

MSC Fatigue 2021.2

User's Guide

HEXAGON | MSC Software

Corporate
MSC Software Corporation
5161 California Ave. Suite 200
University Research Park, Irvine, CA 92617
Telephone:(714) 540-8900
Toll Free Number: 1 855 672 7638
Email: americas.contact@mscsoftware.com

Europe, Middle East, Africa
MSC Software GmbH
Am Moosfeld 13
81829 Munich, Germany
Telephone: (49) 89 431 98 70
Email: europe@mscsoftware.com

Japan
MSC Software Japan Ltd.
Shinjuku First West 8F
23-7 Nishi Shinjuku
1-Chome, Shinjuku-Ku
Tokyo 160-0023, JAPAN
Telephone: (81) (3)-6911-1200
Email: MSCJ.Market@mscsoftware.com

Asia-Pacific
MSC Software (S) Pte. Ltd.
100 Beach Road
#16-05 Shaw Tower
Singapore 189702
Telephone: 65-6272-0082
Email: APAC.Contact@mscsoftware.com

Worldwide Web
www.mscsoftware.com

Support
http://www.mscsoftware.com/Contents/Services/Technical-Support/Contact-Technical-Support.aspx

Disclaimer

This documentation, as well as the software described in it, is furnished under license and may be used only in accordance with the terms of such license.

MSC Software Corporation reserves the right to make changes in specifications and other information contained in this document without prior notice.

The concepts, methods, and examples presented in this text are for illustrative and educational purposes only, and are not intended to be exhaustive or to apply to any particular engineering problem or design. MSC Software Corporation assumes no liability or responsibility to any person or company for direct or indirect damages resulting from the use of any information contained herein.

User Documentation: Copyright ©2021 MSC Software Corporation. All Rights Reserved.

This notice shall be marked on any reproduction of this documentation, in whole or in part. Any reproduction or distribution of this document, in whole or in part, without the prior written consent of MSC Software Corporation is prohibited.

This software may contain certain third-party software that is protected by copyright and licensed from MSC Software suppliers. Additional terms and conditions and/or notices may apply for certain third party software. Such additional third party software terms and conditions and/or notices may be set forth in documentation and/or at http://www.mscsoftware.com/thirdpartysoftware (or successor website designated by MSC from time to time).

The MSC Software Logo, MSC, MSC Nastran, Marc, Patran, Dytran, and Laminate Modeler are trademarks or registered trademarks of MSC Software Corporation in the United States and/or other countries. Hexagon and the Hexagon logo are trademarks or registered trademarks of Hexagon AB and/or its subsidiaries.

NASTRAN is a registered trademark of NASA. PAM-CRASH is a trademark or registered trademark of ESI Group. SAMCEF is a trademark or registered trademark of Samtech SA. LS-DYNA is a trademark or registered trademark of Livermore Software Technology Corporation. ANSYS is a registered trademark of SAS IP, Inc., a wholly owned subsidiary of ANSYS Inc. ACIS is a registered trademark of Spatial Technology, Inc. ABAQUS, and CATIA are registered trademark of Dassault Systemes, SA. FLEXlm and FlexNet Publisher are trademarks or registered trademarks of Flexera Software. PostScript is a registered trademark of Adobe Systems, Inc. PTC and Pro/ENGINEER are trademarks or registered trademarks of Parametric Technology Corporation or its subsidiaries in the United States and/or other countries. Unigraphics, Parasolid and I-DEAS are registered trademarks of Siemens Product Lifecycle Management, Inc. All other brand names, product names or trademarks belong to their respective owners.

FA:V2021.2:Z:Z:Z:DC-USR-PDF

Documentation Feedback

At MSC Software, we strive to produce the highest quality documentation and welcome your feedback. If you have comments or suggestions about our documentation, write to us at: documentation-feedback@mscsoftware.com.

Please include the following information with your feedback:

- Document name
- Release/Version number
- Chapter/Section name
- Topic title (for Online Help)
- Brief description of the content (for example, incomplete/incorrect information, grammatical errors, information that requires clarification or more details and so on).
- Your suggestions for correcting/improving documentation

You may also provide your feedback about MSC Software documentation by taking a short 5-minute survey at: http://msc-documentation.questionpro.com.

Note:	The above mentioned e-mail address is only for providing documentation specific feedback. If you have any technical problems, issues, or queries, please contact Technical Support.

Contents

MSC Fatigue User's Guide

1 Introduction

Purpose . . . 2
Overview of MSC Fatigue . . . 2
Features of MSC Fatigue . . . 3
Architecture of MSC Fatigue . . . 5
Organization of Guide . . . 6
Technology Integration for Durability Management . . . 8

2 Using MSC Fatigue - The DTLib-Based Solver

Introduction . . . 12
Job Setup . . . 13
General Solution Parameters . . . 14
Detailed Solution Parameters . . . 19
Hot-Spot Filtering . . . 23
FE Results Access . . . 25
Fatigue Results Output . . . 27
Job Control . . . 30
Submit Full Analysis . . . 31
Translate Only . . . 33
Save Job Only . . . 33
Monitor Job . . . 34
Abort Job . . . 35
Delete Job . . . 36
Read Saved Job . . . 36
Calculate Normals . . . 37
Compute Stress Gradients . . . 38
Fatigue Preferences . . . 38
Command Line Usage . . . 40

3 Using MSC Fatigue - The Classic Modules

Introduction . . . 44
The Patran Environment . . . 45
Job Setup . . . 47
- General Setup Parameters . . . 47
- Solution Parameters . . . 51
- Materials Information Form . . . 61
- Loading Information Form . . . 70

Job Control . . . 85
- Submit Full Analysis . . . 86
- Submit Partial Analysis . . . 89
- Translate Only . . . 90
- Save Job Only . . . 90
- Monitor Job . . . 91
- Abort Job . . . 93
- Delete Job . . . 93
- Read Saved Job . . . 94
- Calculate Normals . . . 94
- Interactive . . . 95
- Analysis Manager . . . 96

Postprocessing Results . . . 100
- Read Results . . . 101
- List Results . . . 106
- Re-Analyze . . . 106
- Design Optimization . . . 106
- Factor of Safety . . . 107
- Sensitivity Plots . . . 107
- Extract Time History . . . 107
- Extract PSD . . . 108
- Identify Location . . . 108

Other Modes of Job Setup . . . 109

4 Material Management

Introduction to dtmat - The New Material Database Manager . . . 114
Supported Material Types . . . **114**
Material Information Updates . . . **116**
The dtmat User Interface . . . 117
- Opening a Database . . . 118
- dtmat Units Preferences . . . 119
- Creating a New Database . . . 120

Adding Material Data Sets . . . 121
Working with Data Sets . . . 123
Viewing Material Properties Graphs . . . 125
Exporting / Importing Data Sets . . . 126
Adding Multi-Curve Material Data Sets . . . 128
Creation of a new nCode SN Mean Stress Curves S-N material . . . 132
Introduction to PFMAT . . . 137
PFMAT Menu Options . . . 139
Full List Option . . . 140
Search and List Option . . . 141
Tabulate Options . . . 143
Load Option . . . 147
Unload Option . . . 149
Edit Option . . . 149
Create Option . . . 154
Delete Option . . . 160
Weld Classifier Option . . . 162
Graphical Display Option . . . 164
Preferences Option . . . 167
Exit Option . . . 168
Component vs. Material S-N Curves . . . 169
Rules for Changing Young's Modulus . . . 171
PFMAT in BATCH Mode . . . 173
PFMAT Material Listing . . . 177
Grey Cast Iron . . . 190
Characterization of Material Properties for grey cast iron . . . 191
Materials with Temperature Dependent Properties . . . 192
Materials with Multiple Mean S-N Curves . . . 195
Mean Stress Correction . . . 196

5 Loading Management

Automatic Time History Recognition . . . 200
Multi-Channel Time History Support . . . 201
PSD And Cross-Spectra Automation Support . . . 202
Introduction to PTIME . . . 208
PTIME Menu Options . . . 213
Add an Entry Option . . . 213
Change an Entry Option . . . 235
List All Entries Option . . . 242

Search and List Option . . . 243
Plot an Entry Option . . . 245
Delete Entries Option . . . 248
Validate Database Option . . . 249
Multi-Channel . . . 250
New Directory . . . 250
Exit Option . . . 251
Multi-File Display (MMFD) . . . 251
Peak-Valley Extraction (MPVXMUL) . . . 254
Auto Spectral Density (MASD) . . . 260
PTIME Central Database Listing . . . 273
DAC File Format Description . . . 274
Loading and Units . . . 283
Aerospace Spectrum File Support . . . 286
File Definitions . . . 286
Spectrum file . . . 286
Example Spectrum File . . . 287
Load Control file . . . 287
Example Load Control File . . . 287

6 Total Life and Crack Initiation

Introduction . . . 290
Terminal Definition . . . 290
Basic Information . . . 291
Analysis Route . . . 293
Necessary Files . . . 293
The Translator (PAT3FAT or FATTRANS) . . . 294
The FATTRANS Translator . . . 295
Job Setup(dtfat) . . . 297
Solution Parameters (S-N) . . . 297
Solution Parameters (E-N) . . . 300
Material Information Updates . . . 306
Multi-Curve Materials (dtfat) . . . 307
Notes on Mean Stress Correction and the Multi-Curve Methods . . . 308
Temperature Dependent Fatigue (dtfat) . . . 308
Thermal Cycling (dtfat) . . . 310
dtfat Batch Operation . . . 316
FE Fatigue Analysis Options (FEFAT) . . . 317
Fatigue Preprocessing . . . 318

Fatigue Analysis . . . 321
Factor of Safety Analysis . . . 325
Design Optimization . . . 330
Assess Multiaxiality . . . 350
Output Time Histories . . . 353
Graphical Display of Time Histories . . . 355
Matrix Options . . . 356
Results Processing . . . 360
Utilities . . . 360
Reviewing Results (PFPOST) . . . 363
Fast Analysis (FASTAN) . . . 368
Description of Files . . . 369
The Job Information File (jobname.fin) . . . 369
The Submit Script (FatigueSubmit) . . . 383
The Analysis Manager Execution Script (FatigueExecute) . . . 394
The Fatigue Input File (jobname.fes) . . . 401
The Preprocessed File (jobname.fpp) . . . 410
The Results Files (jobname.fef/fos) . . . 414
FEFAT Batch Operation . . . 417

