tool will use default properties if none is specified by the user.

5.6.2 Default analysis settings

The default analysis algorithm and mean stress correction is specified in the material. If the default algorithm is selected in the job file, Quick Fatigue Tool always tries to apply the default algorithm defined in the material.

Material Manager Usage: General region of the *Material Editor* dialogue: Select an option from the **Default analysis algorithm** and **Default mean stress correction** drop-down boxes.

Text File Usage: *DEFAULT ALGORITHM, {UNIAXIAL STRESS | UNIAXIAL STRAIN | SBBM | NORMAL | FINDLEY | INVARIANT | NASALIFE}

*DEFAULT MSC, {MORROW | GOODMAN | SODERBERG | WALKER | SWT | GERBER | RATIO | NONE}

Algorithms in the Material Manager GUI which include the letters (CP) utilize critical plane searching. The BS 7608 algorithm cannot currently be specified as a default analysis algorithm.

5.6.3 Specifying no damage for compressive cycles

The effect of compressive stresses can be ignored during the analysis.

Material Manager Usage: General region of the *Material Editor* dialogue: Check the option **No damage in compression**.

Text File Usage: *NO COMPRESSION

This setting has the following effects:

- fatigue damage is not calculated for fully-compressive cycles (maximum stress in cycle ≤ 0);
- the Fatigue Reserve Factor is reset to the maximum value for fully-compressive cycles; and
- compressive stresses are reset to zero for the yield and composite criteria assessments.

5.7 Mechanical material properties

5.7.1 Overview

Mechanical material properties are used throughout Quick Fatigue tool to facilitate the fatigue analysis calculation.

5.7.2 Mechanical constants

The user specifies the four mechanical constants E , ν , σ_u , σ_c and σ_y .

Material Manager Usage: Mechanical region of the *Material Editor* dialogue: Check the property and fill out the edit box.

Text File Usage: *MECHANICAL
First (and only) line: E , ν , σ_u , σ_c , σ_y

5.8 Fatigue material properties

5.8.1 Overview

Fatigue material properties can be defined for stress-based and strain-based fatigue analysis. Stress-based properties define the Wöhler curve, which the user provides either with the Basquin fatigue constants, or fatigue test data in the form of a stress-life (S-N) curve. Strain-based properties define the strain-life curve, which the user provides with the Manson and Coffin fatigue constants.

The user specifies whether to use fatigue constants or fatigue test data in the job file.

Job File Usage: **USE_SN = {0.0 | 1.0}**

0. Use Basquin fatigue coefficients
1. Use fatigue test data (S-N data points) (default)

5.8.2 Fatigue constants

Overview

Fatigue constants define the fatigue life as a function of the stress (or strain) amplitude using the stress-life and strain-life equations.

$$\text{Stress-Life: } \sigma_a = \sigma_f' (2N_f)^b \quad [5.1]$$

$$\text{Strain-Life: } \varepsilon_a = \frac{\sigma_f'}{E} (2N_f)^b + \varepsilon_f' (2N_f)^c \quad [5.2]$$

Defining fatigue constants

The user specifies the fatigue strength coefficient σ_f' , fatigue strength exponent b , fatigue ductility coefficient ε_f' and fatigue ductility exponent c , which define the stress-life and strain-life curves.

Material Manager Usage: Basquin/Manson and Coffin Coefficients region of the *Material Editor* dialogue: Check the property and fill out the edit box.

Text File Usage: *FATIGUE, CONSTANTS
First (and only) line: $\sigma_f', b, \varepsilon_f', c$

The fatigue strength coefficient σ_f' has units of Megapascals.

Defining a second fatigue strength exponent

An alternative value of the fatigue strength exponent b may be defined beyond a certain life to create a “knee” on the Wöhler curve.

The user specifies the value of the second fatigue strength exponent b_2 , and the life $b_{2,NF}$, beyond which the second fatigue strength exponent takes effect.

Material Manager Usage: Basquin Coefficients region of the *Material Editor* dialogue: Open the Insert Knee dialogue by selecting the  button and fill out the edit box.

Text File Usage: *KNEE
First (and only) line: $b_2, b_{2,NF}$

5.8.3 Fatigue test data

Overview

Fatigue test data defines the fatigue life as a function of stress–life data points. Quick Fatigue Tool interpolates logarithmically between the data points to determine the fatigue life for a given stress amplitude. Fatigue test data can be used only with stress-based fatigue analysis algorithms.

Stress values have units of Megapascals.

Defining fatigue test data using the Material Manager

S–N data format

The S–N curve is read into the material property editor via a text file containing i S–N data points. The data can be tab, space or comma-separated, and is defined according to the table below.

N-values	S-values
N_1	S_1
N_2	S_2
.	.
N_i	S_i

The first column is the list of cycles, N_f . The number of cycles must be increasing down the column. The second column defines the stress amplitude for each N-value. S–N data can also be given as the transpose of the above format.

Material Manager Usage: Test Data region of the *Material Editor* dialogue: Select **S–N Data**. Specify the S–N data file. Select **Open**.

Multiple R-ratios

The S–N curve is read into the material property editor via a text file containing i S–N data points at j load ratios. The data can be tab, space or comma-separated, and is defined according to the table below.

N-values	S-values at R_1	S-values at R_2	...	S-values at R_j
N_1	$S_{1,1}$	$S_{1,2}$.	$S_{1,j}$
N_2	$S_{2,1}$	$S_{2,2}$.	$S_{2,j}$
.
N_i	$S_{i,1}$	$S_{i,2}$.	$S_{i,j}$

The first column is the list of cycles, N_f . The number of cycles must be increasing down the column. Subsequent data columns define the stress amplitude for each N-value at the load ratios R_1-R_j . The stress datasets should be ordered so that the load ratios are monotonically increasing. S–N data can also be given as the transpose of the above format.

The user specifies the R-ratio list in the Material Editor.

Material Manager Usage: Test Data region of the *Material Editor* dialogue: Select **R-Values**. Enter the list of R-values in order of lowest to highest. Select **OK**.

If S–N data is provided for one load ratio, the stress amplitude will be assumed to represent a fully-reversed cycle, irrespective of the specified load ratio.

Plotting S–N data points

S–N data can be plotted using the  button. An example of S–N data for multiple R-ratios is given in Figure 5.3.

Material Manager Usage: Test Data region of the *Material Editor* dialogue: Select **S–N Data**. Navigate to the S–N data file and select **Open**.

Sample S–N data

Example text files containing fatigue test data are found in *Data\material\sn_data*.

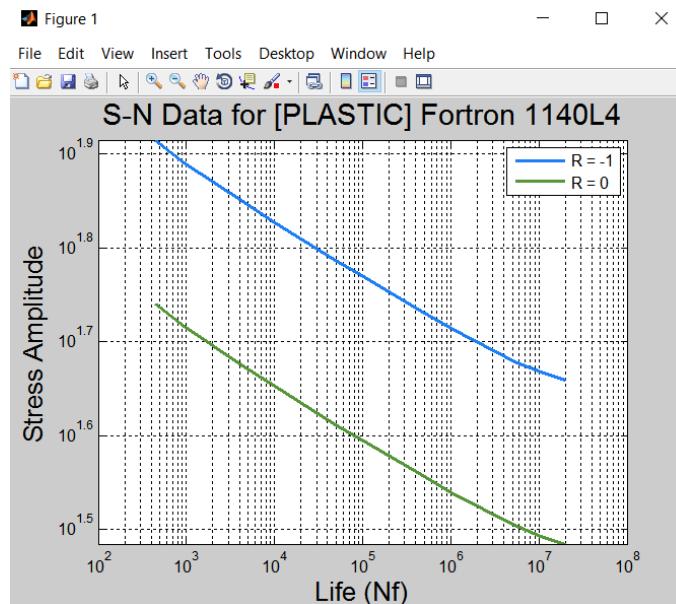


Figure 5.3: S–N data for Fortron 1140L4

Defining fatigue test data inside a material text file

The user specifies the life values N_1-N_i and stress values S_1-S_i for a single load ratio. For j load ratios, the user specifies S_{1-j} for each value of N .

Text File Usage: *FATIGUE, TEST DATA
First line: $N_1, S_{1,1}, S_{1,2}, \dots, S_{1,j}$
Second line: $N_2, S_{2,1}, S_{2,2}, \dots, S_{2,j}$
 i th line: $N_i, S_{i,1}, S_{i,2}, \dots, S_{i,j}$

If the S–N data is defined for multiple R-ratios, these must be specified separately.

Text File Usage: *R RATIOS
First (and only) line: R_1, R_2, \dots, R_j

5.8.4 Defining the fatigue limit for fatigue test data

If the stress cycle is below the fatigue limit, Quick Fatigue Tool may assume zero damage. The fatigue limit is usually calculated from the value of the Constant Amplitude Endurance Limit (defined in Material Manager). However, it is possible to use the fatigue limit directly from the S–N data. When **USE_SN=1** in the job file Quick Fatigue Tool determines the fatigue and endurance limits based on the following logic:

CAEL (Material Manager usage)	CAEL (text file usage)	Fatigue limit definition	Endurance limit definition
n	*CAEL N_∞ 1.0	USE_SN={1 0} : Derived from n using the definition of the fatigueLimitSource environment variable (unless fatigue limit is user-defined)	USE_SN={1 0} : n
2e+07	*CAEL 2e+07 0.0	USE_SN=1 : The S-value corresponding to the last N-value on the user S–N curve USE_SN=0 : Derived from 2×10^7 reversals using the definition of the fatigueLimitSource environment variable (unless fatigue limit is user-defined)	USE_SN=1 : The N-value corresponding to the last S-value on the user S–N curve USE_SN=0 : 2×10^7 reversals

If the fatigue limit is derived from the S–N data and multiple S–N curves are defined, Quick Fatigue Tool uses the $R = -1$ curve (interpolated if necessary).

The behaviour described above only applies to stress-based fatigue analysis.

Not all materials exhibit an endurance limit. Therefore, Quick Fatigue Tool only enforces the limit in certain conditions. Settings related to the endurance limit are described in the document *Quick Fatigue Tool Appendices: A1*.

5.8.5 Cyclic material properties

Cyclic material properties define the hysteresis behaviour of the material when the elastic stress is converted to nonlinear elastic stress and strain using the Ramberg-Osgood cyclic hardening rule.

The user specifies cyclic strain-hardening parameters K' and n' .

Material Manager Usage: Cyclic region of the *Material Editor* dialogue: Check the property and fill out the edit box.

Text File Usage: *CYCLIC
First (and only) line: K', n'

5.9 Composite material properties

5.9.1 Overview

Quick Fatigue Tool can perform failure and damage initiation analyses for fibre-reinforced composite materials. The user must provide the necessary material data for the analysis to evaluate the composite for the desired criterion. Properties are provided for the longitudinal (fibre) and transverse strength of the composite.

When composite material properties are supplied and the composite failure criteria assessment is specified, Quick Fatigue Tool does not perform a subsequent fatigue analysis. Therefore, fatigue properties are not required in conjunction with composite properties.

Refer to Section 12.3 for a detailed overview of each criterion.

5.9.2 Stress-based failure criteria

Overview

Stress-based properties are required for the following criteria:

- maximum stress theory;
- Tsai-Hill theory;
- Tsai-Wu theory; and
- Azzi-Tsai-Hill theory.

These properties are defined using the Fail Stress dialogue in the Material Manager app (Figure 5.4), or in a text file.

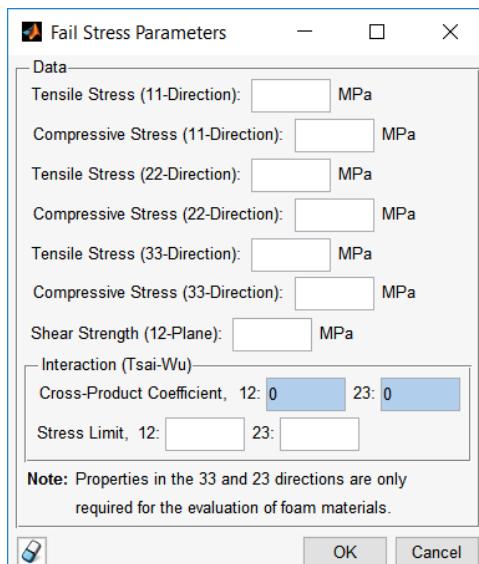


Figure 5.4: Fail Stress dialogue

Defining properties for stress-based failure criteria

The user specifies the material strength in the fibre, transverse and shear directions in addition to optional coupling terms.

Material Manager Usage: Composite Failure Criteria region of the *Material Editor* dialogue:
Select **Fail Stress**. Enter properties into the *Fail Stress* dialogue.

Text File Usage: *COMPOSITE, STRESS
First (and only) line: $X_T, X_C, Y_T, Y_C, Z_T, Z_C, S, f^*_{12}, f^*_{23}, \sigma_{biax,12}, \sigma_{biax,23}$

Usage

Properties can be entered in the three anisotropic directions (11, 22 and 33). The properties Z_T, Z_C, f^*_{23} and $\sigma_{biax,23}$ (directions 33 and 23) are only required for the evaluation of closed cell polyvinyl chloride (PVC) foam in states of plane strain. For the evaluation of fibre-reinforced composites in states of plane stress, only the properties in directions 11, 22 and 12 are required.

Values of f^* and σ_{biax} are only required for the Tsai-Wu theory. If a value of σ_{biax} is not specified, the value of f^* is used instead. The default value of f^* is zero, indicating an uncoupled relationship between the direct and shear stresses.

5.9.3 Maximum strain failure theory

Overview

Strain-based properties are required for the maximum strain failure theory.

These properties are defined using the Fail Strain dialogue in the Material Manager app (Figure 5.5), or in a text file.

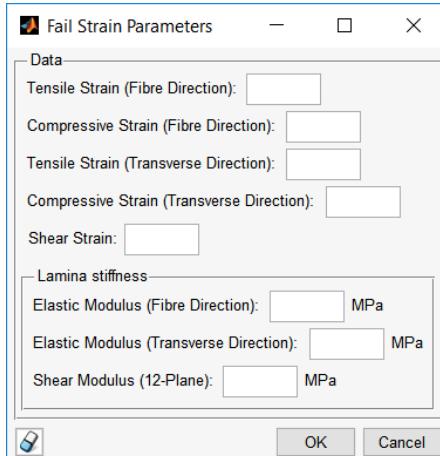


Figure 5.5: Fail Strain dialogue

Defining properties for strain-based failure criteria

The user specifies the limit strain in the fibre, transverse and shear directions and the elastic moduli in the orthotropic directions.

Material Manager Usage: Composite Failure Criteria region of the *Material Editor* dialogue:
Select **Fail Strain**. Enter properties into the *Fail Strain* dialogue.

Text File Usage: *COMPOSITE, STRAIN
First (and only) line: $X_{\varepsilon T}, X_{\varepsilon C}, Y_{\varepsilon C}, Y_{\varepsilon T}, S_{\varepsilon}, E_{11}, E_{22}, G_{\varepsilon,12}$

5.9.4 Hashin's damage initiation criteria

Overview

Properties for Hashin's theory are defined using the Hashin Damage dialogue in the Material Manager app (Figure 5.6), or in a text file.

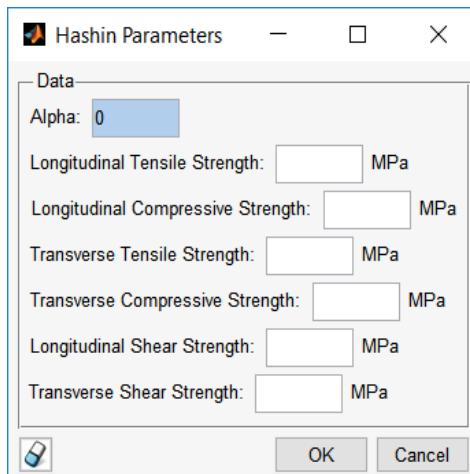


Figure 5.6: Hashin Damage dialogue

Defining properties for Hashin's damage initiation criteria

The user specifies the tensile, compressive and shear material strengths in the longitudinal and transverse directions in addition to an optional coupling term.

Material Manager Usage: Composite Failure Criteria region of the *Material Editor* dialogue:
Select **Hashin Damage**. Enter properties into the *Hashin Damage* dialogue.

Text File Usage: *COMPOSITE, HASHIN
First (and only) line: $\alpha, X_{hT}, X_{hC}, Y_{hT}, Y_{hC}, S_{hT}, S_{hC}$

The default value of the coupling term α is zero, indicating an uncoupled relationship between the direct and shear stresses.

5.9.5 LaRC05 damage initiation criteria

Overview

Properties for the LaRC05 damage initiation criteria are defined using the LaRC05 Damage dialogue in the Material Manager app (Figure 5.7), or in a text file.

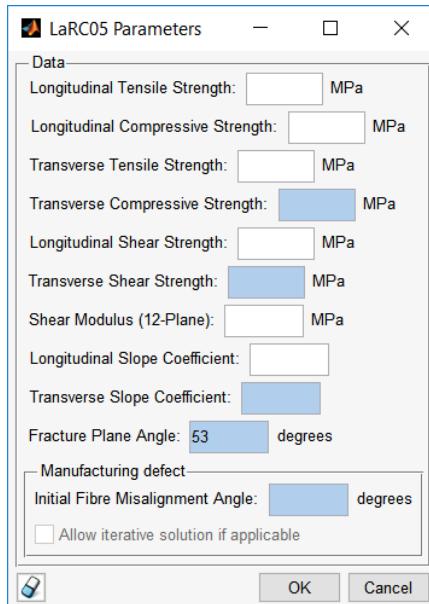


Figure 5.7: LaRC05 dialogue

Defining properties for the LaRC05 damage initiation criteria

The user specifies the tensile, compressive and shear material strengths in the longitudinal and transverse directions in addition to the slope coefficients and the fracture plane angle. Optional manufacturing defect data can also be specified.

Material Manager Usage: Composite Failure Criteria region of the *Material Editor* dialogue:
Select **LaRC05 Damage**. Enter properties into the *LaRC05 Damage* dialogue.

Text File Usage: *COMPOSITE, LARC05
First (and only) line: $X_{lT}, X_{lC}, Y_{lT}, Y_{lC}, S_{lx}, S_{ly}, G_{l,12}, \eta_L, \eta_T, \alpha_0, \varphi^0, I_{\varphi^0}$

5.10 Estimation techniques

5.10.1 Approximating material data

Overview

Fatigue material properties are difficult to find and often only partial data is available. The user can allow Quick Fatigue Tool to approximate material properties using a specified regression algorithm.

In Material Manager, material properties which allow for approximation are distinguished by light blue input fields. If a property is approximated, the method selected in the *Regression* drop-down menu will be used.

Material Manager Usage: Derivation region of the *Material Editor* dialogue: Select the regression method from the Regression drop-down box.

Text File Usage: *REGRESSION, {UNIFORM | UNIVERSAL | MODIFIED | 9050 | NONE}

For detailed information on each regression method, consult the document *Quick Fatigue Tool Appendices: A2*.

Determining the status of the property

A material property has one of four statuses: **Undefined**, **user-defined**, **approximated** or **default**. The material estimation logic is described in the table below.

- The property is unspecified
- The property cannot be approximated
- The property is **undefined**

- The property is specified indirectly
- The property cannot be approximated
- Quick Fatigue Tool will use the **default** value

- The property is unspecified
- Quick Fatigue Tool will attempt to **approximate** the property
- If the property cannot be approximated, it will be **undefined**

- The property is specified indirectly
- Quick Fatigue Tool will attempt to **approximate** the property first
- If the property cannot be approximated, the **user-defined** value will be used

- The property is specified directly
- Quick Fatigue Tool will not attempt to approximate the property
- The property is **user-defined**

5.10.2 Disabling material data approximation

The user specifies that fatigue material data is not approximated.

Material Manager Usage: Derivation region of the *Material Editor* dialogue: Select **None** from the Regression drop-down box.

Text File Usage: *REGRESSION, NONE

5.10.3 Defining material behaviour

The material behaviour is specified in order to instruct Quick Fatigue Tool how to derive certain properties. Specifying the material behaviour:

- controls the default regression method in Material Manager;
- controls the derivation method of the Walker γ -parameter when standard values are used (**walkerGammaSource** = 2.0); and
- controls the default enforcement of the endurance limit (**ndEndurance** = 0.0).

Material Manager Usage: Derivation region of the *Material Editor* dialogue: Select the behaviour from the Material behaviour drop-down box.

Text File Usage: *BEHAVIOR, {STEEL | ALUMINIUM | OTHER}

5.10.4 Defining material classification

The material classification is specified in order to instruct Quick Fatigue Tool how to derive the fatigue shear strength coefficient, τ_f' for Findley's Method.

Material Manager Usage: Stress Sensitivity region of the *Material Editor* dialogue: Select the classification from the Material classification drop-down box.

Text File Usage: *CLASS, {WROUGHT STEEL | DUCTILE IRON | MALLEABLE IRON | WROUGHT IRON | CAST IRON | ALUMINIUM | OTHER}

6. Analysis algorithms

6.1 Background

6.1.1 Overview

Quick Fatigue Tool provides a suite of fatigue analysis algorithms. This section provides a technical background to each algorithm.

6.1.2 Uniaxial analysis algorithms

Uniaxial methods assume that the stresses are only varying in a single direction. Quick Fatigue Tool assumes σ_{11} as the uniaxial direction.

The following uniaxial methods are available:

- Uniaxial Strain-Life; and
- Uniaxial Stress-Life.

Job File Usage: `ALGORITHM = {'UNIAXIAL STRAIN' | 'UNIAXIAL STRESS'}`

Uniaxial methods are recommended for uniaxial stresses only. The user should be aware that uniaxial load cases do not guarantee uniaxial states of stress.

6.1.3 Stress invariant analysis algorithms

Stress invariant methods assume that the damage parameter is represented by an effective stress quantity which is independent of the orientation of the stress tensor. These methods are simple and are intended to work with multiaxial states of stress under certain conditions.

Job File Usage: `ALGORITHM = 'INVARIANT'`

Environment File Usage: `stressInvariantParameter = {0.0 | 1.0 | 2.0 | 3.0 | 4.0}`

0. Program controlled (default)
1. von Mises stress
2. Maximum principal stress
3. Hydrostatic stress
4. Tresca stress

Stress invariant methods are recommended by certain compliance codes such as the British Standards and FKM.

Stress invariant methods are not recommended for non-proportional loads because they cannot reliably track the direction of the most damaging stresses.

6.1.4 Biaxial analysis algorithms

Biaxial methods assume that the stresses of interest are varying primarily on a critical plane where a general combination of shear and normal stress is permitted.

The following biaxial methods are available:

- Stress-based Brown-Miller (brittle/ductile fracture); and
- Normal Stress (brittle fracture only).

Job File Usage: **ALGORITHM** = {'SBBM' | 'NORMAL'}

Biaxial methods are recommended for multiaxial states of stress. Quick Fatigue Tool currently supports stress-based biaxial algorithms—results for low-cycle fatigue applications may be unreliable.

6.1.5 Special purpose analysis algorithms

Special purpose methods are intended to address a specific class of problems described by design and compliance codes.

The following special purpose methods are available:

- Findley's Method (crankshafts);
- BS 7608 (welds/axially-loaded bolts); and
- NASALIFE (aerospace/ceramic matrix composites).

Job File Usage: **ALGORITHM** = {'FINDLEY' | 'WELD' | 'NASALIFE'}

The BS 7608 and NASALIFE algorithms are based on their own design codes. Findley's method is based on literature sources and has been applied to the design of marine crankshafts.

6.1.6 User-defined analysis algorithms

User-defined methods can be specified. Quick Fatigue Tool provides the stress tensor at each point in the load history, for each item in the model. The user writes their own fatigue analysis algorithm and passes the damage values back to Quick Fatigue Tool for results output.

Job File Usage: **ALGORITHM** = 'USER'

6.1.7 Specifying the default analysis algorithm

When the user specifies the default analysis algorithm, Quick Fatigue Tool will use the algorithm specified in the material definition.

Job File Usage: **ALGORITHM** = 'DEFAULT'

6.2 Stress-based Brown-Miller

6.2.1 Overview

The Brown-Miller algorithm postulates that the fatigue damage is dominated by the combination of shear and normal strain [18] [19] [20]:

$$\frac{\Delta\gamma_{max}}{2} + \frac{\Delta\varepsilon_N}{2} = 1.65 \frac{\sigma_f'}{E} (2N_f)^b + 1.75 \varepsilon_f' (2N_f)^c \quad [6.2.1]$$

where $\frac{\Delta\gamma_{max}}{2}$ is the maximum shear strain amplitude, $\frac{\Delta\varepsilon_N}{2}$ is the normal strain amplitude, σ_f' is the tensile fatigue strength coefficient, E is Young's Modulus, b is Basquin's exponent, ε_f' is the fatigue ductility coefficient, c is the fatigue ductility exponent and N_f is the life in repeats (cycles).

The Stress-based Brown-Miller is the same algorithm, but the damage parameter is stress-based. Thus, Equation 6.2.1 becomes:

$$\frac{\Delta\tau_{max}}{2} + \frac{\Delta\sigma_N}{2} = E \left\{ 1.65 \frac{\sigma_f'}{E} (2N_f)^b + 1.75 \varepsilon_f' (2N_f)^c \right\} \quad [6.2.2]$$

Elastic	Plastic
---------	---------

where $\frac{\Delta\tau_{max}}{2}$ is the maximum shear stress amplitude and $\frac{\Delta\sigma_N}{2}$ is the normal stress amplitude. The right-hand side of Equation 6.2.2 is multiplied by Young's Modulus to retain homogeneity.

The Brown-Miller strain-life curve is shown in Figure 6.1.1:

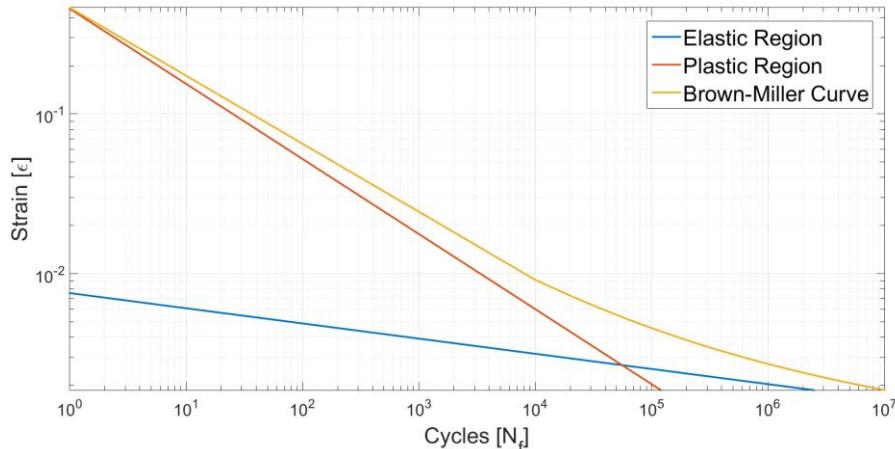


Figure 6.1.1: Brown-Miller strain-life curve, with separate elastic and plastic regions shown.

The Stress-based Brown-Miller algorithm gives the best results for ductile metals. Using the algorithm for brittle materials can result in non-conservative fatigue life predictions.

6.2.2 Nonlinear S-N properties

Overview

For lives greater than one million cycles, the Brown-Miller curve closely resembles its elastic constituent. Therefore, for HCF applications it is usually sufficient to assume that plasticity effects are sufficiently small so that the plastic portion of the equation can be neglected. This is the default behaviour in Quick Fatigue Tool.

Specifying nonlinear S-N properties

The user enables usage of the plastic region of the S-N curve in the environment file.

Environment File Usage: **plasticSN** = {0.0 | 1.0}

0. Use the elastic region of the S-N curve only
1. Use the total region of the S-N curve

6.2.3 Required material properties

The following material properties are required to perform an analysis with the Stress-based Brown-Miller algorithm:

Property	Symbol	Importance
Tensile fatigue strength coefficient	σ_f'	REQUIRED
Tensile fatigue strength exponent	b	REQUIRED
Young's Modulus	E	REQUIRED
Fatigue ductility coefficient	ε_f'	OPTIONAL
Fatigue ductility exponent	c	OPTIONAL

6.2.4 Using stress-life data

Due to the nature of the Stress-based Brown-Miller equation, results obtained from stress-life data can differ significantly compared to the use of material coefficients. When using stress-life data with **USE_SN**=1.0 in the job file, Quick Fatigue Tool uses the damage parameter to interpolate the endurance curve. Since the endurance curve typically arises from stress-based testing and the Stress-based Brown-Miller equation take sit form from its strain-life counterpart, the corresponding life values are not guaranteed to be the same.

6.2.5 Cycle counting

Overview

The Stress-based Brown-Miller algorithm uses the normal and shear stress amplitude to define the total damage parameter. This poses an additional challenge for the cycle counting process, because the formulation in Equation 6.2.2 implies that the damage parameter is the sum of the cycle counted normal and shear stress. However, if these two quantities are cycle counted before being summed, there is no guarantee that the counted histories will still have the same length, and matrix addition may not be immediately possible.

The alternative is to combine the normal and the shear stress beforehand, and cycle count the single combined parameter. This circumvents the issue of matrix addition, but may lead to incorrect fatigue results. For example, consider the normal and shear histories [100, 50] and [20, 50], respectively. Their individual amplitudes are 25 and 15, respectively, meaning that the sum of their amplitudes is 40. If the parameters are combined first to give the history [120, 100], the resulting amplitude is 10. Therefore, cycle counting the combined history may lead to a totally different value of fatigue damage.

Specifying the order of cycle-counted parameters

The user controls the order of operations in the environment file.

Environment File Usage: **`rainflowMode`** = {1.0 | 2.0}

1. Combine the normal and shear parameters, then count the resulting history (default)
2. Count the normal and shear parameters separately, then combine the resulting histories

Usage

The second method is considered by the author to be the most physically correct approach, although it is significantly more time-consuming. Before the cycle counted normal and shear stress histories are combined, Quick Fatigue Tool checks the length of each history and resamples the shorter parameter in order to allow matrix addition. If significant resampling is required, the combined stress parameter may no longer be accurate. In such cases, the user should compare the fatigue life results for both cycle counting methods.

Testing reveals that for the majority of simple load cases, there is little or no difference in the fatigue result between the two methods. For loads where there is a significant phase difference between the normal and shear stresses, the user should compare the fatigue life results for both cycle counting methods and elect the method which offers the favourable accuracy-to-time ratio. The default setting is the first approach.

6.2.6 Proportional loading

Overview

If Quick Fatigue Tool detects that the fatigue loading is proportional, the Stress-based Brown-Miller algorithm is run without critical plane searching, since the direction of the principal stresses does not change. In such cases, Quick Fatigue Tool uses the proportional-load damage parameter.

Specifying the damage parameter for proportional loading

The user specifies the proportional-load damage parameter in the environment file.

Environment File Usage: **`sbbmParameter`** = {1.0 | 2.0}

1. Maximum normal stress
2. Maximum combined (normal + shear) stress (default)

Determining the proportional-load damage parameter

The maximum normal stress is the numerically largest principal stress, $\max\{\sigma_1, \sigma_3\}$.

The maximum combined stress, $\sigma_{c,max}$, is calculated from the principal stress according to Equations 6.2.3-4:

$$\sigma_{c,max} = \sigma_c(\theta_f) \quad [6.2.3]$$

$$2\sigma_c(\theta) = \sigma_1 + \sigma_3 + (\sigma_1 - \sigma_3) \cos(2\theta) - (\sigma_1 - \sigma_3) \sin(2\theta) \cdot \text{sign}(\sigma_h) \quad [6.2.4]$$

where $\sigma_{1,3}$ are the maximum and minimum principal stresses, respectively, σ_h is the hydrostatic stress and θ_f is the fracture transformation angle.

The solution to θ_f is the value of θ_{f0} which produces the numerically largest value of σ_c in the range $\{0 \leq \theta \leq 180\}$:

$$\frac{\partial}{\partial \theta}(\sigma_c) = 0 \xrightarrow{\text{yields}} \theta \equiv \theta_{f0} \quad [6.2.5]$$

Load proportionality is discussed further in Section 4.5.3.

6.3 Normal Stress

6.3.1 Overview

The Normal Stress algorithm uses the normal stress amplitude as the damage parameter in the stress-life equation:

$$\frac{\Delta\sigma_{max}}{2} = \sigma_f' (2N_f)^b \quad [6.3.1]$$

where $\frac{\Delta\sigma_{max}}{2}$ is the maximum normal stress amplitude on the critical plane and the other symbols have their usual meaning. The algorithm predicts that the fatigue strength in torsion and tension is the same. In reality, the allowable normal stress in torsion is approximately 60% of the allowable axial stress. Therefore, the Normal Stress algorithm only provides accurate fatigue life estimates for brittle materials whose crack initiation is dominated by normal stresses. The algorithm is highly non-conservative for ductile metals where the fatigue life is dominated by shear stresses [21].

6.3.2 Required material properties

The following material properties are required to perform an analysis with the Normal Stress algorithm:

Property	Symbol	Importance
Tensile fatigue strength coefficient	σ_f'	REQUIRED
Tensile fatigue strength exponent	b	REQUIRED

6.3.3 Critical plane searching

The principal stress is often used as an effective stress parameter; since it is an invariant quantity, it is tempting to bypass critical plane searching. However, even for simple loadings, relying on the value of the maximum principal stress can yield unexpected results. Take the simple stress tensor given by Equation 6.3.2:

$$\boldsymbol{\sigma} = \begin{bmatrix} \sigma_{11} & 0 & 0 \\ 0 & 0 & 0 \\ 0 & 0 & 0 \end{bmatrix} \quad [6.3.2]$$

The resulting principal stress state when the above tensor is subjected to a fully-reversed loading event, $[1, -1]$, is given by Figure 6.3.1.

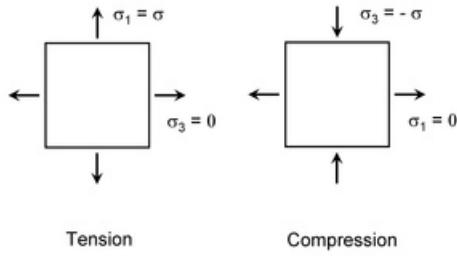


Figure 6.3.1: Principal stress state due to fully-reversed load

Since the principal stresses are ordered such that $\sigma_1 \geq \sigma_2 \geq \sigma_3$, a simple tension-compression event causes the principal direction to rotate by 90 degrees. This problem is avoided if the normal tensile stresses are calculated over a series of planes using critical plane searching [22]. The critical plane is defined as the plane which experiences the largest combination of normal stress range and mean stress.

The Normal Stress algorithm posits that a component subjected to a uniaxial stress cycle, $[\sigma_{11}, -\sigma_{11}]$, will fail on a plane where the shear stress is zero. The normal and shear stress on the critical plane using the Normal Stress algorithm are illustrated in Figure 6.3.2.

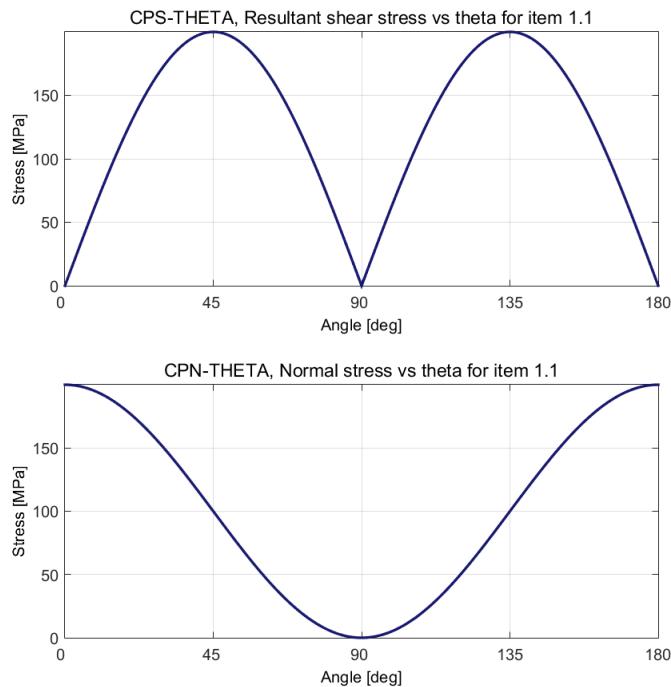


Figure 6.3.2: Normal and resultant shear stress on the critical plane for uniaxial tension

The critical plane occurs when theta is zero, which corresponds to one of the principal planes.

6.4 Findley's Method

6.4.1 Overview

Findley's Method proposes that crack initiation is due to the combined effect of the average normal stress and the alternating shear stress on the critical plane [23] [24] [25]:

$$\frac{\Delta\tau}{2} + k\sigma_n \Big|_{max} = \tau_f^*(N_f)^b \quad [6.4.1]$$

where $\frac{\Delta\tau}{2}$ is the maximum shear stress amplitude on the critical plane, σ_n is the normal stress, k is a material constant describing the sensitivity of the material to normal stresses and τ_f^* is a function of the torsional fatigue strength coefficient.

Findley's Method was originally presented in the form of a safety factor; however, by introducing the stress-life curve defined by N_f and b , the algorithm is also well-suited to finite, HCF life prediction. The value of k is determined experimentally from the tensile and torsional fatigue limit, and determines the influence of the normal stress on the calculated fatigue damage. Consider the example where a specimen under torsional loading experiences a very large mean normal stress. Even if the normal stress amplitude is small (close to static), the predicted fatigue damage from Findley's Method can be very conservative. The normal stress sensitivity constant acts to attenuate the effect of σ_n on the loading and can be considered a form of mean stress correction.

Another advantage of Findley's Method is that it is well-suited to both brittle and ductile metals. The tensile and torsional fatigue limit can be used to "tune" a value of k which accurately characterises the material response.

Work by Kallmeyer et al. has shown that the Findley critical plane method provides the best representation for smooth bar data, which gives the method significance in applications involving shafts under shear loads [26] [27].