7 Multiaxial Fatigue

Introduction . . . 422
Multiaxial Factor of Safety (dtfat) . . . 424
Global Multiaxial Fatigue Life Analyzer . . . 427
Safety Factor Analysis . . . 429
Crack Initiation Life Analysis . . . 433
FEMLF Batch Operation . . . 438
Local Multiaxial Stress/Strain Fatigue Analyzer (MMLF) . . . 438
Summary of Method . . . 439
MMLF Module Operation . . . 453
The Postprocessing Menu . . . 474
MMLF Environment Keywords . . . 478
MMLF Batch Operation . . . 479
Multiaxial Fatigue Theory . . . 481
Multiaxial Stress-Strain State . . . 481
Theories of Multiaxial Fatigue . . . 485
Critical Plane Approaches . . . 490
Multiaxial Low Cycle Fatigue Analysis . . . 503
Multiaxial Safety Factor Analysis . . . 524

8 Crack Growth

Introduction . . . 532
Terminal Definition . . . 533
Analysis Route . . . 533
Necessary Files . . . 534
K Solution Library (PKSOL) . . . 535
Background Information . . . 535
Module Operation . . . 536
Post Analysis Options . . . 551
PKSOL K Solution References . . . 554
Crack Growth Prediction (PCRACK) . . . 556
Module Operation . . . 556
Post Analysis Menu . . . 567
Reviewing Results (PCPOST) . . . 571
PCPOST Module Opera . . . tion 572
Batch Operations . . . 578
Crack Growth Data Analysis (MFCG) . . . 580
MFCG Module Operation . . . 584
NASA/FLAGRO . . . 589
Crack Growth Analysis Theory . . . 593
Basic Principles of Fracture Mechanics . . . 594
MSC Fatigue Crack Growth Models . . . 597
The Service Environment . . . 598
Geometric Description . . . 599
Materials Response . . . 599
Cycle-by-Cycle Approach . . . 600
Applications of Fracture Mechanics . . . 621
Estimation of LEFM Data . . . 622
Multiaxial Stresses and Crack Propagation Methods . . . 623

9 Vibration Fatigue

Introduction . . . 626
Terminal Definition . . . 626
Basic Information . . . 627
Analysis Route . . . 628
Necessary Files . . . 629
The Translator (PAT3FAT or FATTRANS) and Submit Script . . . 629
Job Setup (dtfat) . . . 630
Solution Parameters (Vibration) . . . 630
Superposition of Static Load Case . . . 636

Loading Information (Single PSD) . . . 636
Loading Information (Multiple PSD) . . . 639
Loading Information (Sine Sweep) . . . 641
Loading Information (Sine Dwell) . . . 644
Loading Information (Sine on Random) . . . 646
dtfat Batch Operation . . . 648
Job Setup (Classic Modules) . . . 649
Solution Parameters . . . 650
Materials Information Form . . . 653
Loading Information Form . . . 654
Job Control . . . 663
Postprocessing Results . . . 667
FE Vibration Fatigue Analysis (FEVIB) . . . 670
Global Vibration Fatigue Analysis . . . 671
Design Optimization . . . 674
Output Power Spectrum . . . 690
Graphically Display a PSD . . . 692
Results Postprocessing . . . 692
Utilities . . . 693
FEVIB Batch Operation . . . 693
Frequency Fatigue Life Estimation (MFLF) . . . 695
MFLF Module Operation . . . 696
MFLF Postprocessing Menu . . . 707
MFLF Batch Operation . . . 709
Vibration Fatigue Theory . . . 710
An Introduction to Random Process Theory . . . 717
What Does the FFT Tell Us? . . . 730
How Do We Use FFTs? . . . 731
Time Domain Characterization of Fatigue Life Estimation . . . 740
Characterization of Structural Response in the Frequency Domain . . . 747
Frequency Domain Approaches of Life Estimation . . . 749
Vibration Analysis using Finite Elements . . . 754

10 Weld Analysis

Introduction . . . 772
CWELD Modeling . . . 774
Spot Weld Job Setup (dtfat) . . . 774
Solution Parameters (Spot Weld) . . . 774
dtfat Batch Operation . . . 777
Job Setup (Classic Modules) . . . 777

Solution Parameters . . . 782
Materials Information . . . 783
Loading Information . . . 786
Job Control . . . 787
Results . . . 790
Spot Weld Analyzer (SPOTW) . . . 792
Estimate Fatigue Life . . . 794
Design Optimization . . . 797
List Global Results . . . 814
List .spt File . . . 814
Results Polar Plot . . . 815
Three Sheet Correction . . . 816
Description of Files . . . 817
SPOTW Batch Operation . . . 819
Polar Display (MPOD) . . . 820
MPOD Module Operation . . . 820
MPOD Extra Details Keywords . . . 827
MPOD Batch Operation . . . 827
Spot Weld Analysis Theory . . . 828
Seam Weld Analysis . . . 835
Introduction . . . 835
Grouping and Weld Types . . . 837
Modelling Guidelines and Calculation Points . . . 840
Calculation Points and Stress Recovery . . . 845
Fillet Welds . . . 846
Seam Weld Job Setup (dtfat) . . . 851
Solution Parameters (Seam Weld) . . . 853
Material Properties and Interpolation . . . 854
Weld Groups . . . 855
dtfat Batch Operation . . . 857
Seam Weld Job Setup (MFATFE) . . . 857

11 Rotating Structures Analysis

Introduction . . . 870
Job Setup . . . 871
Solution Parameters . . . 875
Materials Information . . . 875
Loading Information . . . 876
Results . . . 882
Wheels Analyzer (FEROT) . . . 883
Analyze . . . 884

Results Postprocess . . . 885
Extract Time Histories . . . 886
FEROT Batch Operation . . . 887

12 Software Strain Gauges

Introduction . . . 890
The Gauge Tool . . . 892
The Gauge Definition File . . . 893
Creating and Modifying Gauges . . . 897
Extracting the FE Results . . . 901
Creating a Fatigue Input File . . . 903
Software Strain Gauge Module (SSG) . . . 906
Technical Details . . . 909
Correlation with Test . . . 909
SSG Batch Operation . . . 910
Stress-Strain Analysis (MSSA) . . . 911
Strain Gauge Rosettes . . . 911
Calculation of Principal Strains & Stresses . . . 913
MSSA Module Operation . . . 917
Graphical Operations . . . 921
MSSA Batch Operation . . . 932

13 Duty Cycle Analysis

Introduction . . . 936
Duty Cycle Setup (dtfat) . . . 936
dcbuilder . . . 939
Interface (Classic Modules) . . . 945
Duty Cycle Setup . . . 947
Sequence Information . . . 949
Event Information . . . 949
Job Submit . . . 951
Review Results . . . 952
Conclusion . . . 953

14 Fatigue Utilities

Fatigue Preprocessing . . . 956
Low Damage Removal . . . 956

Calculate Stress Gradients 957
. . . . . Utilities
959
Advanced Loading Utilities 963
Arithmetic Manipulation - (MART) 963
Multi-Channel Editor - (MCOE) 970
Rainflow Cycle Counter - (MCYC) 980
Formula Processor (MFRM) 984
File Cut and Paste - (MLEN) 1016
Multi-File Manipulation - (MMFM) 1024
Peak-Valley Extraction - (MPVXMUL) 1031
Simultaneous Values Analysis DAC/RPC - (MSIMMAX) 1032
Amplitude Distribution - (MADA) 1034
Auto Spectral Density - (MASD) 1035
Fast Fourier Filter - (MFFF) 1036
Butterworth Filtration - (MBFL) 1043
Frequency Response Analysis - (MFRA) 1049
Statistical Analysis - (MRSTAT) 1061
Header/Footer Manipulation - (MFILMNP) 1071
Advanced Fatigue Utilities 1083
Single Location S-N Analysis - (MSLF) 1083
Single Location e-N Analysis - (MCLF) 1092
Cycle and Damage Analysis - (MCDA) 1111
Cycles File Lister - (MCYL) 1116
Time Correlated Damage - (MTCD) 1121
Single Location Vibration Fatigue - (MFLF) 1128
Stress-Strain Analysis - (MSSA) 1130
Multi-Axial Life Analysis - (MMLF) 1130
Crack Growth Data Analysis - (MFCG) 1132
Kt/Kf Evaluation - (MKTAN) 1133
Graphics Display Utilities 1143
Graphical Editing - (MGED) 1143
Multi-File Display - (MMFD) 1144
Quick Look Display - (MQLD) 1145
Two Parameter Display - (MTPD) 1146
Polar Display - (MPOD) 1147
Three Dimensional Display - (MP3D) 1148
Linux Based Plotting Utility - (MQPLOT) 1149
Windows-Based Plotting Utility - (MWNPLOT) 1152
Printer/Plotter Definition Module - (MPLTSYS) 1159
Plot/Pen Colors Utility - (MNCPENS) 1177
File Conversion Utilities 1178
Binary/ASCII Convertor - (MDTA/MATD) 1178
Signal Regeneration - (MREGEN) 1195

RPC to DAC - DAC to RPC - (MREMDAC/MDACREM) . . . 1202
Regenerate Time Histories from PSDs. . . . 1210
Cross-Platform Conversion - (MCONFIL) . . . 1213
Waterfall File Create - (MWFLCRE) . . . 1216

15 Validation Problems

Introduction . . . 1222
Problem 1: Analysis of a Keyhole Specimen . . . 1223
S-N Analysis of Keyhole . . . 1225
Crack Initiation of Keyhole . . . 1235
Crack Growth of Keyhole . . . 1243
Keyhole Results . . . 1250
Problem 2: Analysis of an Underwater Pressure Vessel . . . 1262
Crack Initiation of Weld . . . 1263
Crack Growth of Weld . . . 1270
Problem 3: Comparison to Another Code . . . 1276
Problem 4: Ten Simple Notched Geometries . . . 1286
Problem 5: Transient Results . . . 1296
Simple Transient Comparison . . . 1296
MSC Nastran Transient Results . . . 1300
Problem 6: Spot Weld Analysis . . . 1305
H-Profile Spot Weld . . . 1305
T-Beam Specimen . . . 1309
Problem 7: Vibration Fatigue Analysis . . . 1311
A Simple Worked Example . . . 1311
Measured Responses . . . 1319
Finite Element Model Responses . . . 1331

16 Fatigue Theory

Introduction . . . 1340
Background . . . 1340
The History of Fatigue . . . 1341
High Cycle versus Low Cycle Fatigue . . . 1342
Summary . . . 1342
Inputs to Fatigue Life Estimation Models . . . 1343
Total Life (S-N) Analysis . . . 1351
Stress Cycles . . . 1352
The S-N Curve . . . 1354
Procedure for Determining the S-N Curve . . . 1355

Limits of the S-N Curve . . . 1357
Tensile Properties and the S-N Curve . . . 1358
The Influence of Mean Stress . . . 1359
Factors Influencing Fatigue Life . . . 1364
Application in MSC Fatigue . . . 1374
Crack Initiation/Strain-Life (e-N) Analysis . . . 1384
The Microscopic Aspects of Fatigue Failure . . . 1384
The Strain-Life Methodology . . . 1387
Monotonic Stress-Strain Behavior . . . 1388
Cyclic Stress-Strain Behavior . . . 1396
The Strain-Life Curve . . . 1406
Strain-Life vs. Stress-Life . . . 1407
Transition Life . . . 1408
The Effect of Mean Stress . . . 1410
Factors Influencing Fatigue Life . . . 1412
Application in MSC Fatigue . . . 1413
The Statistical Nature of Fatigue . . . 1421
Representation of Fatigue Data on a Statistical Basis . . . 1423
The Statistical Distribution Function . . . 1424
Probability of Failure at a Finite Life . . . 1424
Probability of Failure for Infinite Life . . . 1426
Handling Statistics Under Random Loading Conditions . . . 1427
The Absolute Accuracy of Fatigue Life Estimation . . . 1432
Estimating Material Cyclic Properties From UTS & E . . . 1432

17 References

References . . . 1438
Further Reading . . . 1443

18 Module Operations

Terminal Definition . . . 1446
MASK (X) Mode . . . 1446
Description of the Text Screen . . . 1447
Using the Keypad . . . 1448
Special Commands . . . 1449
Correcting Input Errors on the Text Screen . . . 1450
Inputting a Range of Numbers . . . 1451
List Function Operation . . . 1451
Graphical Screen Manipulation . . . 1453
Graphical Mouse Operations . . . 1453
Graphical Command Line . . . 1454

Font Selection (PFSETFONT) 1455
MOTIF (and Windows) Mode 1455
Graphical Options 1457
Graphical Command Line 1458
Modifying the MSC Fatigue Environment (MENM) 1463

19 Limitations and Error Messages

Program Limitations 1470
Error Messages 1470
PCL Form Messages 1470
Translator (PAT3FAT) Messages 1473
Analyzers (FEFAT, PCRACK) 1479
Analyzer FEVIB 1484

20 MSC Fatigue Information

About MSC Fatigue 1488
Using MSC Fatigue 1488
Common MSC Fatigue Issues 1488
MSC Fatigue and a Corporate Durability Management System 1489
More Information about nCode International 1489

1 Introduction

- Purpose 2
- Overview of MSC Fatigue 2
- Features of MSC Fatigue 3
- Architecture of MSC Fatigue 5
- Organization of Guide 6
- Technology Integration for Durability Management 8

Purpose

MSC Fatigue is an advanced fatigue life estimation software package for use with finite element analysis results. It provides state-of-the-art fatigue analysis tools which can be used to optimize the life of a product early in the design process. MSC Fatigue has been developed jointly by nCode International Ltd. and MSC Software Corporation (MSC). nCode is a recognized leader worldwide in durability management solutions with specialties in fatigue life estimation software systems and consultancy services. MSC in the world market leader in finite element analysis solutions.

MSC Fatigue can run on its own with its own graphical pre- and postprocessor but is also tightly integrated into the Patran environment. All of MSC Fatigue's analysis capabilities and generated results are available from within the Patran environment. However, it is flexible enough to be run outside of the Patran or its own graphical environment. With the tight integration within Patran, the casual user will never need to be aware that separate programs are actually used.

For the expert user, there are various components that make up MSC Fatigue: a PCL (PATRAN Command Language) library to provide the customization of Patran for MSC Fatigue, a translator to convert model data from Patran into the fatigue analysis code and to bring results from the fatigue analysis code back into the Patran database, as well as the various fatigue analysis modules that make up MSC Fatigue itself.

With the addition of the nCode DTLib library, future versions of MSC Fatigue will contain additional customization paths. This includes custom user customizable S-N methods and integration of products such as MSC Digimat. DTLib also allows the support for new time history file formats.

MSC Fatigue is used most effectively within its own pre- and postprocessor or within the Patran environment and not in isolation. It requires loading data and material properties which will often be supplied by measurement and materials test laboratories respectively and also requires preprocessing of finite element stress data. The fatigue life estimation process embodied in a MSC Fatigue analysis can also be correlated with test results. The integration of the measurement, test, and design analysis functions, to develop optimized products, is known as Integrated Durability Management (IDM). IDM is the key to the effective use of MSC Fatigue. See Technology Integration for Durability Management.

Overview of MSC Fatigue

In today's competitive marketplace, companies are seeking to improve product quality, performance, and durability, as well as shorten time-to-market. The complexity of modern products demands the use of computerized engineering analysis to optimize the product design and to increase competitiveness.

Analysis reduces development costs by allowing the effects of varying design parameters to be explored quickly and efficiently in the computer, without building prototypes. For the structural analyst whose goal is to predict stresses and displacements, these parameters typically include geometry changes, loading, and choice of materials. However, the prediction of stresses is only part of product design optimization. An additional critical requirement of computerized engineering analysis in the product's design process is to estimate the product's serviceable life. Analytically predicting fatigue life is, therefore, an important tool.

Designers and engineers often use simple handbook calculations to address product durability during the conceptual design phase. Frequently however, product durability is not examined in detail until the prototype and testing phases of the development cycle. Furthermore, considerable expense can be incurred in reaching the prototype and testing phase.

The use of testing alone cannot evaluate all design parameters and oftentimes fatigue-related failures are not discovered until the product has been in service for a length of time. At this stage, fatigue problems can have disastrous effects that include costly recalls, damaged product reputations, and possible loss of life.

The ability to assess fatigue related problems and to perform durability analysis to estimate product life early in the design phase offers great benefits to a company in reducing development and testing costs, shortening time-to-market, and improving product life. MSC Fatigue should be a part of any engineering organization interested in these benefits.

Features of MSC Fatigue

MSC Fatigue allows the designer or analyst to carry out fatigue calculations early in the design process. It takes the detailed stress distributions produced by a finite element analysis and a description of the variation of the loading with time, carries out fatigue calculations using state-of-the-art fatigue life estimation methods, and produces fatigue life results in the form of contour plots. These life plots can be displayed using standard Patran imaging tools as well as imaging tools from other popular pre- and post-processing systems.

MSC Fatigue offers a unique integration between finite element analysis (FEA) and fatigue life estimation by enabling the user to select areas of the finite element (FE) model for fatigue life analysis with specific tools provided in the display postprocessor. All three commonly used fatigue life estimation techniques are supported: total life or nominal stress-life (including analysis of welded structures), crack initiation (otherwise known as strain-life), and crack growth or Linear Elastic Fracture Mechanics (LEFM) life estimation techniques.

Both the crack initiation and total life approaches provide the capability to investigate the effect of local changes in surface finish and treatment. A comprehensive materials database is provided that has sophisticated search facilities. Full elastic-plastic transformations are carried out in the crack initiation modeling. Multiple or single static and/or offset load cases may be defined. Fatigue results may be presented in a number of different ways including scaling the results in terms of user-defined units such as hours, miles, flights, etc. Full-color life contour plots may be produced providing a rapid assessment of fatigue critical areas.

Detailed analysis may be carried out to establish sensitivities to variation in material, manufacturing process or loading, using the Design Optimization analyzer. Crack growth may be investigated using the cycle-by-cycle linear elastic growth analysis facilities.

The link between finite elements and fatigue introduces a control on fatigue life performance at an early stage in the design cycle. This integrated approach promotes a total quality policy enhancing the design/development process within a company.

Beginning with the 2011 release of MSC Fatigue, a new solver based on the nCode DTLib library was added. The DTLib-based solver offers a multi-disciplinary fatigue solution, which is actively being updated and extended. A number of new material models is available, as well as updated methods for seam and spot weld and vibration fatigue analysis. All new development is focused on the DTLib-based solution.

Key Features of MSC Fatigue

1. **Total Life** analysis (S-N) based on the nominal stress-life method using rainflow cycle counting and Palmgren-Miner linear damage summation. Various analysis parameters may be chosen such as mean stress correction methods and confidence parameters. Both component and material S-N curves may be accessed. Material S-N curves allow for specification of material surface finish and treatment. Multi-Curve material models are available, such as Muti-Mean Curve and Multi R-Ratio Curve to incorporate material test information.
2. **Crack Initiation** analysis (ε-N) or the local strain method using cyclic stress-strain modeling and Neuber elastic-plastic correction. The mean stress correction method, surface finish and treatment factors may be adjusted to investigate the effect of these fatigue dependent parameters. Multi-Curve material models are available, such as Muti-Mean Curve and Multi R-Ratio Curve to incorporate material test information.
3. **Crack Growth** analysis using linear elastic fracture mechanics (LEFM) and cycle-by-cycle modeling of crack closure due to overloads, the effect of chemical environment, the loading rate and history effects. On-line displays of crack progress report the rate of crack growth, and postprocessing menus enable interpolation of results.
4. **Factor of Safety** analysis for structures designed for infinite life (such as powertrain and engine components) is available also for the crack initiation and the total life methods. A Dang Van multi-axial factor of safety analysis is also available.
5. Fatigue analysis of steel or aluminum welded structures using the total life approach as defined in the British Standard, BS7608, design code including a **Weld Classifier.**
6. A state-of-the-art **Spot Weld** analyzer is also available module where spot welds are modeled in MSC Nastran using stiff bar, CWELD, or CHEXA elements between two or three sheets. The forces in the weld elements are then converted to stress and used in a S-N analysis using the Rupp-Stoerzel-Grubisic method. The method calculates fatigue life on the basis of structural stresses around each spot weld which are then calculated on the basis of the cross-sectional forces and moments in the weld elements. A relatively coarse mesh is required, and results can be visualised to good effect using INSIGHT.
7. An advanced **Seam Weld** solver is available where thin shell structures are modeled with fillet, overlap, and other weld type connections. The Volvo method is used where the 2-D stress tensor on the top and bottom shell surface is decomposed into flexible and stiff components. The fatigue material properties interpolated based on the proportion of flexible to stiff components.
8. **Vibration Fatigue** aanalysis calculates fatigue lives directly from Power Spectral Density Functions (PSDF or PSD) using the S-N method. This is a very powerful capability when it is not convenient to analyze a structure in the time domain, making it necessary to do a random vibration analysis. Partially of fully deterministic methods are also available, such as Sine on Random, Sine Sweep, and Sine Dwell.
9. Global life estimates presented as color fringe contour plots enable the rapid assimilation of the **Results** and easy identification of the fatigue critical areas.
10. Interactive **Design Optimization** allowing the rapid assessment of analysis parameters and design options including alternative geometries, surface finishes, surface treatments, weld details, or materials.These features are typically available in the Classic solver modules.