6.4.2 Required material properties

The following material properties are required to perform an analysis with the Findley algorithm:

Property	Symbol	Importance
Tensile fatigue strength coefficient	σ_f'	REQUIRED
Tensile fatigue strength exponent	b	REQUIRED
Normal stress sensitivity constant	k	REQUIRED ³
Modified fatigue shear strength coefficient	τ_f^*	REQUIRED ⁴
Poisson's Ratio	ν	OPTIONAL
Tensile Fatigue Strength Limit	f	OPTIONAL
Torsional Fatigue Strength Limit	t	OPTIONAL
Ultimate Tensile Strength	σ_U	OPTIONAL

³ If no value is specified, a default value is used.

⁴ This parameter is computed automatically.

6.4.3 Determining the value of τ_f^*

The modified fatigue shear strength coefficient is calculated from the standard equation:

$$\tau_f^* = \sqrt{1 + k^2 \tau_f'} \quad [6.4.2]$$

where τ_f' is the fatigue shear strength coefficient. τ_f' is obtained from the following table:

Material classification	Fatigue Shear Strength Coefficient
Wrought steel and alloys	$\tau_f' \approx 0.75\sigma_f'$
Ductile iron	$\tau_f' \approx 0.90\sigma_f'$
Malleable iron – pearlitic structure	$\tau_f' \approx 1.00\sigma_f'$
Wrought iron	$\tau_f' \approx 0.83\sigma_f'$
Cast iron	$\tau_f' \approx 1.30\sigma_f'$
Aluminium/copper and alloys	$\tau_f' \approx 0.65\sigma_f'$
Other	$\tau_f' \approx \frac{\sigma_f'}{2(1+v)}$

Guidance on defining material classification can be found in Section 5.10.4.

6.4.4 Determining the value of k

The normal stress sensitivity constant is determined by comparing the fatigue limit of a material under tension and torsion fatigue tests. The value of k is specified in the *Stress Sensitivity* region of the material editor (Figure 6.4.1).

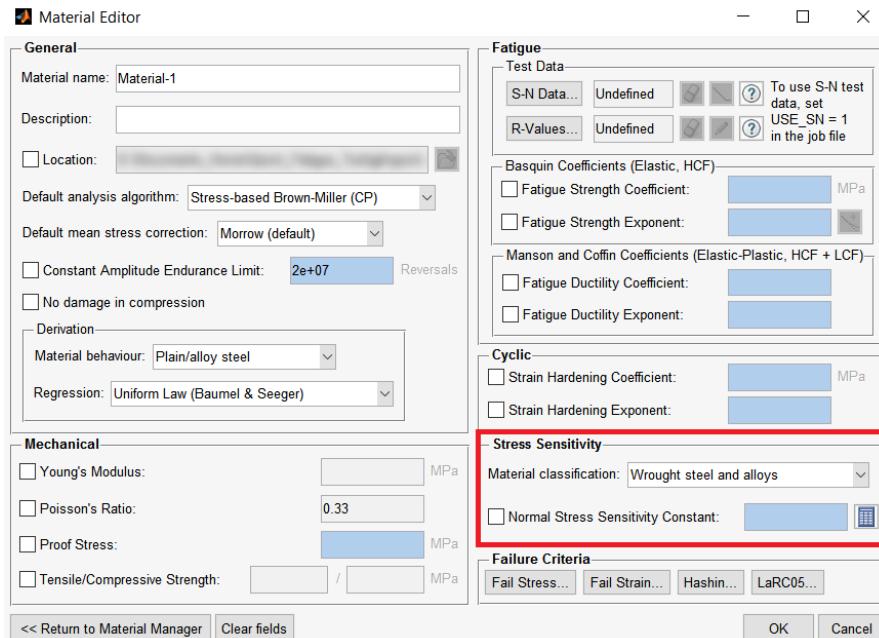


Figure 6.4.1: Definition of the normal stress sensitivity constant in the material editor

The value can either be specified directly by checking the box next to the input box, or a value may be calculated based on the fatigue limit. This is done by clicking on the calculator button. The resulting dialogue box is shown in Figure 6.4.2.

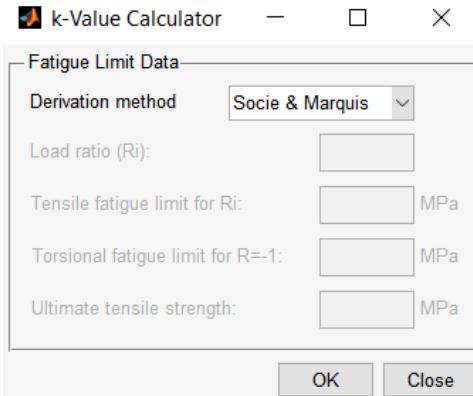


Figure 6.4.2: Calculator tool to estimate the value of k

The value of k is calculated according to the table below. Note that if the input field is left blank, the Socie & Marquis value of 0.2857 will be assumed [28]. For ductile materials k has a solution in the range of 0.2-0.3 [29].

Derivation Method	Solution
Socie & Marquis	$k = 0.2857$
General formula ⁵	$\frac{f_R}{t_{-1}} = \frac{2 \cdot \sqrt{1 + k^2}}{\sqrt{\left(\frac{2k}{1-R}\right)^2 + 1} + \frac{2k}{1-R}}$
Dang van	$k = \frac{3t_{-1}}{f_{-1}} - \frac{3}{2}$
Sines	$k = \frac{3t_{-1}(\sigma_U + f_{-1})}{\sigma_U \cdot f_{-1}} - \sqrt{3}$
Crossland	$k = \frac{3t_{-1}}{f_{-1}} - \sqrt{3}$

Where f_R is the tensile fatigue limit for a load ratio R , f_{-1} is the fully-reversed tensile fatigue limit and t_{-1} is fully-reversed torsional fatigue limit. Since the derivation models attempt to approximate k based on the fatigue limit, they are not guaranteed to find a solution. In such cases, the Socie & Marquis value may be used.

Note that defining k as zero makes Findley's Method a maximum shear stress criterion.

⁵ Requires the Symbolic Math Toolbox

6.4.5 Critical plane searching

Overview

Findley's Method uses critical plane searching to determine the value of $\frac{\Delta\tau}{2} + k\sigma_n \Big|_{max}$. The stress tensor on the critical plane is split into one normal and two shear components:

$$\sigma_n = \sigma'(1,1)$$

$$\tau_{xy} = \sigma'(1,2) \quad [6.4.3]$$

$$\tau_{xz} = \sigma'(1,3)$$

The variable σ' is described in the document *Quick Fatigue Tool Appendices: A1*. The normal and shear stress history is illustrated for a plane Δ of an arbitrary orientation by Figure 6.4.3.

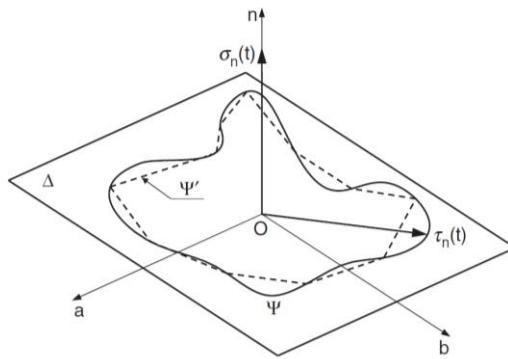


Figure 6.4.3: Normal and shear stress relative to plane Δ

On these planes, the n direction is always perpendicular to the ab plane on which the shear stresses act. The quantity $\tau_n(t)$ is the resultant shear stress history, which scribes the path Ψ . The value of the maximum normal stress is simply the maximum value of the normal stress history, $\sigma_n(t)$. Determining the value of the maximum shear stress is less trivial.

Shear stress on the critical plane

Several methods have been proposed for determining the maximum shear stress. These include—but are not limited to—the longest chord [30], the longest projection [31], the minimum circumscribed circle [32], the minimum circumscribed ellipse [33] [34] and the maximum variance [35] [36] of the path Ψ . It has been noted by Susmel [25] that the longest chord method is not only simple but also very effective when applied in conjunction with the critical plane concept. Quick Fatigue Tool uses the longest chord method to determine the maximum shear stress history on the critical plane. This is given by Equation 6.4.4.

$$\tau_{n,a} = \frac{1}{2} \max_{t_1 \in T} \left[\max_{t_2 \in T} |\tau_n(t_1) - \tau_n(t_2)| \right] \quad [6.4.4]$$

where t_1 and t_2 are two instants of the cyclic load history having period equal to T . The longest chord method requires every shear pair along Ψ to be compared. Therefore, for large histories the analysis may be slowed down significantly. In cases where the calculation time of the maximum shear stress is unreasonable, the maximum resultant shear stress may be used instead:

$$\tau_{n,a} = \sqrt{\tau_{xy}^2 + \tau_{xz}^2} \quad [6.4.5]$$

Specifying the shear stress calculation

The user specifies the shear stress calculation method in the environment file.

Environment File Usage: `cpShearStress = {1.0 | 2.0}`

1. Longest chord method (default)
2. Maximum resultant shear stress method

Usage

The longest chord method suffers from a theoretical set-back. Figure 6.4.4 illustrates how the mean value of the shear stress cannot be defined with certainty when two or more reference chords having the same length can be defined. However, Susmel states that from a practical point of view, this ambiguity should not affect the accuracy in estimating high-cycle fatigue since the torsional mean stress can be neglected provided that the maximum shear stress in the loading does not exceed the material torsional yield strength.

The longest chord method is only applied to the calculation of the maximum shear stress history on the critical plane (output variable **CS**). For the critical plane analysis, the cycle counted shear quantity is the resultant shear stress given by Equation 6.4.5.

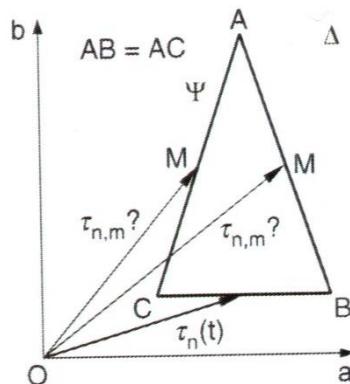


Figure 6.4.4: Limitation of the longest chord method

Normal stress on the critical plane

The maximum shear stress history on each plane is cycle counted using the Rainflow method described in the document *Quick Fatigue Tool Appendices: A1*. It is not obvious how the maximum normal stress should be combined with the shear stress cycles. This issue is illustrated by Figure 6.4.5.

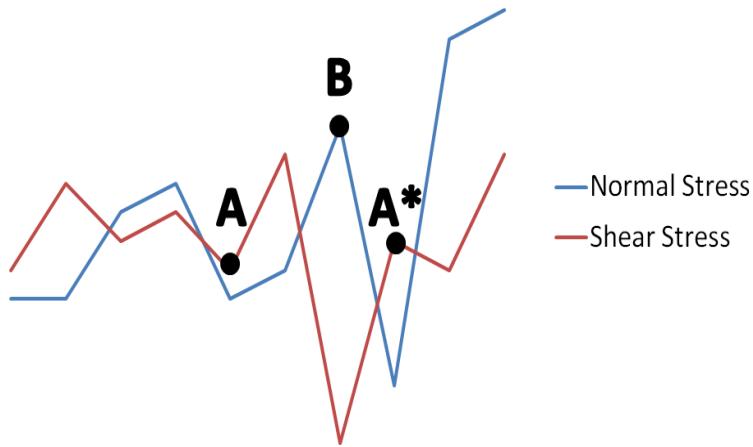


Figure 6.4.5: A shear stress cycle with start and end points A and A*, respectively. By default, the normal stress associated with the shear cycle is the maximum normal stress, B, occurring in the interval A-A*

Specifying the normal stress calculation

The user specifies the calculation of the normal stress in the environment file.

Environment File Usage: `findleyNormalStress = {1.0 | 2.0 | 3.0}`

1. Each shear stress cycle is combined with the maximum normal stress over the whole loading.
This is the most conservative approach
2. Each shear stress cycle is combined with the maximum (positive) normal stress in the shear cycle interval. This is the most balanced approach (default)
3. Each shear stress cycle is combined with the average normal stress in the shear cycle interval.
This is the most non-conservative approach

6.4.6 Output

The variables **WCM** and **SFA** have a slightly different meaning when using Findley's Method:

Variable	Usual meaning	Findley's Method
WCM	Mean value of the damage parameter on the critical plane (algorithm-dependent)	Mean value of the resultant shear stress on the critical plane
SFA	Ratio between the material fatigue limit and the maximum stress in the loading	Ratio between the material fatigue limit and the Findley parameter

6.4.7 Limitations

The critical plane search algorithm used by Findley's Method is found to be very sensitive to the search increment when compared to other algorithms which use critical plane searching. Therefore, Findley's Method is not compatible with load proportionality checking. The step size will always be the value defined in the environment file.

Literature sources reference the shear stress in conjunction with Rainflow cycle counting. However, it is not clear which shear stress quantity should be cycle counted. Currently, Quick Fatigue Tool uses the resultant shear stress history as the cycle counting parameter. This has the drawback that the resultant shear stress is always positive. In order to circumvent this problem, a scheme similar to that used by the Stress Invariant Parameter algorithm is used, whereby the shear stress history is multiplied by a factor based on a pre-determined sign convention (Section 6.5.5).

6.5 Stress Invariant Parameter

6.5.1 Overview

The Stress Invariant Parameter analysis algorithm uses a Cauchy stress invariant term as the damage parameter in the stress-life relationship:

$$\frac{\Delta\sigma_{eff}}{2} = \sigma_f' (2N_f)^b \quad [6.5.1]$$

where $\frac{\Delta\sigma_{eff}}{2}$ is the effective stress amplitude.

6.5.2 Specifying the stress invariant parameter

The user specifies the stress invariant parameter in the environment file.

Environment File Usage: `stressInvariantParameter = {0.0 | 1.0 | 2.0 | 3.0 | 4.0}`

0. Program controlled (default)
1. von Mises
2. Principal
3. Hydrostatic
4. Tresca

6.5.3 Required material properties

The following material properties are required to perform an analysis with the Stress Invariant Parameter algorithm:

Property	Symbol	Importance
Tensile fatigue strength coefficient	σ_f'	REQUIRED
Tensile fatigue strength exponent	b	REQUIRED

6.5.4 Effective stress parameters

von Mises stress

The von Mises stress is based on the second invariant stress, and provides an estimate of the onset of yielding. The von Mises stress is given by Equation 6.5.2:

$$\sigma_{eff} = \sqrt{\frac{3}{2} s_{ij} s_{ij}} \quad [6.5.2]$$

where s_{ij} are the components of the stress deviator tensor σ^{dev} :

$$\sigma^{dev} = \sigma - \frac{1}{3} (tr \sigma) I \quad [6.5.3]$$

where I is the identity matrix.

Principal stress

The principal stress parameter defines the load history as the largest (positive or negative) principal stress at each point in the loading. For example, if the absolute value of the minimum principal stress is larger than the value of the maximum principal stress at a given loading point, then the minimum principal stress is used for that point. The following load history illustrates the use of the principal stress as the invariant parameter.

σ_1	349	294	174	441
σ_3	-294	-349	-147	-523
Load history	349	-349	174	-523

The principal stress is valid for uniaxial test data. Loadings which exhibit a high degree of biaxiality do not correlate well to the principal stress, since failure is not guaranteed to occur at the locations of maximum stress. For loadings which exhibit a high degree of non-proportionality, the direction of the principal stress will change throughout the history; in such cases, the Normal Stress algorithm is recommended instead.

Hydrostatic stress

The hydrostatic stress defines the load history in terms of the equivalent pressure stress.

$$\sigma_{eff} = -\frac{1}{3}(tr \boldsymbol{\sigma}) = -\frac{1}{3}\sigma_{ii} \quad [6.5.4]$$

The hydrostatic stress is an isotropic parameter given by the average of the direct pressure forces acting on a body. Deformation states dominated by expansion correlate well with a hydrostatic criterion. The hydrostatic stress parameter is less conservative than the principal stress parameter.

Tresca stress

The Tresca stress is defined as the maximum difference between the first and third principal stress.

$$\sigma_{eff} = \sigma_1 - \sigma_3 \quad [6.5.5]$$

The Tresca stress is the maximal shear stress and is used as a yield criterion for ductile metals. As a fatigue criterion, the Tresca stress assumes that crack initiation is driven by states of pure shear. This can provide reasonable estimates for shear-dominated loads with a high degree of proportionality. In all other cases, a balanced shear-normal biaxial criterion such as the Stress-based Brown-Miller algorithm or Findley's method is recommended instead.

Program controlled

Quick Fatigue Tool can attempt to choose a suitable stress invariant parameter. The applicability of a given stress invariant parameter depends on the biaxiality ratio, α :

$$\alpha = \frac{\sigma_2}{\sigma_1} \quad [6.5.6]$$

Stress invariants are applicable to uniaxial ($\alpha = 0$) and equibiaxial ($\alpha = 1$) loads, as well as proportional biaxial loads ($-1 \leq \alpha \leq 1$) [37]. The table below shows the applicable range of the biaxiality ratio for each stress invariant parameter.

Invariant	Range of α
von Mises	$\alpha = 0$ and $\alpha = 1$
Principal	$-1 \leq \alpha \leq 0$
Hydrostatic	$-1 \leq \alpha \leq 0$
Tresca	$0 \leq \alpha \leq 1$

The stress invariant parameter is chosen on the basis of the applicable range of α . If no suitable parameter can be found, the principal stress is used by default.

The range of α over the loading is printed in the message file at the item with the largest principal stress, for each analysis group.

6.5.5 Sign convention

Overview

The von Mises and Tresca stresses are always positive, meaning that damage in compression is neglected. A material element in a state of pure hydrostatic compression appears to experience zero effective stress, even if the volumetric deformation is large enough to cause fatigue damage. The solution is to correct the stresses by using a criterion which determines the correct sign of the effective stress parameter.

Specifying the sign convention

The user specifies the sign convention in the environment file.

Environment File Usage: `signConvention = {1.0 | 2.0 | 3.0}`

1. Take the sign from the hydrostatic stress (default)
2. Take the sign from the largest stress
3. Take the sign from Mohr's circle

6.5.6 Additional guidance

The stress invariant parameters do not correlate well to multiaxial stress states. Although the algorithm can be used to quickly locate the region of expected maximum stress, the location of maximum damage can often be elsewhere due to the fact that the in-plane principal directions can change during the loading. Thus, the Stress Invariant Parameter analysis algorithm is included for completeness only; none of the invariants are recommended as a damage parameter except for the simplest cases.

The user should check the validity of the selected stress invariant parameter before the analysis. This can be done by setting **DATA_CHECK=1** in the job file and inspecting the message file for feedback regarding the selected parameter. The following points should be observed:

- for $-1 \leq \alpha < 0$, the Tresca stress is very conservative; the von Mises stress is conservative; the principal stress is acceptable;
- for $\alpha = 0$, the Tresca, von Mises and principal stresses are all acceptable;
- for $0 < \alpha < 1$, the Tresca stress is acceptable; the von Mises stress is non-conservative; and
- for $\alpha = 1$, both the Tresca and von Mises stresses are acceptable.

Critical plane searching is not required, and rough estimates of life can be obtained very quickly. However, the Stress Invariant Parameter algorithm is not considered to be a valid durability assessment criterion for general fatigue analysis problems [6].

6.6 BS 7608 Fatigue of Welded Steel Joints

6.6.1 Overview

Quick Fatigue Tool includes an implementation of the British Standard BS 7608:1993 code of practice for fatigue design and assessment of steel structures [38]. The standard is applicable to the following:

- Parent material remote from joints;
- Welded joints (in air or sea water) in such material;
- Bolted or riveted joints in such material; and
- Shear connectors between concrete slabs and steel girders acting compositely in flexure.

The standard offers a family of S_r-N curves based on weld geometry criteria, spanning ten weld classifications. The damage parameter is the stress range acting on the critical plane. The code stipulates that the stress range is whichever of the two in-plane principal stresses lie within +/- 45 degrees of an axis perpendicular to the weld toe. However, Quick Fatigue Tool performs a full critical plane search in a spherical coordinate space. The user may choose between the normal and the shear stress as the damage parameter.

The standard is based on the assumption that the exact stresses at the weld toe cannot be determined analytically. The provided S_r-N curves account for the effect of the stress concentration, thus the analyst need only compute the stresses as if the weld feature did not exist. If the stress solution is obtained from finite element analysis and the weld detail is modelled, the calculated stress range may be greatly overestimated and could result in highly conservative fatigue life predictions. Therefore, in such cases the analyst should choose the stress a short distance away from the weld toe, an example of which is given in Figure 6.6.1.

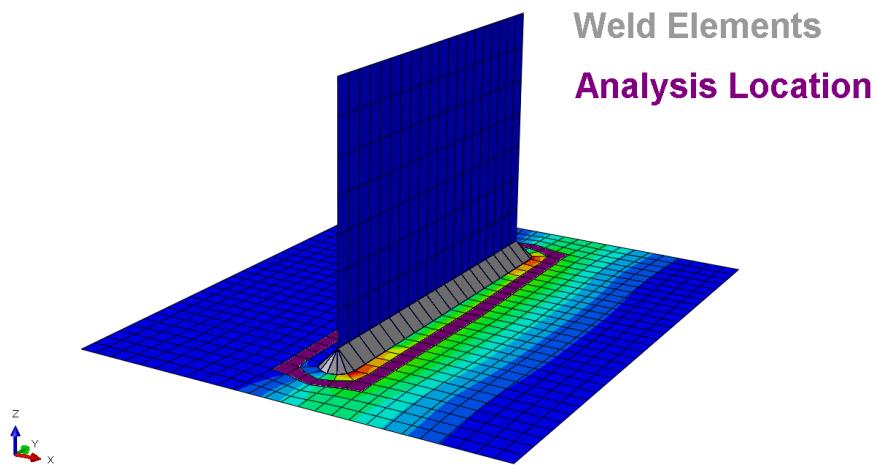


Figure 6.6.1: Example analysis location for BS 7608

6.6.2 Derivation of the $S_r - N$ curve

Overview

The $S_r - N$ curve is defined explicitly as Equation 6.6.1

$$S_r^m N = C_d \quad [6.6.1]$$

where S_r is the stress range, m is the Paris Law exponent related to the energy release rate of a crack, N is the number of cycles to failure and C_d is a constant relating to the weld classification. The resulting $S_r - N$ curves are shown in Figure 6.6.2. The curves are material-independent and as such, no material needs to be specified in the job file.

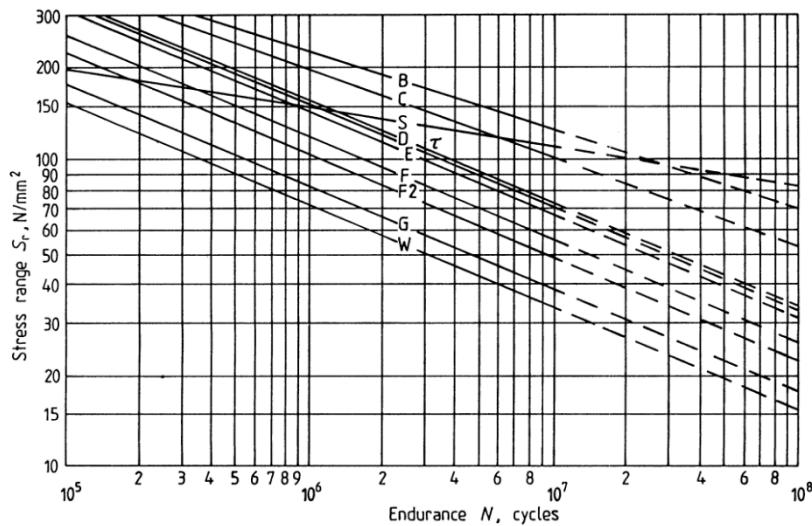


Figure 6.6.2: $S_r - N$ curve family for BS 7608

Probability of failure

The weld classification constant, C_d , is related to the probability of failure by the number of standard deviations from the mean $S_r - N$ curve. For example, two standard deviations imply that there is a 97.7% probability that the component will fail before the predicted life. Some values of d and their corresponding probabilities of failure are given as

Probability of failure (%)	d
50	0.0
69	0.5
84	1.0
97.7	2.0
99.86	3.0

The user specifies the number of standard deviations d in the job file.

Job File Usage: `DEVIATIONS_BELOW_MEAN = d (0)`

A value of $d = 0$ corresponds to the mean-line $S_r - N$ curve, while a value of $d = 2$ corresponds to the standard design curve.

6.6.3 Analysis of axially loaded bolts

Overview

In addition to welded joints, BS 7608 also offers a set of $\frac{S_r}{\sigma_u} - N$ curves for axially loaded bolts with cut, ground or rolled threads up to 25mm in diameter. These curves belong to class X. The curves are defined by Equations 6.6.2-3 and illustrated by Figure 6.6.3.

$$\text{Mean curve: } \left(\frac{S_r}{\sigma_u}\right)^3 N = 800 \quad [6.6.2]$$

$$\text{Mean} - 2SD: \left(\frac{S_r}{\sigma_u}\right)^3 N = 400 \quad [6.6.3]$$

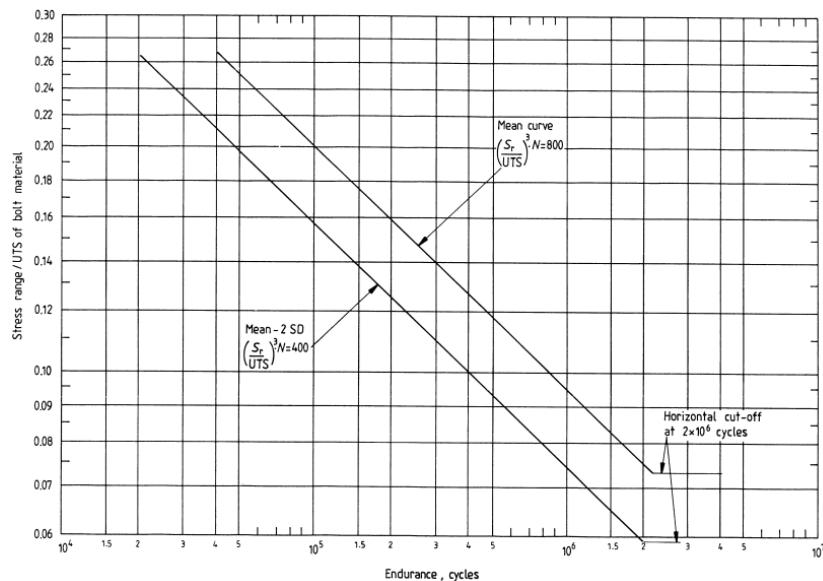


Figure 6.6.3: $\frac{S_r}{\sigma_u} - N$ curve family for axially loaded bolts

Location of failure

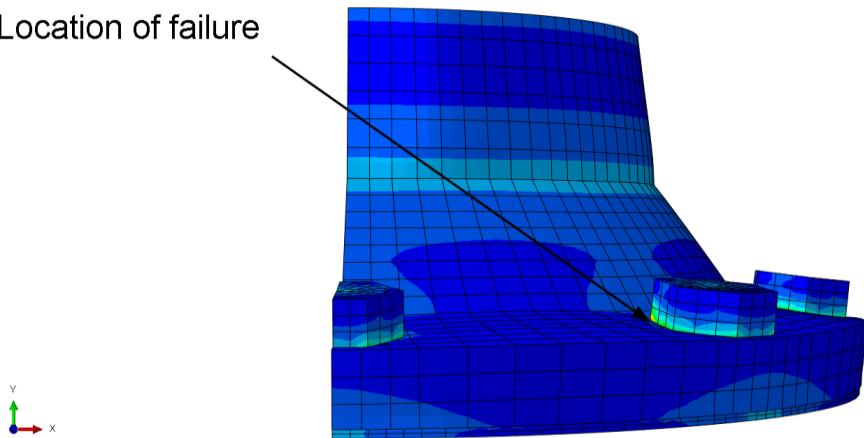


Figure 6.6.4: Example of a bolt failure in FEA

The $\frac{S_r}{\sigma_u} - N$ curves in this classification are only defined for the mean line or two standard deviations from the mean. The ultimate tensile strength of the bolt feature must be provided. The $\frac{S_r}{\sigma_u} - N$ curves are valid only for values of the ultimate tensile strength up to 785MPa.

If the bolt is modelled in FEA such as the one shown in Figure 6.6.4, the effective stress should be taken a small distance away from the stress concentration at the location of crack initiation otherwise the fatigue analysis will produce conservative results.

Specifying the analysis of axially loaded bolts

The user specifies the analysis of axially loaded bolts in the job file.

Job File Usage: **WELD_CLASS = 'X'**

A value of the ultimate tensile strength σ_u of the bolt material must be specified.

Job File Usage: **UTS = σ_u**

6.6.4 Effect of the characteristic length

Overview

For welded joints, the characteristic length is the plate thickness. For class X (axially loaded bolts), the bolt diameter is used instead.

The fatigue life of welded joints and bolts reduces with increasing characteristic length. If a value for the weld length is specified in the job file, the S_r-N curve may be scaled according to Equation 6.6.4

$$S_F = \frac{S}{S_B \left(\frac{16}{t} \right)^{0.25}} \quad [6.6.4]$$

where S is the fatigue strength of a weld (or bolt) of thickness (or diameter) t and S_B is the fatigue strength of the weld without considering the effect of thickness (or diameter). The characteristic length is given in units of mm. The S_r-N curves are already valid for the lengths given by the table below. Thus, the correction is only performed if the specified length lies outside the pre-defined range.

Classification	Range of characteristic length
Nodal joints (Class T)	16mm only
Non-nodal joints (Classes B to G)	Up to 16mm
Bolts (Class X)	Up to 25mm diameter
All other weld classes	16mm only

Specifying the characteristic length

The user specifies the characteristic length L in the job file. It is not compulsory to define a characteristic length—if the option is left empty, a value of 1mm is assumed.

Job File Usage: **CHARACTERISTIC_LENGTH = L (1)**

6.6.5 Effect of stress relief

BS 7608 assumes that the stress range is the sum of the tensile and 60% of the compressive component of the cycle. For example, if a cycle has a range of -100 to 50, the effective range will be calculated as $abs(0.6 \times -100) + 50 = 110$.

6.6.6 Effect of small cycles

BS 7608 stipulates that earlier fatigue failure could be predicted if it is assumed that all stress ranges below the fatigue limit are non-damaging. Thus, the Paris Law exponent is changed from m to $(m + 2)$ for cycles below the fatigue limit, at which the calculated life is 1e7 cycles.

6.6.7 Effect of large cycles

BS 7608 stipulates that the S_r - N curves may be extrapolated no further than twice the material's yield strength. Cycles exceeding this value will result in non-fatigue failure.

6.6.8 Effect of exposure to sea water

Overview

Unprotected welds situated in sea water accumulate fatigue damage faster than the same weld in fresh air. If the effect of sea water is specified in the job file, the correction for small cycles is ignored and the fatigue strength of the weld is reduced by a factor of two.

Specifying the effect of exposure to sea water

The user enables the treatment of exposure to sea water in the job file.

Job File Usage: **SEA_WATER = {0.0 | 1.0}**

0. Disable treatment of sea water exposure (default)
1. Enable treatment of sea water exposure

6.6.9 Failure mode

Overview

According to BS 7608, the damage parameter is taken as the principal stress acting on the critical plane. However, the implementation in Quick Fatigue Tool allows the user to choose between a pure normal, pure shear and combined normal-shear stress criterion, depending on how the crack is expected to propagate.

Specifying the failure mode

The user specifies the failure mode in the job file.

Job File Usage: **FAILURE_MODE = {'NORMAL' | 'SHEAR' | 'COMBINED'}**

- ''NORMAL'': Normal-only failure (default)
- ''SHEAR'': Shear-only failure
- ''COMBINED'': Combined normal-shear failure

The damage parameter quoted in the field output corresponds to either the normal, shear or combined (normal + shear) stress on the critical plane, according to the definition of the above option.

6.6.10 Specifying the $S_r - N$ curve as a BS 7608 weld class

Overview

The $S_r - N$ curve can be defined by one of the ten standard BS 7608 weld classes. Choice of weld class requires knowledge of the weld features and access to *Section 2. Classification of details* from document BS 7608:1993.

Specifying the weld class

The user specifies the weld class in the job file.

Job File Usage: **WELD_CLASS = {'B' | 'C' | 'D' | 'E' | 'F' | 'F2' | 'G' | 'W' | 'S' | 'T'}**

6.6.11 Specifying the $S_r - N$ curve as user data

Overview

The $S_r - N$ can be defined as tabulated data. The format of the user $S_r - N$ data file should meet the requirements explained in Section 5.8.3. The S_r values in the data file must be provided in terms of the stress range. S_r values are assumed to be provided at a load ratio of $R = -1$; S_r values defined over multiple load ratios are not accepted. The fatigue limit is taken as the last S_r value in the data.

By default, Quick Fatigue Tool processes the user $S_r - N$ in a column-wise fashion:

<i>N – Values</i>	<i>S_r – Values at R = -1</i>
N_1	S_1
N_2	S_2
.	.
N_i	S_i

Specifying a user-defined weld class

The user specifies the user-defined weld class in the job file.

Job File Usage: **WELD_CLASS** = {'*user-weld-curve-file-name.sn*', ['ROW' | 'COL']}

The optional 'ROW' and 'COL' parameters instruct Quick Fatigue Tool to read the user-defined weld curve in a row-wise or column-wise fashion, respectively.

6.6.12 Compatibility with other features

The BS 7608 method is not compatible with the following analysis features:

- mean stress correction;
- nodal elimination;
- plasticity correction,
- S–N data (**USE_SN**);
- S–N Scale factors (**SN_SCALE**);
- S–N knockdown curves (**SN_KNOCK_DOWN**); and
- fatigue notch factors.

The following output variables are not available:

- **FOS, SFA, FRFH, FRFV, FRFR, FRFW**

6.6.13 Required material properties

Material properties are not required to perform analyses with BS 7608—the user configures the weld attributes from the *BS 7608 WELD DEFINITION* region of the job file. The available options are summarised as

Option	Meaning	Importance
WELD_CLASS	S_r – N curve for analysis	REQUIRED
YIELD_STRENGTH	Used to set extrapolation limit	OPTIONAL
UTS	Used to define $\frac{S_r}{\sigma_u} - N$ curve	REQUIRED ⁶
DEVIATIONS_BELOW_MEAN	Standard deviations below mean S_r – N curve	REQUIRED
FAILURE_MODE	Failure criterion (normal or shear)	OPTIONAL
CHARACTERISTIC_LENGTH	Plate thickness or bolt diameter	OPTIONAL
SEA_WATER	Fatigue strength correction for sea water exposure	OPTIONAL

⁶ For Class X welds only.

6.7 NASALIFE

6.7.1 Overview

NASALIFE is a fatigue life prediction software developed by General Electric Aircraft Engines and the NASA Enabling Propulsion Materials program, to assess the durability of ceramic matrix composites (CMCs) subject to varying thermo-mechanical loads. The methodology is required by some regulatory bodies in the aviation sector for the validation of aero engine components [39].

6.7.2 Methodology

The NASALIFE method has been partially implemented in Quick Fatigue Tool as a stress-based, HCF fatigue analysis algorithm. The analysis procedure is as follows:

1. Explore all possible stress pair (cycle) combinations in the load history
2. For each cycle, calculate the effective mean stress and stress amplitude based on the effective stress parameter
3. Using the Walker mean stress correction, find the cycle pair with the largest damage. This is the most damaging major cycle (MDMC)
4. For the MDMC, find the principal directions and orientation of the octahedral shear plane
5. Align the stress tensor history with the octahedral plane of the MDMC and convert the stress tensor history into the octahedral shear stress history
6. Rainflow cycle count the shear stress history and record the position index of each cycle in the stress history
7. Convert the effective stress history into a matrix of cycles based on the indexes from step 6
8. Repeat steps 2 and 3 to calculate the damage of each cycle

6.7.3 Theoretical background

The stress tensor history is organized into all possible pairs using the combination formula given by Equation 6.7.1:

$$C_r = \frac{n!}{2!(n-2)!} \quad [6.7.1]$$

where C_r is the number of stress tensor pair combinations from a loading consisting of n history points.

Each cycle is calculated from an effective stress parameter. This parameter is explained in more detail in Section 6.7.4. For the case of the Manson McKnight parameter, the equivalent mean stress and stress amplitude are calculated by Equations 6.7.2-3, respectively.

$$\sigma_m = SIGN(I_1) \times \frac{\sqrt{2}}{2} \sqrt{(\sigma_{xm} - \sigma_{ym})^2 + (\sigma_{ym} - \sigma_{zm})^2 + (\sigma_{xm} - \sigma_{zm})^2 + 6(\sigma_{xym}^2 + \sigma_{yzm}^2 + \sigma_{xzm}^2)} \quad [6.7.2]$$

$$\sigma_a = \frac{\sqrt{2}}{2} \sqrt{(\sigma_{xa} - \sigma_{ya})^2 + (\sigma_{ya} - \sigma_{za})^2 + (\sigma_{xa} - \sigma_{za})^2 + 6(\sigma_{xya}^2 + \sigma_{yza}^2 + \sigma_{xza}^2)} \quad [6.7.3]$$

Where I_1 is the first stress invariant (hydrostatic stress).

Each stress cycle is corrected for the effect of mean stress using the Walker mean stress correction in Equation 6.7.4:

$$\sigma_{aw} = \sigma_a \left(\frac{2}{1 - R} \right)^{1-\gamma} \quad [6.7.4]$$

where σ_{aw} is the effective stress amplitude due to cycle σ_a , the load ratio R and the Walker parameter γ . The method of calculating γ is discussed in Section 7.7. The A -ratio is calculated as Equation 6.7.5:

$$A = \frac{\sigma_a}{\sigma_m} \quad [6.7.5]$$

If the A -ratio is less than 0 or greater than 10^7 , the Walker mean stress correction is modified to Equation 6.7.6:

$$\sigma_{aw} = \sigma_a [0.5(1 - R)]^{\gamma-1} \quad [6.7.6]$$

The mean stress correction is limited to positive mean stress only; negative mean stress will not increase the fatigue life of the component.