11. **Materials Database** loaded with standard fatigue data sets. Access to the database is provided by a sophisticated materials database which offers loading, editing, creating, searching, and data visualization. Multi-Curve material models are also supported such as Multi-Mean Curve, Multi R-Ratio Curve, Haigh Curve, and Multi-Temperature Curve. A number of multi-curve materials are pre-loaded in the materials database. The Multi-Curve materials are typically available with the DTLib-based solver.
12. Loading **Time History Database** manager provides a method of archiving loading time histories together with their details. In addition, full graphical editing and signal creation facilities offer the ability to prepare time histories, spectrums and PSDs from measured field data or artificially synthesized data. Multi-Channel loading time histories are supported in the MSC Fatigue User interface.
13. The combined effect of **Multiple Loading** histories may be explored together with **Multiple Materials** datasets on one structure. Substructures may be analyzed by selecting specific geometric entities within Patran.Modeling of a number loading regimes can be performed using the Duty Cycle analysis feature. An aero-spectrum loading feature is also available to evaluate fatigue life under one or more mission profiles.
14. A **Biaxiality Analysis** feature helps in determination of necessary fatigue analysis methods when complex multi-axial loadings are involved and the validity of an associated fatigue analysis. Corrections can be made for proportional loading and if non-proportional loading is determined a separate module allows for **Multiaxial Fatigue** life calculations.A multiaxial Dang Van factor of safety analysis is also available.
15. Finite element stress/strain results may be used from either **Linear Static/ Transient Dynamic/Forced Vibration or Frequency Response/Random Vibration** analyses. Results are read directly from the Patran database or from external FE results files from analysis codes such as MSC Nastran, MSC Marc, Abaqus, and Ansys.
16. An interface to **NASA/FLAGRO** is also featured within MSC Fatigue via the Patran PCL forms. NASA/FLAGRO is a two dimensional crack growth code developed by NASA which is complimentary to the crack growth module featured in MSC Fatigue.
17. A **Software Strain Gauge** module allows the MSC Fatigue user within the Patran environment to simulate an actual strain gauge. This allows for extraction of time varying strain results from a fatigue analysis in the coordinate system(s) of the strain gauge for test/analysis comparison and correlation.
18. A number of useful **Utilities** are include as a separate module of MSC Fatigue. These include many time history manipulation and display utilities and additional fatigue life analysis features such as calculations from measured stress or strain data and time correlated damage.

Architecture of MSC Fatigue

MSC Fatigue is organized into three distinct analysis modes.

The first mode is the **Global** multi-location analysis which allows selection of a region or regions containing nodes and/or elements of the FE model in which to carry out a fatigue life analysis. The life estimates may be contoured to provide a visual indication of the fatigue critical areas. This mode of operation relies on a tight integration between Patran and the MSC Fatigue system, or use of MSC Fatigue's own graphical display.

Retrieval of the FE results is generally achieved by direct access to the Patran database. The definition of MSC Fatigue job parameters is achieved through the use of forms and menus accessible directly within Patran or MSC Fatigue's own graphical display. The fatigue analysis may be submitted, monitored, and aborted, directly from these forms and menus. All fatigue analysis methods may be accessed through this Global method.

Another mode of operation provides a **Design Optimization** capability based on calculations at a localized node or element. This mode of operation provides the engineer with a fast and efficient method of assessing alternative materials, surface finishes/treatments, weld procedures, etc. In general terms, when the engineer has located the fatigue problem areas using the Global multi-node/element analysis, the design may then be optimized in terms of fatigue performance using the Design Optimization facilities. These analysis modules use information from the Global analysis to provide the ability to carry out fatigue life estimation rapidly in a localized region, assessing various design optimization options until a solution has been found which meets the fatigue life criteria. A back calculation facility allows the definition of a desired life and the analysis code will find the optimum value of a chosen parameter to meet that life requirement.

Having located the region where a crack will initiate the progress of the crack as it grows can be modeled using the **Crack Growth** analysis capabilities. This is the third mode of operation.

In addition to these three modes there are additional modules for more specialized analyses such as mentioned in the previous section. These include things such as spot welds, software strain gauges, factors of safety, multi-axial assessments, and much more.

Organization of Guide

This document describes the use of MSC Fatigue to solve fatigue problems using finite element analysis results. The guide includes the following chapter topics:

Introduction (Ch. 1) provides an overview of the features of MSC Fatigue.Overviews of the DTLib-based solver and the Classic modules are provided.

Using MSC Fatigue - The DTLib-Based Solver (Ch. 2) describes the details involved in setting up, submitting, monitoring, and aborting a MSC Fatigue DTLib-based solver. An overview of the differences between the DTLib-based and Classic module solvers is also given.

Using MSC Fatigue - The Classic Modules (Ch. 3) describes the details involved in setting up, submitting, monitoring, and aborting a MSC Fatigue job both within the Patran environment and in its own stand alone mode. Also the graphical postprocessing of results is described including two modules, PFPOST and PCPOST, used for tabular viewing of fatigue results.

Material Management (Ch. 4) describes in detail the creation and manipulation of cyclic material properties using the materials database managers: dtmat and PFMAT. The purpose and limitations of each utility is described. Each can be accessed directly from the MSC Fatigue setup forms or from the computer's operating system

Loading Management (Ch. 5) describes in detail the management, creation and manipulation of loading time histories, power spectral density functions (PSDs), and rainflow matrices (spectrums). The tools available for both the DTLib-based and Classic module solvers is described. Usage of the database manager, PTIME, which can be accessed directly from the MSC Fatigue setup forms or from the computer's operating system is described. Other modules for multiple graphical displays, MMFD, and peak-valley-extraction, MPVXMUL, are also described.

Total Life and Crack Initiation (Ch. 6) describes the operation of the S-N and E-N fatigue solvers. This includes dtfat, the DTLib-based solver and the Classic modules, MFATFE and FEFAT. A description of features including the fatigue preprocessing (rainflow cycle counting), fatigue analysis, sensitivity studies, and factor of safety calculations is given. The details of the Classic modules as both an interactive and batch operation, which offers a fast route for processing multiple MSC Fatigue jobs, is explained also.

Multiaxial Fatigue (Ch. 7) addresses multiaxial consideration when loading conditions are nonproportional. Standard accepted uniaxial methods break down when these conditions exist.

Crack Growth (Ch. 8) describes the crack growth module, PCRACK, both its interactive and batch operations. The K solution or compliance function preparation module, PKSOL, for defining crack geometries is also described in this chapter as well as postprocessing options. The interface to NASA/FLAGRO is also explained in this chapter.

Vibration Fatigue (Ch. 9) describes the operation of the fvibration fatigue analyzers dtfat and FEVIB, where loading is in the form of PSDFs and the finite element results are from frequency response or random vibration analyses.

Weld Analysis (Ch. 10) describes the DTLib-based fatigue spot weld and seam weld analyzers, dtfat and the Classic modules SPOTW and MFATFE. Display of scalar and polar results is also explained in this chapter.

Rotating Structures Analysis (Ch. 11) describes the operation of the WHEELS module

Software Strain Gauges (Ch. 12) describes the creation of software strain gauges, directly on to locations of the FE model, the extraction of FE results from these gauges, and the subsequent fatigue analysis, SSG, of these results with possible correlation to actual data from hardware strain gauges.

Duty Cycle Analysis (Ch. 13) describes the capabilities available for modeling loading under various regimes. The tools available for the DTLib-based solver are described as well as those available for the Classic modules. A description of the aero-spectrum loading approach is also given.

Fatigue Utilities (Ch. 14) describes the collection of utility modules for advanced loading and cycle/damage display and manipulation as well as other useful utilities such as cross-platform file transfers, hardcopy plotting, and fatigue calculations from measured stress and/or strain data.

Validation Problems (Ch. 15) shows detailed examples of each of the fatigue analyzers available in MSC Fatigue including how to set up jobs, submit and monitor them and evaluate the results. Some of these problems are used for benchmarking and validation of the MSC Fatigue application.

Fatigue Theory (Ch. 16) covers the basic fatigue theories for S-N and crack initiation analysis used in MSC Fatigue. Theory for other analysis types are covered in their respective chapters. This chapter in not intended to be complete technical text on fatigue analysis, but rather a fairly comprehensive overview of the techniques adopted by MSC Fatigue. Note that a separate document, The MSC Fatigue Theory Guide, is available which gives details specifically on the theory behind the DTLib-based solver.

Various appendices cover technical References (Ch. 17), different aspects of Module Operations (Ch. 18), and Limitations and Error Messages (Ch. 19).

A comprehensive Quick Start Guide is also available separately from this guide to quickly get you started as a productive MSC Fatigue user including several useful example problems.

Technology Integration for Durability Management

Durability Management is the control and organization of design, test, and production, to ensure products are developed to meet the required life within cost and on time. The process has evolved over the last 150 years since fatigue failures were first recognized. While there are many technologies that have contributed to the understanding of fatigue and to the solution of fatigue problems, two major procedures are used in durability management: fatigue testing and fatigue modelling.

Fatigue Testing

The first fatigue tests were carried out on full-scale components to establish their safe working stress. Later, the more complete relationship between cyclic stress or strain and fatigue life was established. Small scale specimens were tested to study component life and also fatigue mechanisms. In more recent times, as tests had to become increasingly realistic, special test techniques were developed such as Remote Parameter Control. Today, testing is still the most common way of confirming the fatigue life of a product prior to releasing it onto the market. However, testing often reveals weaknesses which necessitate re2wdesign. Assessing the suitability of particular design modifications using fatigue testing alone can be time consuming and cost far more than just a delayed product.

Fatigue Modelling

The estimation of fatigue life using mathematical modelling techniques was developed to assist the engineer in solving fatigue problems without always having to physically test all the options. For this reason, techniques such as local strain or crack initiation modelling have become widely used. Improvements in the power of computers have enabled the effective use of these techniques. Today, most major companies designing mechanical structures will use a fatigue life estimation tool such as MSC Fatigue in conjunction with testing. By the late 1980s the use of finite element analysis (FEA) had become established as a tool for stress analysis. At the same time the integration of FEA and fatigue life estimation through the MSC Fatigue product began to provide new benefits by assessing fatigue earlier in the development process.

Integrated Durability Management

Understanding and effective implementation of durability management strategies require a partnership between test and design analysis. It can reduce product lead time by focusing the use of fatigue testing to the essential correlation and sign-off tests. The use of fatigue modeling, at the design analysis stage, allows more options to be assessed for little incremental cost. Integrated durability management can produce better products more quickly and cheaply.

Figure 1-1 illustrates some of the links and benefits from this integration. In summary they are:

- Correlation of test and analysis to improve the accuracy of analytical design optimization
- Early fatigue life estimates in the design process leading to better products earlier
- Analytical fatigue design optimization based on realistic descriptions of service environments.

With reference to Figure 1-1, service environment descriptions can be edited to produce drive files for fatigue test or nominal loading data for use in FE based fatigue analysis. Fatigue optimization analysis may be carried out on local structural response histories extracted from the FEA based fatigue simulation or from measurements made on structures under test. Early estimates of life at the design stage can be made using

nominal stress data and calibration. Correlation between component test lives and calculated lives form an important step in developing appropriate analytical procedures for specific components.

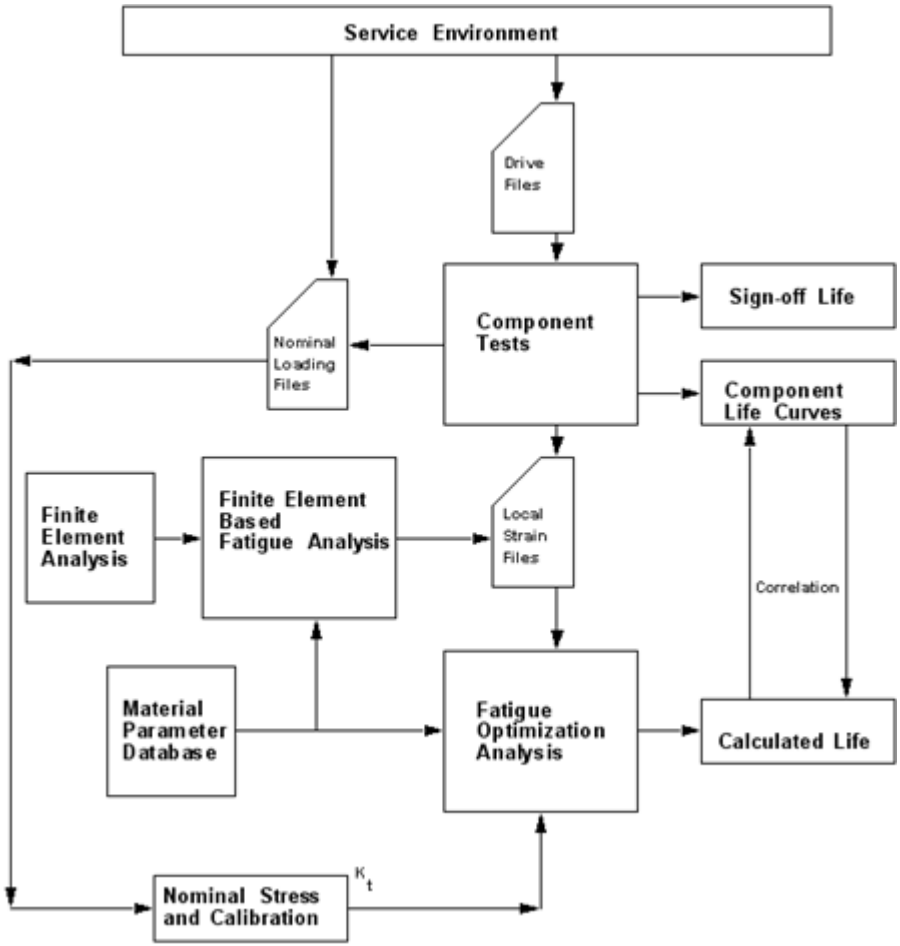


Figure 1-1 Schematic for Integrated Durability Analysis

2 Using MSC Fatigue - The DTLib-Based Solver

- Introduction 12
- Job Setup 13
- General Solution Parameters 14
- Detailed Solution Parameters 19
- Hot-Spot Filtering 23
- FE Results Access 25
- Fatigue Results Output 27
- Job Control 30
- Fatigue Preferences 38
- Command Line Usage 40

Introduction

This chapter introduces the latest addition to MSC Fatigue, the DTLib-based solver dtfat. The solver has new capabilities and options not available in the Classic modules. Most fatigue analysis methods are available in the dtfat solver, except for crack growth, software strain gauge, multi-axial E-N, rotating structures analysis, and the optimization tools. In the chapters that follow, the features and usage of the dtfat solver is described first and then approaches using the Classic modules is described.

A MSC Fatigue analysis is initiated in one of three ways. The most useful and practical way is either through its own graphical interface or directly through the Patran environment. The stand alone operation is described in Job Setup and the Patran environment is described in General Solution Parameters, 14.

A MSC Fatigue job can also be setup outside of any type of graphical environment directly through the MSC Fatigue modules themselves, although this is limiting.

Regardless of how a fatigue analysis is set up there are always three basic inputs that must be specified before an analysis can proceed. Only setup for the basic analysis types (Total Life and Crack Initiation (Ch. 6) and Crack Growth (Ch. 8)) are explained in this chapter. For other more advanced fatigue analysis setup (Vibration Fatigue (Ch. 9), Weld Analysis (Ch. 10), Multiaxial Fatigue (Ch. 7)), see the appropriate chapter.

Note:	DTFatigue forms have not been upgraded to the latest version of QT (QT5) and are still in the old version (QT3).

Preparing for a MSC Fatigue Analysis

The three basic pieces of information needed to compute a fatigue life estimation using MSC Fatigue are:

- **Materials Information:** This is information describing the cyclic fatigue properties of the component or material. This can be accomplished using MSC Fatigue's materials database managers dtmat or PFMAT. The materials data may already be found in the materials database, or it may be generated from the UTS of the material based on empirical formulations, or it may be obtained from materials tests or other references and input manually into the materials database. See Material Management (Ch. 4).
- **Loading Information:** Loads can be in the form of time histories, power spectral density functions (PSDFs) or cycle definitions (rainflow matrices). The loading is defined externally to the FE model in the case of static and frequency response FE analysis or else must be an integral part of the FE analysis for transient and random vibration analyses. External loading can be defined using MSC Fatigue's time history database manager, PTIME or other external tools for multi-channel time histories. See Loading Management (Ch. 5) and Duty Cycle Analysis () for more information.
- **Geometry Information:** For most fatigue analyses, this entails the finite element model and the stress or strain results. In the case of a crack growth analysis, a comprehensive library of compliance functions is provided in the module, PKSOL, to define the crack geometry. To obtain the stress/strain information, an appropriate finite element (FE) analysis must have been performed. Any fatigue analysis begins where a finite element analysis ends. Therefore, this guide does not go into any detail concerning model creation or finite element analysis itself.

The fatigue analysis can be thought of as a “five box trick” (as shown in Figure 2-1) where the first three are the inputs as described above, the fourth is the actual fatigue analysis, and the fifth is the interpretation and/or postprocessing of the results which could lead to a loop back to the first three inputs for sensitivity studies.

This is an important concept in MSC Fatigue in that it allows you to build a “fatigue model.” Fatigue analysis is a logarithmic process, thus amplifying any errors or discrepancies of the three inputs through the analysis process. Sensitivity studies allow for easy and quick interpretation of resulting life predictions to see how sensitive your model is to variations in any of the inputs.

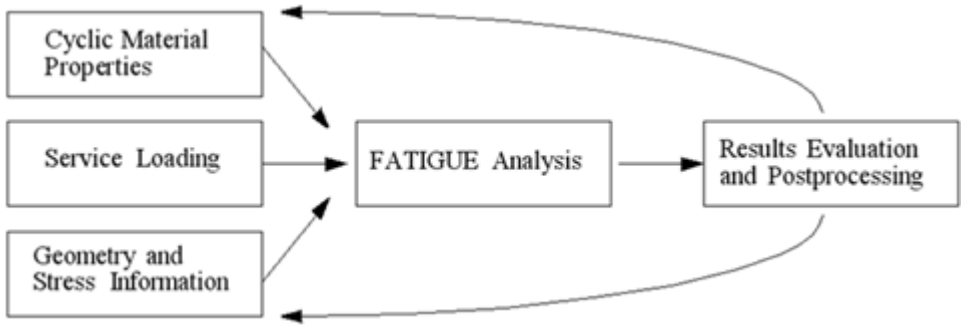


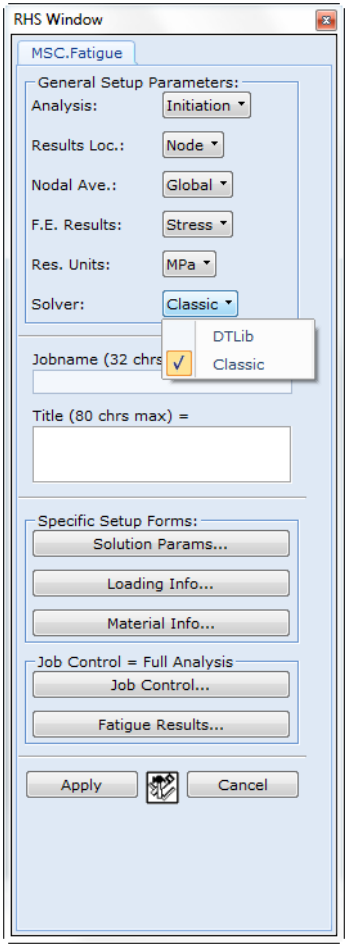
Figure 2-1 The MSC Fatigue “five-box trick”

Job Setup

The MSC Fatigue DTLib-based solver is integrated with Patran. A brief description of the environment is given in **The Patran Environment** section of this document (Ch. 3). A full description of the job setup for each analysis type is given in later chapters of this document. Several aspects of the job setup are common for most analysis types. These are described in the following sections.