The MDMC is resolved onto octahedral planes, shown by Figure 6.7.1. The octahedral shear stress, τ_{oct} , is given by Equation 6.7.7.

$$\tau_{oct} = \frac{1}{3} \sqrt{2I_1^2 - 6I_2^2} \quad [6.7.7]$$

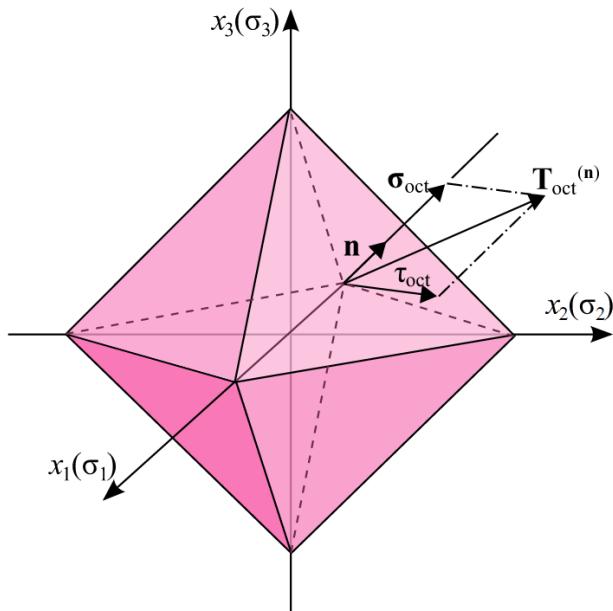


Figure 6.7.1: Octahedral planes showing the unit normal and shear directions

where I_2 is the second stress invariant, and is related to the von Mises stress, σ_e , by Equation 6.7.8.

$$\sigma_e = \sqrt{3I_2} \quad [6.7.8]$$

After identifying the octahedral shear plane, the original stress history is transformed to the principal directions of the MDMC using the rotation matrix and tensor transform given by Equations 6.7.9-10, respectively:

$$R = \begin{bmatrix} \sqrt{\frac{\sigma_1}{\sigma_{11}}} & 0 & 0 \\ 0 & \sqrt{\frac{\sigma_2}{\sigma_{22}}} & 0 \\ 0 & 0 & \sqrt{\frac{\sigma_3}{\sigma_{33}}} \end{bmatrix} \quad [6.7.9]$$

where $\sigma_{i,j,k}$ are the principal and $\sigma_{ii,jj,kk}$ are the normal stresses of the MDMC tensor.

$$\sigma' = R^T \sigma R \quad [6.7.10]$$

Where σ' and σ are the rotated and un-rotated stress tensors for each point in the load history, respectively.

The transformed load history is resolved into its octahedral shear component and then cycle counted. The indices of the octahedral shear cycles are then used with the original tensor history to locate stress cycles from the effective stress. The damage per cycle is calculated using Equation 6.7.11:

$$\sigma_{aw} = \sigma_f' (2N_f)^b \quad [6.7.11]$$

6.7.4 The effective stress parameter

Overview

The NASALIFE algorithm uses one of five effective stress parameters for the calculation of the cycle combinations. The user specifies the effective stress parameter in the environment file.

Environment File Usage: `nasalifeParameter = {1.0 | 2.0 | 3.0 | 4.0 | 5.0}`

1. Manson-McKnight (default)
2. Sines
3. Smith-Watson-Topper
4. R-ratio Sines
5. Effective method

Manson-McKnight

The effective mean stress and stress amplitude are based on the concept of a signed von Mises stress, given by Equations 6.7.12-13. The sign is taken from the hydrostatic stress of the current cycle.

$$\sigma_m = \text{SIGN}(\sigma_{xm} + \sigma_{ym} + \sigma_{zm}) \times \frac{\sqrt{2}}{2} \sqrt{(\sigma_{xm} - \sigma_{ym})^2 + (\sigma_{ym} - \sigma_{zm})^2 + (\sigma_{xm} - \sigma_{zm})^2 + 6(\sigma_{xym}^2 + \sigma_{yzm}^2 + \sigma_{xzm}^2)} \quad [6.7.12]$$

$$\sigma_a = \frac{\sqrt{2}}{2} \sqrt{(\sigma_{xa} - \sigma_{ya})^2 + (\sigma_{ya} - \sigma_{za})^2 + (\sigma_{xa} - \sigma_{za})^2 + 6(\sigma_{xya}^2 + \sigma_{yza}^2 + \sigma_{xza}^2)} \quad [6.7.13]$$

For situations where the loading is shear-dominated, the sign of the mean stress can be unreliable. Therefore, if the signs of the maximum and the minimum principal stresses differ, a modified version of the Manson-McKnight method is used instead, given by Equation 6.7.14:

$$\sigma_m = \text{SIGN}\left(\frac{\sigma_1 + \sigma_3}{\sigma_1 - \sigma_3}\right) \times \frac{\sqrt{2}}{2} \sqrt{(\sigma_{xm} - \sigma_{ym})^2 + (\sigma_{ym} - \sigma_{zm})^2 + (\sigma_{xm} - \sigma_{zm})^2 + 6(\sigma_{xym}^2 + \sigma_{yzm}^2 + \sigma_{xzm}^2)} \quad [6.7.14]$$

Sines

The Sines method determines the effective mean stress as the hydrostatic stress [40]. The effective stress amplitude is then modified by the mean stress. These are given by Equations 6.7.15-16:

$$\sigma_m = \sigma_{xm} + \sigma_{ym} + \sigma_{zm} \quad [6.7.15]$$

$$\sigma_a = \frac{\sqrt{2}}{2} \sqrt{(\sigma_{xa} - \sigma_{ya})^2 + (\sigma_{ya} - \sigma_{za})^2 + (\sigma_{xa} - \sigma_{za})^2 + 6(\sigma_{xya}^2 + \sigma_{yza}^2 + \sigma_{xza}^2) - C\sigma_m} \quad [6.7.16]$$

The constant C is intended to have a value of 0.5 for uniaxial loads and 1.0 for multiaxial loads. However, the loading is assumed to be multiaxial and a value of 1.0 is always used.

The A -ratio is assumed to be infinite, hence the Walker mean stress correction takes the modified form of Equation 6.7.6.

Smith-Watson-Topper

The Smith-Watson-Topper method assumes that the effective mean stress is zero, therefore for loadings with a load ratio of $R \neq -1$, use of this parameter will produce highly non-conservative results. The effective stress amplitude is a function of the maximum and the minimum value of the first principal stress of the current cycle [41]. These are given by Equations 6.7.17-18:

$$\sigma_m = 0 \quad [6.7.17]$$

$$\sigma_a = \frac{1}{2} \sqrt{\sigma_{1,max}(\sigma_{1,max} - \sigma_{1,min})} \quad [6.7.18]$$

R-ratio Sines

The R-Ratio Sines Method uses the hydrostatic stress as the effective mean stress, but keeps the original definition of the effective stress amplitude. These are given by Equations 6.7.19-20:

$$\sigma_m = \sigma_{xm} + \sigma_{ym} + \sigma_{zm} \quad [6.7.19]$$

$$\sigma_a = \frac{\sqrt{2}}{2} \sqrt{(\sigma_{xa} - \sigma_{ya})^2 + (\sigma_{ya} - \sigma_{za})^2 + (\sigma_{xa} - \sigma_{za})^2 + 6(\sigma_{xya}^2 + \sigma_{yza}^2 + \sigma_{xza}^2)} \quad [6.7.20]$$

Effective Method

The Effective Method defines the effective mean stress as twice the distortion energy minus the effective stress amplitude. The effective stress amplitude takes the same form as that from the Manson McKnight method. These are given by Equations 6.7.21-22:

$$\sigma_m = \sqrt{2} \sqrt{(\sigma_{xm} - \sigma_{ym})^2 + (\sigma_{ym} - \sigma_{zm})^2 + (\sigma_{xm} - \sigma_{zm})^2 + 6(\sigma_{xym}^2 + \sigma_{yzm}^2 + \sigma_{xzm}^2)} - \sigma_a \quad [6.7.21]$$

$$\sigma_a = \frac{\sqrt{2}}{2} \sqrt{(\sigma_{xa} - \sigma_{ya})^2 + (\sigma_{ya} - \sigma_{za})^2 + (\sigma_{xa} - \sigma_{za})^2 + 6(\sigma_{xya}^2 + \sigma_{yza}^2 + \sigma_{xza}^2)} \quad [6.7.22]$$

6.7.5 Required material properties

NASALIFE analyses require the following material parameters:

Property	Symbol	Importance
Tensile fatigue strength coefficient	σ_f'	REQUIRED
Tensile fatigue strength exponent	b	REQUIRED
Ultimate tensile strength	S_u	OPTIONAL
Walker gamma parameter	γ	REQUIRED ⁷

⁷ This parameter can be user-defined or computed automatically. Consult Section 7.7.2 for detailed information about the Walker gamma parameter.

6.7.6 Guidance for load history gating

The NASALIFE algorithm locates the MDMC by considering every stress tensor combination in the load history. If tensor gating is enabled then the original, un-gated load history is used, and the resulting analysis time can become very large. If the load history contains many data points, it may be expedient to pre-gate the load histories.

The user specifies gating criteria in the environment file.

Environment File Usage: `gateTensors` = 0.0
 `gateHistories` = 1.0

Care should be taken when pre-gating multiple load histories, as the phase relationship between the loading channels may be lost and the accuracy of the fatigue result may be adversely affected.

6.8 Uniaxial Stress-Life

6.8.1 Overview

Uniaxial Stress-Life is the most basic fatigue analysis technique. The method is ideal for simple loading conditions where fatigue damage is caused primarily by stresses in a single direction. The algorithm is especially useful for measured stress data from plane stress specimens. Since the algorithm does not require critical plane searching, it is computationally much less expensive than the biaxial methods such as the Stress-based Brown-Miller fatigue algorithm.

The Uniaxial Stress-Life algorithm is defined by Equation 6.8.1:

$$\frac{\Delta\sigma}{2} = \sigma_f' (2N_f)^b \quad [6.8.1]$$

where $\frac{\Delta\sigma}{2}$ is the uniaxial stress amplitude.

6.8.2 Defining a uniaxial stress-life analysis

The Uniaxial Stress-Life algorithm only requires a single stress history—stress datasets are not recognised by the program.

Job File Usage: **DATASET** = []
 HISTORY = '*history-file-name.**'

6.8.3 Required material properties

The following material properties are required to perform an analysis with the Uniaxial Stress-Life algorithm:

Property	Symbol	Importance
Fatigue strength coefficient	σ_f'	REQUIRED
Fatigue strength exponent	b	REQUIRED

6.9 Uniaxial Strain-Life

6.9.1 Overview

Uniaxial Strain-Life is a method for analysing simple unidirectional strain histories. The uniaxial elastic stress history is converted into an inelastic strain history using the Ramberg-Osgood nonlinear elastic strain-hardening model. The same model is used to convert the elastic principal stress histories into inelastic principal strain histories.

The Uniaxial Strain-Life algorithm is defined by Equation 6.9.1:

$$\frac{\Delta\varepsilon}{2} = \frac{\sigma_f'}{E} (2N_f)^b + \varepsilon_f' (2N_f)^c \quad [6.9.1]$$

where $\frac{\Delta\varepsilon}{2}$ is the uniaxial strain amplitude.

6.9.2 Defining a uniaxial strain-life analysis

The Uniaxial Strain-Life algorithm only requires a single stress history—stress datasets are not recognised by the program.

Job File Usage: **DATASET** = []
 HISTORY = '*history-file-name.**'

6.9.2 Required material properties

The following material properties are required to perform an analysis with the Uniaxial Strain-Life algorithm:

Property	Symbol	Importance
Fatigue strength coefficient	σ_f'	REQUIRED
Fatigue strength exponent	b	REQUIRED
Fatigue ductility coefficient	ε_f'	REQUIRED
Fatigue ductility exponent	c	REQUIRED
Young's Modulus	E	REQUIRED
Cyclic strain-hardening coefficient	K'	REQUIRED
Cyclic strain-hardening exponent	n'	REQUIRED

6.10 User-defined algorithms

6.10.1 Overview

The Quick Fatigue Tool framework allows the user to create their own fatigue analysis algorithm. Information is passed into the class *algorithm_user*, which is used to evaluate the fatigue damage at each analysis item.

The user algorithm class file *algorithm_user.m* is located in *Application_Files\code\main*. The file contains a single function called *main*. This is the function which Quick Fatigue Tool calls iteratively for each analysis item.

The class *algorithm_user* can be expanded so that the function *main* calls other functions within *algorithm_user*.

6.10.2 Variables passed in for information

Quick Fatigue Tool passes the following arguments into *algorithm_user.main*:

Argument	Description	Notes
S11	Stress tensor history in the normal Cartesian 1-direction.	
S22	Stress tensor history in the normal Cartesian 2-direction.	
S33	Stress tensor history in the normal Cartesian 3-direction.	
S12	Stress tensor history in the shear Cartesian 12-direction.	
S23	Stress tensor history in the shear Cartesian 23-direction.	
S13	Stress tensor history in the shear Cartesian 13-direction.	
N	Current analysis item number.	N = 1 unless the stress dataset contains more than one analysis item.
MSC	Identifier defining the selected mean stress correction.	Consult Section 7.1 for a table relating the value of MSC to the mean stress correction.

6.10.3 Variables to be defined

The user must define the following output arguments:

Argument	Description	Format	Example usage (per analysis item)
DPARAMI	Maximum damage parameter at the current analysis item.	$1 \times L$ numeric array.	$\text{DPARAMI}(N) = X;$ X is the maximum damage parameter.
AMPI	Stress amplitude of each cycle for the loading at the current analysis item.	$1 \times L$ cell array.	$\text{AMPI}\{N\} = [A_1, A_2, \dots, A_N];$ A_1 to A_N are the amplitudes over the load history.
PAIRI	Cycle pairs for the loading at the current analysis item.	$C \times 2$ cell array.	$\text{PAIRI}\{N\} = [P_{1,min}, P_{1,max}; \dots; P_{C,min}, P_{C,max}];$ P_{min} and P_{max} are the minimum and maximum cycle values for each pair in the loading.
DAMI	Total damage for the loading at the current analysis item.	$1 \times L$ numeric array.	$\text{DAMI}(N) = D;$ D is the total damage over the load history.

L is the number of analysis items; C is the number of cycle in the load history; N is the current analysis item.

6.10.4 Material properties

Material properties are accessed using the `getappdata()` method:

```
property = getappdata(0, 'identifier');
```

A complete list of material properties and their identifiers is given below:

Property	Identifier	Property	Identifier
Young's modulus	E	Default analysis algorithm	defaultAlgorithm
Poisson's ratio	poisson	Default mean stress correction	defaultMSC
Ultimate tensile strength	uts	Constant amplitude endurance limit	cael
Proof stress	twops	S-N data points: S-values (interpolated at R=-1)	s_values_reduced
S-N data points: S-values	s_values	S-N data points: Number of curves	nSNDataSets
S-N data points: N-values	n_values	Fatigue strength exponent at knee-point	b2
S-N data points: R-values	r_values	Life at knee-point	b2Nf
Fatigue strength coefficient	Sf	Fatigue limit	fatigueLimit
Fatigue strength exponent	b	Residual stress	residualStress
Fatigue ductility coefficient	Ef	Surface finish	kt
Fatigue Ductility exponent	c	Notch sensitivity constant	notchRootRadius

Cyclic strain hardening coefficient	<code>kp</code>	Notch root radius	<code>notchSensitivityConsta nt</code>
Cyclic strain hardening exponent	<code>np</code>		
Normal stress sensitivity constant	<code>k</code>		

If analysis groups are used, material properties will depend on which group the current analysis item belongs to.

6.10.5 Utility functions

User-defined algorithms can call to other Quick Fatigue Tool functions to facilitate the analysis.

Rainflow cycle counting

A stress history can be cycle counted using the following function.

M-File Usage: `rfData = analysis.rainflow(history);`

The variable `rfData` is a $C \times 4$ matrix where C is the number of counted cycles. The first two columns are the cycle points. The last two columns are the indexes in the stress history corresponding to the cycle.

The amplitude of each cycle is extracted using the following function:

M-File Usage: `pairs = rfData(:, 1.0:2.0);
amplitudes = analysis.getAmps(pairs);`

Mean stress correction

The variables `pairs` and `amplitudes`, as defined above, can be corrected for the effect of mean stress.

M-File Usage: `[mscAmplitudes, ~, ~] = analysis.msc(amplitudes, pairs, MSC);`

The variable `MSC` is the identifier which is passed into `algorithm_user.main`.

6.10.6 Additional stress histories

Before the analysis, Quick Fatigue Tool calculates the principal stress and von Mises stress history. These can be accessed at the current analysis item using `getappdata()`:

History variable	Definition
First principal stress	<code>getappdata(0, 'S1');</code>
Second principal stress	<code>getappdata(0, 'S2');</code>
Third principal stress	<code>getappdata(0, 'S3');</code>
Von Mises stress	<code>getappdata(0, 'VM');</code>

7. Mean stress corrections

7.1 Background

7.1.1 Overview

Mean stress occurs when the maximum and minimum stress in a cycle are not equal and opposite. The mean stress is given by Equation 7.1.

$$\sigma_m = \frac{\sigma_{max} - \sigma_{min}}{2} \neq 0 \quad [7.1]$$

The mean stress is also expressed in terms of the load ratio R in Equation 7.2.

$$R = \frac{\sigma_{min}}{\sigma_{max}} \quad [7.2]$$

Fatigue material data usually assumes a mean stress of zero, ($R = -1$). In practice, the mean stress in the fatigue loading is rarely zero everywhere in the history. Therefore, methods are required which correct for the presence of mean stress by modifying the stress amplitude to account for the change in fatigue life when non-zero mean stresses are present.

There are three methods which can account for the mean stress:

- specify S–N data at multiple R -values;
- use σ_m – σ_a envelopes to compute the effective stress amplitude; and
- correct the fatigue life equation analytically.

7.1.2 Specifying the mean stress correction

The available mean stress correction depends on the specified fatigue analysis algorithm.

Uniaxial Strain-Life

The Uniaxial Strain-Life algorithm is compatible with the following mean stress corrections:

- Morrow (default); Smith-Watson-Topper; Walker.

Job File Usage: `MS_CORRECTION = {'MORROW' | 'SWT' | 'WALKER'}`

Uniaxial Stress-Life

The Uniaxial Stress-Life algorithm is compatible with the following mean stress corrections:

- Goodman (default); Gerber; Walker; R-ratio S–N curves; User-defined envelopes.

Job File Usage: `MS_CORRECTION = {'GOODMAN' | 'GERBER' | 'WALKER' | 'SODERBERG' | 'RATIO' | 'filename.msc'}`

Stress Invariant Parameter

The Stress Invariant Parameter algorithm is compatible with the following mean stress corrections:

- Goodman (default); Gerber; Walker; Soderberg; R-ratio S–N curves; User-defined envelopes.

Job File Usage: `MS_CORRECTION = {'GOODMAN' | 'GERBER' | 'WALKER' | 'RATIO' | 'filename.msc'}`

Stress-based Brown-Miller

The Stress-based Brown-Miller algorithm is compatible with the following mean stress corrections:

- Morrow (default); Goodman; Gerber; Soderberg; R-ratio S–N curves; User-defined envelopes.

Job File Usage: `MS_CORRECTION = {'MORROW' | 'GOODMAN' | 'GERBER' | 'SODERBERG' | 'RATIO' | 'filename.msc'}`

Normal Stress

The Normal Stress algorithm is compatible with the following mean stress corrections:

- Morrow (default); Smith-Watson-Topper; Goodman; Gerber; Walker; R-ratio S–N curves; User-defined envelopes.

Job File Usage: `MS_CORRECTION = {'MORROW' | 'SWT' | 'GOODMAN' | 'GERBER' | 'WALKER' | 'RATIO' | 'filename.msc'}`

Findley's Method

The Findley algorithm is not compatible with mean stress corrections.

Job File Usage: The **MS_CORRECTION** option is not supported for Findley's Method.

BS 7608

The BS 7608 algorithm is not compatible with mean stress corrections.

Job File Usage: The **MS_CORRECTION** option is not supported for BS 7608.

NASALIFE

The NASALIFE algorithm uses the Walker mean stress correction.

Job File Usage: The **MS_CORRECTION** option is not supported for NASALIFE.

Default

When the user specifies the default mean stress correction, Quick Fatigue Tool will use the mean stress correction specified in the material definition.

Job File Usage: `MS_CORRECTION = 'DEFAULT'`

7.1.3 Comparison of methods

Material-based corrections

If R-ratio S-N data is supplied, Quick Fatigue Tool corrects the loading for the effect of mean stress by computing the equivalent S-N curve for the current load ratio, R .

Analytical corrections

For analytical mean stress corrections, the stress-life equation is modified by inserting σ_m as an independent variable in the damage calculation.

The following mean stress corrections in Quick Fatigue Tool are analytical:

- Morrow; Smith-Watson-Topper; Walker.

Envelope-based corrections

Envelopes are used to calculate the effective stress amplitude, σ_{eff} , due to the mean stress. The cycle is represented on a Haigh diagram such as that in Figure 7.1. The effective stress is calculated as the equivalent stress amplitude as if the mean stress were zero.

The following mean stress corrections in Quick Fatigue Tool are based on envelopes:

- Goodman; Soderberg; Gerber; User-defined envelopes.

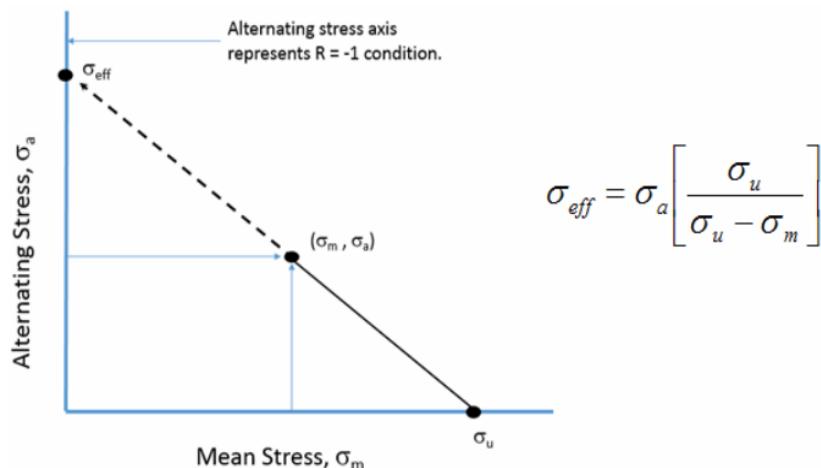


Figure 7.1: Haigh diagram showing the Goodman envelope.

Image courtesy of CAE Associates.

7.2 Goodman

7.2.1 Overview

The Goodman mean stress correction assumes that, for mirror polished specimens, the relationship between the allowable stress amplitude and tensile mean stress is linear [42]:

$$\frac{\sigma_a}{\sigma_0} + \frac{\sigma_m}{\sigma_u} = 1 \quad [7.3]$$

Equation 7.3 can be plotted on a Haigh diagram to give Figure 7.2.

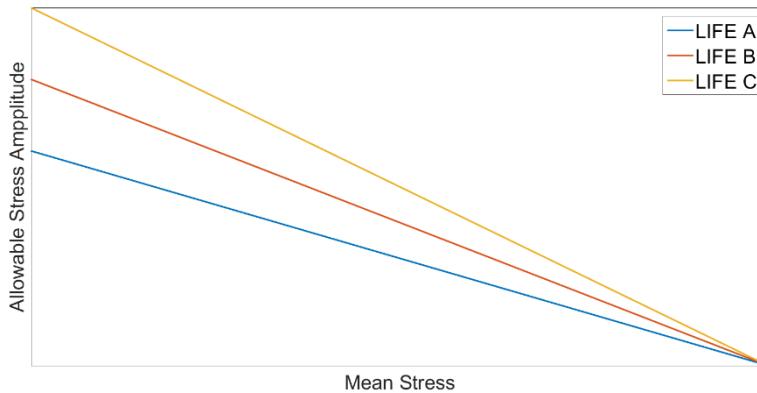


Figure 7.2: Haigh diagram representation of Goodman relationship for different reference lives. The curves meet the material's ultimate tensile strength.

For each cycle in the loading, Quick Fatigue Tool evaluates the mean stress and finds the equivalent stress amplitude as if the cycle had zero mean stress. This is achieved by re-arranging Equation 7.3 into Equation 7.4:

$$\sigma_0 = \sigma_a \left(1 - \frac{\sigma_m}{\sigma_u}\right)^{-1} \quad [7.4]$$

Quick Fatigue Tool assumes that the equivalent stress amplitude has zero mean stress. Therefore, if custom S-N data is being used which was measured at an R-ratio other than -1.0 then the Goodman correction will not produce a reliable solution.

7.2.2 Modified Goodman envelopes

Overview

The standard implementation of the Goodman mean stress correction is non-conservative for large values of mean stress and assumes no change in the allowable stress amplitude for negative mean stress. A modified version of the Goodman correction combines the standard slope with the Buch line, shown in Figure 7.3.

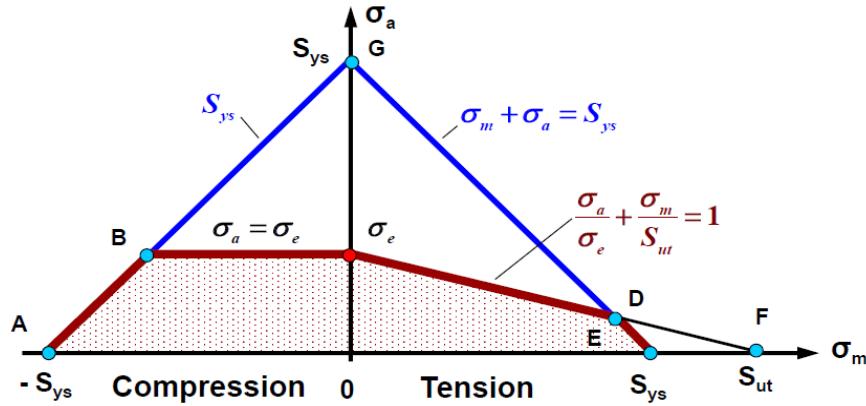


Figure 7.3: Modified Goodman line (red), defined by the intersection of the standard line (black) with the Buch line (blue); copyright © Professor Grzegorz Glinka [63]; reproduced with permission.

Specifying modified Goodman envelopes

The user specifies modified Goodman envelopes in the environment file.

Environment File Usage: `modifiedGoodman = {0.0 | 1.0}`

0. Do not use modified Goodman envelopes
1. Use modified Goodman envelopes (default—if available)

The modified Goodman envelope requires a value of the proof stress. If the proof stress is undefined, the standard Goodman envelope is used instead.

7.2.3 Limit stress for Goodman envelopes

The limit stress is the horizontal intercept of the $\sigma_m - \sigma_a$ axes. By default, the intercept between the Goodman envelope and the horizontal (mean stress) axis is the material ultimate tensile strength.

The user specifies the limit stress in the environment file.

Environment File Usage: `goodmanMeanStressLimit = {n | 'UTS' | 'PROOF' | 'S-N'}`

- n : User-defined
- '`UTS`': Computed from the material ultimate tensile strength, σ_u (default)
- '`PROOF`': Computed from the material proof stress, σ_y
- '`S-N`': Computed from the vertical intercept of the S-N axes (stress amplitude at $N_f = 1$)

When the modified Goodman envelope is enabled, the Goodman limit stress is taken as the yield strength and cannot be modified by the user.

7.2.4 Guidance

Care must be taken when using the Goodman mean stress correction. Since the stress amplitude is zero along the horizontal axis, there is no fatigue in this regime. The assumption that the limiting fatigue strength is equal to the static tensile strength is therefore incorrect, since static and fatigue failure are driven by physically distinct mechanisms. The Goodman correction can often over predict fatigue lives by a factor of three to four. In cases where the Goodman correction fails to produce acceptable results, the Walker correction is recommended as an alternative.

7.3 Soderberg

The Soderberg mean stress correction is similar to Goodman, but uses the proof stress as the maximum allowable mean stress [43]:

$$\frac{\sigma_a}{\sigma_0} + \frac{\sigma_m}{\sigma_y} = 1 \quad [7.5]$$

Quick Fatigue Tool assumes that the equivalent stress amplitude has zero mean stress. Therefore, if custom S–N data is being used which was measured at an R-ratio other than -1.0, then the Soderberg correction will not produce a reliable solution.

7.4 Gerber

The Gerber mean stress correction is a non-linear version of Goodman:

$$\frac{\sigma_a}{\sigma_0} + \left(\frac{\sigma_m}{\sigma_u} \right)^2 = 1 \quad [7.6]$$

The Haigh diagram representation is shown in Figure 7.4:

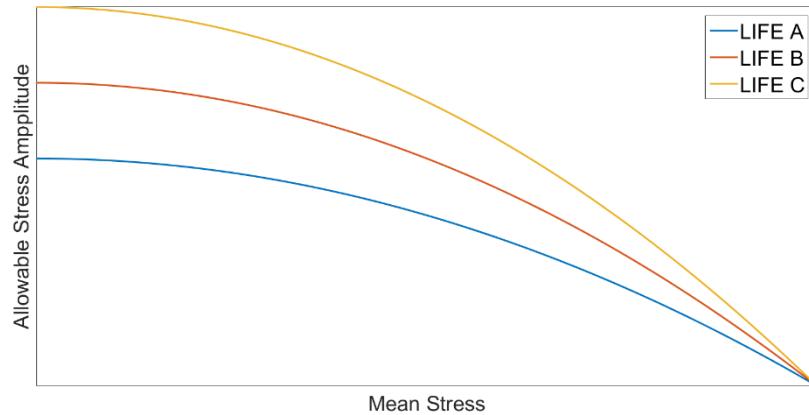


Figure 7.4: Haigh diagram representation of Gerber relationship for different reference lives. The curves meet the material's ultimate tensile strength.

The Gerber correction is more conservative than Goodman; however, for compressive mean stresses, the Gerber correction increases the fatigue life, whereas the Goodman correction has no effect in compression.

7.5 Morrow

7.5.1 Strain-life

The Morrow mean stress correction modifies the elastic component of the strain-life curve [44]. This reflects the observation that the mean stress has the most noticeable effect in the HCF regime. The correction is expressed in its original form as Equation 7.7:

$$\frac{\Delta\varepsilon}{2} = \frac{(\sigma_f' - \sigma_m)}{E} (2N_f)^b + \varepsilon_f' (2N_f)^c \quad [7.7]$$

Although the Morrow mean stress correction gives reasonable results, it does not reflect the material's true behaviour. The correction postulates that the ratio between the elastic and plastic strain components varies with mean stress. In fact, the hysteresis behaviour of metals is a function of strain only.

7.5.2 Stress-life

Stress-life algorithms which use the Morrow mean stress correction only have their fatigue strength term modified:

$$\frac{\Delta\sigma}{2} = \frac{(\sigma_f' - \sigma_m)}{E} (2N_f)^b \quad [7.8]$$

7.6 Smith-Watson-Topper

7.6.1 Strain-life

The Smith-Watson-Topper (SWT) relationship [41] is obtained by multiplying the damage parameter by the maximum stress in the load cycle, and the right-hand side by the tensile fatigue strength coefficient:

$$\frac{\Delta\varepsilon}{2}\sigma_{max} = \frac{\sigma_f'^2}{E}(2N_f)^b + \sigma_f'\varepsilon_f'(2N_f)^{b+c} \quad [7.9]$$

The SWT correction gives acceptable results for a wide range of materials and load types, and is less conservative than the Morrow correction for compressive mean stress.

7.6.2 Stress-life

The stress-life implementation of the SWT mean stress correction is applied to the stress cycle, rather than the stress-life equation. This is achieved using the Walker mean stress correction with $\gamma = 0.5$.

7.7 Walker

7.7.1 Overview

The Walker mean stress correction is similar to the Smith-Watson-Topper correction, with the addition of the material parameter γ . The Walker equation corrects the stress amplitude term in the stress-life equation:

$$\frac{\Delta\sigma_a}{2} = \frac{\Delta\sigma}{2} \left(\frac{2}{1-R} \right)^{1-\gamma} \quad [7.10]$$

where $\frac{\Delta\sigma_a}{2}$ is the equivalent stress amplitude at zero mean stress, $\frac{\Delta\sigma}{2}$ is the stress amplitude of the cycle and R is the load ratio of the cycle.

7.7.2 Walker γ -parameter

Overview

The user specifies the calculation technique of the Walker γ -parameter in the environment file.

Environment File Usage: `walkerGammaSource = {1.0 | 2.0 | 3.0}`

1. Walker regression fit (default)
2. Standard values for steel and aluminium
3. User-defined

When either the Walker regression fit or standard values are specified, the value of γ is selected based on the material behavior. Guidance on defining material behaviour can be found in Section 5.10.

γ -parameter from Walker regression fit

The value of γ is approximated using the regression fit in Figure 7.5.

The value of γ correlates well with steels [45]. However, there exists no such correlation between the Walker parameter and aluminium, as shown by Figure 7.6. It is therefore recommended that the Walker regression fit is not used with non-steels.

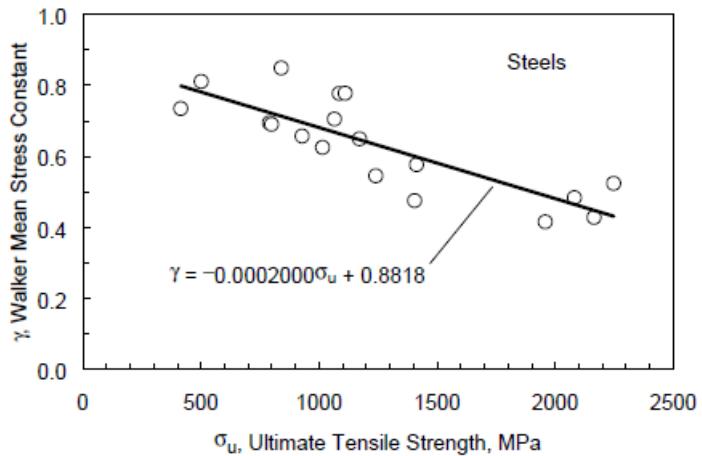


Figure 7.5: Approximation of γ based on the ultimate tensile strength of steel

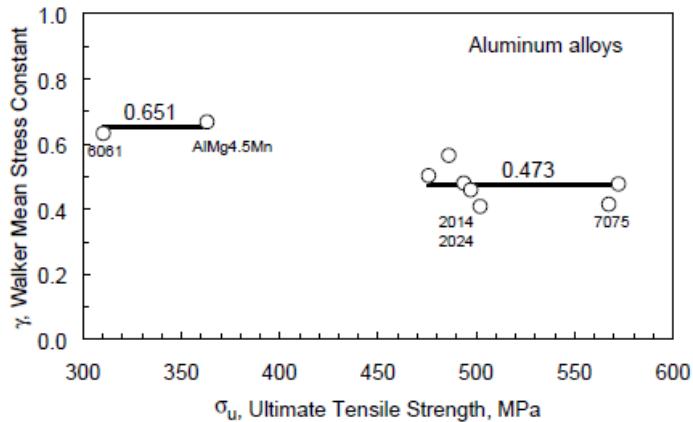


Figure 7.6: Approximation of γ based on the ultimate tensile strength of aluminium

γ -parameter from standard values

The value of γ is found from standard values. Work by Dowling [46] has shown that the value of γ can be approximated for most steel and aluminium specimens. If the material is neither steel nor aluminium, γ is calculated as a function of the load ratio, R .

The value of γ is calculated according to the following table:

Material behaviour	γ -Solution
Steel	0.65
Aluminium	0.45
Other	$1, R < 0$ $0.5, R \geq 0$

Specifying the material behaviour

The user specifies the material behaviour in the material data.

Material Manager Usage: Derivation region of the *Material Editor* dialogue: Select the behaviour from the *Material behaviour* drop-down menu.

Text File Usage: *BEHAVIOR, {STEEL, ALUMINIUM, OTHER}

γ -parameter from user input

The user specifies the value of γ in the environment file.

Environment File Usage: userWalkerGamma = γ

7.7.3 Strain-life

The strain-life implementation of the Walker mean stress correction has been described by Dowling [47] and Ince [48], and is given in the form of Equation 7.11:

$$\frac{\Delta\varepsilon}{2} = \frac{\sigma_f'}{E} \left[2N_f \left(\frac{1-R}{2} \right)^{(1-\gamma)/b} \right]^b + \varepsilon_f' \left[2N_f \left(\frac{1-R}{2} \right)^{(1-\gamma)/b} \right]^c \quad [7.11]$$

7.8 R-ratio S–N curves

7.8.1 Overview

The loading can be corrected for the effect of mean stress using experimental stress-life data, where the endurance curve for the material has been measured at more than one load ratio. An example of a multi R-ratio S–N curve from the Material Manager is shown in Figure 7.7. Creating S–N data with multiple load ratios is explained in Section 5.8.3.

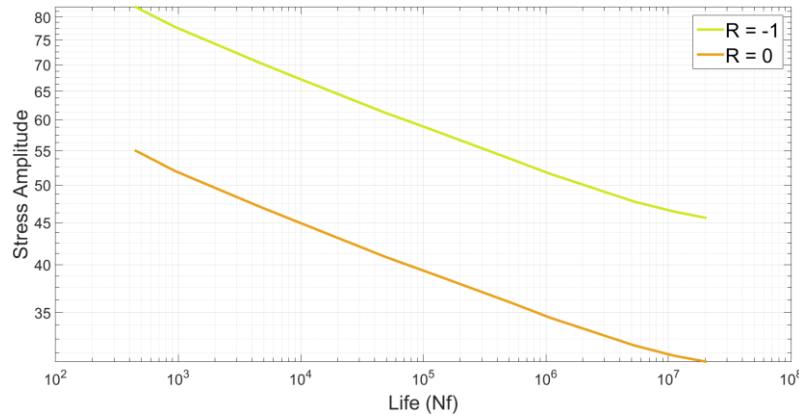


Figure 7.7: Example S–N data from the Material Manager, for a load ratio of -1 and 0

7.8.2 Calculation of the S–N curve

If R-ratio S–N curves are selected, Quick Fatigue Tool interpolates between the two curves that envelope the load ratio of the current cycle. The standard interpolation formula is used:

$$S_{R_i} = S_2 - \frac{S_2 - S_1}{R_{upper} - R_{lower}} \cdot (R_i - R_{lower}) \quad [7.12]$$

where S_{R_i} is the stress-life curve at the calculated load ratio R_i . R_{upper} and R_{lower} are the upper and lower load ratios corresponding to the curves S_2 and S_1 , respectively, which envelope the required curve.

If the required S–N curve lies outside the range of S–N data (i.e. the required curve is not bound by a curve to either side), then the stress-life data is extrapolated. In such cases, accuracy is not guaranteed, and a warning will be written to the message file.

R-ratio S–N curves may only be used for $R < 1$. Fully-compressive cycles ($R > 1$) will use the S–N curve at $R = -1$. S–N curves may not be defined for infinite-valued R-ratios.

7.8.3 Treatment of the fatigue limit

When the R-ratio S–N Curves mean stress correction is used, the fatigue limit depends on the load ratio of the current cycle. Therefore, the fatigue limit is re-calculated for each cycle based on the interpolated S–N curve; the values of the environment variables `fatigueLimitSource` and `userFatigueLimit` are ignored.

Under certain conditions, Quick Fatigue Tool reduces the fatigue limit such that previously non-damaging cycles may become damaging. Modification of the fatigue limit is enabled with the environment variable `modifyEnduranceLimit`, and is discussed in detail in the document *Quick Fatigue Tool Appendices: A1.5*. If modification of the fatigue limit is enabled then the instantaneous fatigue limit, σ_{f_i} , is based on the interpolated S–N curve for the current cycle. Subsequent cycles are compared either to the reduced fatigue limit, σ_{f_r} , or σ_{f_i} (whichever is smaller). If the subsequent cycles are non-damaging, σ_{f_r} will recover towards σ_{f_i} . Hence, the rate of recovery may change depending on the load ratio of the current cycle.

It is not clear how best to treat this “bouncing” endurance limit in combination with multiple R-ratio S–N curves; the above outlined methodology is a suggestion only. In case the user has doubt as to the applicability of the modified endurance limit, it can be enabled or disabled in the environment file.

Environment File Usage: `modifyEnduranceLimit = {0.0 | 1.0}`

0. Damaging cycles do not affect the endurance limit
1. Damaging cycles can reduce the endurance limit (default)

7.9 User-defined mean stress corrections

7.9.1 Overview

User-defined mean stress corrections can be provided for an analysis. The data describes the allowable stress amplitude at the material's endurance limit as a function of the mean stress. The result is a Haigh envelope which Quick Fatigue Tool uses to calculate a mean stress correction factor.

7.9.2 Specifying user-defined mean stress corrections

The user specifies the user-defined mean stress correction as a mean stress correction (.msc) file in the job file.

Job File Usage: `MS_CORRECTION = 'msc-file-name.msc'`

Material Manager Usage: User-defined mean stress corrections are not supported in Material Manager.

7.9.3 Technical background

The mean stress correction factor is calculated by comparing the allowable stress amplitude at zero mean stress to the allowable stress amplitude corresponding to the mean stress of the cycle. Consider the following user-defined mean stress data:

Normalised mean stress, σ_m	Normalised allowable stress amplitude, σ_a
0.2	0.8
0.3	0.7

Now consider a stress cycle which has a mean stress of $88MPa$ and stress amplitude of $100MPa$. If the material's UTS is $400MPa$, then the normalised mean stress is $88/400 = 0.22$. Quick Fatigue Tool determines the corresponding allowable stress amplitude by assuming that the envelope follows a linear relationship and hence fits the equation of a straight line ($y = mx + c$). The normalised stress amplitude corresponding to the mean stress of the cycle is then $-(0.22 - 0.3) + 0.7 = 0.78$. The mean stress correction factor is therefore $1/0.78 \cong 1.282$. The equivalent stress amplitude at zero mean stress is $100 \times 1.282 = 128.2MPa$. In general, the mean stress correction factor is given by Equation 7.13:

$$MSC = \left[\frac{d\sigma_a}{d\sigma_m} (\sigma_m - \sigma_{mi}) + \sigma_{ai} \right]^{-1} \quad [7.13]$$

where $\frac{d\sigma_a}{d\sigma_m}$ is the local gradient of the Haigh data, σ_m is the cycle mean stress and σ_{mi} and σ_{ai} are the allowable mean stress and stress amplitude at point i , respectively. The equivalent stress amplitude at zero mean stress is given by Equation 7.14:

$$\sigma_{a,e} = \sigma_a \times MSC \quad [7.14]$$

7.9.4 Creating user-defined mean stress correction files

User-defined mean stress correction data is usually obtained from existing Haigh diagrams for the given material. The mean stress and stress amplitude are typically normalised by the ultimate tensile strength and the tensile fatigue limit, respectively. For cycles with negative mean stress, Quick Fatigue Tool will normalise those cycles by the compressive strength. If the compressive strength is not defined, the tensile strength will be used for all cycles.

The following file format must be obeyed for user-defined mean stress data:

First column: Normalised mean stress values

Second column: Normalised stress amplitude values

The values of the normalised mean stress must be decreasing.

If the mean stress of the cycle is less than the minimum, or greater than the maximum value of mean stress defined in the .msc file, Quick Fatigue Tool will calculate the mean stress correction factor directly from the first or last stress amplitude value in the data, respectively.

7.9.5 Examples

Quick Fatigue Tool includes .msc files for Gray Iron and Inconel 718 Steel. These are shown in Figure 7.8. The files are located in *Data\msc*.

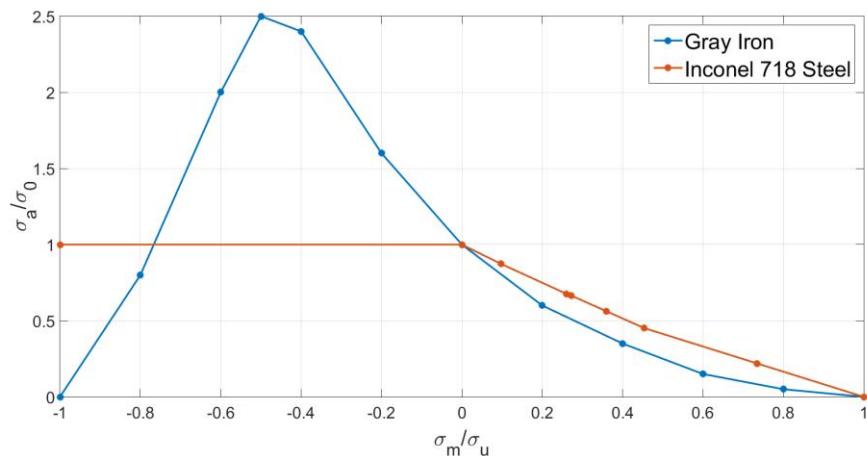


Figure 7.8: Allowable stress amplitude as a function of mean stress for Gray Iron and Inconel 718 Steel

According to the Haigh data, Inconel 718 has an allowable stress amplitude in compression which remains constant, hence compressive cycles are not corrected for the mean stress.

Gray Iron has increased strength in compression up to 50% of its compressive strength, hence compressive cycles within this region are corrected in a sense that reduces the resulting fatigue damage. Beyond 50% of the compressive strength, the fatigue strength reduces to zero, at which point crushing is expected.

In tension, both materials have decreasing fatigue strength until the mean stress equals the ultimate tensile strength, at which point static failure is expected.

8. Safety factor analysis

8.1 Background

Some designs require that the fatigue performance of a component is expressed as a safety factor, rather than a number of cycles. The finite life calculation is substituted for a pass/fail criterion, such that a value greater than unity indicates that safety has been achieved by a certain margin; values less than unity indicate that design and/or load modification is required. Quick Fatigue Tool offers two design factor calculations:

1. Fatigue Reserve Factor (FRF)
2. Factor of Strength (FOS)

The two parameters are similar in their objectives. However, their methodologies have some fundamental differences which should be considered before the analysis.

8.2 Fatigue Reserve Factor

8.2.1 Overview

The FRF is a linear scale factor which considers the worst stress cycle in the loading and compares the mean stress and stress amplitude to a given target life. There are three types of calculation:

1. Horizontal
2. Vertical
3. Radial

Figure 8.1 shows the Goodman and Gerber life envelopes which can be used for the FRF calculation. The FRF is calculated for the worst cycle in the loading.

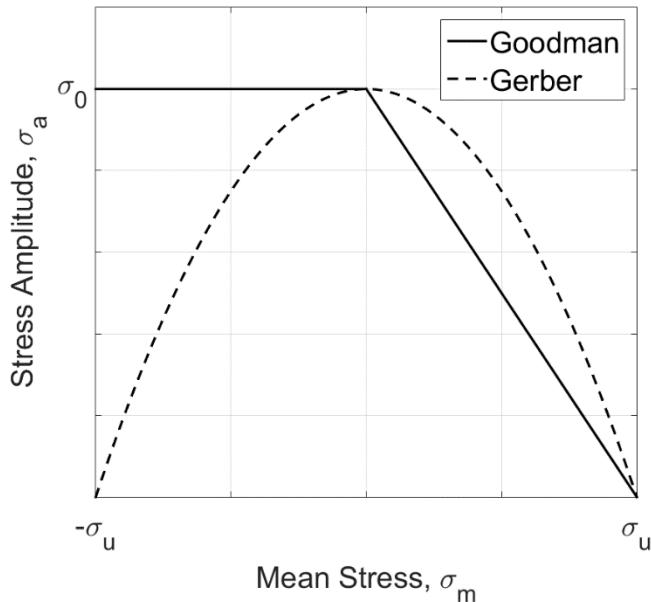


Figure 8.1: Goodman and Gerber envelopes for FRF calculation

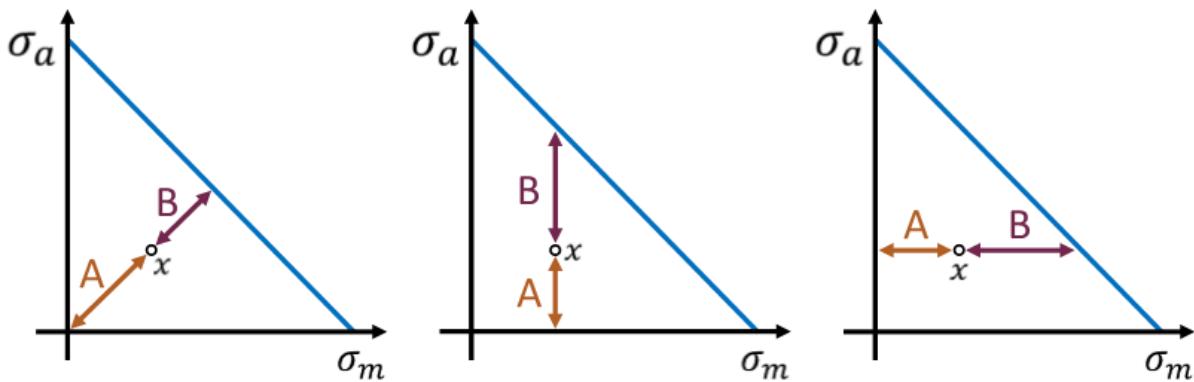


Figure 8.2: Radial, vertical and horizontal fatigue reserve factor relative to an arbitrary cycle, x , on a Haigh diagram. The blue line represents an arbitrary design envelope

An arbitrary cycle, x , is plotted on a Haigh diagram in Figure 8.2. The value of the FRF in the radial, vertical and horizontal direction is given by Equation 8.1. The terms σ_m and σ_a correspond to the mean stress and stress amplitude, respectively.

$$FRF = \frac{A + B}{A} \quad [8.1]$$

The pros and cons of the FRF calculation are as follows:

Pros:

- the calculation is fast; and
- results are accurate for simple, constant amplitude loading.

Cons:

- the choice of design envelope is usually subjective;
- results are inaccurate for variable amplitude loading;
- since the FRF is not calculated from the complete loading, it cannot be used to reliably scale the stresses; and
- the number of loading repeats set by **REPEATS** in the job file will not affect the FRF calculation.

8.2.2 Specifying the FRF envelope

Quick Fatigue Tool includes the Goodman, Goodman B and Gerber design envelopes. The user specifies the FRF envelope in the job file.

Job File Usage: `FATIGUE_RESERVE_FACTOR = {1.0 | 2.0 | 3.0 | 'msc-file-name.msc'}`

- 1: Goodman
- 2: Goodman B (default—if available)
- 3: Gerber
- '`msc-file-name.msc`': User-defined envelope

8.2.3 Theoretical background

Goodman envelope

The Goodman envelope is defined by Equation 8.2:

$$\frac{\sigma_a}{\sigma_0} + \frac{\sigma_m}{\sigma_u} = 1 \quad [8.2]$$

The mean stress and stress amplitude is given by σ_m and σ_a , respectively. The envelope is limited by the ultimate tensile strength and the endurance limit, σ_u and σ_0 , respectively.

By rearranging Equation 8.2, the values of the Goodman fatigue reserve factors in the horizontal, vertical and radial directions are given by Equations 8.3-5, respectively:

$$FRF_H = \frac{\sigma_u}{\sigma_m} \left(1 - \frac{\sigma_a}{\sigma_0} \right) \quad [8.3]$$

$$FRF_V = \frac{\sigma_0}{\sigma_a} \left(1 - \frac{\sigma_m}{\sigma_u} \right) \quad [8.4]$$

$$FRF_R = \frac{\alpha}{\beta} \quad [8.5]$$

The terms α and β are given by Equations 8.6-7:

$$\alpha = \sqrt{\gamma^2 + \delta^2} \quad [8.6]$$

$$\beta = \sqrt{{\sigma_m'}^2 + {\sigma_a'}^2} \quad [8.7]$$

where σ_m' and σ_a' is the mean stress and stress amplitude of the cycle, respectively. The terms γ and δ are given by Equations 8.8-9:

$$\gamma = \frac{\sigma_0}{\frac{\sigma_a'}{\sigma_m'} + \frac{\sigma_0}{\sigma_u}} \quad [8.8]$$

$$\delta = \sigma_a' + \frac{\sigma_a'}{\sigma_m'} (\gamma - \sigma_m') \quad [8.9]$$

Goodman B envelope

Equations 8.2-9 apply to the standard Goodman envelope. By default, Quick Fatigue Tool uses a modified version of this envelope called Goodman B, which is the intersection between the standard Goodman and Buch envelopes. The Goodman B envelope is illustrated in Figure 7.3. If the proof stress is not defined in the material, the standard Goodman envelope is automatically used by default.

Gerber envelope

The Gerber envelope is the quadratic equivalent of the Goodman envelope, and is defined by Equation 8.10:

$$\left(\frac{\sigma_a}{\sigma_0}\right)^2 + \left(\frac{\sigma_m}{\sigma_u}\right)^2 = 1 \quad [8.10]$$

By rearranging Equation 8.10, the values of the Gerber fatigue reserve factors in the horizontal, vertical and radial directions are given by Equations 8.11-13, respectively:

$$FRF_H = \frac{\sigma_u}{\sigma_m} \sqrt{1 - \left(\frac{\sigma_a}{\sigma_0}\right)^2} \quad [8.11]$$

$$FRF_V = \frac{\sigma_0}{\sigma_a} \sqrt{1 - \left(\frac{\sigma_m}{\sigma_u}\right)^2} \quad [8.12]$$

$$FRF_R = \frac{\alpha}{\beta} \quad [8.13]$$

The terms α and β are given by Equations 8.14-15:

$$\alpha = \sqrt{\gamma^2 + \delta^2} \quad [8.14]$$

$$\beta = \sqrt{{\sigma_m'}^2 + {\sigma_a'}^2} \quad [8.15]$$

The terms γ and δ are given by Equations 8.16-17:

$$\gamma = \sqrt{\frac{{\sigma_0}^2}{\left(\frac{{\sigma_a}'}{{\sigma_m}'}\right)^2 + \left(\frac{\sigma_0}{\sigma_u}\right)^2}} \quad [8.16]$$

$$\delta = {\sigma_a}' + \frac{{\sigma_a}'}{{\sigma_m}'} (\gamma - {\sigma_m}') \quad [8.17]$$

Fatigue reserve factors calculated from the Goodman envelope are slightly more conservative than those calculated from the Gerber envelope.

8.2.4 Specifying the target life

The FRF target life, N_T , can either be the infinite life envelope defined by the material's endurance limit, or a user-defined design life. The user specifies the target life calculation mode in the environment file.

Environment File Usage: `frfTarget = {1.0 | 2.0}`

1. Target the FRF towards the design life ($N_T = N_D$)
2. Target the FRF towards infinite life ($N_T = N_\infty$) (default)

The user specifies the design life in the job file.

Job File Usage: `DESIGN_LIFE = N_D`

Infinite life is governed by the constant amplitude endurance limit, N_∞ . The user specifies the constant amplitude endurance limit in the material data.

Material Manager Usage: General region of the *Material Editor* dialogue: Check the option **Constant amplitude endurance limit** and fill out the edit box.

Text File Usage: *CAEL
First (and only) line: N_∞

8.2.5 Specifying the FRF limits

The user specifies the maximum and minimum FRF values, FRF_{max} and FRF_{min} , in the environment file.

Environment File Usage: `frf.MaxValue = FRF_{max} (10)`
`frf.MinValue = FRF_{min} (0.1)`

If the user specifies the option **No damage in compression** (*NO COMPRESSION) in the material definition and the cycle is purely compressive, then a value of FRF_{max} is always assumed for that cycle.

8.2.6 Output

Specifying FRF output

The user specifies FRF output in the job file.

Job File Usage: `OUTPUT_FIELD = {0.0 | 1.0}`

0. Do not write field output
1. Write field output—including the FRF (default)

FRF error output

Although the radial FRF is defined for any combination of stress amplitude and mean stress, the horizontal and vertical FRFs are only defined within certain regimes. For example, with respect to the Goodman B envelope the horizontal FRF is only defined for stress amplitudes below the fatigue limit, whereas the vertical FRF is only defined for mean stresses less than the proof stress. If the FRF cannot be evaluated, a value of -1.0 is reported to the field output file.

8.2.7 User-defined FRF envelopes

Overview

The FRF can be calculated with user-defined data. This is a file containing a material life envelope as Haigh diagram ($\sigma_m - \sigma_a$) data. FRF data files are created in the same way as user-defined mean stress correction (.msc) files. Quick Fatigue Tool uses the .msc file for both user mean stress correction and FRF calculations. For guidance on creating custom FRF data, see Section 7.9.

Specifying user-defined FRF envelopes

The user specifies user-defined FRF envelopes in the job file.

Job File Usage: `FATIGUE_RESERVE_FACTOR = 'msc-file-name.msc'`

Specifying the interpolation method

Quick Fatigue Tool interpolates the user-defined data to find the point on the envelope which forms the intersecting line with the stress cycle coordinate (radial, horizontal and vertical).

The user specifies the interpolation method in the environment file.

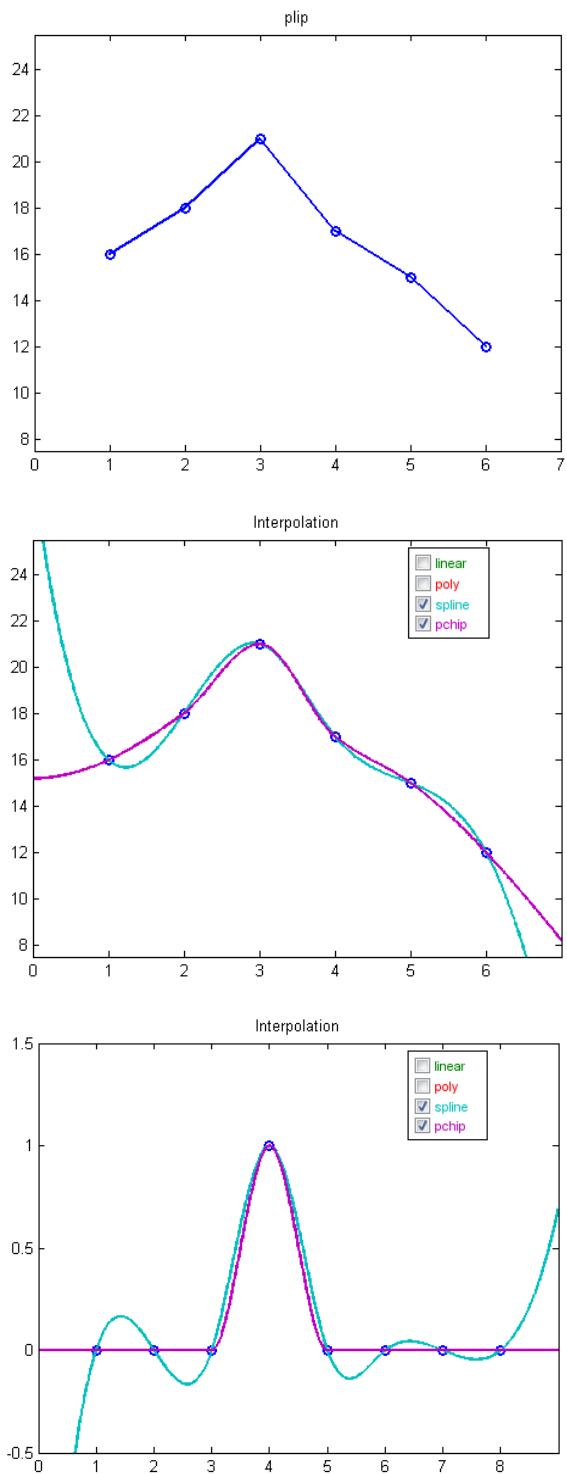
Environment File Usage: `frfInterpOrder = {'NEAREST' | 'LINEAR' | 'SPLINE' | 'PCHIP'}`

- '**NEAREST- '**LINEARy = mx + c) (default)**
- '**SPLINE- '**PCHIP******

The *linear* method is shown in Figure 8.3. A comparison between the *spline* and *pchip* methods are shown in Figures 8.4-5.

The *linear* method is appropriate in most cases, and especially when few data points are available. The *spline* method is globally very smooth, while *pchip* only considers local curvature between data points. Although the *pchip* method results in coarser interpolation, its behaviour is preferable over the *spline* method in cases where the data contains sudden changes in gradient, as shown by Figure 8.5.

The *Nearest neighbour* method is not recommended as it can produce highly inaccurate results.



Figures 8.3-5: From top to bottom: Linear; spline vs PCHIP for smooth data; spline vs PCHIP for discontinuous data.

Image credit: Cleve Moler, “Splines and Pchips”, July 16 2012, The Mathworks.

Modelling compressive behaviour with user-defined FRF envelopes

Metals tend to show greater fatigue strength when loaded in compression, compared with tension. This behaviour is often represented by a plateau on the FRF curve as shown by Figure 8.1, in the case of the Goodman envelope. Care should be taken when defining these plateaus with user-defined FRF data, since this can result in undesirable behaviour from the interpolation algorithm in MATLAB.

For an endurance envelope with a compressive plateau, the FRF data may be defined as

Mean stress	Stress amplitude
1	0
0	1
-1	1
-1	0

Since the mean stress values must be monotonically decreasing, the definition above will result in an error. The problem is resolved by modifying the mean stress values by a very small amount, as shown below.

Mean stress	Stress amplitude
1	0
0	1
-0.9999	1
-1	0

Quick Fatigue Tool automatically adjusts the stress amplitude values to ensure the correct behaviour of the interpolation algorithm.

Specifying normalisation parameters for user-defined FRF envelopes

Quick Fatigue Tool normalises the mean stress and stress amplitude of each fatigue cycle to the limits of the design envelope before performing an FRF calculation with user-defined FRF data. By default, the design envelope limits are those defined by the Goodman FRF: Tensile and compressive mean stresses are normalised by the ultimate tensile and ultimate compressive strength, respectively, while the stress amplitude is normalised by the fatigue limit stress.

The user specifies the normalisation parameters in the environment file.

Tensile/compressive mean stress normalisation

Environment File Usage: `frfNormParamMeanT = {'UTS' | 'UCS' | 'PROOF' | n}`
 `frfNormParamMeanC = {'UTS' | 'UCS' | 'PROOF' | n}`

- '`UTS`': Normalise by the ultimate tensile strength, σ_u (default)
- '`UCS`': Normalise by the ultimate compressive strength, σ_c
- '`PROOF`': Normalise by the proof stress, σ_y
- '`n`': User-defined normalisation parameter

Stress amplitude normalisation

Environment File Usage: `frfNormParamAmp = {'LIMIT' | n}`

- '`LIMIT`': Normalise by the fatigue limit, σ_0 (default)
- n : User-defined normalisation parameter

Visualising user-defined FRF envelopes

The user-defined FRF envelope can be plotted for a chosen analysis item with its respective radial, horizontal and vertical projections.

The user specifies user-defined FRF envelopes at the requested items n as a MATLAB figure in the environment file. MATLAB figures must also be requested.

Environment File Usage: `frfDiagnostics = n`

Job File Usage: `OUTPUT_PICTURE = 1.0`

Any number of analysis items can be requested. However, if the requested item was removed by nodal elimination then it will not be exported. An example diagnostic output is shown in Figure 8.6 for the uniaxial cycle [28.3, -348.3], with the envelope 'Gray Iron.msc'.

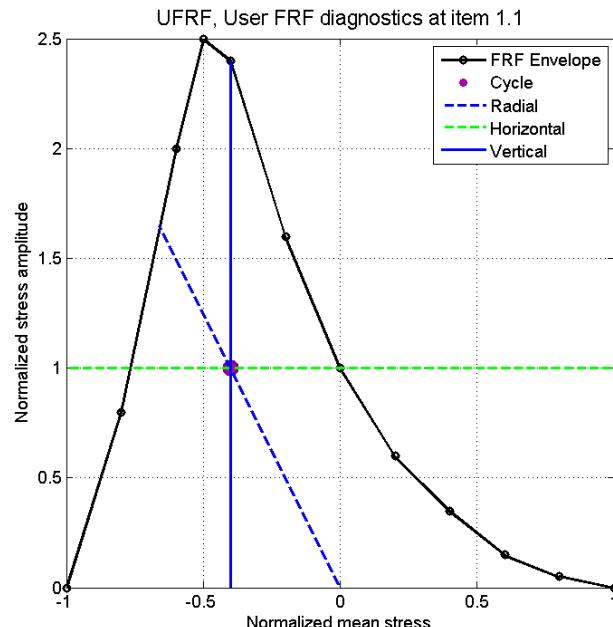


Figure 8.6: An example FRF diagnostics with 'Gray Iron.msc'. Mean stress and stress amplitude values are normalised by 400 and 188.3, respectively.

Precautions for user-defined FRF envelopes

The user should ensure that the user-defined FRF data is defined in such a way that valid FRF calculations are possible. If Quick Fatigue Tool is unable to calculate the FRF based on the user data, a value of -1 will be assigned. If the data is considered unusable, the analysis will exit with an error. User FRF data is checked against the following conditions before the start of the analysis:

- there must be exactly two columns of data;
- there must be at least two FRF data pairs;
- mean stress values must be decreasing down the column;
- stress amplitude values must be positive;
- the radial from the origin, through the cycle, must not be able to cross the FRF envelope more than once; and
- duplicate stress amplitude values on a given side of the amplitude axis are not permitted (the same amplitude value can be specified as long as it is on the other side of the amplitude axis).

The following observations apply to user-defined FRF data:

- mean stress values should not be over unity;
- adjacent stress amplitude values are automatically adjusted to prevent zero gradients;
- the FRF envelope should be closed at both ends (zero amplitude at mean stress limits); and
- horizontal FRF values will default to -1.0 if there is more than one possible solution.

8.2.8 Treatment of residual stresses

If a residual stress is defined with **RESIDUAL**, the FRF calculation is modified such that the origin of the Haigh diagram is shifted from $(0, 0)$ to $(\sigma_R, 0)$.

8.3 Factor of Strength

8.3.1 Overview

The FOS is a linear scale factor which, when applied to the fatigue loading, results in the specified design life. Although traditionally the FRF was preferred over the FOS due to it being relatively inexpensive, modern computers are able to perform FOS calculations in a reasonable amount of time and as such the FOS is generally preferred over the FRF.

The FOS takes into account the nonlinear relationship between the damage parameter and the fatigue life by iterating to recalculate the life until a stop condition is met.

Quick Fatigue Tool performs the following procedure when calculating the FOS:

1. Assume an initial FOS of 1.0
2. Compare the calculated fatigue life with the target life:
 - a. If the calculated life is less than the target life, decrease the FOS by a fixed increment
 - b. If the calculated life is greater than the target life, increase the FOS by a fixed increment
3. Apply the current FOS to the original loading and re-calculate the life
4. Repeat steps 2 and 3 until one of the following stop conditions is met:
 - a. The maximum number of iterations has been reached
 - b. The specified tolerance has been achieved
 - c. The target life lies between the last two computed values of fatigue life
 - d. The current FOS is less than or equal to the minimum specified FOS
 - e. The current FOS is greater than or equal to the maximum specified FOS

The FOS is far more computationally expensive than the FRF because it repeats the damage calculation at every analysis item. Furthermore, for the multiaxial algorithms such as the Stress-based Brown-Miller, a new critical plane analysis must be performed each time. This is because changes in the mean stress caused by re-scaling of the load history can affect the orientation of the critical plane.

For simple, constant amplitude loading, the FOS does not provide additional accuracy over the FRF. In such cases, the FRF should be used instead.

The pros and cons of the FOS calculation are as follows:

Pros:

- subjective choice of design envelope is not required;
- results are accurate for both simple and complex, variable amplitude loading; and
- since the whole damage calculation is repeated, the FOS takes into account changes in the critical plane orientation, mean stress, surface finish definition, plasticity correction etc.

Cons:

- the calculation is computationally expensive; and

- since the calculation accounts for the global load state, the FOS is only valid when used to scale the stress at every point in the model. For example, models containing bolts where the preload level is fixed may yield overly conservative FOS values.

8.3.2 Enabling the FOS calculation

The user enables the FOS calculation in the job file.

Job File Usage: **FACTOR_OF_STRENGTH** = {0.0 | 1.0}

0. Disable FOS iterations (default)
1. Enable FOS iterations

Field output must also be requested.

Job File Usage: **OUTPUT_FIELD** = 1.0

8.3.3 Specifying the target life

The FOS target life, N_T , can either be the infinite life envelope defined by the material's endurance limit, or a user-defined design life. The user specifies the target life calculation mode in the environment file.

Environment File Usage: **fosTarget** = {1.0 | 2.0}

1. Target the FOS towards the design life ($N_T = N_D$) (default)
2. Target the FOS towards infinite life ($N_T = N_\infty$)

The user specifies the design life in the job file.

Job File Usage: **DESIGN_LIFE** = N_D

Infinite life is governed by the constant amplitude endurance limit, N_∞ . The user specifies the constant amplitude endurance limit in the material data.

Material Manager Usage: General region of the *Material Editor* dialogue: Check the option **Constant amplitude endurance limit** and fill out the edit box.

Text File Usage: *CAEL
First (and only) line: N_∞

8.3.4 FOS band definitions

Specifying the envelope values

The user specifies the maximum and minimum FOS values, FOS_{max} and FOS_{min} , in the environment file.

Environment File Usage: **fos.MaxValue** = FOS_{max} (2)
fos.MinValue = FOS_{min} (0.5)

Specifying the incrementation regions

During each iteration, the FOS is increased or decreased by a fixed increment. The value of the increment depends on whether the current FOS lies within the coarse or the fine incrementation region.

The user specifies the maximum and minimum FOS values, FB_{max} and FB_{min} , in the fine incrementation band in the environment file.

Environment File Usage: $\text{fosMaxFine} = FB_{max}$ (1.5)
 $\text{fosMinFine} = FB_{min}$ (0.8)

The FOS band values must be decreasing:

$$FOS_{max} \geq FB_{max} \geq FB_{min} \geq FOS_{min}$$

Furthermore,

$$FOS_{max} > FOS_{min}$$

FOS values within this band are incremented with a smaller value. This fine value must be smaller than or equal to the coarse value.

Specifying the incrementation values

The user specifies the increment size for the coarse and fine bands, FI_{coarse} and FI_{fine} , in the environment file.

Environment File Usage: $\text{fosCoarseIncrement} = FI_{coarse}$ (0.1)
 $\text{fosFineIncrement} = FI_{fine}$ (0.01)

8.3.5 Specifying FOS stop conditions

The user specifies the FOS stop conditions in the environment file.

Specifying FN_{mc} , the maximum number of coarse iterations

Environment File Usage: $\text{fosMaxCoarselterations} = FN_{mc}$ (8)

If FN_{mc} is reached, the FOS calculation is abandoned and the last FOS solution accepted.

Specifying FN_{mf} , the maximum number of fine iterations

Environment File Usage: $\text{fosMaxFinelterations} = FN_{mf}$ (12)

If FN_{mf} is reached, the FOS calculation is abandoned and the last FOS solution accepted.

Specifying F_{tol} , the FOS tolerance

Environment File Usage: $\text{fosTolerance} = F_{tol}$ (5)

If F_{tol} is reached, the FOS calculation is converged and the last FOS solution is accepted.

Specifying FOS bracketing

If the target life is below one million cycles, linearly scaling the loading by even very small amounts can have a dramatic effect on the fatigue life. This can cause numerical difficulties when Quick Fatigue Tool attempts to find a solution to the FOS. An example of “chattering” is shown in Figure 8.9, where the FOS never converges on the target life. This can be prevented by enabling a bracketing condition which ends the FOS calculation if the current calculated life crosses the target life.

The user specifies FOS bracketing in the environment file.

Environment File Usage: `fosBreakAfterBracket = {0.0 | 1.0}`

0. Continue FOS calculations after the calculated life crosses the target life (default)
1. Accept the solution if the calculated life crosses the target life

If the FOS algorithm detects a chattering condition, bracketing is automatically enabled to prevent additional iterations. Quick Fatigue Tool will automatically accept the best solution based on the tolerance.

8.3.6 Additional guidance and FOS diagnostics

Overview

Quick Fatigue Tool estimates the solution to the FOS by linearly scaling the original loading by successive estimates of the FOS. If the target life is very far away from the calculated life, or if the target life is very small, the FOS calculation can become unreliable. This is due to the finite allowable increments in the first case and the severe nonlinearity of the S–N curve at low lives in the second case. In fact, there is no single configuration of the FOS parameters which is guaranteed to achieve a reliable solution for all target lives.

The user is strongly encouraged to review the accuracy of the FOS calculation, which is printed to the log file (*Project\output\<jobName>\<jobName>.log*).

FOS diagnostics

Detailed information about the calculation can be obtained by limiting the analysis to the worst item using the **ITEMS** option in the job file, and requesting FOS diagnostics.

The user specifies FOS diagnostics in the environment file.

Environment File Usage: `fosDiagnostics = {0.0 | 1.0}`

0. Do not write diagnostic output
1. Create a MATLAB figure of the FOS history (default)

This creates a MATLAB figure showing the successive FOS and life values over each iteration, which allows the user to easily check whether the scaled loading results in a fatigue life sufficiently close to the target value. The iteration history is printed to the log file, and the FOS accuracy for each analysis item is written to the file *Project\output\<job>\Data Files\fos_accuracy.log*. If automatic export is enabled, the FOS accuracy per analysis item is written to the *.odb* file as an additional results field.

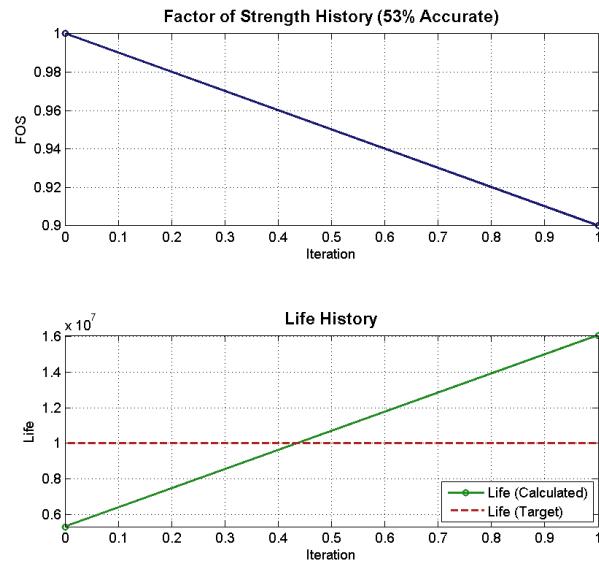


Figure 8.7: Example FOS diagnostics for *tutorial_intro* with a target life of 1E7 repeats. The coarse and fine increments are 0.1 and 0.01, respectively.

The user must enable MATLAB figures in the job file in order to generate the FOS diagnostic figure.

Job File Usage: `OUTPUT_PICTURE = 1.0`

Example FOS calculation

An example FOS diagnostic output is shown by Figure 8.7 for the job *tutorial_intro*. In this case only one iteration was performed because the value of the FOS crossed the target life. The value of the FOS is reported as 1.0 because the original value of life is closer to the design target than the initial FOS calculation would produce. This represents an accuracy of 53%. It is clear that the default FOS configuration for this analysis failed to achieve an acceptable value.

The coarse and fine increments are reduced to allow Quick Fatigue Tool to perform more iterations before the target life is crossed.

Environment File Usage: `fosCoarseIncrement = 0.01`
`fosFineIncrement = 0.001`

The maximum number of iterations is also increased.