When **Analysis** is picked from **Tools/MSC Fatigue** pull-down menu or **Durability** is selected from the ribbon, the form shown below appears. Since the job setup differs significantly between the DTLib-based solver and the Classic solver modules, a Solver Type selection is available on the MSC Fatigue Main form. To setup and manage a job for the DTLib-based solver, select **DTLib** in the pull-down shown below.

The Patran Environment and the implementation of the DTLib-based solver is essentially the same as for the Classic modules. This is described in The Patran Environment (Ch. 3).



General Solution Parameters

All the Fatigue analysis types that use DTLib support a number of common solution parameters. These can be accessed by clicking the **General Solution Parameters** button on the MSC Fatigue Main form. The form is divided in several sections, as can be seen in the figure below. A detailed description of each setting is given in **Table 2-1**.

This first section contains several useful features. The **Number of Processors** option sets the number of processing threads used in the analysis. For machines with multi-core processors, this can speed up processing almost by a multiple of the number chosen. Choosing a number greater than the number of available cores can result in reduced performance.

The **Log File Detail** option allows the user to increase or decrease the output from the dtfat solver. Setting the detail to **High** allows the user to view a great deal of information about the job, which can be viewed in

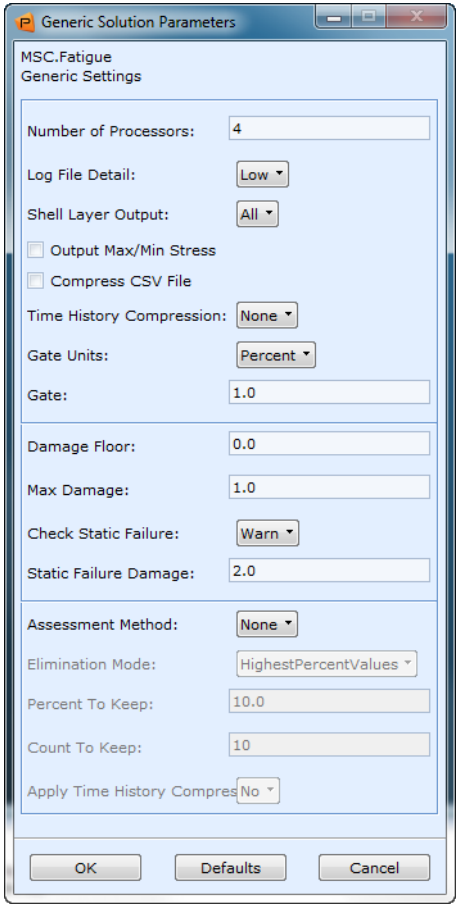
the **$jobname.log** file. At the **High** setting, the log file contains detailed information concerning material property values, stress/strain tensor values, stress cycles, etc. For large models, using the **High** detail setting should be avoided due to performance degradation in writing large amounts of data to the log file.

The **Shell Layer Output**, **Output Max/Min Stress**, and **Compress CSV File**, all are available to maximize or minimize the amount of detail in the fatigue results files. The **Shell Layer Output** and **Output Max/Min Stress** options affect all results output. The **Compress CSV File** option only affects the $jobname.csv output file. Limiting the amount of output to the results files can be useful if only the worst-case results are of interest, or if the total duty cycle results are of interest.

The **Time History Compression** option can be useful for omitting smaller stress cycles from the cycle counting process. This can be especially useful for large models with time histories containing a large number of points. The Time History Compression is best used in comparisons of one design against another, since the removal of stress cycles can lead to non-conservative life estimates. For a detailed explanation of Time History Compression, please see the **Fatigue Theory Guide**.

The **Damage Floor**, **Max Damage**, **Check Static Failure**, and **Static Failure Damage** options are used to control how damage is computed and presented in the fatigue results. The **Max Damage** and **Static Failure Damage** settings are used to differentiate between failure due to fatigue an failure due to static rupture over a single cycle. The **Check Static Failure** option allows the user to ignore or stop processing when a static failure is encountered.

The **Assessment Method** is described further in the **Hot-Spot Filtering** section below.



The table below lists the General Solution Parameters, gives a description of the available options for each, gives a description of the purpose of each, and gives the default value for each parameter

Table 2-1 DTLib General Solution Parameters

Parameter	Options	Description	Default
Number of Processors	**1<= Value <= 100**	The DTLib-based fatigue solver is multi-threaded. This option allows the user to control the number of threads the fatigue solver uses. If the user machine contains multiple processors, a larger number of threads can be specified to enhance performance..	**4**
Log File Detail	**Low, Medium, High**	This option allows the user to control the amount of information to be output from the fatigue solver, which is written to the "$jobname.log" file. The details can be limited to error and information messages or increased to contain detailed information about the solvers settings, material properties, and stress and cycle counting information. Please note that setting the log file detail to "High" can slow performance and result in very large log files.	**Low**
Shell Layer Output	**All, Worst**	This option applies to all fatigue results files, and allows only worst-case results to be output rather than both Top and Bottom. **NOTE:** Setting to **Worst** also has the affect of outputting only the worst-case Critical Plane results.	**All**
Output Max/Min Stress	**Enabled, Disabled**	This option is used to request that the post-processing files (fef, csv) contain the maximum and minimum cycle stress for each entity.	**No**
Compress CSV File	**True, False**	This option allows for the suppression of all but the worst case results from Critical Plane and Duty Cycle to be output to the CSV file.	**No**
Time History Compression	**None, Peak-Valley, Limits**	This option can be used to reduce the time history cycle counting effort in the solver. By default, no time history filtering is performed. The **Peak-Valley** option extracts peak/valley time history values subject to the Gate and Gate Units described below. The **Limits** option retains only the maximum and minimum values of the time histories, and eliminates all other values. Although the **Limits** option is the most extreme filtering mechanism, both options should be used with caution since some damaging cycles are removed from the analysis.	**None**

Table 2-1 DTLib General Solution Parameters

Parameter	Options	Description	Default
Gate Units	**Percent, Data**	This option is only applicable with the **Peak-Valley** Time History Compression option.With the **Percent** option, the **Gate** value is set as a percentage of the range of the input data. The percent gate is applied to each load case independently. With the **Data** option, the **Gate** value is applied directly in the units of the time series data.	**Percent**
Gate	**Value >= 0**	The **Gate** value can be expressed either as an absolute value in the same units as the time history, or as a percentage of the maximum range of the time history.	**0.**
Damage Floor	**Value >= 0**	Sets a minimum value under which the calculated damage is reported as zero.	**0.**
Max Damage	**0<=Value<=1e30**	Sets a maximum value for reported damage.	**1.0**
Check Static Failure	**Warning, Stop, None**	This option allows the user to control job processing in the event that a static failure is predicted. In such an event, the fatigue job can be instructed to ignore, continue with a warning, or completely stop the solution."	**Warning**
Static Failure Damage	**1 <= Value <= 1e30**	Sets a value for reported static failure damage. This allows the user to distinguish between failures due to accumulated cycles and static failure.	**2.0**
Assessment Method	**None, FE Data Only, Simple Range, Actual Analysis**	Controls when and how the hot-spot filtering methods are applied. **None** No filtering is performed **FE Data Only** The filtering is based on stress calculated by summing the von Mises stresses from each loadcase **Simple Range** The filtering is based on stress is calculated by summing the von Mises stresses from each loadcase, taking the range of loading, divisor, and scale factor associated with each loadcase into account **Actual Analysis** The filtering is based on damage values from an actual fatigue analysis	**None**

Table 2-1 DTLib General Solution Parameters

Parameter	Options	Description	Default
Elimination Mode	**Highest Percent Values, Highest Percent Points, Highest Count**	Controls how entities are filtered out during the assessment phase. **Highest Percent Values** Only entities above this percentage of the highest assessed value are kept **Highest Percent Points** This percentage of entities with the highest assessed values are kept **Highest Count** This number of entities with the highest assessed values are kept	**Highest Percent Values**
Percent to Keep	**0. < Value < 100.**	Only valid for Elimination Mode = Highest Percent Values or Highest Percent Points. The Default is: 90. (Elimination Mode = Highest Percent Values) 10. (Elimination Mode = Highest Percent Points)	**90.** or **10.**
Count to Keep	**0 < Value**	Only valid for Elimination Mode = Highest Count. The Default value is 25.	25
Apply Time History Compression	**True, False**	Determines whether Time History Compression is maintained after the assessment phase.	**False**

Detailed Solution Parameters

Another set of common parameters is what is referred to as the Detailed Solution Parameters. This set of parameters is located on a sub-form within the main solution parameters form for the S-N, E-N, Seam Weld, and Vibration solver types. The form contains setting for multiaxial assessment and back-calclulation. A description of each parameter is given in the table below.