Environment File Usage: `fosMaxCoarselIterations = 16.0`
`fosMaxFinIterations = 24.0`

The FOS diagnostic for this run is shown in Figure 8.8. The calculated FOS is 0.94 and scaling the loading by this value results in a life $1.017e7$ repeats. This solution is within 1.7% of the target life. Therefore, the value is considered to be acceptable.

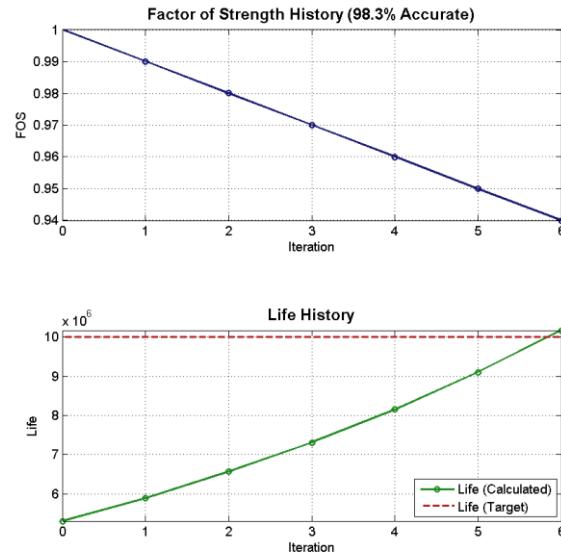


Figure 8.8: Example FOS diagnostics for *tutorial_intro* with a target life of $1E7$ repeats. The coarse and fine increments are 0.01 and 0.001, respectively.

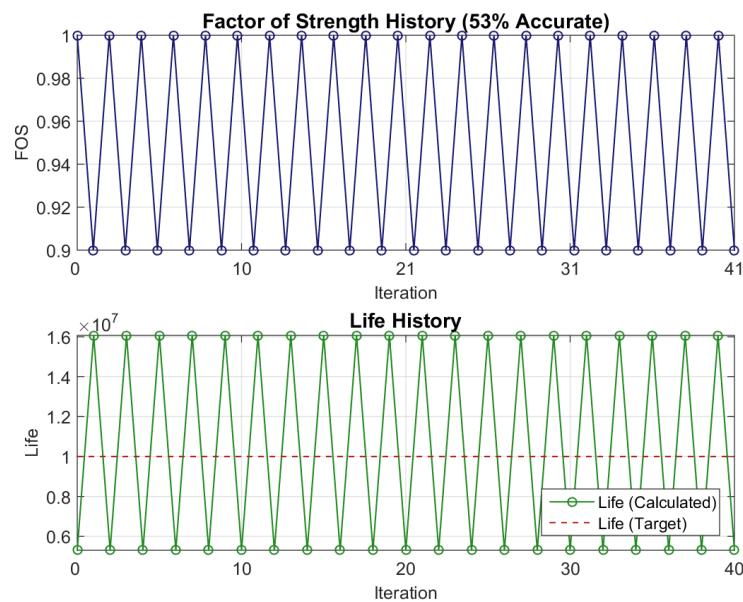


Figure 8.9: An example of FOS “chattering” where the algorithm is unable to converge on the target life.

8.3.7 FOS augmentation

Overview

Selection of the correct incrementation settings is often unobvious and can lead to a time-consuming process of trial and error. Even with well-tuned settings, the default incrementation scheme is linear, and it is easy to unnecessarily spend many iterations. FOS augmentation is a scheme which attempts to accelerate FOS convergence by modifying the user-defined incrementation parameters if the current iteration is judged to cause unsatisfactory convergence behaviour. Thus, FOS augmentation aims to alleviate the manual aspect of the FOS algorithm by applying a certain level of automation to the incrementation scheme.

Since the effect of estimating the FOS depends on a number of factors, such as loading, critical plane analysis, material properties (and so on), even the augmented FOS scheme is required to make guesses as to the appropriate incrementation.

Specifying FOS augmentation

The user specifies FOS augmentation in the environment file.

Environment File Usage: **fosAugment** = {0.0 | 1.0}

0. Disable FOS augmentation
1. Enable FOS augmentation (default)

Methodology

When augmentation is enabled, Quick Fatigue Tool compares N_l , the difference in fatigue life between the two most recent iterations, with N_t , the difference between the previously calculated fatigue life and the target life. If the ratio between N_l and N_t is less than a threshold value, k_{thresh} , the next FOS increment is increased by a factor k_{factor} .

The user specifies the threshold value and augmentation factor, k_{thresh} and k_{factor} , in the environment file.

Environment File Usage: **fosAugmentThreshold** = k_{thresh} (0.2)
 fosAugmentFactor = k_{factor} (5)

Figures 8.10-11 illustrate the improvement in convergence behaviour when FOS augmentation is enabled with the default settings using *tutorial_intro.m* as an example (note that the load factor and algorithm have been modified to improve the illustration).



Figure 8.10: FOS diagnostics with augmentation disabled

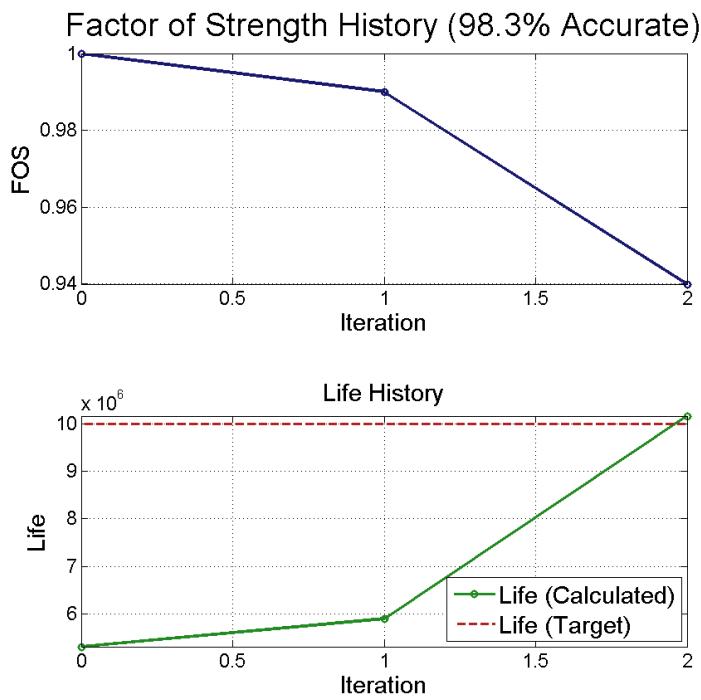


Figure 8.11: FOS diagnostics with augmentation enabled

9. Job and environment files

This section has been released in the document *Quick Fatigue Tool User Settings Reference Guide*.

10. Output

10.1 Background

Quick Fatigue Tool reports extensive analysis output in addition to the fatigue life and safety factors at the worst analysis item.

There are four categories of output:

1. Worst item log file summary
2. Field variables
3. History variables (worst analysis item)
4. History variables (all analysis items)

The worst item is defined as the item with the largest fatigue damage. In cases where more than one item has the largest damage, the item with the largest principal stress over the set of worst items is taken.

Field and history data is written to a set of text files. Pre-selected history variables can be plotted automatically and saved in the results directory as MATLAB figures. If FEA stresses were analysed from an Abaqus model, field data can be written to an output database (.odb) file and viewed on the finite element mesh with Abaqus/Viewer. This functionality is discussed in Section 10.5.

The user requests field and history output in the job file.

Job File Usage: **OUTPUT_FIELD** = {0.0 | 1.0}
 OUTPUT_HISTORY = {0.0 | 1.0}

The user requests pre-selected MATLAB figures in the job file.

Job File Usage: **OUTPUT_PICTURE** = {0.0 | 1.0}

When an analysis runs to completion, a compact summary of a selection of variables is displayed in the command window and written to the log file, an example of which is shown in Figure 10.1. If extensive output is requested in the job file, the fields and histories are written to separate output files.

```
FATIGUE RESULTS SUMMARY:  
=====
```

Worst Life-Repeats at item 1.1	: 5.3e+06
Number of cycles in loading	: 1
Worst FRF at Item 1.1	: 0.9415
Maximum stress (MPa) at Item 1.1	: 200
Maximum stress/yield at Item 1.1	: 0.6154
Maximum stress/UTS at Item 1.1	: 0.5
Worst cycle mean stress (MPa) at Item 1.1	: 0
Worst cycle stress amplitude (MPa) at Item 1.1	: 200
Worst cycle damage parameter (MPa) at Item 1.1	: 200
Analysis time	: 0:00:0.398

```
=====
```

Job tutorial_intro completed successfully (14-Jul-2016 15:59:42)

Figure 10.1: Example fatigue results summary from the log file

10.2 Output files

10.2.1 Overview

Quick Fatigue Tool can create the following output files during an analysis:

- a log file containing printed output of the model and analysis definition;
- a series of data files containing printed output of field and history variables;
- a series of MATLAB figure files containing plots of selected history variables;
- a message file containing diagnostic messages about the analysis;
- a status file containing information about the status of the analysis;
- an Abaqus output database file containing selected field variables; and
- MATLAB binary files containing model definition data for use during the analysis.

10.2.2 The data files

The data file is a text file that contains printed field and history output variables.

File name	Contents
<i>Project\output\<job-name>\Data Files\f-output-all.dat</i>	<ul style="list-style-type: none">• Field output variables at each analysis item
<i>Project\output\<job-name>\Data Files\f-output-analysed.dat</i>	<ul style="list-style-type: none">• Field output variables at each analysis item which were not removed by nodal elimination
<i>Project\output\<job-name>\Data Files\h-output-cycle.dat</i>	<ul style="list-style-type: none">• Worst cycle information at each analysis item• Cycle history at the worst analysis item
<i>Project\output\<job-name>\Data Files\h-output-load.dat</i>	<ul style="list-style-type: none">• Stress invariant, normal and shear stress histories at the worst analysis item
<i>Project\output\<job-name>\Data Files\h-output-tensor.dat</i>	<ul style="list-style-type: none">• Stress tensor history at the worst analysis item
<i>Project\output\<job-name>\Data Files\h-output-angle.dat</i>	<ul style="list-style-type: none">• Stress components and damage on each plane at the worst analysis item⁸

Controlling which data files are written

The user controls which data files are written in the environment file.

Environment File Usage: `file_F_OUTPUT_<name> = {0.0 | 1.0}`
 `file_H_OUTPUT_<name> = {0.0 | 1.0}`

A complete list of these variables is found in Section 2.5 of the document *Quick Fatigue Tool User Settings Reference Guide*.

⁸ Only for analyses which utilise critical plane searching

10.2.3 The message file

The message file is a text file that contains diagnostic messages about the progress of the analysis. If any of the messages describe warnings, the message is prepended with ***WARNING.

The message file contains information about the worst stress range in the model, results of the nodal elimination algorithm, analysis-specific guidance, as well as various other diagnostic notes to inform the user about the analysis.

The user is strongly advised to review the message file at the end of each analysis.

File name	<i>Project\output\<job-name>\<job-name>.msg</i>
------------------	---

10.2.4 The MATLAB binary files

The MATLAB binary files contain model definition data for use during the analysis. The main purpose of these files is to cache information which is likely to be reused in order to save time.

File name	Contents
<i>Data\library\[J]<job-name>_fd.mat</i>	<ul style="list-style-type: none">• Analysis definition data
<i>Data\surfaces\<model-ID>_surface.mat</i>	<ul style="list-style-type: none">• Mesh surface extracted by the surface detection algorithm

These files are generated automatically and accessed when necessary—the user is not required to modify these files.

10.3 Output variables

10.3.1 Fatigue analysis field variable identifiers

L	Fatigue life (linear scale) at each item in the model. The units depend on the string value set by LOAD_EQ in the job file.
LL	The \log_{10} value of L . Note that values of LL are capped at the material's endurance limit. For example, if the endurance limit is $2E+07$ reversals, then the maximum reported value of LL will be 7.0.
D	Fatigue damage (L^{-1}) at each item in the model.
DDL	Fatigue damage at design life. Calculated by multiplying the damage, D , by the user-specified design life.
FOS	Factor of strength at design life. A linear scale which, when multiplied by the loading, results in the design life. This field has a value of -1.0 if the FOS calculation was not requested. The FOS calculation is enabled by setting FACTOR_OF_STRENGTH=1.0 in the job file.
SFA	Endurance safety factor. The ratio between the material fatigue limit and the maximum stress/strain amplitude, WCA , at each item in the model. The endurance limit is discussed in the document <i>Quick Fatigue Tool Appendices: A1</i> . For strain-life analysis, the fatigue limit is converted to elastic strain.
FRFH	Horizontal fatigue reserve factor. For user-defined FRF data, a value of -1.0 is returned if the calculation was unsuccessful.

FRFV	Vertical fatigue reserve factor. For user-defined FRF data, a value of -1.0 is returned if the calculation was unsuccessful.
FRFR	Radial fatigue reserve factor. For user-defined FRF data, a value of -1.0 is returned if the calculation was unsuccessful.
FRFW	Fatigue reserve factor (worst of above three).
SMAX	Largest stress in loading. If the absolute value of the third principal stress is greater than the absolute value of the first principal stress, the maximum stress will be the third principal. The calculation of the maximum stress does not include the effect of residual stress.
SMXP	SMAX divided by the proof stress.
SMXU	SMAX divided by the material's ultimate tensile strength.
TRF	Stress triaxiality factor (ratio between hydrostatic and von Mises stress): $TRF = -\frac{tr(\sigma)I_1}{3\sqrt{3J_2}}$ <p>where σ is the Cauchy stress tensor, I_1 is the first stress invariant and J_2 is the second deviatoric stress invariant.</p>
WCM	Worst cycle mean stress/strain. Note that the mean stress/strain is taken as the mean value of the worst cycle at each item in the model defined by the damage parameter. Therefore, the value of WCM depends on the selected fatigue analysis algorithm.

WCA Worst cycle stress/strain amplitude. The stress/strain amplitude is the cycle counted quantity according the selected analysis algorithm:

Uniaxial Stress/Strain-Life: Uniaxial stress/strain $\sigma_{11}/\varepsilon_{11}$ (defined directly as the stress history).

Stress-based Brown-Miller: Sum of the shear and normal stress on the critical plane.

Normal Stress: Normal stress on the critical plane.

Findley's Method: Shear stress on the critical plane (note that the normal stress is not included in the definition of the stress amplitude).

Stress Invariant Parameter: von Mises equivalent stress.

BS 7608: Normal, shear or (normal + shear) stress on the critical plane, depending on the value of **FAILURE_MODE** in the job file

NASALIFE: Effective stress, defined by the environment variable **nasalifeParameter**. The effective stress variables are described in Section 6.7.4.

WCATAN Worst cycle arctangent between **WCM** and **WCA**.

WCDP Worst cycle damage parameter. The damage parameter is the stress used in the fatigue damage calculation and is usually the stress/strain amplitude.

The damage parameter includes the effect of the mean stress correction (except in the case of the Morrow and R-ratio S–N curves corrections, since these are applied indirectly). Therefore, if mean stress correction is applied to the loading, the values of **WCDP** and **WCA** will differ.

If no mean stress correction is applied, or if the mean stress in the loading is zero, the value of **WCDP** and **WCA** is the same. The exception is with the use of Findley's Method with $k > 0$, since the amplitude parameter is different from the damage parameter.

10.3.2 Static analysis field variable identifiers

Yield criteria

YIELD	Items in the model which are yielding.
	This field has a value of 1.0 if the items have yielded according to the selected criterion, 0.0 if the item has not yielded, -2.0 if the yield criterion could not be evaluated and -1.0 if the yield calculation was not requested.
	If the variable YIELD has a value of 1.0 anywhere in the model, the energy densities are written to 'yield_assessment.dat' .
VMCRT	von Mises criterion yield index.
TRCRT	Tresca criterion yield index.
SSCRT	Maximum shear strain energy theory yield index.
TSCRT	Total strain energy theory yield index.
PEEQ	Normalised equivalent plastic strain energy density.

Composite criteria

MSTRS	Maximum stress theory failure measure.
MSTRN	Maximum strain theory failure measure.
TSAIH	Tsai-Hill theory failure measure.
TSAIW	Tsai-Wu theory failure measure.
TSAIWTT	Tsai-Wu theory failure measure for closed cell PVC foam.
AZZIT	Azzi-Tsai-Hill theory failure measure.
HSNFTCRT	Hashin's fibre tensile damage initiation criterion.

HSNFCCRT	Hashin's fibre compression damage initiation criterion.
HSNMTCRT	Hashin's matrix tensile damage initiation criterion.
HSNMCCRT	Hashin's matrix compression damage initiation criterion.
LARPFCRT	LaRC05 polymer damage initiation criterion.
LARMFCRT	LaRC05 matrix damage initiation criterion.
LARKFCRT	LaRC05 fibre kink damage initiation criterion.
LARSFCRT	LaRC05 fibre split damage initiation criterion.
LARTFCRT	LaRC05 fibre tensile damage initiation criterion.

10.3.3 Fatigue analysis history variable identifiers

History output represents the load-varying quantities for the most damaged analysis item in the model. The following histories are written, with their respective identifiers:

ST*	Stress tensor components at worst item for the loading.
HD*	Haigh diagram for the worst cycle.
VM	von Mises stress at the worst item.
PS	Principal stresses (maximum, middle and minimum).
PE	Principal strains (maximum, middle and minimum).
CN	Maximum normal stress history for the loading. For Uniaxial Stress-Life, Uniaxial Strain-Life and von Mises analyses, the maximum normal stress is the numerically largest principal stress. For multiaxial analyses, the maximum normal stress is that maximum normal stress history on the critical plane.
CS	Maximum shear stress history for the loading. For Uniaxial Stress-Life, Uniaxial Strain-Life and von Mises analyses, the maximum shear is the Tresca stress. For NASALIFE analysis, the maximum shear is the maximum octahedral shear stress. For multiaxial analyses, the maximum shear is the maximum shear stress history on the critical plane, determined either by the maximum chord method or the maximum resultant shear stress. This is specified by the environment variable cpShearStress .
DP	Damage vs. plane angle.
DPP	Damage parameter vs. plane angle.
LP	Life vs. plane angle.
DAC	Damage accumulation at worst analysis item. Only available if more there is more than one cycle in the loading.
RHIST	Rainflow histogram of cycle counted stresses.

RC	Stress range distribution.
SIGS	Uniaxial stress history (before and after gating if applicable).
SIGE	Uniaxial inelastic strain history.

* Output is for the critical plane if applicable.

10.3.4 Whole model variable identifiers

Whole model histories are load histories at every item in the model:

ANHD	Worst cycle Haigh diagram for each item in the model.
-------------	---

10.3.5 MATLAB figure variable identifiers

Certain history data can also be plotted to a series of figures. The default figure type is the MATLAB *.fig* file. The user specifies the MATLAB figure format in the environment file.

Environment File Usage: `figureFormat = '<format>'`

Any valid file format is accepted, e.g. '`png`', '`jpeg`', '`jpg`' etc.

ANHD + HD	Worst cycle Haigh diagram for all items and the critical plane.
KDSN	S-N curves for materials using knock-down factors.
CN + CS	Normal and shear stress history on the critical plane at the worst item.
DP	Damage vs. angle at the worst item.
DPP	Damage parameter vs. angle at the worst item.
LP	Life vs. angle at the worst item.
CPS	Normal stress and resultant shear stress vs. plane angle at the worst item.
P(S/E)	Principal stresses and/or strains at the worst item.
VM	von Mises stress at the worst item.
DAC	Cumulative damage at the worst item.
RHIST⁹	Rainflow cycle histogram at the worst item.
RC	Stress range distribution at the worst item.
SIG(S/E)	Uniaxial stress and/or strain load history (before and after gating, if applicable).
LH	Tensor load histories at the worst item.
FOS	Factor of strength diagnostics.

⁹ Requires the Statistics Toolbox

10.4 Viewing output

10.4.1 Output location

Field and history output for a particular analysis is stored under *Project\output\<jobName>\Data Files*. The output directory has the file structure shown in Figure 10.2.

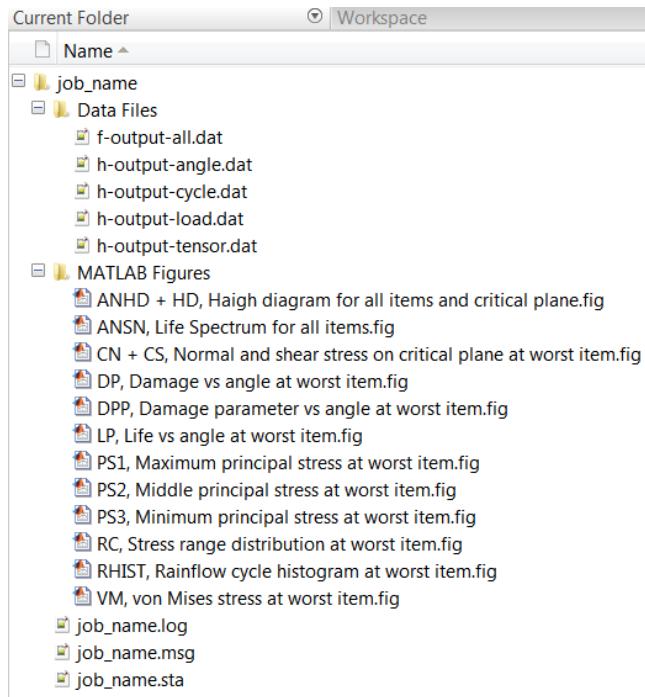


Figure 10.2: Output directory structure

The name of the output directory shares the name of the job with which the output is associated. If a job is submitted with the same name as an existing output directory, Quick Fatigue Tool will overwrite the previous results files. Therefore, care should be taken when choosing the job name in the job file.

Output is stored as human-readable ASCII text, and can be viewed in MATLAB or with any text editor.

10.4.2 Changing the output format

By default, floating-point values are converted to text using fixed-point notation ('%f'). The user specifies the field and history output format in the environment file.

Environment File Usage: `fieldFormatString = '<format>'`
 `historyFormatString = '<format>'`

For example, '`.2f`' represents two digits after the decimal mark, '`12f`' represents twelve characters in the output and '`0f`' represents integer output. The user is not required to specify the '%' format identifier. More information on formatting text can be found in the official MATLAB documentation.

10.4.3 Limitations of critical plane output

If the user specifies `OUTPUT FIGURE=1.0` and the analysis utilises critical plane searching, Quick Fatigue Tool exports MATLAB figures of pertinent analysis quantities (including the damage parameter of the chosen algorithm) on the analysed planes. The critical plane is denoted by a red marker on each curve.

The following precautions should be taken into account when viewing critical plane plots:

- the plots are written for the worst analysis item only
- if critical plane smoothing is specified with `cpSample`>0.0 then the location of the indicated critical plane may not appear on the curve, and the plotted values may not match those printed to the '*h-output-angle.dat*' history file; and
- if the worst cycle mean stress at the worst analysis item is non-zero then the indicated critical plane may not coincide with the plane of maximum damage parameter.

10.5 The ODB Interface

10.5.1 Overview

Quick Fatigue Tool includes an interface which is capable of writing fatigue results to an Abaqus output database (*.odb*) file. This allows the user to visualize selected field output variables in Abaqus/Viewer. There are two methods for accessing the ODB interface:

1. via the job and environment files (text-based); and
2. via the Export Tool application (GUI-based).

The first method allows the user to configure the interface to automatically write fatigue results to the *.odb* file immediately after the analysis, whereas the second method allows the user to write results to an *.odb* file based on a selected field data file, after the analysis has already been performed.

Both methods represent the same functionality; the differences lie in how and when the interface is accessed.

The ODB interface requires that Abaqus is installed on the host machine. If an installation cannot be found, the interface will exit with an error.

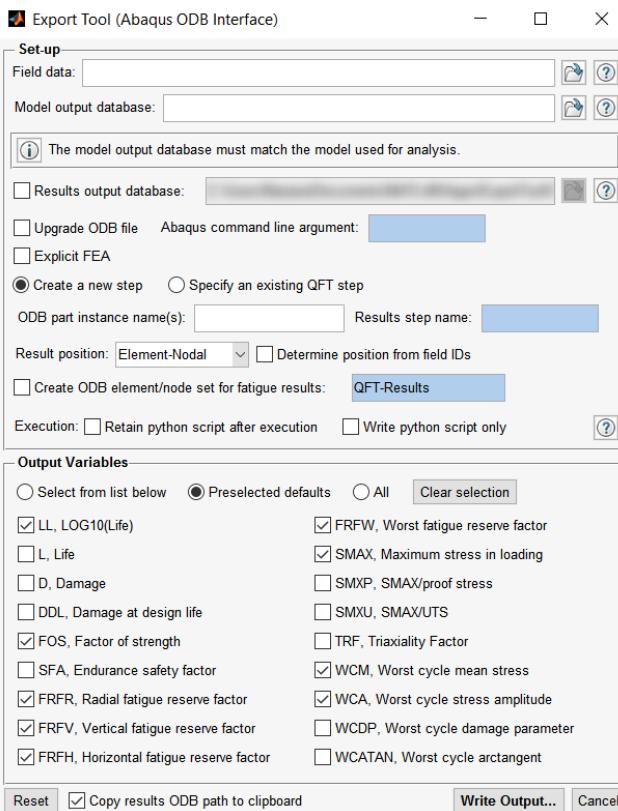


Figure 10.3: Export Tool interface

10.5.2 Accessing the ODB interface via the Export Tool

The Export Tool is a MATLAB GUI application which gives the user interactive access to functions of the ODB interface.

The Export Tool is launched either by running the file *ExportTool.m* in the *Application_Files\code\odb_interface* directory, or by installing and running the tool as a MATLAB app. The app installer can be found in *Application_Files\toolbox*. The Export Tool interface is shown in Figure 10.3.

10.5.3 Enabling the ODB interface via the environment file

As long as the ODB interface is enabled, Quick Fatigue Tool will automatically export field results to an output database file if it finds a valid definition in the job file.

The user enables the ODB interface in the environment file.

Environment File Usage: `autoExport_ODB = {0.0 | 1.0}`

0. Do not export fatigue results to an *.odb* file
1. Automatically export fatigue results to an *.odb* file if applicable (default)

10.5.4 Configuring the ODB interface

Specifying the field data to export

The ODB interface copies results data from the field output file in the output directory. This is typically located in *\Project\output\<jobName>\Data Files\f-output-all.dat*. The field output file must be specified by the user in advance.

Export Tool Usage: Set-up region: Select the field data file either by entering the absolute path in the file window, or by selecting the file via the file browser using the  button.

Job/Environment file Usage: The field data file is created during the job and automatically located after the analysis. The user is not required to specify the field data file.

Specifying the model ODB file

The ODB interface must be able to locate the original (model) *.odb* file so that it can make a copy of the file and append the result step.

Export Tool Usage: Set-up region: Select the model output database file either by entering the absolute path in the file window, or by selecting the file via the file browser using the  button.

Job File Usage: `OUTPUT_DATABASE = 'model-odb-file-name.odb'`

The user must specify the absolute path of the *.odb* file.

Specifying the results ODB file

Quick Fatigue Tool automatically chooses the location of the results .odb file. By default, the file is stored in the *Project\output\<jobName>\Data Files* directory. This behaviour may be overridden with the Export Tool.

Export Tool Usage: Set-up region: Check the option **Results output database**. Select the results output database location either by entering the absolute path in the directory window, or by selecting the location via the file browser using the  button.

Job/Environment File Usage: The results .odb file location cannot be modified by the user. It will always be stored under *Project\output\<jobName>\Data Files*.

Specifying the FE procedure

Due to rules defined within the Abaqus modelling framework, general static procedures are not permitted directly after an Abaqus/Explicit procedure. By default, the ODB interface will attempt to append a static step to the model .odb file for fatigue results. Therefore, if the previous FEA step originated from an Abaqus/Explicit analysis, the user must indicate this directly.

Export Tool Usage: Set-up region: Check the option **Explicit FEA**.

Job File Usage: **EXPLICIT_FEA = {0.0 | 1.0}**

0. Write fatigue results to a standard results step (default)
1. Write fatigue results to an Abaqus/Explicit-compatible results step

Specifying the Abaqus API version

The user can specify which version of the Abaqus API will be used to create the results .odb file. This is done by specifying the Abaqus command line argument. The command line argument is the name of the batch file corresponding to the Abaqus version. These files are typically located in *<Abaqus_installation_directory>\Commands*.

Export Tool Usage: Set-up region: Fill out the **Abaqus command line argument** edit box.

Environment File Usage: **autoExport_abqCmd = 'abaqus-command'**

By default, the command '**abaqus**' is used. Assuming a standard Abaqus installation exists on the host machine, this argument points to the most recently installed Abaqus version.

If the installed Abaqus version is more recent than the model .odb file, the upgrade utility can be used to upgrade the file to a more recent version. The command line syntax for this is as follows:

```
>> <abaqus_command> upgrade -job <jobName> -odb <oldOdbFileName>
```

The user specifies whether to automatically upgrade the model .odb file.

Export Tool Usage: Set-up region: Check the option **Upgrade ODB file**.

Environment File Usage: `autoExport_upgradeODB = {0.0 | 1.0}`

0. Do not upgrade the model .odb file
1. Automatically upgrade the model .odb file (default—if applicable)

If the user specified to upgrade the .odb file, the ODB interface chooses the Abaqus version specified by the `autoExport_abqCmd` environment variable.

Specifying the part instance

Quick Fatigue Tool can usually only recognise stress datasets originating from a single part instance, since Abaqus may assign duplicate element-node numbers for each instance. Therefore, the part instance name must be specified to resolve potential ambiguity. Part instance names are inherited from the part itself. For example, if the part was named *BOLT*, then the part instance will be named *BOLT-n*. To check the name of the part instance, from Abaqus/Viewer, query an element on a region of the model from where stress data was exported. In the message window, the element type and corresponding part instance name is shown.

Export Tool Usage: Set-up region: Fill out the **ODB part instance name(s)** edit box.

Job File Usage: `PART_INSTANCE = 'part-instance-name'`

If the Abaqus job was run from a flat input file, there is only a single part instance in the output database called *PART-1-1*. If a flat input file was used, this name should be specified.

If the field data spans multiple part instances, these can be specified together.

Export Tool Usage: Set-up region: Fill out the **ODB part instance name(s)** edit box.
Separate part instance names with double quotes ("").

Job File Usage: `PART_INSTANCE = {'part-instance-1', ..., 'part-instance-n'}`

Specifying the results step name

The ODB interface can either append a new step to the results .odb file, or it can write fatigue results to an existing step that was created by the interface on a previous occasion.

Export Tool Usage: Set-up region: Choose either **Create a new step** or **Specify an existing QFT step** from the radio button selector. If a new step is being created, the name can be specified; otherwise a default name will be used. If an existing step is specified, the name of the step must be given.

The step name is specified by filling out the **Results step name** edit box.

Job File Usage: `STEP_NAME = 'step-name'`

Environment File Usage: `autoExport_stepType = {1.0 | 2.0}`

1. Export fatigue results to a new ODB step (default)
2. Export fatigue results to an existing ODB step

If a new step is chosen, the step name is optional. If no name is specified, Quick Fatigue Tool will choose a name automatically based on a combination of the job name and the part instance.

If an existing step is chosen, the name of the step must be specified. The following limitations apply:

- the step must have been written by the ODB interface on a previous occasion;
- the same field output variables must be written as when the step was created

Specifying the element result position

The result position is required because the ODB interface must tell the Abaqus API where on the element to write the fatigue results.

Export Tool Usage: Set-up region: Select the position from the **Result position** drop-down menu.

Job File Usage: `RESULT_POSITION = {'ELEMENT NODAL' | 'UNIQUE NODAL' | 'INTEGRATION POINT' | 'CENTROID'}`

If the result position is unknown, the ODB interface can choose the most suitable position based on the format of the field data.

Export Tool Usage: Set-up region: Check the option **Determine position from field IDs**

Environment File Usage: `autoExport_autoPosition = {0.0 | 1.0}`

0. User specifies the results position (default)
1. The ODB interface chooses the results position

It is possible for the code to determine the incorrect position, so the user should ensure that the selected result position matches the selection made when exporting the RPT file.

Figure 10.4 is an example of a plane stress element. If the result position is unique or element-nodal, data is written to the outer points of the element. Integration point data is written to the Gauss points in the element subsurface. Centroidal data is written to the geometric centre of each element.

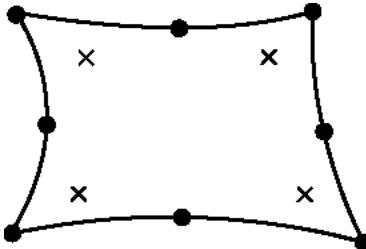


Figure 10.4: S8R Shell element with four integration points and nine nodes

Creating an element/node set

The ODB interface can create an element or node set in the results ODB file to facilitate easier results visualization in Abaqus/Viewer.

Export Tool Usage: Set-up region: Check the option **Create ODB element/node set for fatigue results** and fill out the edit box.

Environment File Usage: `autoExport_createODBSet = {0.0 | 1.0}`

0. Do not write an ODB element/node set to the `.odb` file (default)
1. Write an ODB element/node set to the `.odb` file

Environment File Usage: `autoExport_ODBSetName = 'ODB-set-name'`

The format of the ODB set is given as

Result position	ODB set type
Element-nodal	Node and element
Unique nodal	Node
Integration point	Element
Centroidal	Element

If an ODB set is written to the same ODB file by a subsequent analysis, the API will return the following error:

`OdbError: Duplicate set or surface name QFT_PART-1-1_QFT`

ODB sets can only be written to the output database if the selected field data file exactly matches the element-nodes for the selected part instance. ODB sets will not be written if more than one part instance name is specified.

Specifying how the python script is handled

The Export Tool works by writing a Python script containing all the instructions necessary to create a copy of the model ODB with an additional step containing the fatigue analysis result data. The script is submitted to the Python interpreter within the Abaqus/CAE framework, which then creates the output database. The process schematic is shown in Figure 10.5. The Python script used for the export operation may be retained, or the user can specify that only the python script should be written and it should not be submitted to the Abaqus API. In this case a results .odb file will not be created.

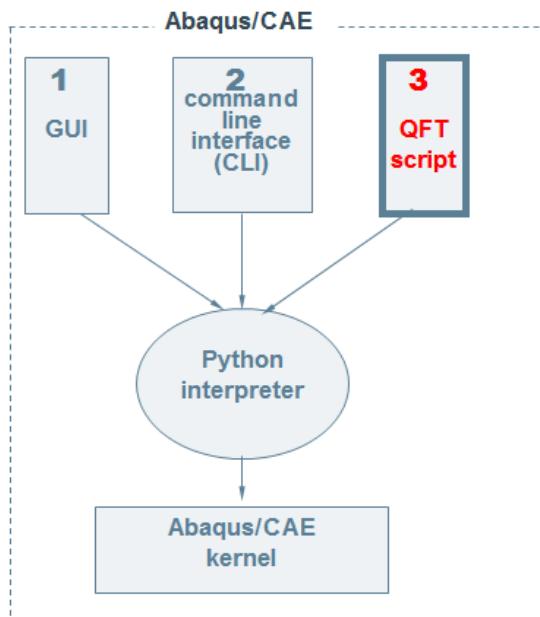


Figure 10.5: Interface between Quick Fatigue Tool and Abaqus/CAE

Export Tool Usage: Set-up region: Check the option **Retain python script after execution** or **Write python script only**. The options are mutually exclusive.

Environment File Usage: `autoExport_executionMode = {1.0 | 2.0 | 3.0}`

1. Export fatigue results and discard the python script (default)
2. Export fatigue results and retain the python script
3. Write the python script only—no results are exported

Specifying the field output variables

The user can choose which field output variables to export to the output database file.

Export Tool Usage: Output Variables region: Choose either **Select from list below**, **Preselected defaults** or **All** from the radio button selector.

If **Select from the list below** is selected, check the field output variables you wish to export to the output database file.

Environment File Usage: `autoExport_selectionMode = {1.0 | 2.0 | 3.0}`

1. Request output variables from a list
2. Request preselected defaults (default)
3. Request all available output variables

Environment File Usage: `autoExport_<field> = {0.0 | 1.0}`

0. Do not export the selected field variable
1. Export the selected field variable if available

Exporting yield criterion variables to the output database

If the user enabled the yield criterion assessment, the results can be exported to the results output database file. The user specifies the relevant field output variable in the environment file.

Export Tool Usage: Yield assessment output is not supported in Export Tool.

Environment File Usage: `autoExport_YIELD = {0.0 | 1.0}`

0. Do not export yield assessment results to the `.odb` file (default)
1. Export yield assessment results to the `.odb` file if available

10.5.5 Mismatching ODB files

While it is important that the source model matches the results data, Quick Fatigue Tool will attempt to match field data to corresponding elements and nodes in the model ODB, even if the finite element mesh is different. If the user selects the “Unique Nodal” or “Centroidal” options, Quick Fatigue Tool will write field data directly to the ODB based on the position labels provided in the field output file. If the user selects the “Element-Nodal” or “Integration Point” options, Quick Fatigue Tool will request the nodal connectivity matrix from the Abaqus API. This is required due to the fact that the order of the element-node or element-integration point data in the field output file will not necessarily match the order required in the ODB.

If the ODB interface finds nodes in the ODB file which cannot be found in the field data, that node and all others belonging to the same element will be ignored. If the Export Tool finds elements in the ODB which cannot be found in the field data, that element and all of its nodes or integration points will be ignored. As such, it is not necessary to export stress datasets from every element in the part instance when generating the RPT file(s), since the ODB interface will automatically ignore regions of the part instance which weren’t included in the fatigue analysis.

10.5.6 Large ODB files

For very large ODB files, it is unnecessary (and often impractical) to copy all of the FEA data to the results ODB. In cases where copying the ODB would consume a significant amount of time, the user is advised to create a *data check* ODB. These files contain the mesh from the FE model, but no FE results data. Hence, *data check* ODBs are more efficient as a container for fatigue analysis results.

Abaqus/CAE Usage: Expand the jobs container from the model tree. Right-click on the job and select **Data Check**.

Command Line Usage: *abaqus job=<jobName> datacheck*

10.5.7 Exporting field data to multiple Abaqus ODB part instances

Overview

Since Quick Fatigue Tool only uses the element-node numbers written to the field report file, it does not distinguish between individual part instances. Although it is possible to write field data to multiple part instances, the user must ensure that element-node numbers are not reused between instances, otherwise results will not be written to the output database correctly. There are four approaches for writing field data to multiple part instances:

1. Write multiple steps to the same *.odb* file, each step containing results at one part instance
2. Append field output from subsequent part instances to a previously created results step
3. Run the FEA from a flat input file
4. Manually set node numbering in Abaqus/CAE to enforce unique element-node numbers

Example: Write multiple steps to the same *.odb* file

If the user only has access to the *.odb* file and is unable to modify the attributes of the original model, field data may be written to multiple part instances by running a separate fatigue analysis job for each instance. Upon completion of each job, field data is written back to the previous result *.odb* file as a new step.

Individual part instances are selected in Abaqus/Viewer using the **Create Display Group** tool. In order to create the necessary stress dataset file, only the required part instance should be displayed in the viewport. This is achieved by selecting the required part instance and selecting **Replace**, as shown in Figure 10.6.

The stress dataset file is then created by following the steps outlined in Section 3.2, corresponding to the first part instance of interest. For the first job, the Abaqus ODB options are configured in the job file.

Job File Usage:

```
DATASET = 'dataset-A.rpt'
OUTPUT_DATABASE = '<absolutePath>\model-ODB.odb'
PART_INSTANCE = 'part-instance-name-A'
STEP_NAME = 'step-name-A'
```

A second job is run for the dataset corresponding to the second part instance of interest. The model output database is specified as the results output database from the previous analysis and the name of the second part instance is specified.

Job File Usage:

```
DATASET = 'dataset-B.rpt'
OUTPUT_DATABASE = '<absolutePath>\results-ODB.odb'
PART_INSTANCE = 'part-instance-name-B'
STEP_NAME = 'step-name-B'
```

This will append a new step to the existing results output database containing field data for the second part instance. The **STEP_NAME** option adds an additional string to the name of the results step and is

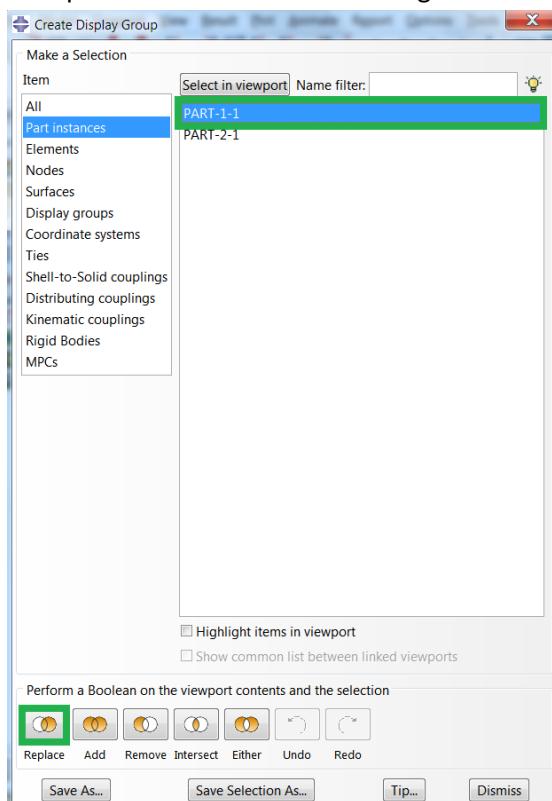


Figure 10.6: Create display group tool showing how to replace the viewport contents with the required part instance

not compulsory. However, adding a step name for each analysis minimizes the risk of Quick Fatigue Tool attempting to create two steps with the same name in a single .odb file, which would result in an error.

The advantage of writing field output to individual steps is that the field output variable selection can differ between part instances. However, the drawback of this method is that it is not possible to view a field over all part instances simultaneously; rather, the user must switch between steps in order to view results at different regions in the model.

Example: Append field output from subsequent part instances to a previously created results step

An alternative to the previous method is to append results from subsequent analyse to a results ODB step created from a previous analysis. Output is written to the first part instance.

Job File Usage:

```
DATASET = 'dataset-A.rpt'  
OUTPUT_DATABASE = '<absolutePath>\model-ODB.odb'  
PART_INSTANCE = 'part-instance-name-A'  
STEP_NAME = 'step-name-A'
```

The analysis is then repeated for the second part instance.

Job File Usage:

```
DATASET = 'dataset-B.rpt'  
OUTPUT_DATABASE = '<absolutePath>\results-ODB.odb'  
PART_INSTANCE = '<partName_B>'  
STEP_NAME = '<stepName>'
```

In this case, the path to the output database is defined as the path to the results output database from the first analysis. The step names must match between the two models. By default, Quick Fatigue Tool creates a new step for the results data. Therefore, in order to instruct the program to append results to the previous step, the following environment variable must be set in the environment file.

Environment File Usage: `autoExport_stepType` = 2.0

The following caveats must be observed when appending output to an existing step:

- The previous step must have been written by Quick Fatigue Tool
- The fields must exactly match those written in the original step, otherwise the Python API will exit with errors

The advantage of appending results data to a previous step is that fields can be visualized over all part instances simultaneously, without having to switch between steps. The results position of the elements does not have to be the same between part instances, although exporting data at different positions will result in duplicate field variables being written to the ODB frame. Results which are appended may also overwrite results which were written during previous analyses.

Example: Run the FEA from a flat input file

The alternative solution is to avoid writing individual part instances and assemblies to the input file. This is achieved by selecting **Do not use parts and assemblies in input files** in the **Edit Model Attributes** dialogue in Abaqus/CAE.

After the analysis, Quick Fatigue Tool will write to the message file the number of regions found in the **.rpt** file.

10.5.8 Minimum requirement for output

The Abaqus API imposes minimum requirements for the amount of field data which must be written to a given result frame. These requirements are detailed in the table below:

Result position	Requirement
Element-nodal	All the nodes defining a single element
Unique Nodal	At least two nodes
Integration Point	All the integration points defining a single element
Centroid	At least two centroids

Failure to adhere to above requirements will cause the ODB interface to exit with the following error message:

Element-nodal/Integration Point	Error: No matching position labels were found from the model output database
Unique Nodal/Centroid	ODBgetSeqSeqDoubleFromArray() num dims (1) != 2

11. FEA Modelling techniques

11.1 Background

This section applies to users who wish to use stress datasets from a finite element analysis. Detailed guidance and illustrations focus on SIMULIA Abaqus, but the concepts can be applied to any FEA package.

11.2 Preparing an FE model for fatigue analysis

Consider the model in Figure 11.1. We wish to determine the fatigue life of the shaft when subject to a fully-reversed bending load. The first consideration is whether it is viable to analyse the whole model. The shaft is made from 56,720 elements and 60,867 nodes. Analyses with Quick Fatigue Tool can be cumbersome for models containing more than a few thousand nodes, so it would be unadvisable to analyse every element. At this stage, the user should ask the following questions:

- a) Is analysing the whole FE model reasonable considering the size of the mesh?

If the answer to a) is no, then ask the following question:

- b) Are the stresses concentrated at a particular location? i.e. are there regions in the model where the stresses are clearly non-damaging?

If the answer to b) is yes, then reduce the number of elements for analysis

In the example of the shaft model, the bending load has resulted in a stress concentration at one of the fillet radii, so it is only necessary to analyse this region.

Using the Display Group Manager, the elements at the notch containing the stress concentration are isolated, shown by Figure 11.2. The model could be cut once more in the x-z plane to exploit the symmetry of the result. However, for this example the full notch geometry will be considered.

Quick Fatigue Tool defines “failure” as the complete propagation of a crack across a material surface defined by critical plane analysis. In most cases, these cracks initiate on the component surface, therefore subsurface elements may be excluded in cases where the stress gradient is sufficiently large. Subsurface elements are excluded from the analysis by creating a new display group and choosing to select elements “by angle”, shown in Figure 11.3. The final element set for the analysis is shown in Figure 11.4.

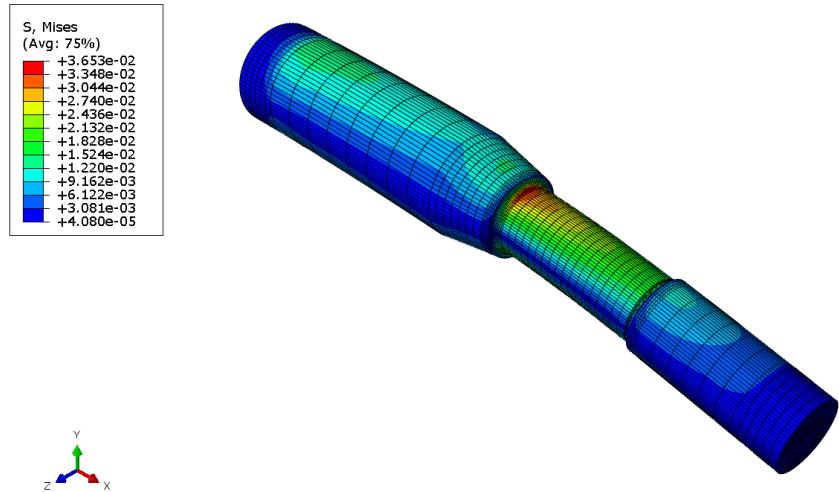


Figure 11.1: SAE shaft model in bending

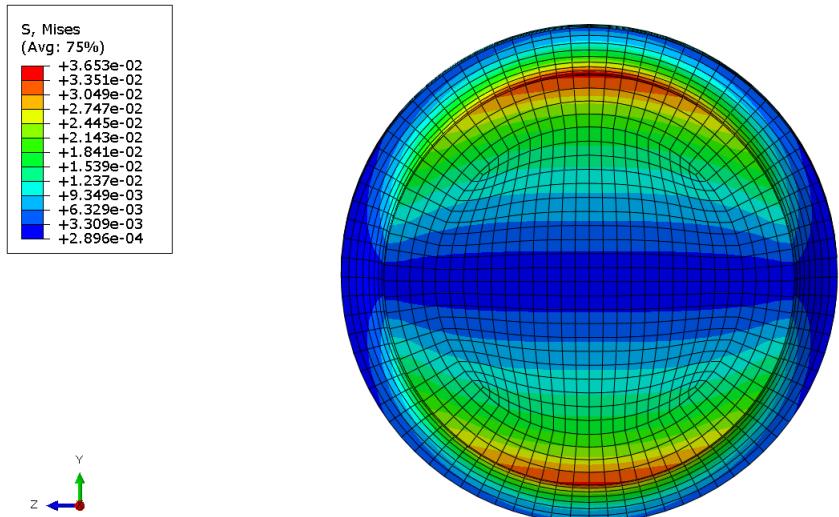
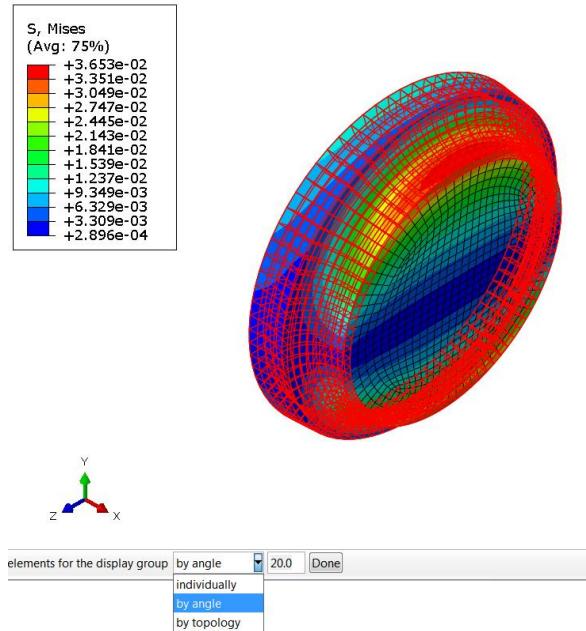


Figure 11.2: Isolating the stress concentration

Before exporting the stress tensors for analysis, it is recommended that result averaging is turned off so that the stresses written to the *.rpt* file are a closer representation of the calculated gauss point values. From the main menu bar, go to **Result → Options...** and make sure that “Average element output at nodes” is unchecked (Figure 11.5).



11.3: Picking surface elements by angle

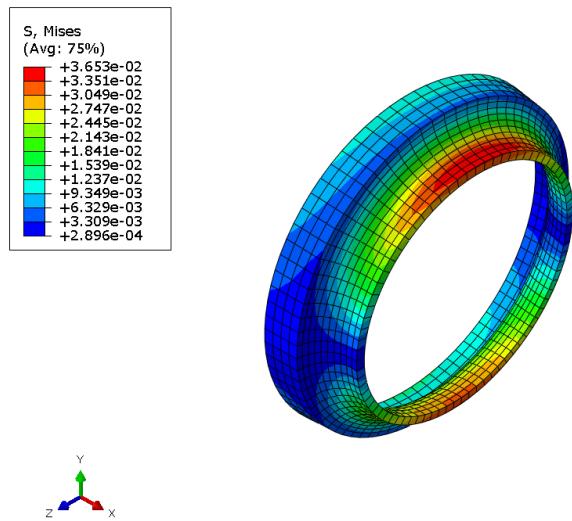


Figure 11.4: Final element analysis group

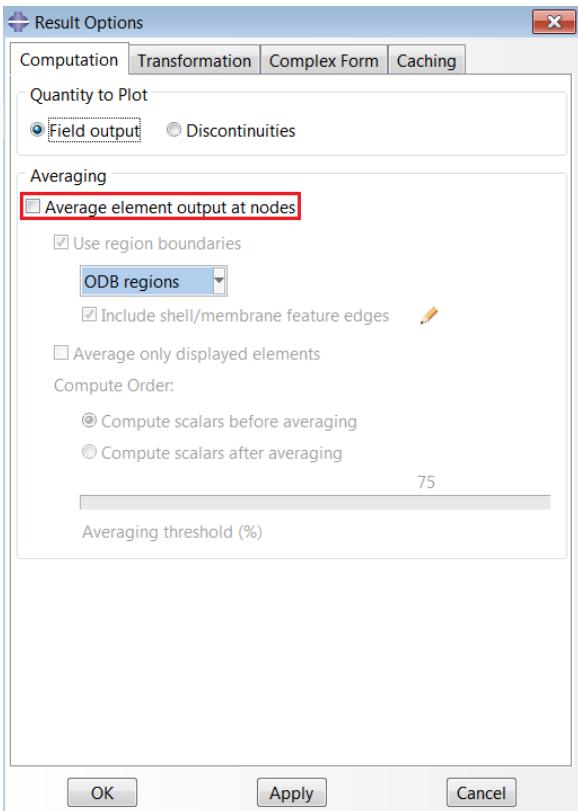


Figure 11.5: Result options dialogue in Abaqus/CAE

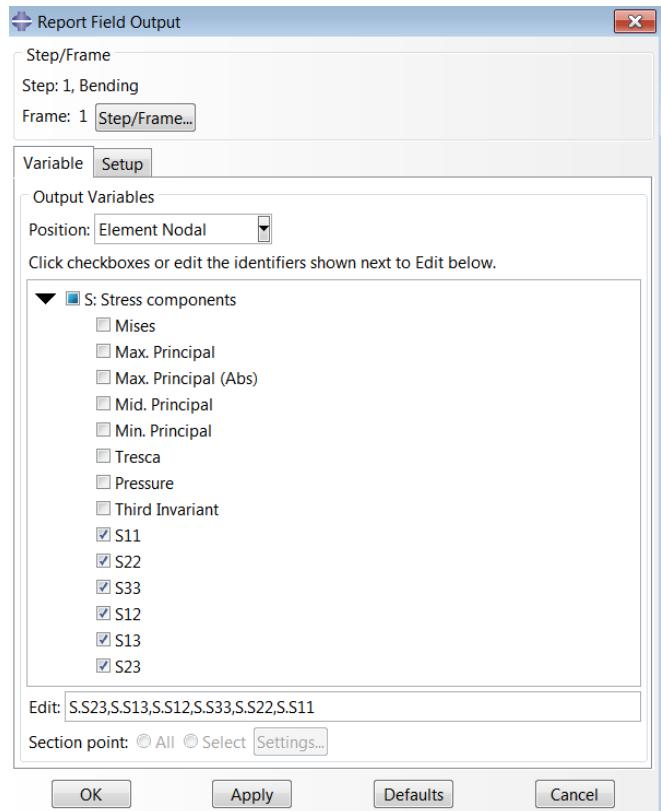


Figure 11.6: Selecting stresses for the .rpt file

The stresses are then exported by going to **Report -> Field Output...** and choosing the stress components, as shown in Figure 11.6. The user has the option to select the position for the field output. The choices of positions compatible with Quick Fatigue Tool are described below.

Position	Description
Integration Point	<ul style="list-style-type: none"> • “True” stress solution • Size of output depends on the integration order • Produces accurate results for brittle materials where crack initiation is in the element sub-surface • Not recommended for ductile metals
Centroid	<ul style="list-style-type: none"> • Single, averaged value at the geometric centre of each element • Size of output depends only on the size of the mesh. Generates the least amount of output • Good for quick fatigue estimates • Offers the worst solution accuracy
Element Nodal	<ul style="list-style-type: none"> • Un-averaged result for each node of each element (nodes belonging to N elements have N solutions) • Size of output depends on the element geometric order • Produces accurate results for ductile materials where crack initiation is on the element free surface • Not recommended for brittle metals
Unique Nodal	<ul style="list-style-type: none"> • Single, averaged value at each node • Size of output depends on the element geometric order • Recommended for ductile metals • Offers slightly shorter analysis time than Element Nodal at the expense of some solution accuracy

If the stresses are written using the Unique Nodal position, the number of nodes written for analysis for the example shaft model is 2,240. Since this is approximately 27 times less nodes than originally present, the expected analysis should also be approximately 27 times faster.

12. Supplementary analysis procedures

12.1 Background

Quick Fatigue Tool includes a set of validation procedures to assess the static strength of components undergoing either static or varying stress histories:

- yield criteria for homogeneous metallic components; and
- failure and damage initiation criteria for fibre-reinforced composites.

The yield and composite criteria assessments are mutually exclusive procedures and thus cannot both be specified in the same job.

Yield criteria

The yield criterion assessment is used to check for the onset of permanent deformation of statically-loaded structures where a fatigue analysis would usually not be required. However, the assessment can be performed in tandem with a fatigue analysis.

If only the yield assessment is required without subsequent fatigue analysis, the user can run the job as a data check analysis.

Job File Usage: **DATA_CHECK** = 1.0

The yield criterion assessment is intended to address small-scale yielding—gross yielding of load-bearing sections must be addressed with finite element analysis techniques.

Failure and damage initiation criteria for fibre-reinforced composites

The composite criteria assessment is used to check for failure and damage initiation of fibre-reinforced composites loaded in plane stress.

The orthotropic plane stress failure measures and damage initiation criteria:

- provide a “pass-fail” result only;
- cannot predict progressive damage or fatigue failure; and
- take values that are greater than or equal to 0.0, with values that are greater than or equal to 1.0 implying failure or damage initiation.

The composite criteria assessment cannot be performed in tandem with a fatigue analysis.

12.2 Yield criteria

12.2.1 Overview

The yield criteria assessment determines whether permanent deformation is likely to develop based on the specified loading. The user specifies the yield criterion in the job file.

Job File Usage: **YIELD_CRITERION** = {0.0 | 1.0 | 2.0 | 3.0 | 4.0}

0. Do not perform yield criterion assessment (default)
1. Total strain energy theory
2. Maximum shear strain energy theory
3. Tresca yield criterion
4. von Mises yield criterion

12.2.2 Material properties

Proof stress

The yield criteria compare the general state of stress in the loading to the proof stress, σ_y . This is usually the yield point of the material in uniaxial tension and is often measured by convention using the 0.2% offset strain method.

For the total strain energy theory and the maximum shear strain energy theory criteria, the proof stress is expressed in terms of the tensile strain limit energy, σ_y^2 .

The user specifies the proof stress in the material definition.

Material Manager Usage: Mechanical region of the *Material Editor* dialogue: Check the Proof Stress property and fill out the edit box.

Text File Usage: *MECHANICAL
First (and only) line: , , , σ_y

Poisson's Ratio

The total strain energy theory yield criterion requires a value of Poisson's ratio, ν .

Material response

Prior to yield, the material response can be assumed to be either linear elastic or nonlinear elastic. The user specifies the material response in the environment file.

Environment File Usage: **materialResponse** = {1.0 | 2.0}

1. Linear elastic (default)
2. Nonlinear elastic

If cyclic the stress-strain material properties are not available, a linear elastic response is assumed by default.

The nonlinear elastic response accounts for the effect of hysteresis and material memory. Therefore, the instantaneous elastic-plastic stress is a function of all previous stress states in the history. Stress hardening due to small amounts of ratcheting may result in delayed yielding as the yield surface is shifted along the σ -axis.

For yield criteria with a nonlinear elastic material response, the following material properties are required in addition to fatigue properties:

Symbol	Definition
E	Young's Modulus
K'	Cyclic strain-hardening coefficient
n'	Cyclic strain-hardening exponent

Treatment of compressive stresses

If the user specifies the option **No damage in compression** (*NO COMPRESSION) in the material definition, then compressive values in the stress history are reset to zero.

12.3.3 Model definition

Stress datasets are created in the same fashion as a standard fatigue analysis. Instructions on creating stress datasets are found in Section 3.2.

12.3.4 Loading definition

The loading definition is created in the same fashion as a standard fatigue analysis. All of the available loading methods in Quick Fatigue Tool are described in Section 3.1.

Static loading

For static loading problems, the load history is expected to be a single point. The load history must consist of at least two points. Therefore, if the loading is static the load history can be defined as follows.

Job File Usage: **DATASET** = '*dataset-file-name.**'
HISTORY = [0, 1]

This guarantees that the first point in the load history cannot cause yield, and only the second "static" point is considered.

Dynamic loading

For dynamic loading problems, the load history is defined by two or more points. Quick Fatigue Tool evaluates the yield criterion at every point in the load history to determine if yield occurs.

Dynamic load histories may cause the yield surface to evolve. Therefore, it is recommended that for dynamic loading problems the nonlinear elastic material response is specified.

Environment File Usage: **materialResponse** = 2.0

12.2.5 Total strain energy theory

The total strain energy theory is based on work by Beltrami and Haigh [49]. The stored energy associated with elastic deformation at the point of yield is independent of the specific stress tensor. Thus, yield occurs when the strain energy density is greater than the strain energy at the elastic limit in simple tension. For the three-dimensional stress state this is given by Equation 12.2.1:

$$I_Y = \sigma_1^2 + \sigma_2^2 + \sigma_3^2 - 2v(\sigma_1\sigma_2 + \sigma_2\sigma_3 + \sigma_1\sigma_3) < \sigma_y^2 \quad [12.2.1]$$

The onset of permanent deformation occurs when the equivalent total strain energy density exceeds the tensile strain limit energy.

The total strain energy theory is recommended for ductile materials [50].

12.2.6 Maximum shear strain energy theory

The maximum shear strain energy theory states that yield occurs when the shear strain energy in the actual case exceeds the shear strain energy in a simple tension test [51]. For the three-dimensional stress state this is given by Equation 12.2.2:

$$I_Y = 0.5 \times [(\sigma_1 - \sigma_2)^2 + (\sigma_2 - \sigma_3)^2 + (\sigma_1 - \sigma_3)^2] < \sigma_y^2 \quad [12.2.2]$$

The onset of permanent deformation occurs when the equivalent shear strain energy density exceeds the tensile strain limit energy.

The shear strain energy theory is more conservative than the total strain energy theory.

12.2.7 Tresca yield criterion

The Tresca yield criterion—otherwise known as the maximum shear stress theory—describes the yield surface according to Equation 12.2.3.

$$I_Y = \max(|\sigma_1 - \sigma_2|, |\sigma_2 - \sigma_3|, |\sigma_3 - \sigma_1|) < \sigma_y \quad [12.2.3]$$

The onset of permanent deformation occurs when the maximum shear stress on an element reaches the same level exhibited in an element in a uniaxial tensile test at the yield point [52].

The Tresca yield criterion provides the most conservative estimate of permanent deformation.

12.2.8 von Mises yield criterion

The von Mises yield criterion—otherwise known as the maximum distortion energy criterion—describes the yield surface according to Equation 12.2.4.

$$I_Y = \sqrt{\sigma_1^2 + \sigma_2^2 + \sigma_3^2 - \sigma_1\sigma_2 - \sigma_2\sigma_3 - \sigma_3\sigma_1} < \sigma_y \quad [12.2.4]$$

The onset of permanent deformation occurs when the distortion energy on an element reaches the same level exhibited in an element in a uniaxial tensile test at the yield point [53].

Norton describes the distortion energy as the uniaxial tensile stress that would create the same distortion energy as is created by the actual combination of applied stresses [54].

The von Mises yield criterion is non-conservative compared to the Tresca yield criterion.

12.2.9 Output

Output yield indices

The yield measure, I_Y , is first normalised by the tensile strain limit energy. It is then quoted in terms of the yield index R , so that for the given principal stress state $\{\sigma_1, \sigma_2, \sigma_3\}$:

$$\left\{ \frac{\sigma_1}{R}, \frac{\sigma_2}{R}, \frac{\sigma_3}{R} \right\} \rightarrow I_Y = 1.0 \quad [12.2.5]$$

That is, $1/R$ is the scaling factor by which all of the stress components must be simultaneously multiplied to hit the yield surface. The term $1/R$ can be used with the **SCALE** option in order to make $R = 1$ for the specified yield criterion.

The calculation of R depends on the yield criterion, and is summarised according to the following table:

Yield criterion	Yield index, R
• Total strain energy theory	$\sqrt{I_Y}$
• Maximum shear strain energy theory	$\frac{\sqrt{I_Y}}{\sigma_y}$
• Tresca yield criterion	$\frac{I_Y}{\sigma_y}$
• von Mises yield criterion	

The values of R should be interpreted according to the following table:

Range of values of R	Meaning
$0 \leq R < 1$	Structure has not yielded
$R \geq 1$	Structure has yielded

Yield assessment criteria analysis field variable identifiers

The criterion variable names take their meaning from the following table:

Yield index, R	Definition
YIELD	Status flag: 0: The item has not yielded 1: The item has yielded -1: The yield assessment was not requested -2: Yield criterion could not be evaluated
VMCRT	von Mises criterion yield index
TRCRT	Tresca criterion yield index
SSCRT	Maximum shear strain energy theory yield index
TSCRT	Total strain energy theory yield index
PEEQ	Normalised equivalent plastic strain energy density for all yield criteria

Normalised equivalent plastic strain energy density

The normalised equivalent plastic strain energy density, **PEEQ**, is the yield index relative to the yield envelope when $R \geq 1$, and is calculated according to Equation 12.2.6:

$$PEEQ = \begin{cases} 0, & R \leq 1 \\ R^2 - 1, & R \geq 1 \end{cases} \quad [12.2.6]$$

The value of **PEEQ** is the amount of strain energy in excess of the tensile strain limit energy, and is expressed in terms of the yield criterion according to the following table:

Yield criterion	Calculation of PEEQ
• Total strain energy theory	$\frac{I_Y}{\sigma_Y^2} - 1$
• Maximum shear strain energy theory	$\frac{I_Y^2}{\sigma_Y^2} - 1$
• Tresca yield criterion	
• von Mises yield criterion	

Output files

Field output

The results for each analysis item are written to *Project\output\<job-name>\Data Files\yield_assessment.dat*. This file contains the value of the specified yield criterion for each item in the model.

The **YIELD** field output variable is copied to the field output file *Project\output\<job-name>\Data Files\f-output-all.dat*.

Whole model summary

In addition to tabulated results, a whole model summary of the analysis is written to the message file, indicating the number of items in the model which have yielded and the worst value of the active yield index.

Exporting to an Abaqus .odb file

Quick Fatigue Tool can automatically write the results of a yield criterion assessment to an Abaqus output database (.odb) file.

The user enables the ODB interface in the environment file and specifies the model output database in the job file.

Environment File Usage: `autoExportODB = 1.0`

Job File Usage: `OUTPUT_DATABASE = 'model-odb-file-name.odb'`

Quick Fatigue Tool automatically writes yield assessment results to the output database by default. If manual field output selection is enabled, the user must specify the relevant field output variable in the environment file.

Environment File Usage: `autoExport_YIELD = 1.0`

Export Tool Usage: Yield assessment output is not supported in Export Tool.

If the yield criterion assessment is run during a data check analysis, the results of the assessment are still written to the output database at the end of the analysis.

The ODB interface is discussed in Section 10.5.

12.3 Failure and damage initiation criteria for fibre-reinforced composites

12.3.1 Overview

Composite failure criteria can be used to assess the strength of fibre-reinforced composite laminate components subject to biaxial loading. The criteria have also been extended to the analysis of closed cell PVC foam.

A composite can fail in two basic failure modes:

1. Failure of individual plies in tension, compression, or shear (fibre and matrix)
2. Delamination between plies

These composite failure modes are illustrated by Figure 12.1. Quick Fatigue Tool is able to assess the first mode of failure.

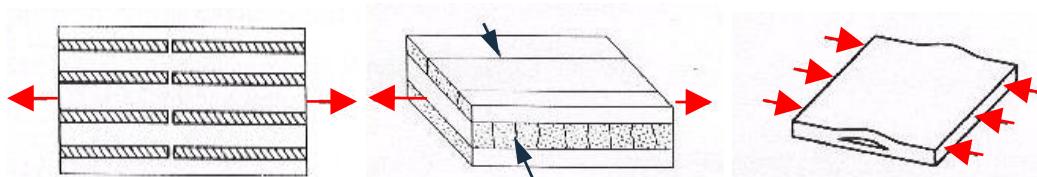


Figure 12.1: (L-R) Fibre, matrix and delamination failure

The user specifies composite the criteria assessment in the job file.

Job File Usage: **COMPOSITE_CRITERIA** = {0.0 | 1.0}

0. Do not evaluate failure/damage initiation criteria (default)
1. Evaluate failure/damage initiation criteria based on available material data

12.3.2 Conventions for fibre-reinforced composites

The composite failure criteria assessment assumes that the loading is in a state of orthotropic plane stress. The ply directions are given by Equation 12.3.1.

$$\boldsymbol{\sigma} = \begin{bmatrix} \sigma_{11} & \sigma_{12} & \sigma_{13} \\ \sigma_{21} & \sigma_{22} & \sigma_{23} \\ \sigma_{31} & \sigma_{32} & \sigma_{33} \end{bmatrix} \equiv \begin{bmatrix} \sigma_{fibre} & \sigma_{shear} & 0 \\ \sigma_{shear} & \sigma_{transverse} & 0 \\ 0 & 0 & 0 \end{bmatrix} \quad [12.3.1]$$

Quick Fatigue Tool requires stress components σ_{11} , σ_{22} and σ_{12} for the fibre, transverse and shear directions, respectively.

12.3.3 Conventions for closed cell PVC foam

For closed-cell PVC foam, the stress components σ_{11} and σ_{22} represent the in-plane directions, while the stress components σ_{33} and σ_{23} represent the through-thickness of the foam.

12.3.4 Material properties

Stress-based failure criteria

For stress-based failure criteria, the following material properties are required:

Symbol	Definition
X_T	Tensile stress (11-direction)
X_C	Compressive stress (11-direction)
Y_T	Tensile stress (22-direction)
Y_C	Compressive stress (22-direction)
Z_T	Tensile stress (33-direction)
Z_C	Compressive stress (33-direction)
S	Shear strength (12-plane)
f_{12}^*	Cross-product coefficient (12-plane)
f_{23}^*	Cross-product coefficient (23-plane)
$\sigma_{biax,12}$	Stress limit (12-plane)
$\sigma_{biax,23}$	Stress limit (23-plane)

Maximum strain failure theory

For the maximum strain failure theory, the following material properties are required:

Symbol	Definition
$X_{\varepsilon T}$	Tensile strain (fibre direction)
$X_{\varepsilon C}$	Compressive strain (fibre direction)
$Y_{\varepsilon T}$	Tensile strain (transverse direction)
$Y_{\varepsilon C}$	Compressive strain (transverse direction)
S_ε	Shear strain
E_{11}	Modulus of elasticity (fibre direction)
E_{22}	Modulus of elasticity (transverse direction)
G_{12}	Shear modulus (12-plane)

Hashin's damage initiation theory

For Hashin's damage initiation theory, the following material properties are required:

Symbol	Definition
α	Shear influence parameter
X_{hT}	Longitudinal tensile strength
X_{hc}	Longitudinal compressive strength
Y_{hT}	Transverse tensile strength
Y_{hc}	Transverse compressive strength
S_{hx}	Longitudinal shear strength
S_{hy}	Transverse shear strength

LaRC05 damage initiation theory

For the LaRC05 damage initiation theory, the following material properties are required:

Symbol	Definition
X_{LT}	Longitudinal tensile strength
X_{LC}	Longitudinal compressive strength
Y_{LT}	Transverse tensile strength
Y_{LC}	Transverse compressive strength
S_{LX}	Longitudinal shear strength
S_{LY}	Transverse shear strength
$G_{L,12}$	Shear modulus (12-plane)
η_L	Longitudinal slope coefficient
η_T	Transverse slope coefficient
α_0	Fracture plane angle for pure compression
ϕ^0	Initial fibre misalignment angle

Defining composite material properties

Detailed guidance on creating composite material data can be found in Section 5.9. A folder containing several example materials using composite properties from Pinho et al. [55] can be found in *Data\material\composites*.

Treatment of compressive stresses

If the user specifies the option **No damage in compression** (*NO COMPRESSION) in the material definition, then compressive values in the stress history are reset to zero. This setting is not recommended for composite criteria analysis.

12.3.5 Model definition

Stress datasets

Stress datasets are created in the same fashion as a standard fatigue analysis. Instructions on creating stress datasets are found in Section 3.2.

If the stress datasets originate from an Abaqus output database (.odb) file, then it is important to ensure that stresses are reported only at the location of interest. By default, Abaqus will write stresses for all available section points. However, Quick Fatigue Tool can only read stresses from a single location.

For composite analysis, stress dataset (.rpt) files from Abaqus/Viewer are created as follows:

1. Make sure that the regions of the model you wish you export are displayed in the active viewport
2. From the main menu in Abaqus/Viewer, select **Report → Field Output...**
3. From the Output Variables region of the Report Field Output dialogue, select **Select** as the section point, then select **Settings....**
4. From the Field Report Section Point Settings dialogue that appears, select either **Categories** or **Plies** as the selection method. For composite structures, selecting results by ply is usually more convenient
5. From the Categories/Plies region, select the location/section point, or the ply/result location of interest, then select **OK**

Stress datasets are not required if the composite failure analysis is defined with uniaxial histories.

Element type

Composite structures often utilise plane stress elements such as conventional and continuum shells. In such cases, the treatment of plane stress elements must be indicated in the job file.

Job File Usage: **ELEMENT_TYPE = {0.0 | 1.0}**

0. Do not allow plane stress elements (default)
1. Allow plane stress element assumption

12.3.6 Loading definition

The loading definition is created in the same fashion as a standard fatigue analysis. All of the available loading methods in Quick Fatigue Tool are described in Section 3.1. The load history can have a length of one.

The composite failure assessment is intended for static problems where only the maximum stress is of interest, thus the load history is usually expected to be a single point. If the user defines a load history with more than one point, the criteria are evaluated separately for each point in the history. The quoted value for each criteria is the maximum value computed over the entire load history for each analysis item. Cycle counting is not employed for composite failure assessment.

Composite failure criteria are only compatible with linear elastic stress datasets.

12.3.7 Maximum stress theory

The maximum stress theory predicts composite failure by comparing each stress component to its respective strength, according to Equation 12.3.2.

$$I_F = \max\left(\frac{\sigma_{11}}{X}, \frac{\sigma_{22}}{Y}, \left|\frac{\sigma_{12}}{S}\right|\right) < 1 \quad [12.3.2]$$

The value of the tensile (fibre) strength, X , and transverse strength, Y , is determined according to Equations 12.3.3-4.

$$\sigma_{11} > 0 \rightarrow X = X_T; \sigma_{11} \leq 0 \rightarrow X = X_C \quad [12.3.3]$$

$$\sigma_{22} > 0 \rightarrow Y = Y_T; \sigma_{22} \leq 0 \rightarrow Y = Y_C \quad [12.3.4]$$

This theory does not account for the interaction between the stress components in different directions. Therefore, its accuracy is limited.

12.3.8 Tsai-Hill theory

Tsai proposed a composite failure criterion as an extension of Hill's anisotropic plasticity model [56]. This simple model defines the failure criterion as a function of the three stress components according to Equation 12.3.5.

$$I_F = \max\left(\frac{\sigma_{11}^2}{X^2} - \frac{\sigma_{11}\sigma_{22}}{X^2} + \frac{\sigma_{22}^2}{Y^2} + \frac{\sigma_{12}^2}{S^2}\right) < 1 \quad [12.3.5]$$

The value of the tensile (fibre) strength, X , and transverse strength, Y , is determined according to Equations 12.3-4.

The Tsai-Hill theory is a piecewise-continuous failure surface, shown by Figure 12.2.

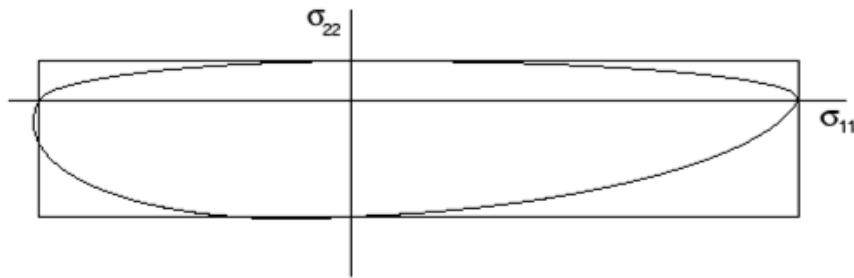


Figure 12.2: Tsai-Hill piecewise-continuous failure surface

12.3.9 Tsai-Wu theory for fibre-reinforced composites

The Tsai-Wu composite failure criterion is a smooth form of the Tsai-Hill criterion [57]. The direct and shear stress is coupled by the Tsai-Wu by the parameter F_{12} , which is a function of either the cross-product coefficient, f_{12}^* or the stress limit, $\sigma_{biax,12}$.