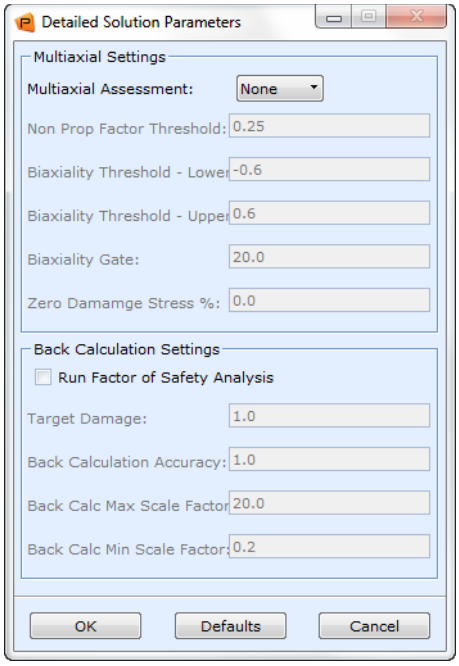

Detailed Solution Parameters
Multiaxial Settings
Multiaxial Assessment:
None
Non Prop Factor Threshold:
0.25
Biaxiality Threshold - Lower
-0.6
Biaxiality Threshold - Upper
0.6
Biaxiality Gate:
20.0
Zero Damamge Stress %:
0.0
Back Calculation Settings
Run Factor of Safety Analysis
Target Damage:
1.0
Back Calculation Accuracy:
1.0
Back Calc Max Scale Factor
20.0
Back Calc Min Scale Factor
0.2
OK
Defaults
Cancel

Table 2-2 Parameters for Multiaxial Assessment and Back-Calculation

Solution Parameter	Description	Enum Options/Range	Default Value
MultiAxialAssessment	Whether to use a multi-axial assessment method.	**None** No multi-axial assessment **SimpleBiaxiality** Calculates a simple biaxiality ratio only **Standard** Standard nCode method of assessment, which merely reports the results. **Auto** Performs the standard method, but then may recalculate depending on the results of the assessment.	**None**
ZeroDamageStressPercent	Used when MultiAxialAssessment is set to Auto. Stress range below which damage is assumed zero, and no re-calculation is performed. Specified as a percentage of UTS.	0 <= Value	**0**
NonPropFactorThreshold	Used when MultiAxialAssessment is set to Auto. The threshold used to check if the loading is proportional or not. This is used in combination with the biaxiality ratio threshold.	0 < Value	**0.25**
BiaxialityThresholdLower	Used when MultiAxialAssessment is set to Auto. The threshold used to check if the loading is proportional or not. This is used in combination with the non-proportionality factor.	Value < 0	**-0.6**

Table 2-2 Parameters for Multiaxial Assessment and Back-Calculation

Solution Parameter	Description	Enum Options/Range	Default Value
BiaxialityThresholdUpper	Used when MultiAxialAssessment is set to Auto. The threshold used to check if the loading is proportional or not. This is used in combination with the non-proportionality factor.	0 < Value	**0.6**
BiaxialityGate	Used if MultiAxialAssessment is set to SimpleBiaxiality. This gate value is used to prevent small stresses from adversely affecting the biaxiality calculation. Stresses below this value are not included in the biaxiality calculation, their biaxiality ratio will be set to 0. The value is set as a percentage of UTS.	0 < Value	**20**
Factor of Safety Analysis	This option performs a back-calculation on stress to compute a scale factor, which when applied, results in the damage value specified below. For cases where the scale factor is greater than 1.0, the damage target is exceeded and the local stress is "safe".Note that the full sets of fatigue results from this analysis are based on the application of the computed scaling factor. A separate analysis should always be performed without the "Factor of Safety Analysis" option turned on.Note that the full set of fatigue results from this analysis are based on the application of the computed scaling factor. A separate analysis should always be performed without the "Factor of Safety Analysis" option turned on.	**True/False**	**False**

Table 2-2 Parameters for Multiaxial Assessment and Back-Calculation

Solution Parameter	Description	Enum Options/Range	Default Value
Target Damage	The target damage for back calculation.	0 < **Value** <= 1E30	**1**
BackCalcAccuracy	This defines a percentage error on the target damage. The back calculation will iterate until the calculated damage matches the target damage within this tolerance.	0.01 <= **Value** < 100	**1**
BackCalcMaxScale	This defines the maximum value of the scale factor that will be reported in the analysis results.	20 <= **Value** <= 5.e6	**5**
BackCalcMinScale	This defines the minimum value of the scale factor that will be reported in the analysis results.	0 <= **Value** <= 0.5	**0.2**

Hot-Spot Filtering

The **Assessment Method** can be used to focus the fatigue analysis on only the most damaged elements and/or nodes.Because fatigue is a non-linear phenomenon, much of a model experiences no appreciable damage and using this option can appreciably speed up the fatigue analysis.The Assessment Method is a two-phase process, where: 1) a subset of nodes and/or elements are found which are estimated to be subject to the most damaging stess/strain, and 2) a full fatigue analysis is performed on this subset of entities. By default, no assessment is performed and all elements and/or nodes are evaluated in the fatigue analysis.

Using the **FE Data Only** option, only the raw stress/strain tensor is used in the evaluating which entities are retained. No superposing of the stress/strain tensor with the time histories is performed. Using the **Simple Range** option performs a superposition of the stress/strain tensor with the time histories, but only the largest and smallest values are in evaluating which entities are to be retained.

The **Actual Analysis** option is best employed in conjunction with Time History Compression. A fatigue analysis is used in the option to determine the most damaged set of elements and/or nodes. The Time History Compression is turned-off by default in the final full analysis. However, it can be kept on by setting the **Apply Time History Compression** to **True.**

The Hot-Spot Filtering methods are available based on the analysis type and loading type. This is summarized in the table below.

Analysis Type	Loading Type	Assessment Method	Comments
S-N and E-N	Time History	**None, FE Data Only, Simple Range, Actual Analysis**	All assessment methods supported
S-N and E-N	Transient	**None, Actual Analysis**	The **FE Data Only** and **Simple Range** assessment methods not applicable for transient FE results
Vibration		**None, Actual Analysis**	The **FE Data Only** and **Simple Range** assessment methods not applicable for frequency response FE results
Seam Weld		**None, Actual Analysis**	The **FE Data Only** and **Simple Range** assessment methods not applicable for seam weld FE results
Spot Weld		**None**	Hot-Spot Filtering is not applicable for spot weld

Assessment Method = FE Data Only

Using this option, the von Mises stress for each load case is summed for each entity and the entities ranked in decreasing order. Only the higher-ranked entities are retained for the fatigue analysis.

$$\sigma_{Assessed} = \sum_{k} \sigma_{vonMises,\,K}$$

Assessment Method = Simple Range

Using this option, the von Mises stress for each load case is similar to the FE Data Only option. However, the stress is also superposed with the maximum and minimum values from the time histories and scaled with the load case scale factor and divider. The entities are ranked in decreasing order using these values and only the higher-ranked entities are retained for the fatigue analysis. For a time history-related loading condition, the stress computation is as follows:

$$\sigma_{Assessed} = \sum_{k} \frac{(Max_k - Min_k) \times ScaleFactor_k \times \sigma_{vonMises,\,K}}{Divider_k}$$

The Max and Min values for each load case represent the time history maxima and minima. For duty cycle analysis, the largest value from each event is considered.

Assessment Method = Actual Analysis

Using this option, a fatigue analysis is performed to determine the highest damage entities. The analysis type used in the filtering phase is the same type as used in the final analysis phase. This method is most useful if the filtering phase uses a “cheaper” analysis, such as one which uses Time History Compression to speed up the filtering phase. A more “expensive” analysis method can be used in the second phase, such as critical plane or multi-axial assessment.

Elimination Mode

A note on elimination mode is warranted. The option Highest Percent Values considers the actual assessed values, rather than only the relative values for each entity. Using this option, entities with an assessed value higher than a given percentage (Percent to Keep) of highest assessed value are retained. Using the option Highest Percent Points, a percentage of the total number of entities (Percent to Keep) with the highest assessed values are retained.

FE Results Access

The dtfat solver has different options available for access to FE results that are not available to the Classic modules. The Classic modules rely on the fatigue translators PAT3FAT or FATTRANS to create an intermediate (fes) file, which contains the job setup and FE results data. For several reasons, the dtfat solver is not reliant on the fes file.

The dtfat solver can use several FE results types in a single fatigue analysis. For example, an analysis type may require both temperature step and stress/strain step data. Another analysis type may require both static FE results and frequency response results. The fes file is not compatible with such FE results requirements. For this reason, an alternative approach is required. The fes file also contains limitations on the length of certain data such are material data set names, file names, etc.

One additional benefit in not relying on the fes file is that job setup data is read directly from the fin file, rather than from the fes file Thus, there is no limitation on text string length, such as with file or other names.

The Direct Access FE results access method is available for use with dtfat. Use of the fes file is still available for legacy sake, but not all analysis options are supported wtih the fes file. The Direct Access method allows FE results to be accessed directly from the Patran database or any FE results files attached to the Patran database, without use of an intermediate file. An intermediate file can be created, if desired. The intermediate (fei) file can be used for subsequent analyses. This can be convenient, when models are large, and data access time is significant.

To use the fei file again in another analysis, pick the **Use existing fei file** on the Job Control form and select the file with the browser button. This file can be used safely with different analysis settings under the following conditions:

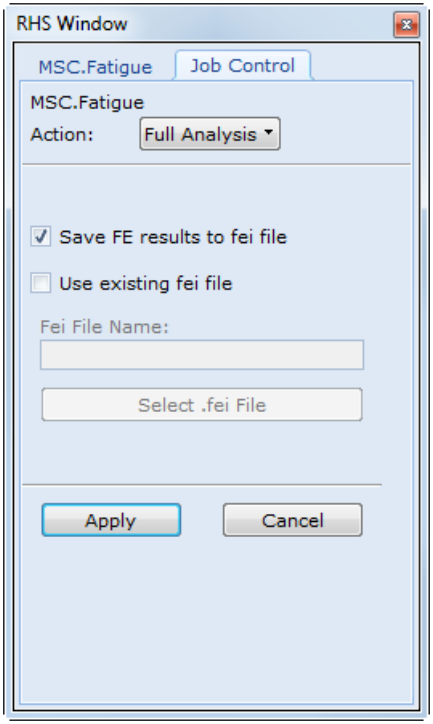
- any settings on the Generic Parameters form may be modified
- any settings on the Solution Parameters form may be modified
- many settings on the Materials Info form may be modified, with the exception of:
  - number of material groups

- group names
- layers selection
- temperature dependence

- Loading Info form settings should not be modified
- The analysis type can be changed between S-N and E-N, but only for stress-based E-N

In order to use the Direct Access method in MSC Fatigue, the FE Results Access option on the Fatigue Preferences form should be set to Direct Access. See Figure 1-15 and the section on Fatigue Preference Settings for more information. For analysis types that require the use of the Direct Access, this method is selected automatically when the job is launched from MSC Fatigue.

To support large models, the Direct Access method also provides a means to save the FE results to an intermediate file, which can be used in subsequent analyses. The intermediate file uses the binary nCode fei format. By default, the FE results are not saved to the intermediate file when using Direct Accessmethod. In order to do so, the Job Control form has the option **Save FE results to fei file**. If this option is picked, a binary format file will be created during the fatigue analysis with the name $jobname.fei.



To use an existing file, select the **Use existing fei file** option on the Job Control form, and select the desired file using the **Select .fei file file** browser.

The Direct Access FE results access method is also required to support the following features in MSC Fatigue. Note further that these features are only available using the DTLib-based solver.