The criterion is defined according to Equation 12.3.6.

$$I_F = F_1\sigma_{11} + F_2\sigma_{22} + F_{11}\sigma_{11}^2 + F_{22}\sigma_{22}^2 + F_{66}\sigma_{12}^2 + 2F_{12}\sigma_{11}\sigma_{22} < 1 \quad [12.3.6]$$

The Tsai-Wu coefficients are defined according to Equation 12.3.7.

$$F_1 = \frac{1}{X_T} + \frac{1}{X_C}, F_2 = \frac{1}{Y_T} + \frac{1}{Y_C}, F_{11} = -\frac{1}{X_T X_C}, F_{22} = -\frac{1}{Y_T Y_C}, F_{66} = \frac{1}{S^2} \quad [12.3.7]$$

If the stress limit, $\sigma_{biax,12}$, is defined, the coupling parameter, F_{12} , is given by Equation 12.3.8.

$$F_{12} = \frac{1}{2\sigma_{biax,12}^2} \left[1 - \left(\frac{1}{X_T} + \frac{1}{X_C} + \frac{1}{Y_T} + \frac{1}{Y_C} \right) \sigma_{biax,12} + \left(\frac{1}{X_T X_C} + \frac{1}{Y_T Y_C} \right) \sigma_{biax,12}^2 \right] \quad [12.3.8]$$

If the stress limit is not defined, the coupling parameter is defined by Equation 12.3.9.

$$F_{12} = f_{12}^* \sqrt{F_{11} F_{22}}, \{-1 \leq f_{12}^* \leq 1\} \quad [12.3.9]$$

The default value of f_{12}^* is zero. If $F_{12} = 0$, the Tsai-Wu failure surface is given by Figure 12.3.

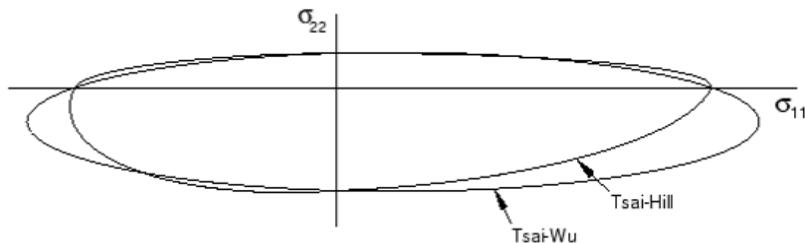


Figure 12.3: Tsai-Wu piecewise-continuous failure surface ($F_{12} = 0$)

12.3.10 Tsai-Wu theory for closed cell PVC foam

The Tsai-Wu failure criterion has been applied to closed cell PVC foam under plain strain conditions, according to Equation 12.3.10.

$$I_F = F_2\sigma_{22} + F_3\sigma_{33} + F_{22}\sigma_{22}^2 + F_{33}\sigma_{33}^2 + 2F_{23}\sigma_{22}\sigma_{33} < 1 - k^2 \quad [12.3.10]$$

The Tsai-Wu coefficients are defined according to Equation 12.3.11.

$$F_3 = \frac{1}{Z_T} + \frac{1}{Z_C}, F_{33} = \frac{1}{Z_T Z_C}, k = \frac{\sigma_{12}}{\sigma_{13}} \quad [12.3.11]$$

If the stress limit, $\sigma_{biax,23}$, is defined, the coupling parameter, F_{12} , is given by Equation 12.3.12.

$$F_{23} = \frac{1}{2\sigma_{biax,23}^2} \left[1 - \left(\frac{1}{Y_T} + \frac{1}{Y_C} + \frac{1}{Z_T} + \frac{1}{Z_C} \right) \sigma_{biax,23} + \left(\frac{1}{Y_T Y_C} + \frac{1}{Z_T Z_C} \right) \sigma_{biax,23}^2 \right] \quad [12.3.12]$$

If the stress limit is not defined, the coupling parameter is defined by Equation 12.3.13.

$$F_{23} = f^{*}_{23} \sqrt{F_{22} F_{33}}, \{-1 \leq f^{*}_{23} \leq 1\} \quad [12.3.13]$$

The default value of f^{*}_{23} is zero.

The foam is assumed to be in a state of plane strain, such that $\varepsilon_{33} = 0$.

12.3.11 Azzi-Tsai-Hill theory

The Azzi-Tsai-Hill composite failure criterion is the same as the Tsai-Hill theory, except that the absolute value of the cross-product term is used, according to Equation 12.3.14.

$$I_F = \max \left(\frac{\sigma_{11}^2}{X^2} - \left| \frac{\sigma_{11}\sigma_{22}}{X^2} \right| + \frac{\sigma_{22}^2}{Y^2} + \frac{\sigma_{12}^2}{S^2} \right) < 1 \quad [12.3.14]$$

The Azzi-Tsai-Hill failure surface is given by Figure 12.3.4.

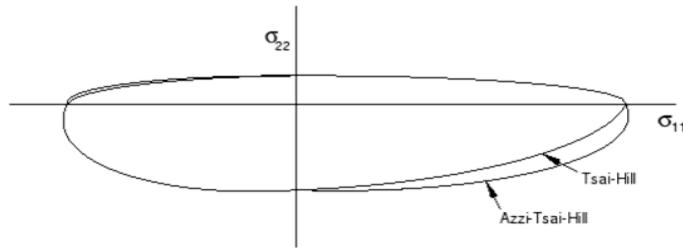


Figure 12.4: Azzi-Tsai-Hill piecewise-continuous failure surface

12.3.12 Maximum strain failure theory

The maximum failure strain theory predicts composite failure by comparing each strain component to its respective strain limit, according to Equation 12.3.15.

$$I_F = \max \left(\frac{\varepsilon_{11}}{X_\varepsilon}, \frac{\varepsilon_{22}}{Y_\varepsilon}, \left| \frac{\gamma_{12}}{S_\varepsilon} \right| \right) < 1 \quad [12.3.15]$$

The value of the tensile (fibre) strain, X_ε , and transverse strain, Y_ε , is determined according to Equations 12.3.16-17.

$$\varepsilon_{11} > 0 \rightarrow X_\varepsilon = X_{\varepsilon T}; \varepsilon_{11} \leq 0 \rightarrow X_\varepsilon = X_{\varepsilon C} \quad [12.3.16]$$

$$\varepsilon_{22} > 0 \rightarrow Y_\varepsilon = Y_{\varepsilon T}; \varepsilon_{22} \leq 0 \rightarrow Y_\varepsilon = Y_{\varepsilon C} \quad [12.3.17]$$

Experimental data suggests that the maximum failure strain theory is not as accurate as the maximum stress theory.

The maximum failure strain theory assumes that the stresses in the composite are linear elastic until $I_F = 1$. The elastic stress is converted into elastic strain using the shear modulus, $G_{\varepsilon,12}$, according to Equation 12.3.18.

$$\varepsilon_{11} = \frac{\sigma_{11}}{E_{11}}, \varepsilon_{22} = \frac{\sigma_{22}}{E_{22}}, \gamma_{12} = \frac{\sigma_{12}}{G_{\varepsilon,12}} \quad [12.3.18]$$

12.3.13 Hashin's damage initiation criteria

The damage initiation criteria for fibre-reinforced composites is calculated using Hashin's theory [58].

The model considers the following modes of damage initiation:

- I. Fibre tension ($I_{D,ft}$)
- II. Fibre compression ($I_{D,fc}$)
- III. Matrix tension ($I_{D,mt}$)
- IV. Matrix compression ($I_{D,mc}$)

The criteria for each mode are calculated according to Equations 12.3.19-22.

$$I_{D,ft} = \left(\frac{\hat{\sigma}_{11}}{X_{hT}} \right)^2 + \alpha \left(\frac{\hat{\sigma}_{12}}{S_{hL}} \right)^2 = 1, \{0 \leq \alpha \leq 1; \hat{\sigma}_{11} \geq 0\} \quad [12.3.19]$$

$$I_{D,fc} = \left(\frac{\hat{\sigma}_{11}}{X_{hc}} \right)^2 = 1, \{\hat{\sigma}_{11} < 0\} \quad [12.3.20]$$

$$I_{D,mt} = \left(\frac{\hat{\sigma}_{22}}{Y_{hT}} \right)^2 + \left(\frac{\hat{\sigma}_{12}}{S_{hL}} \right)^2 = 1, \{\hat{\sigma}_{22} \geq 0\} \quad [12.3.21]$$

$$I_{D,mc} = \left(\frac{\hat{\sigma}_{22}}{2S_{hT}} \right)^2 + \left[\left(\frac{Y_c}{2S_{hT}} \right)^2 - 1 \right] \frac{\hat{\sigma}_{22}}{Y_{hc}} + \left(\frac{\hat{\sigma}_{12}}{S_{hL}} \right)^2 = 1, \{\hat{\sigma}_{22} < 0\} \quad [12.3.22]$$

The effective stress tensor, $\hat{\sigma}$, is given by Equation 12.3.23.

$$\hat{\sigma} \equiv \begin{bmatrix} \sigma_{11} & 0 & 0 \\ 0 & \sigma_{22} & 0 \\ 0 & 0 & \sigma_{12} \end{bmatrix} \quad [12.3.23]$$

12.3.14 LaRC05 damage initiation criteria

Overview

An alternative damage initiation criterion has been proposed by Pinho et al. [55] [59], wherein the damage in the composite is based on a pressure-dependent, three-dimensional constitutive law. The model considers the following modes of failure:

- I. Polymer ($I_{F,p}$)
- II. Matrix ($I_{F,m}$)
- III. Fibre kinking/splitting ($I_{F,k/s}$)
- IV. Fibre tensile ($I_{F,t}$)

The model has been used to make predictions of the test cases presented from the second World-Wide Failure Exercise [60].

Polymer failure

The polymer damage initiation criterion is calculated according to Equation 12.3.24.

$$I_{D,p} = \frac{3[k^2 - (X_{lT}X_{lC})\sigma_h]}{X_{lT}X_{lC}} \quad [12.3.24]$$

The hydrostatic stress is given by Equation 12.3.25.

$$3\sigma_h = \sigma_{11} + \sigma_{22} + \sigma_{33} \quad [12.3.25]$$

The parameter, k , is given by Equation 12.3.26.

$$6k^2 = (\sigma_1 - \sigma_2)^2 + (\sigma_2 - \sigma_3)^2 + (\sigma_3 - \sigma_1)^2 \quad [12.3.26]$$

The maximum, middle and minimum principal stresses are given by σ_1 , σ_2 and σ_3 , respectively.

Matrix failure

The matrix damage initiation criterion is calculated according to Equation 12.3.27.

$$I_{D,m} = \left(\frac{\tau_T}{S_{lY} - \eta_T \sigma_N} \right)^2 + \left(\frac{\tau_L}{S_{lX} - \eta_L \sigma_N} \right)^2 + \left(\frac{\langle \sigma_N \rangle_+}{Y_{lT}} \right)^2 \quad [12.3.27]$$

The stresses τ_T , τ_L and σ_N are the traction components on each trial fracture plane, and are calculated according to Equations 12.3.28-30.

$$\sigma_N = \frac{\sigma_2 + \sigma_3}{2} + \frac{\sigma_2 + \sigma_3}{2} \cos(2\alpha) + \sigma_{23} \sin(2\alpha) \quad [12.3.28]$$

$$\tau_T = -\frac{\sigma_2 - \sigma_3}{2} \sin(2\alpha) + \sigma_{23} \cos(2\alpha) \quad [12.3.29]$$

$$\tau_L = \sigma_{12} \cos(\alpha) + \sigma_{13} \sin(\alpha) \quad [12.3.30]$$

Fibre kinking/splitting failure

The fibre kinking and splitting damage initiation criteria are calculated according to Equation 12.3.31.

$$I_{D,k/s} = \left(\frac{\tau^m_{23}}{S_{LY} - \eta_T \sigma^m_2} \right)^2 + \left(\frac{\tau^m_{12}}{S_{LY} - \eta_L \sigma^m_2} \right)^2 + \left(\frac{\langle \sigma^m_2 \rangle_+}{Y_{LT}} \right)^2 \quad [12.3.31]$$

The stresses on the kink band plane are given by Equations 12.3.32-35.

$$\sigma^\psi_2 = \cos(\psi \sigma_2)^2 + \sin(\psi \sigma_3)^2 + 2 \sin(\psi) \cos(\psi \sigma_{23}) \quad [12.3.32]$$

$$\tau^\psi_{12} = \sigma_{12} \cos(\psi) + \sigma_{13} \sin(\psi) \quad [12.3.33]$$

$$\tau^\psi_{23} = -\sin(\psi) \cos(\psi \sigma_2) + \sin(\psi) \cos(\psi \sigma_3) + [\cos(\psi)^2 - \sin(\psi)^2] \sigma_{23} \quad [12.3.34]$$

$$\tau^\psi_{13} = \sigma_{13} \cos(\psi) - \sigma_{12} \sin(\psi) \quad [12.3.35]$$

The stresses on the fibre misalignment frame are given by Equations 12.3.36-38.

$$\sigma^m_2 = \sin(\varphi \sigma_1)^2 + \cos(\varphi \sigma^\psi_2)^2 - 2 \sin(\varphi) \cos(\varphi \tau^\psi_{12}) \quad [12.3.36]$$

$$\tau^m_{12} = -\sin(\varphi) \cos(\varphi \sigma_1) + \sin(\varphi) \cos(\varphi \sigma^\psi_2) + [\cos(\varphi)^2 - \sin(\varphi)^2] \tau^\psi_{12} \quad [12.3.37]$$

$$\tau^m_{23} = \tau^\psi_{23} \cos(\varphi) - \tau^\psi_{13} \sin(\varphi) \quad [12.3.38]$$

The misalignment angle, φ , is the sum of the initial misalignment angle, φ^0 , and the shear strain, γ_{m^0} , and is given by Equation 12.3.39.

$$\varphi = sign(\tau^\psi_{12}) \varphi^0 + \gamma_{m^0}, \quad \gamma_{m^0} = \frac{\sigma_{12}}{G_{l,12}} \quad [12.3.39]$$

Fibre tensile failure

The tensile damage initiation criterion is calculated according to Equation 12.3.40.

$$I_{D,t} = \frac{\langle \sigma_1 \rangle_+}{X_{LT}} \quad [12.3.40]$$

Derived properties

If the value of S_{lY} is not specified, it is derived from Equation 12.3.41.

$$S_{lY} = Y_C \cos(\alpha_0) \left[\sin(\alpha_0) + \frac{\cos(\alpha_0)}{\tan(2\alpha_0)} \right] \quad [12.3.41]$$

If the value of η_t is not specified, it is derived from Equation 12.3.42.

$$\eta_T = -\frac{1}{\tan(2\alpha_0)} \quad [12.3.42]$$

The parameter α_0 represents the fracture plane angle for pure transverse compression. Several sources report that $51^\circ \leq \alpha_0 \leq 55^\circ$ is typical for glass and carbon composites [55] [61] [62]. A default value of 53 degrees is assumed.

If the value of φ^0 is not specified then it can be derived iteratively from Equations 12.3.43-44, otherwise a value of 0 degrees is assumed.

$$\varphi^0 = \varphi^c - \gamma_{m^0} \left[\frac{1}{2} \sin(2\varphi^0) X_{lc} \right] \quad [12.3.43]$$

$$\varphi^c = \tan^{-1} \left[\frac{1 - \sqrt{1 - 4 \left(\frac{S_{lx}}{X_{lc}} + \eta_L \right) \frac{S_{lx}}{X_{lc}}}}{2 \left(\frac{S_{lx}}{X_{lc}} + \eta_L \right)} \right] \quad [12.3.44]$$

This calculation is enabled by checking the option **Allow iterative solution if applicable**¹⁰ in the LaRC05 Parameters dialogue box (Figure 5.7). This option may be time-consuming if the assessment is run for more than 100 analysis items.

Critical plane searching

The values of α and ψ which maximise the functions $F_{I,m}$ and $F_{I,k/s}$ are found by performing critical plane analyses for $0 \leq \alpha, \psi \leq 180$, respectively.

Quick Fatigue Tool searches 19 planes by default. The user specifies the step size n for the critical plane search algorithm in the environment file.

Environment File Usage: **stepSize** = n

¹⁰ Requires the Symbolic Math Toolbox

12.3.15 Output

Output failure indices

The failure and damage initiation criteria, $I_{F/D}$, are evaluated based on the available material data. The values of I_F are quoted in terms of the failure index R , so that for the given plane stress state $\{\sigma_{11}, \sigma_{22}, \sigma_{12}\}$:

$$\left\{ \frac{\sigma_{11}}{R}, \frac{\sigma_{22}}{R}, \frac{\sigma_{12}}{R} \right\} \rightarrow I_F = 1.0 \quad [12.3.45]$$

That is, $1/R$ is the scaling factor by which all of the stress components must be simultaneously multiplied to hit the failure surface. The term $1/R$ can be used with the **SCALE** option in order to make $R = 1$ for the specified failure criterion.

The calculation of R depends on the failure criterion, and is summarised according to the following table:

Failure criterion	Failure index, R
Maximum stress theory	I_F
Maximum strain theory	I_F
Tsai-Hill theory	$\sqrt{I_F}$
Tsai-Wu theory for fibre-reinforced composites	$\left(\frac{-B \pm \sqrt{B^2 + 4A}}{2A} \right)^{-1}$
Tsai-Wu theory for closed cell PVC foam	$\left(\frac{-B \pm \sqrt{B^2 - 4AC}}{2A} \right)^{-1}$
Azzi-Tsai-Hill theory	$\sqrt{I_F}$

For the Tsai-Wu theory for fibre-reinforced composites:

$$A = F_{11}\sigma_{11}^2 + F_{22}\sigma_{22}^2 + F_{66}\sigma_{12}^2 + 2F_{12}\sigma_{11}\sigma_{22} \quad [12.3.46]$$

$$B = F_1\sigma_{11} + F_2\sigma_{22} \quad [12.3.47]$$

For the Tsai-Wu theory for closed cell PVC foam:

$$A = F_{22}\sigma_{22}^2 + F_{33}\sigma_{33}^2 + 2F_{23}\sigma_{22}\sigma_{33} \quad [12.3.48]$$

$$B = F_2\sigma_{22} + F_3\sigma_{33} \quad [12.3.49]$$

$$C = \frac{\sigma_{12}}{\sigma_{23}} - 1 \quad [12.3.50]$$

For Hashin's theory and LaRC05, R is not evaluated; output for these criteria is given directly as the value of I_D .

The values of R and I_D should be interpreted according to the following table:

Range of values of R/I_D	Meaning
$0 \leq R/I_D < 1$	Composite is safe, or damage has not initiated
$R/I_D \geq 1$	Composite has failed, or damage has initiated
$R/I_D = -1$	Criterion could not be evaluated

Composite failure criteria analysis field variable identifiers

The criterion variable names take their meaning from the following tables:

Failure index, R	Definition
MSTRS	Maximum stress theory failure measure
MSTRN	Maximum strain theory failure measure
TSAIH	Tsai-Hill theory failure measure
TSAIW	Tsai-Wu theory failure measure
TSAIWT	Tsai-Wu theory failure measure for closed cell PVC foam
AZZIT	Azzi-Tsai-Hill theory failure measure

Damage initiation criterion, I_D	Definition
HSNFTCRT	Hashin's fibre tensile damage initiation criterion
HSNFCCRT	Hashin's fibre compressive damage initiation criterion
HSNMTCRT	Hashin's matrix tensile damage initiation criterion
HSNMCCRT	Hashin's matrix compressive damage initiation criterion
LARPPCRT	LaRC05 polymer failure measure
LARMFCRT	LaRC05 matrix failure measure
LARKFCRT	LaRC05 fibre kinking failure measure
LARSFCRT	LaRC05 fibre splitting failure measure
LARTFCRT	LaRC05 fibre tensile failure measure

Output files

Field output

If there are sufficient material properties to evaluate at least one criterion, the results for each analysis item are written to *Project\output\<job-name>\Data Files\composite_criteria.dat*. This file contains the value of each criterion for each item in the model.

The user controls which composite failure criteria are evaluated in the environment file.

Environment File Usage: `compositeFile_stress = {0.0 | 1.0}`
`compositeFile_strain = {0.0 | 1.0}`
`compositeFile_hashin = {0.0 | 1.0}`
`compositeFile_larc05 = {0.0 | 1.0}`

Whole model summary

In addition to tabulated results, a whole model summary of the analysis is written to the message file with the status of each criterion and the number of items at which failure is detected.

Exporting to an Abaqus .odb file

Quick Fatigue Tool can automatically write the results of a composite criteria assessment to an Abaqus output database (.odb) file.

The user enables the ODB interface in the environment file and specifies the model output database in the job file.

Environment File Usage: `autoExportODB = 1.0`

Job File Usage: `OUTPUT_DATABASE = 'model-odb-file-name.odb'`

Export Tool Usage: Composite assessment output is not supported in Export Tool.

The ODB interface is discussed in Section 10.5.

12.3.16 Example Usage

Composite failure assessments are run as jobs in much the same way as standard fatigue analyses. Example input files can be found in *Data\tutorials\composite_tutorial*. Damage in the fibre and the matrix are each considered as separate jobs, given by the text files *composite_load_fibre.inp* and *composite_load_matrix.inp*, respectively. The composite assessment is specified using the *COMPOSITE CRITERIA keyword.

The job is submitted by executing the following command:

```
>> composite_tutorial
```

Results are written to *Project\output\composite_tutorial\Data Files\composite_criteria.dat*.

Since the composite failure criteria assessment is always run as a data check analysis, the user is not required to specify **ALGORITHM** or **MS_CORRECTION** in the job file.

12.4 Compatibility with other features

12.4.1 Nodal elimination

Since the fatigue limit is not calculated for either the yield or composite assessments, the nodal elimination algorithm is not utilised and Quick Fatigue Tool analyses every item in the model.

12.4.2 Analysis regions

The user can specify the analysis region of the model using the **ITEMS** option. This includes surface detection if the an Abaqus .odb file is specified.

For composite criteria assessment on models which use layered solid elements, the user should exercise caution since the surface detection algorithm might remove elements belonging to interior plies of the composite. If results are needed at plies which do not contain any elements on the free surface, then surface detection should be disabled.

Job File Usage: **ITEMS** = {**'ALL'** | **'MAXPS'** | **'file-name.*'** | [**ID₁,..., ID_n**]}

12.4.3 Analysis algorithms

The yield criterion assessment cannot be performed in tandem with a weld fatigue analysis using BS 7608, since the algorithm does not read any material data.

13. Tutorial A: Analysis of a welded plate with Abaqus

13.1 Background

This tutorial outlines the procedure for analysing a welded structure using an Abaqus output database file with Quick Fatigue Tool. The analysis is based upon the continuum shell model of a welded T-joint in bending, shown in Figure 13.1.

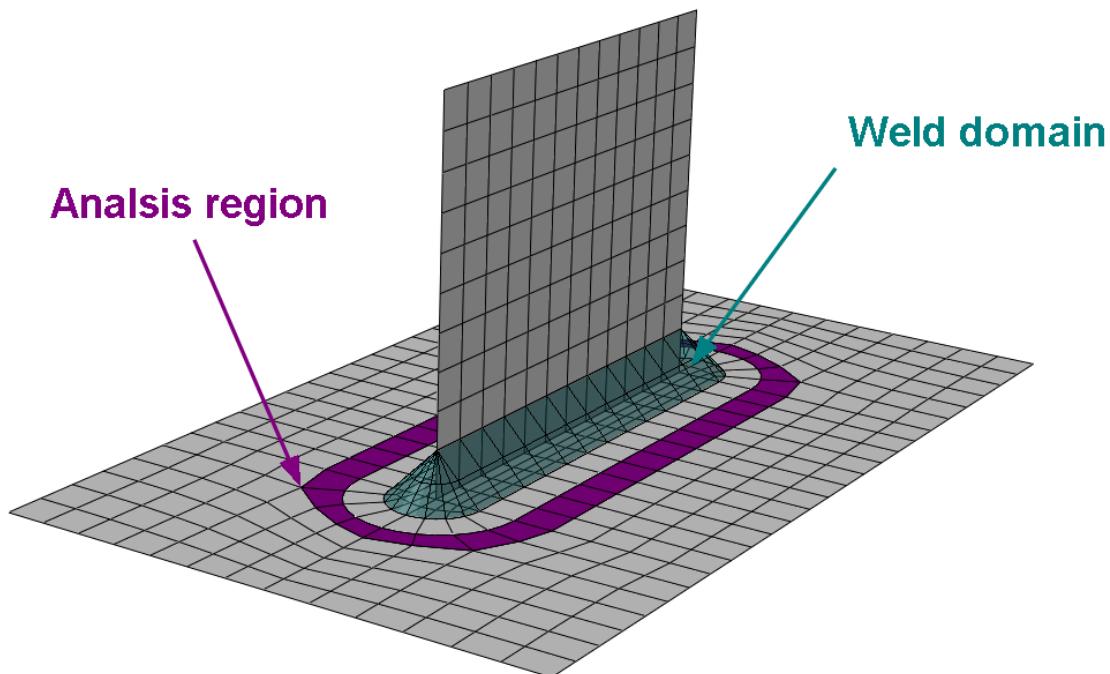


Figure 13.1: Weld plate model

The model will be analysed using the British Standard BS 7608 method for the analysis region indicated by the magenta elements in Figure 13.1. The methodology corrects the Stress-Life curve to account for the presence of a stress concentration at the weld toe; therefore, the stresses used for analysis should be a small distance away from the weld line, in order to avoid excessively conservative results. For more information about the BS 7608 method, refer to Section 6.6.

The stress datasets are first extracted from the output database and written to an RPT file. A fatigue analysis is then performed with Quick Fatigue Tool and the results are written back to the ODB using the Export Tool.

13.2 Preparing the RPT file

If you do not have Abaqus installed, you can skip this step; the file *weldPlate.m* has already been added to the *Data\datasets* folder. You must copy this file into the *Project\input* folder before continuing.

To generate an RPT file for the analysis region, open the Create Display Group dialogue box. From the Item region, select Elements. Select the element group PLATE-1_ANALYSE and select Replace from the list of Boolean operations.

The exported stress tensors should not have any averaging applied to them. From the main menu, select **Result → Options....** From the Averaging region, deselect “Average element output at nodes” and select **OK**. From the main menu, select **Report → Field Output...** From the Variable tab, select Element-Nodal as the result position and select all four stress tensor components S11, S22, S33 and S12. From the Setup tab, specify the absolute path to the Quick Fatigue Tool input folder *Project\input* and name the file *weldPlate.rpt*. Deselect *Column totals* and *Column min/max* from the data region and select **OK**.

13.3 Running the analysis

From the *Project\input* folder, open the job file named *tutorial_A.m* and review the contents. Ensure that the DATASET option points to the correct file. A summary of the other pertinent options are listed below.

Option	Details
ELEMENT_TYPE =1.0	The elements are plane stress
ALGORITHM =8.0	The algorithm is set to BS 7608
OUTPUT_FIELD =1.0	Field output is required to write results to the ODB
WELD_CLASS ='F2'	Weld classification. For guidelines on choosing the weld class, consult the document BS 7608
YIELD_STRENGTH =325 UTS =400	Mechanical properties of the weld plate material (MPa)
DEVIATIONS_BELOW_MEAN =2.0	Confidence interval (95% probability of failure)
FAILURE_MODE ='NORMAL'	Failure criterion
CHARACTERISTIC_LENGTH =1.0	Plate thickness (mm)
SEA_WATER =0.0	Atmospheric condition (fresh air)

Run the analysis by right-clicking the job file and selecting *Run*. The analysis may take a few minutes depending the computer’s hardware specification. The result summary is displayed in the command window (Figure 13.2).

```

FATIGUE RESULTS SUMMARY:
=====
Worst Life-Repeats : 3.73e+04
at item 299.49

Number of cycles in loading : 1

Maximum stress (MPa) : -226.1
at Item 299.49

Worst cycle mean stress (MPa) : 18.08
at Item 351.442

Worst cycle stress amplitude (MPa) : 113.1
at Item 299.49

Worst cycle damage parameter (MPa) : 113.1
at Item 299.49

Analysis time : 0:00:0.457

=====
Job tutorial_A completed with warnings. See message file for details. (15-Jul-2016 18:55:37)

```

Figure 13.2: Fatigue results summary

The predicted life is 37,300 repeats at element 299, node 49. Open the message file (located in *Project\output\tutorial_A*) to view information about the analysis. Take note of which element face Quick Fatigue Tool analysed. By default, the negative shell face is used for analysis.

13.4 Post processing the results

This step can only be completed if the user has access to Abaqus/Viewer.

The field data for the entire analysis region can be written back to the Abaqus output database. Start the Export Tool either by clicking on the App icon, or by running the file *ExportTool.m* in *Application_Files\code\python*. Configure the dialogue so that it appears as shown in Figure 13.3.

For the *Field Data* input, select the file *f-output-all.dat* from *Project\output\Tutorial_A\Data Files*.

For the *Model output database* input, select *weldPlate_614.odb* from *Data\abaqus*.

Check **Results output database** and select *Project\output\Tutorial_A* as the result directory.

Accept the default field output and select **Write Output....** The progress of the export is displayed in the command window.

The contents of the export log file are shown in Figure 13.4. Since the fatigue analysis only considered a subset of elements, 615 of the 655 total elements in the ODB were ignored.

The fatigue results ODB path is copied to the clipboard. Open Abaqus/CAE and press Ctrl+O. In the Open Database dialogue, paste the ODB path into the file box and press Enter. The fatigue results are shown in Figure 13.5.

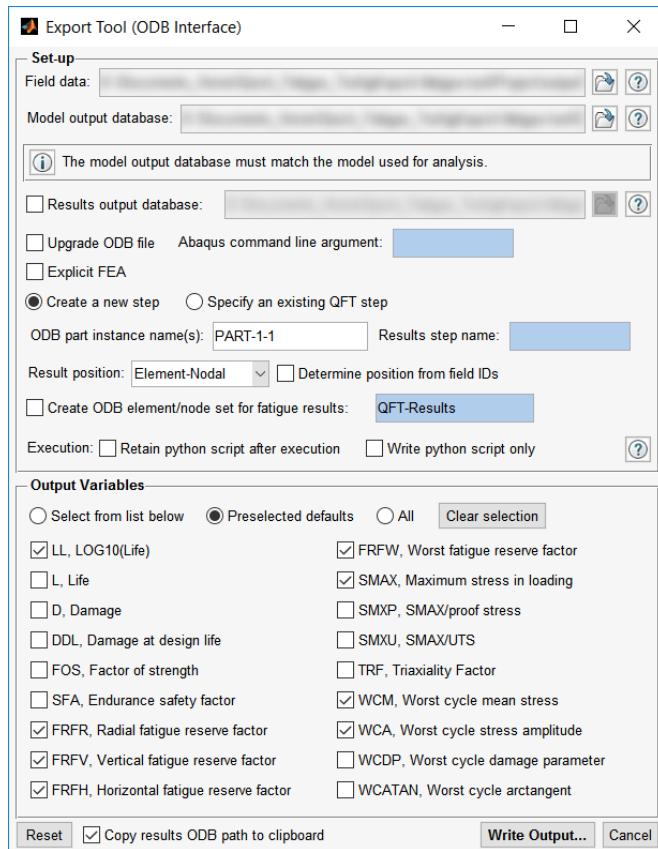


Figure 12.3: Export Tool setup

```

Quick Fatigue Tool 6.8-00 ODB Interface Log

User-selected results position: Element-Nodal
Allow Quick Fatigue Tool to determine results position based on field IDs: NO

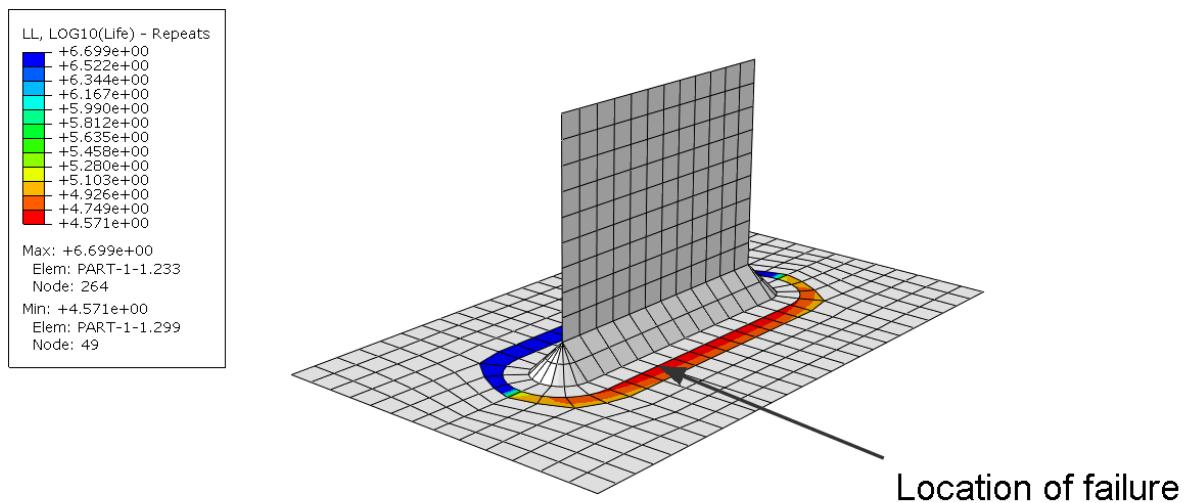
Collecting field data...
    Based on user selection, field data will be read as ELEMENT_NODAL
    Requesting nodal connectivity matrix... Success
        Detected 1983 nodes belonging to 655 elements
        Checking for redundant nodes... 615 connected node groups removed
        Checking for redundant elements... 0 elements removed
    Taking position labels from ODB element listing
    Generating position labels... 1 fields requested

Preparing field data...
    (1 of 1)

Writing field data to ODB... Success

```

Figure 12.4: ODB export log file



ODB: weldPlate_614Results.odb Abaqus/Standard 6.14-2 Sun Oct 04 12:15:04 W. Europe Daylight Time 2015

Step: Quick Fatigue Tool, version 6.4-00; Job: tutorial_A, Loading: 1 Repeats

Fatigue results field data

Primary Var: LL, LOG10(Life) - Repeats

Deformed Var: not set Deformation Scale Factor: not set

Figure 12.5: Fatigue results ODB

14. Tutorial B: Complex loading of an exhaust manifold

14.1 Background

This tutorial outlines the procedure for analysing an exhaust manifold using an Abaqus output database file with Quick Fatigue Tool. The manifold model used is that shown in Figure 14.1.

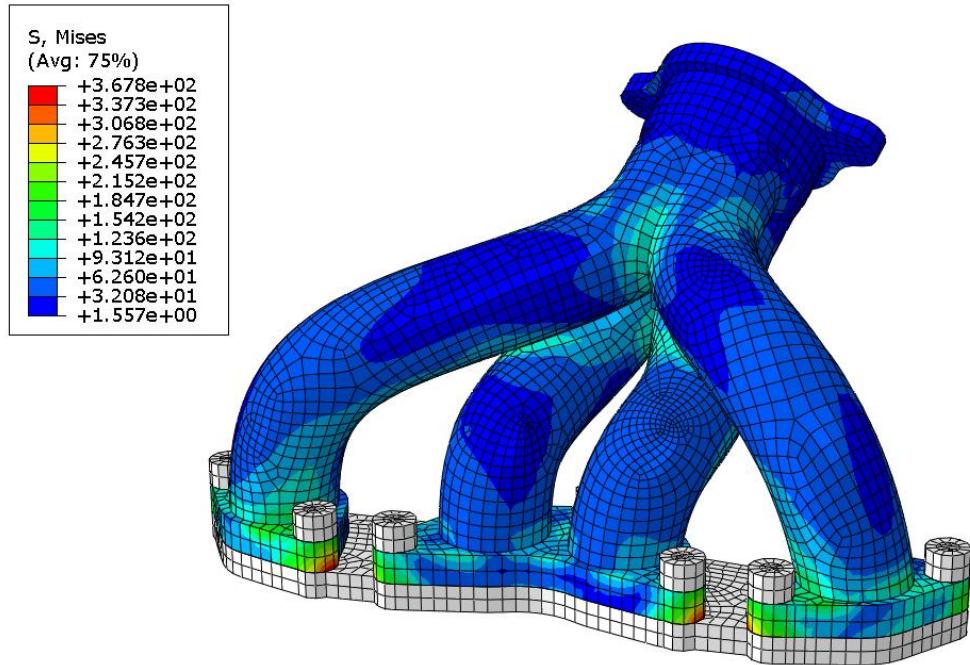


Figure 14.1: Exhaust manifold model

The analysis consists of three loading steps. First, a pre-tension is applied to the bolts. The manifold is then subjected to a transient thermal load. The load is then removed and the model is allowed to return to ambient temperature. The stresses are obtained at each load step and analysed as a stress dataset sequence. The peak stress history is shown in Figure 14.2. In addition to the thermal loading, the mechanical load history shown in Figure 14.3 is superimposed onto the thermal stress as a high frequency stress dataset. The mechanical load is defined in Quick Fatigue Tool as a simple loading of the stress data from the pre-tension step with a user-defined load history.

This tutorial demonstrates the use of several features in Quick Fatigue Tool. However, the model data itself is arbitrary. Sections of the tutorial use Abaqus/Viewer for results post-processing. If the user does not have access to Abaqus/Viewer, these sections may be skipped.

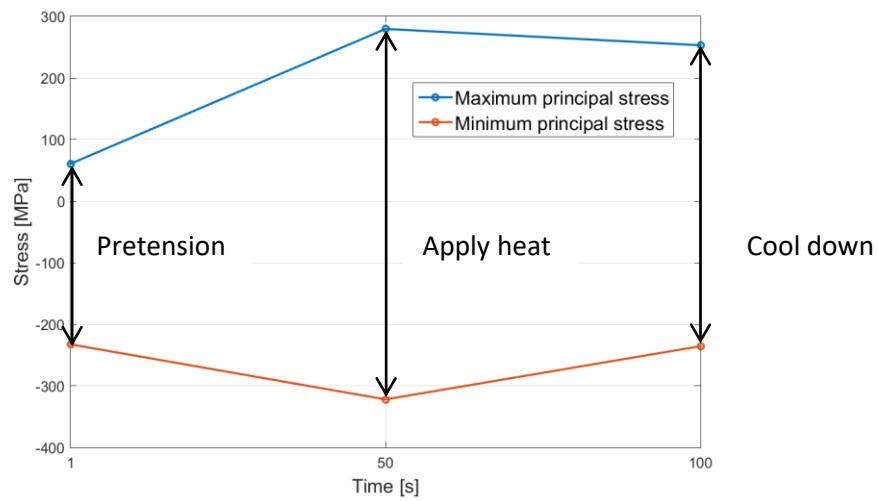


Figure 14.2: Peak thermal stress

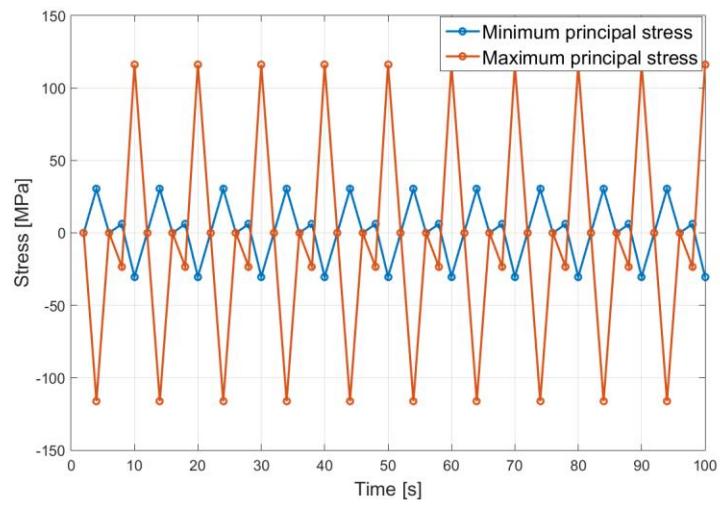


Figure 14.3: Peak mechanical load

14.2 Preparation

Analysing finite element models with complex load histories with Quick Fatigue Tool can sometimes be time-consuming; therefore, the analysis will be split into two procedures. First, the whole model will be analysed with a simplified loading configuration to find the location of maximum damage. Afterwards, a second analysis will be performed on the node which experiences maximum damage with the complete loading definition.

Before running the analysis, verify that the following files exist in the *Project\input* directory:

'manifold_1.rpt'	Stress data for step 1 (pretension)
'manifold_2.rpt'	Stress data for step 2 (thermal load)
'manifold_3.rpt'	Stress data for step 3 (cool-down)
'manifold_history_hf.dat'	Normalised history data for the mechanical load

Copy the datasets and history file from *Data\datasets* and *Data\histories*, respectively, into the *Project\input* folder. The input folder should appear as in Figure 14.4.

The datasets may be created in Abaqus/CAE following the procedure outlined in Section 3.2, using unique nodal as the result position.

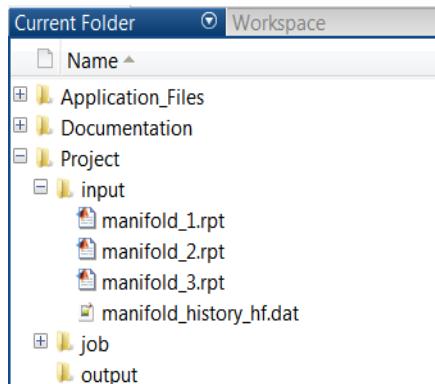


Figure 14.4: Required input files for
Tutorial B

In order to customise the analysis, setting in the environment file can be changed. However, future analyses should not be affected by this change, so a separate environment file will be stored locally, which corresponds to the analysis in Tutorial B. To create a local environment file, copy the file *environment.m* from *Application_Files\default* and paste it into *Project\job*. Rename the file to *tutorial_B_env.m*. If the file is named differently, it will be ignored during analysis. File names are case-sensitive. The job folder should appear as in Figure 14.5.

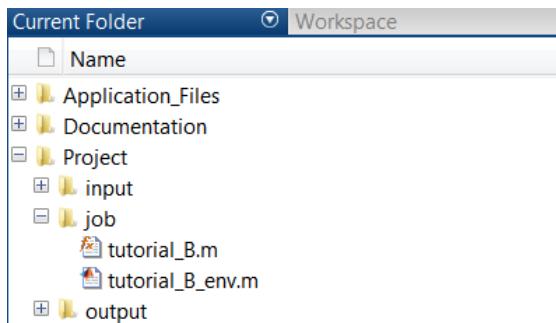


Figure 14.5: Required job files for Tutorial B

14.3 Defining the material

The analysis job file *tutorial_B.m* references a material called *material_tutorial_B.mat*. This material needs to be created using the material manager. To launch the material manager, run the file *materialManager.m* from *Application_Files\code\material_manager* or run the material manager app from the app menu. The main dialogue is shown in Figure 14.6.

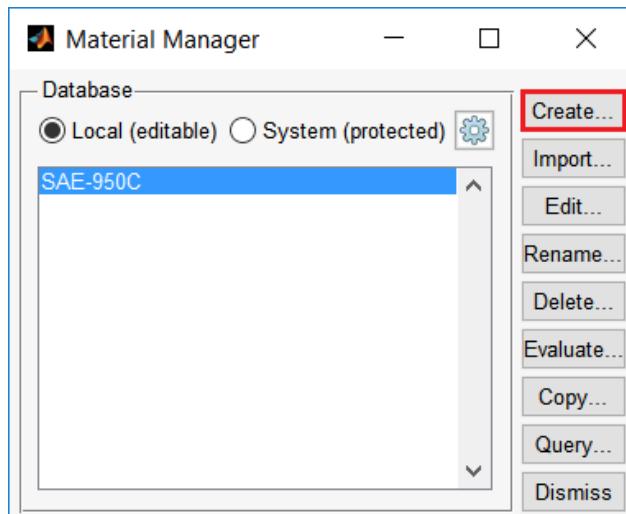


Figure 14.6: Material Manager dialogue

Click **Create...** to create a new material. The material editor is shown in Figure 14.7. Enter the parameters as shown by the green boxes. The material properties are as follows:

Material name	material_tutorial_B
Young's Modulus	200GPa
Fatigue Strength Coefficient	1050MPa
Fatigue Strength Exponent	-0.085
Strain Hardening Coefficient	1200MPa
Strain Hardening Exponent	0.19

Press **OK** to save the material to the workspace. The newly created material should now appear in the list of workspace materials in the Material Manager main dialogue box. Exit the Material Manager by clicking **Dismiss**.

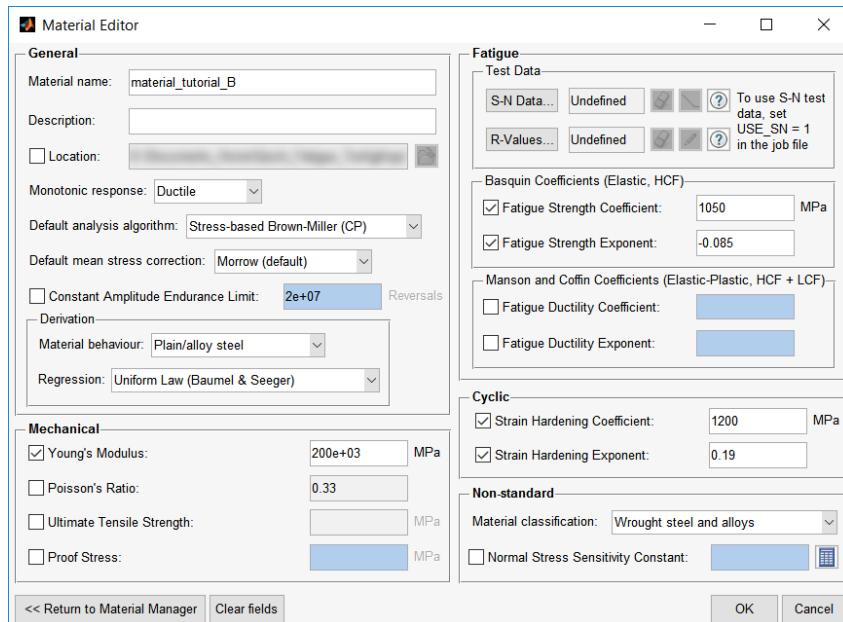


Figure 14.7: Material editor dialogue

14.4 Running the first analysis

The first analysis is run to identify the location of maximum damage. The loading is restricted to the thermal stress and the critical plane step size is increased to reduce the analysis time.

Close the environment file and open *tutorial_B.m* from the job folder and review the settings:

- The thermal load is defined by **DATASET** as a sequence of stress datasets representing each loading step from the finite element analysis
- The high frequency mechanical loading defined by **HF_DATASET** and **HF_HISTORY** is commented out for this analysis
- The default algorithm and mean stress correction are set by **ALGORITHM** and **MS_CORRECTION**, respectively
- **ITEMS** is used to indicate that all items in the model will be analysed
- A residual stress of 10MPa is specified using **RESIDUAL**
- Field output is requested using **OUTPUT_FIELD**

For a complete description of analysis options, consult the document *Quick Fatigue Tool User Settings Reference Guide*.

To run the analysis, right-click on *tutorial_B.m* and select *Run*, or press F5 while the file is open in the editor. Analysis progress is displayed in the command window. Figure 14.8 shows the result of the analysis.

```
FATIGUE RESULTS SUMMARY:  
=====  
Worst Life-Repeats : 4.3e+06  
at item 21413.1  
  
Number of cycles in loading : 1  
  
Maximum stress (MPa) : -562.4  
at Item 20077.1  
  
Maximum stress/yield : -1.527  
at Item 20077.1  
  
Worst cycle mean stress (MPa) : 243.1  
at Item 21345.1  
  
Worst cycle stress amplitude (MPa) : 338.7  
at Item 21413.1  
  
Worst cycle damage parameter (MPa) : 338.7  
at Item 21413.1  
  
Analysis time : 0:00:6.765  
=====
```

[NOTICE] Results have been written to [REDACTED]
[NOTICE] Job test completed with warnings. See message file for details. (17-Nov-2017 16:12:06)

Figure 14.8: Fatigue analysis results indicating node 21413 as the location of failure

The stress data from Abaqus was extracted at the nodes and averaged, therefore the worst life is quoted at the unique nodal position. Since the mechanical stresses were not included for analysis the life result is unimportant. However, the analysis result indicates that failure will occur at node 21413 on the finite element model.

14.5 Viewing the results with Abaqus/Viewer

This step may be skipped if the user does not have Abaqus 6.14 or later installed on their machine.

To view the life data on the manifold model, launch the Export Tool by running the file *exportTool.m* from *Application_Files\code\python*, or by running the app from the app bar. Configure the dialogue box so that it appears as in Figure 14.9.

1. Select the field data file *f-output-all.dat* from *Project\output\tutorial_BData Files*
2. Select the model output database *manifold_614.odb* from *Data\abaqus*
3. Specify the part instance name *PART-1-1*
4. Uncheck *Determine position from field IDs* and select *Unique Nodal* as the result position
5. Click *Clear selection* to deselect all fields, then select *LL, LOG10(Life)*
6. Click *Write Output...* to write the life data to an Abaqus ODB file

For a detailed explanation of the Export Tool, consult Section 10.5.

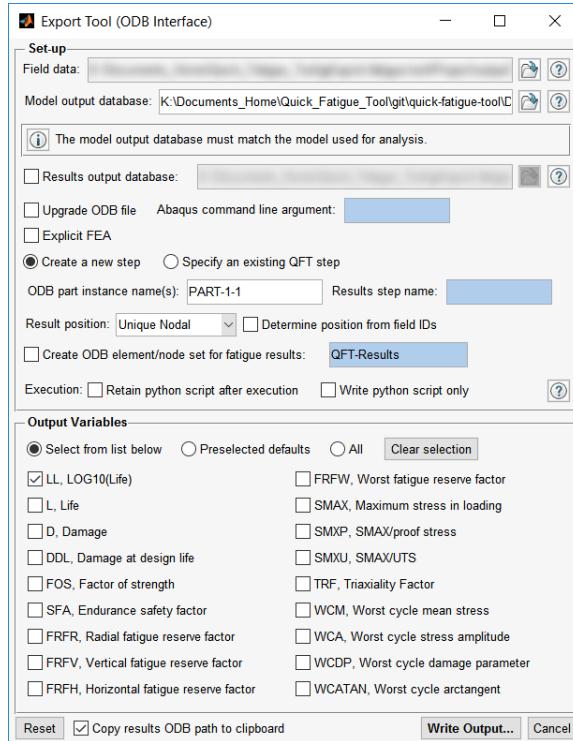


Figure 14.9: Export Tool settings for the results ODB

To view the ODB file in Abaqus/Viewer, start Abaqus and open the ODB in the usual way. The full path to the results ODB file has been copied to the clipboard, so pressing *Ctrl+V* can be used. Isolate the location of failure by selecting *Create Display Group*. Select *Elements* as the item and *Element Sets* as the method of creating the display group. Select *PART-1-1.FL2* and click *Replace*. The location of failure is shown in Figure 14.10.

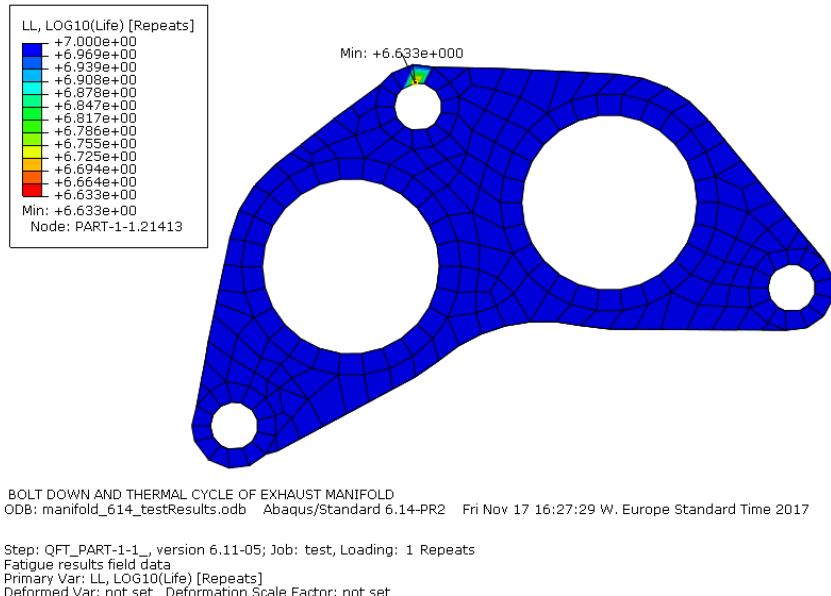


Figure 14.10: Location of failure shown in Abaqus/Viewer

14.6 Running the second analysis

The previous analysis determined that the manifold will fail at node 21426. By inspecting the message file, Quick Fatigue Tool tells us that this node corresponds to item number 10505 in the dataset. A second analysis will be performed on this item only. From the job file, specify this item as follows:

ITEMS=10505;

The mechanical load must now be considered. Define the high frequency data as follows:

```
HF_DATASET='manifold_1.rpt'  
HF_HISTORY='manifold_history_hf.dat'
```

Commented versions of these entries are already provided below the previous definitions.

Request histories and MATLAB figures in addition to fields:

OUTPUT_FIELD=1.0

OUTPUT_HISTORY=1.0

OUTPUT_PICTURE=1.0

MATLAB should be restarted before running the analysis, to ensure that all data from the previous analysis is cleared. After running the analysis, a summary of the fatigue results are displayed in the command window, as in Figure 14.11.

The analysis indicates a life of 146,000 repeats of the loading until failure.

```
FATIGUE RESULTS SUMMARY:  
=====  
Worst Life-Repeats : 1.46e+05  
at item 21413.1  
  
Number of cycles in loading : 4  
  
Maximum stress (MPa) : 477.8  
at Item 21413.1  
  
Maximum stress/yield : 1.297  
at Item 21413.1  
  
Worst cycle mean stress (MPa) : 107.6  
at Item 21413.1  
  
Worst cycle stress amplitude (MPa) : 527.9  
at Item 21413.1  
  
Worst cycle damage parameter (MPa) : 527.9  
at Item 21413.1  
  
Analysis time : 0:00:3.294  
=====
```

[NOTICE] Results have been written to [REDACTED]
[NOTICE] Job test completed with warnings. See message file for details. (17-Nov-2017 16:20:28)

Figure 14.11: Fatigue analysis results from the second analysis

14.7 Post processing the results

The results of the analysis are stored in *Project\output\tutorial_B*.

- field and history data is written to a set of tabulated data files;
- certain results data is automatically plotted to a set of MATLAB figures;
- an overview of the analysis is written to *tutorial_B.log*; and
- warnings and messages are written to *tutorial_B.msg*.

Open the message file to view possible issues with the analysis:

MESSAGES:

=====

```
***NOTE: The proof stress for material material_tutorial_B.mat (group 1)
was not specified
-> A derived value of 368.4MPa will be used
```

```
***WARNING: In at least one group, the UTS is undefined. The following
fields are unavailable:
-> FRFR, FRFH, FRFV, FRFW, SMXU
```

```
***NOTE: Worst damage at design life (1e+07) is 68.6
```

```
***WARNING: 1 items have lives less than 1e+06 Repeats
```

- the proof stress was derived based on the cyclic material properties using the 0.2% strain offset method;
- certain variables are not available because the ultimate tensile strength was not defined;
- the manifold will not survive the default design life of 10 million loading repeats; and
- the stress-life methodology is not well-suited to low-cycle fatigue problems (lives below 1 million repeats).

A summary of the cycle counting process can be viewed either as a Haigh diagram or a rainflow histogram. The rainflow histogram is shown by Figure 14.12. This indicates that the damage is primarily caused by one cycle in the loading where the stress range is significantly higher than every other location in the history.

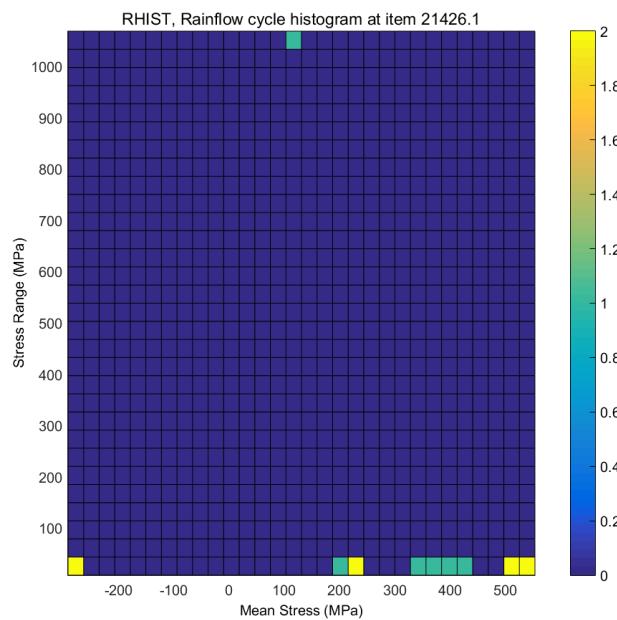


Figure 14.12: Rainflow histogram of cycles

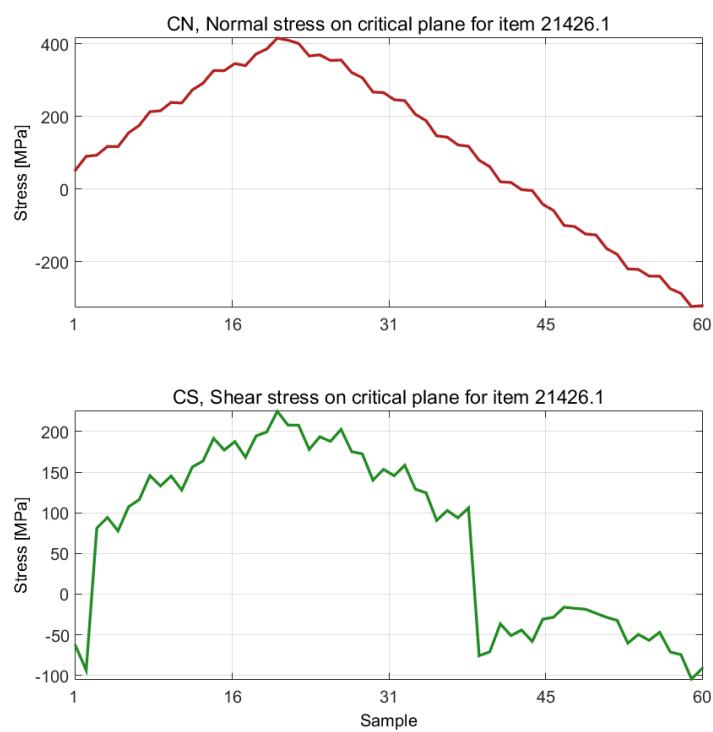


Figure 14.13: Normal and shear stress on the critical plane

The normal and shear stress on the critical plane is shown by Figure 14.13. It is clear from the data that the mechanical stress was superimposed onto the thermal stress correctly.

The results of the critical plane analysis are shown by Figures 14.15-16. Planes of maximum shear stress occur at 45 degrees, which is expected since ductile metals in pure tension fail along planes at 45 degrees to the free surface. According to the log file, the critical plane occurs where theta is 110 degrees (Figure 14.14). This represents the angle at which the combination of shear and normal stress is maximised.

```
CP SUMMARY AT WORST ITEM:  
=====  
CP Step Size: 10 degrees  
361 planes searched  
Coordinates (degrees):  
THETA = 70, PHI = 90
```

Figure 14.14: Sample log file output indicating the orientation of the critical plane

By inspecting Figure 14.16, it is clear that the damage parameter is maximised when theta equals 110 degrees. In fact, the damage profile is symmetrical about 90 degrees. This is the point at which the shear stress is zero.

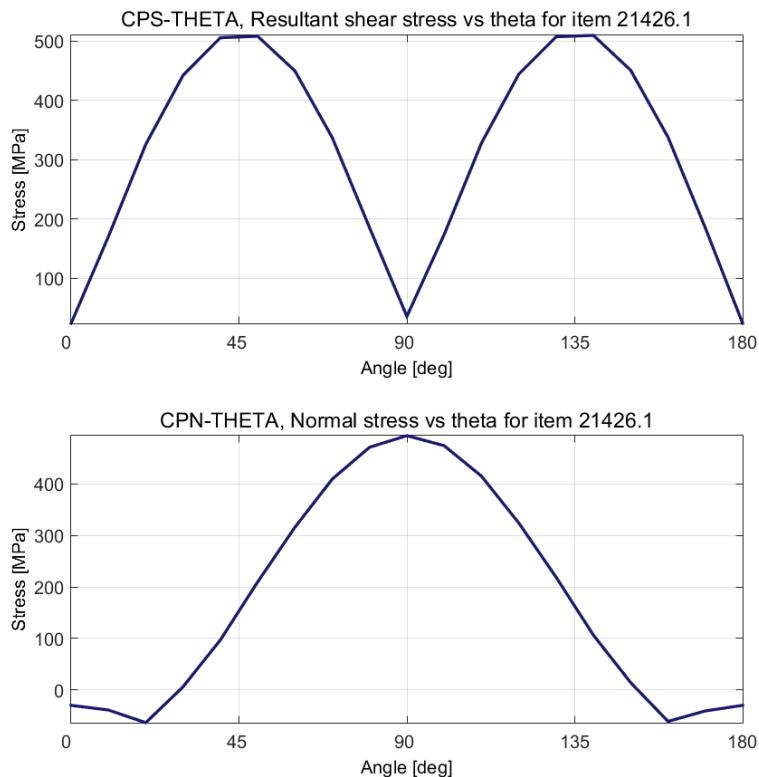


Figure 14.15: Critical plane analysis results

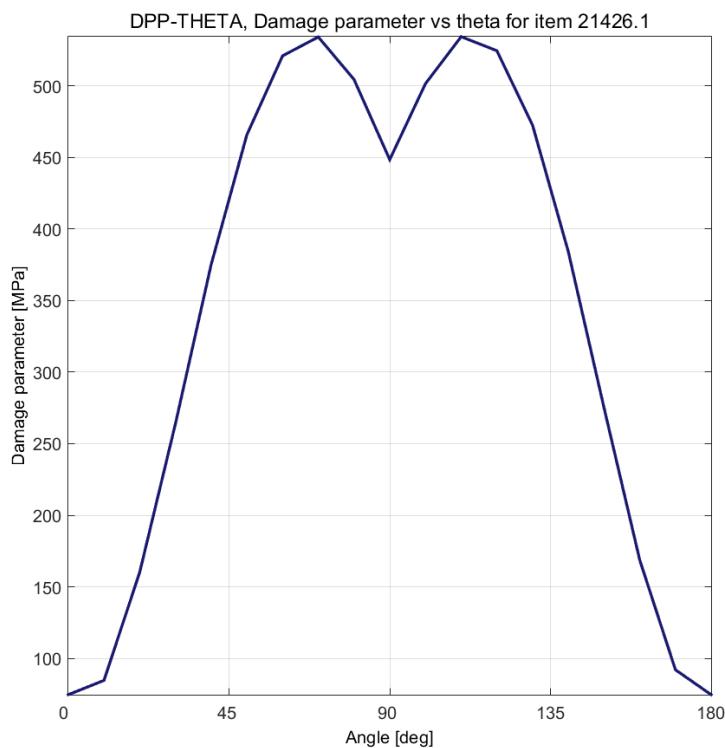


Figure 14.16: Damage parameter on the critical plane

Appendix I. Fatigue analysis techniques

This section has been released in the document *Quick Fatigue Tool Appendices*.

Appendix II. Materials data generation

This section has been released in the document *Quick Fatigue Tool Appendices*.

Appendix III. Gauge fatigue toolbox

This section has been released in the document *Quick Fatigue Tool Appendices*.

References

- [1] J. A. Bannantine, J. J. Comer and J. L. Handrock, *Fundamentals of Metal Fatigue Analysis*, Prentice Hall, 1989.
- [2] R. I. Stephens, A. Fataemi, R. R. Stephend and H. O. Fuchs, *Metal Fatigue in Engineering*, Jon Wiley & Sons, 2001.
- [3] D. N. W. M. Bishop and D. F. Sherratt, *Finite Element Based Fatigue Calculations*, Glashow: The International Association for the Engineering Analysis Community (NAFEMS), 2000, p. 18.
- [4] D. Taylor, *The theory of critical distances: a new perspective in fracture mechanics*, 1st ed., Elsevier, 2007, pp. 11, 164-165.
- [5] L. Vallance, A. Winkler and A. Belles Meseguer, "An Engineering Approach to Advanced Fatigue of Welded Joints," Graz, 2015.
- [6] J. Draper, *Modern Metal Fatigue Analysis*, East Sussex: EMAS Publishing, 2008.
- [7] M. A. Miner, "Cumulative Damage in Fatigue," *Journal of Applied Mechanics Vol 12, Trans ASME Vol 67*, vol. 12, pp. A159-A164, 1945.
- [8] W. Ramberg and W. R. Osgood, "Description of Stress-Strain Curves by Three Parameters," *National Advising Committee for Aeronautics*, no. Technical Note no. 902, 1947.
- [9] N. E. Dowling, *Mechanical Behavior of Materials*, 4th ed., Pearson, 2013, pp. 644-648.
- [10] L. Pook, "Metal Fatigue: What It Is, Why It Matters: Preliminary Entry No. 1525," *Solid Mechanical and its Applications*, vol. 145, p. 25, 9 March 2007.
- [11] R. E. Peterson, "Analytical Approach to Stress Concentration Effect in Fatigue of Aircraft Structures," *WADS Symposium*, 1959.
- [12] R. E. Peterson, *Notch Sensitivity*, McGraw-Hill Book Co., In., 1959, p. Metal Fatigue Chapter 13.
- [13] P. Kuhn and H. F. Hardrath, "An Engineering Method for Estimating Notch-Size Effect in Fatigue Tests on Steel," *National Advisory Committee for Aeronautics*, p. Technical Note 2805, October 1952.
- [14] H. J. Harris, New York: Pergamon Press, 1961.
- [15] R. B. Heywood, *Design by photoelasticity*, London: Chapman and Hall Ltd., 1952, p. 348.
- [16] J. E. Shigley and C. R. Mischke, *Mechanical Engineering Design*, 5th ed., New York: McGraw-Hill, Inc., 1989, pp. Fig. 5-16 and Fig. 5-17.

- [17] L. Susmel, "La progettazione a fatica in presenza di stati complessi di sollecitazione (PhD thesis)," Padova, 2001.
- [18] M. W. Brown and K. J. Miller, "A Theory Of Fatigue Under Multiaxial Strain Conditions," *Proc Inst Mech Eng*, vol. 187, pp. 745-755, 1973.
- [19] K. J. Miller, "Fatigue Under Complex Stress," *Metal Science*, pp. 482-488, August-September 1977.
- [20] M. W. Brown and K. J. Miller, "A Theory for Fatigue Failure under Multiaxial Stress-Strain Conditions," *Proceedings of the Institution of Mechanical Engineering*, vol. 187, no. 1, pp. 745-755, June 1973.
- [21] C. Lipson and R. C. Juvenal, *Handbook of Stress And Strength - Design And Material Application*, MacMillan, 1963.
- [22] A. Carpinteri and A. Spagnoli, "Multiaxial high-cycle fatigue criterion for hard metals," *International Journal of Fatigue*, vol. 23, no. 2, pp. 135-145, 2001.
- [23] W. N. Findley, "A Theory for the Effect of Mean Stress on Fatigue of Metals Under Combined Torsion and Axial Loading or Bending," *Journal of Engineering and Industry*, vol. 81, pp. 301-306, 1959.
- [24] W. N. Findley, B. Hanley and J. J. Coleman, "Theory for Combined Bending and Torsion Fatigue with Data for SAE 4349 Steel," *Proceedings for the International Conference on Fatigue of Metals*, September 1956.
- [25] L. Susmel, *Multiaxial Notch Fatigue*, Oxford: Woodhear Publishing, 2009, p. 101.
- [26] A. R. Kallmeyer, "Mutiaxial Fatigue Life Prediction Methods for Notched Bars of Ti-6AL-4V," *Fargo, Urbana*.
- [27] A. Winkler, S. Holt and L. Vallance, "Concerning the Synergy of Stress and Strain-based Methods in Modern Metal Fatigue Analysis," in *AVL AST User Conference*, Graz, 2013.
- [28] D. Socie and G. Marquis, *Multiaxial Fatigue*, SAE International, 1999.
- [29] G. Marquis and D. Socie, "Long-life torsion fatigue with normal mean stress," *Fatigue & Fracture of Engineering Materials & Structures*, vol. 23, no. 4, 2000.
- [30] J. Lemaitre and J. L. Chaboche, *Mechanics of Solid Materials*, Cambridge: Cambridge University Press, 1990.
- [31] V. Grubisic and A. Simbürger, "Fatigue under combined out of phase multiaxial stresses," *Processdings of International Conference on Fatigue Testing and Design*, pp. 27.1-27.8, 1976.

- [32] I. V. Papadopoulos, "Critical plane approaches in high-cycle fatigue: on the definition of the amplitude and mean value of the shear stress acting on the critical plane," *Fatigue and Fracture of Engineering Materials and Structures*, vol. 21, pp. 269-285, 1997.
- [33] B. Li, J. L. T. Santos and M. de Freitas, "A computerized procedure for long-life fatigue assessment under multiaxial loading," *Fatigue and Fracture of Engineering Materials and Structures*, vol. 24, pp. 165-177, 2001.
- [34] N. Zouain and E. N. Mamiya, "Using enclosing ellipsoids in multiaxial fatigue strength criteria," *European Journal of Mechanics and Solids*, vol. 25, pp. 51-71, 2006.
- [35] K. Bel Knani, D. Benasciutti, A. Signorini and R. Tovo, "Fatigue damage assessment of a car body-in-white using a frequency-domain approach," *International Journal of Materials and Product Technology*, vol. 30, pp. 172-198, 2007.
- [36] T. Lagoda, E. Macha and A. Dragon, "Influence of correlations between stresses on calculated fatigue life of machine elements," *International Journal of Fatigue*, vol. 18, pp. 547-555, 1996.
- [37] P. Heyes, "Multiaxial Fatigue," 1 May 2012. [Online]. Available: http://www.ncode.com/fileadmin/mediapool/nCode/downloads/events/Multiaxial_Fatigue_UGM_May_2012_Heyes__Compatibility_Mode_.pdf. [Accessed 21 November 2016].
- [38] British Standard, "Code of practice for Fatigue design and assessment of steel structures," British Standard, 1993.
- [39] J. Z. Gyekenyesi, P. L. N. Murthy and S. K. Mital, "NASALIFE - Component Fatigue and Creep Life Prediciton Program," NASA Center for Aerospace Information; National Technical Information Service, Cleveland, Ohio 44135, 2014.
- [40] G. Sines and G. Ohgi, "Fatigue Criteria Under Combined Stresses and Strains," *Journal of Engineering Materials and Technology*, vol. 103, pp. 82-90, 1981.
- [41] K. N. Smith, P. Watson and T. H. Topper, "A Stress-Strain Function for the Fatigue of Metals," *Journal of Materials*, vol. 5, no. 4, pp. 767-778, December 1970.
- [42] J. Goodman, *Mechanics Applied to Engineering*, London: Longmans Green, 1899.
- [43] C. R. Soderberg, "Factors of Safety and Working Stress," *Trans. ASME*, 1939.
- [44] J. Morrow, *Fatigue Design Handbook*, Society of Automotive Engineers, 1968, pp. 21-29.
- [45] K. Walker, "The Effect of Stress Ratio during Crack Propagation and Fatigue for 2024-T3 and 7075-T6 Aluminum," *American Society of Testing and Materials*, no. STP 462, pp. 1-14, 1970.
- [46] N. E. Dowling, "Mean Stress Effects in Stress-Life and Strain-Life Fatigue," 2004.

- [47] N. E. Dowling, "Mean Stress Effects in Stress-Life and Strain-Life Fatigue," *SAE Technical Paper*, no. No. 2004-01-2227, 2004.
- [48] A. Ince and G. Glinka, "A modification of Morrow and Smith-Watson-Topper mean stress correction models," *Fatigue & Fracture of ENgineering Materials & Structures*, vol. 34, no. 11, pp. 854-867, 17 February 2011.
- [49] K. Golos and F. Ellyin, "A Total Strain Energy Density Theory for Cumulative Fatigue Damage," *J. Pressure Vessel Technol*, vol. 110, no. 1, pp. 36-41, 1st Feb 1988.
- [50] Learn Engineering, "Theories of Failure," [Online]. Available: <http://www.learnengineering.org/2012/12/theories-of-failure.html>. [Accessed 28 March 2018].
- [51] E. Roos, U. Eisele and H. Silcher, "Effect of stress state on the ductile fracture behavior of large-scale specimens," *Constraint effects in fracture*, 1993.
- [52] H. Tresca, "Mémoire sur l'écoulement des corps solides soumis à de fortes pressions," *C. R. Acad. Sci.*, vol. 59, p. 754, 1864.
- [53] R. von Mises, "Mechanik der festen Körper im plastisch deformablen Zustand," *Nachr. Math. Phys.*, vol. 1, pp. 582-592, 1913.
- [54] J. Norton and C. Arraez, Machine Design, 2000.
- [55] S. Pinho, P. Camanho and L. Lannucci, "Failure Models and Criteria for FRP Under In-Plane or Three-Dimensional Stress States Including Shear Non-Linearity," NASA, 2005.
- [56] W. C. Cui, M. R. Wisnom and M. Jones, "A comparison of failure criteria to predict delamination of unidirectional glass/epoxy specimens waisted through the thickness," *Composites*, vol. 23, no. 3, pp. 158-166, May 1992.
- [57] S. W. Tsai and E. M. Wu, "A general theory of strength for anisotropic materials," *Journal of Composite Materials*, vol. 5, pp. 58-80, 1971.
- [58] Z. Hashin and A. Rotem, "A Fatigue Failure Criterion For Fiber Reinforced Materials," Isreal Institute of Technology, Haifa, 1973.
- [59] S. Pinho, G. Vyas and P. Robinson, "Material and structural response of polymer-matrix fibre-reinforced composites: Part B," *Journal of Composite Materials*, vol. 47, no. 6-7, pp. 679-696, 26 March 2013.
- [60] P. D. Soden, A. S. Kaddour and M. J. Hinton, "Recommendations for designers and researchers resulting from the world-wide failure exercise," *Composites Science and Technology*, vol. 64, no. 3-4, pp. 589-604, March 2004.

- [61] S. T. Pinho, L. Iannucci and P. Robinson, "Physically-based failure models and criteria for laminated fibre-reinforced composites with emphasis on fibre kinking: Part I: Development," *Composites A: Applied Science and Manufacturing*, vol. 37, no. 1, pp. 63-73, 2006.
- [62] A. Puck and H. Schuermann, "Failure analysis of FRP laminates by means of physically based phenomenological models.,," *Composites Science and Technology*, vol. 62, no. 12-13, pp. 12-13, 2002.
- [63] G. Glinka, "Fatigue and Fracture of Materials and Structures (A practical approach)," August 2014. [Online]. [Accessed 2015].