1. Multi-layer models, such as those using Nastran PCOMP
2. Multi-region seam weld
3. Vibration fatigue with static-offset loading

An ascii version of the fei intermediate file can be created using the utility bin2asc, located in `$p3_home/mscdtfat_files/bin`. Alternatively, an ascii version can be converted to binary format using the utility asc2fei. The usage of the fei file conversion utilities is as follows:

```
$p3_home/mscdtfat_files/bin/bin2asc /INP=$fei_name.fei /OUT=$fei_name.asc
$p3_home/mscdtfat_files/bin/asc2fei /INP=$fei_name.asc /OUT=$fei_name.fei
```

It is only in the most unusual circumstances that a user would need to edit the ascii fei file and subsequently use with MSC Fatigue. The above information is given for completeness sake.

Fatigue Results Output

The dtfat solver gnerally creates a larger set of fatigue results than the Classic modules. This is because the results can include event results for duty cycle analysis and plane angle results for critical plane and biaxial assessment. The dtfat solver is also able to return intermediate layer results from multi-layered elements.

The binary fatigue results (fer) file is typically the file imported back to Patran after the fatigue job is complete. At import time, a layer is created for each layer used in a multi-layered element fatigue analysis. A layer is also created for each event in a duty cycle analysis, if event results are requested,

The csv results file can be useful when viewed with a spreadsheet-type applications, where results can be sorted or filtered as needed. Since the csv file is an ASCII file, it can also be viewed with any text editor.

Filename	Description
***jobname*.fer**	Binary fatigue results file. Typically, this file contains results for all Duty Cycle events and all plane angles in a critical plane analysis.
***jobname*.csv**	Human-readable fatigue results file. Typically, this file contains results for all Duty Cycle events and all plane angles in a critical plane analysis. This file is compatible with most spreadsheet programs.
***jobname*.fef**	Fatigue worst-case results file. If shell element results are included, jobname_Top.fef or jobname_Bottom.fef files may be produced.
***jobname*.tml**	Template file, giving a list of results column names included in the fef file. If shell element results are included, jobname_Top.tml or jobname_Bottom.tml files may be produced.

The types of fatigue results output from each analysis type is shown in the table below.

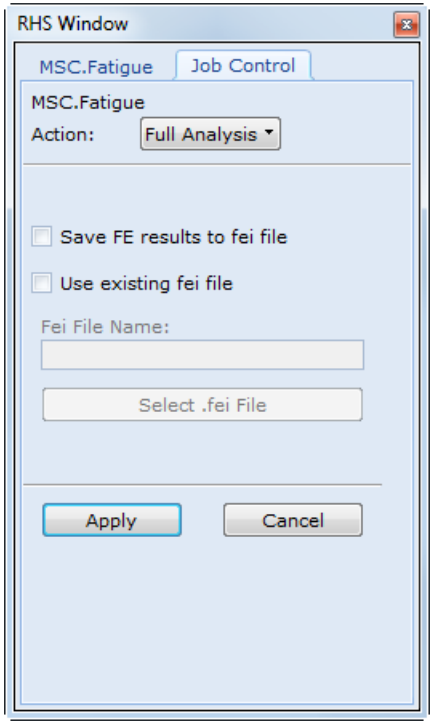
Filename	Description
All Analysis Types	NodeId, ElementId ShellLayer, Layer, Section MaterialGroup, PropertyId, MaterialId, Material Life, LifeEquivUnits
S-N	Damage PlaneAngle MeanBiaxRatio, SDBiaxRatio, MaxAPS, AngleSpread, PopularAngle, BiaxGate, NonProp, DominantDirection, AbsMaxStressDom, CalcMethod ScaleFactor EventName, RepeatCount MaxStress, MinStress Temperature
E-N	Damage, PlaneAngle MeanBiaxRatio, SDBiaxRatio, MaxAPS, AngleSpread, PopularAngle, BiaxGate, NonProp, DominantDirection, AbsMaxStressDom, CalcMethod ScaleFactor EventName, RepeatCount MaxStress, MinStress, MaxStrain, MinStrain Temperature

Filename	Description
Spot Weld	Damage Sheet1Material, Sheet2Material, NuggetMaterial Sheet1TorsionMaterial, Sheet2TorsionMaterial, SheetCount, PlaneAngle Torsion, Location, MaxForce SpotWeldDiameter, SpotWeldNumber ScaleFactor EventName MaxStress, MinStress

Filename	Description
Seam Weld	Damage ToeMaterial, RootMaterial, ThroatMaterial PlaneAngle MeanBiaxRatio, SDBiaxRatio, MaxAPS, AngleSpread, PopularAngle, BiaxGate, NonProp, DominantDirection, AbsMaxStressDom, CalcMethod ScaleFactor EventName WABendingRatio, MinBendingRatio, MaxBendingRatio, BendingRatioStdDev SeamWeldType, SeamWeldLocation MaxStress, MinStress
Vibration	Damage PlaneAngle MeanBiaxRatio, SDBiaxRatio, MaxAPS, AngleSpread, PopularAngle, BiaxGate, NonProp, DominantDirection, AbsMaxStressDom, CalcMethod ScaleFactor EventName, RepeatCount RMSStress, MeanStress, ReturnPeriodToExceedUTS, IrrFactor, LargestStressCycle MaxStress, MinStress Temperature
Dang Van	SafetyFactor, TauCritical, PhCritical, TauZero, DangerFactor EventName

Job Control

By selecting the *Job Control* button located on the main MSC Fatigue setup form, a Job Control form will appear. This form is generic for any of the fatigue analysis types. The form allows for job submission and monitoring, as well as other functions explained below. The form updates itself depending on the action required. In all cases, except when reading an old job setup, the action is linked to the jobname entered in the main MSC Fatigue setup form. This simple form appears as:



The actions that can be invoked from this form include submission of a Submit Full Analysis, 31, Translate Only, Save Job Only, 33 Monitor Job, 34, Abort Job, 35, Delete Job, 36, Read Saved Job, 36, Calculate Normals, 37, and Analysis Manager. (The action that is invoked from this form uses the current jobname on the main setup form. Click on Apply to invoke the action.)

Submit Full Analysis

When the action is set to *Full Analysis* and the user clicks the Apply button the following occurs:

- The job begins the submission process by checking to see if an existing job of the same name exists. If it does, overwrite permission will be requested.
- All of the information requested in the main setup form and the subordinate solution parameter, materials, and loading information forms are written to a MSC Fatigue input file called `jobname.fin` as well as to the Patran database. The fin file includes the fatigue input parameters. The fin file is an ASCII text file whose lines consist of "Parameter = Value". Some portion of the fatigue input information is contained in the database and consists of the region application information (nodes or elements) for the material and surface finish/treatment combinations and, in some cases, the FE results. If any information is not complete, the user will be notified and the submission process will terminate.

- Information is extracted from the database and attached results files such as the region or group data and FE results via the FATTRANS translator or Direct Access facility. Patran is suspended while this action is in progress. If an intermediate file is created, a `jobname.fes or jobname.fei` file results from this action. The intermediate file and is a binary file. It can be translated to ASCII form and edited if required.
- A background process is started in , 383which the actual fatigue analysis begins. The analysis goes through two or three basic steps. The fatigue input file is preprocessed, via the dtfat solver. This consists of reading the `jobname.fin` file, and a validation of the input data before the actual analysis is handed over to the DTLib library. The results of this operation is a file named jobname.config.
- The next phase consists of superpositioning of load cases, rainflow cycle counting, and performing the actual fatigue calculations, which is performed by the DTLib library. The results of this operation is are files named, jobname.log, jobname.fer, jobname.csv, and jobname.fef.
- If MSC's Analysis Manager is installed and licensed, the job as described above will be submitted via the Analysis Manager as opposed to a Linux shell script (although the script is still executed). It is important that this module be configured properly for proper execution. See Analysis Manager for details.

When the job has been completed the results can be read into the database under the Results form. See Command Line Usage, 40. For more information about the files created from a MSC Fatigue analysis and the actual operation of the MSC Fatigue solvers, see Total Life and Crack Initiation (Ch. 6), .

The following files are generated during a typical dtfat fatigue analysis:

Filename	Description
***jobname*.fin**	Fatigue job parameter data.
***jobname*.fei or *jobname*.fes**	FE stress, if an intermediate file is created. Otherwise these files are not generated in the process.
***jobname*.config**	Data file containing a listing of the DTLib job setup. This is for information purposes only. The file is human-readable, but reflects the architecture of DTLib.
***jobname*.log**	A human-readable file containing detailed information about the fatigue analysis.
***jobname*.fer**	Binary fatigue results file. Typically, this file contains results for all Duty Cycle events and all plane angles in a critical plane analysis.
***jobname*.csv**	Human-readable fatigue results file. Typically, this file contains results for all Duty Cycle events and all plane angles in a critical plane analysis. This file is compatible with most spreadsheet programs.

Filename	Description
***jobname*.fef**	Fatigue worst-case results file. If shell element results are included, jobname_Top.fef or jobname_Bottom.fef files may be produced.
***jobname*.tml**	Template file, giving a list of results column names included in the fef file. If shell element results are included, jobname_Top.tml or jobname_Bottom.tml files may be produced.
***jobname*.msg**	Message file.
***jobname*.sta**	Job status file.

Translate Only

This action is only used for the **fes** FE Results Access method. When the action is set to *Translate Only* and the user selects the **Apply** button the following occurs:

- The job will begin the submission process by checking to see if an existing job of the same name exists. If it does, overwrite permission will be requested.
- All of the information requested in the main setup form and the subordinate solution parameter, materials, and loading information forms are written to a MSC Fatigue input file called `jobname.fin` as well as to the Patran database. The fin file includes the fatigue input parameters. The fin file is an ASCII text file whose lines consist of "Parameter = Value". Some portion of the fatigue input information is contained in the database and consists of the region application information (nodes or elements) for the material and surface finish/treatment combinations and, in some cases, the FE results. If any information is not complete, the user will be notified and the submission process will terminate.
- Information is extracted from the database and attached results files such as the region or group data and results via the FATTRANS translator. Patran is suspended while the translation is in progress. A `jobname.fes` file results from this translation.

At this point, the translation stops and no fatigue analysis is performed. This might be convenient if it is desired to edit the fatigue input deck.

For more information about the files created from a MSC Fatigue analysis and the actual operation of the MSC Fatigue modules, see Total Life and Crack Initiation (Ch. 6) or Crack Growth (Ch. 8). Also see The MSC Fatigue Jobname.

Save Job Only

When the action is set to *Save Job Only* and the user selects the Apply button the following occurs:

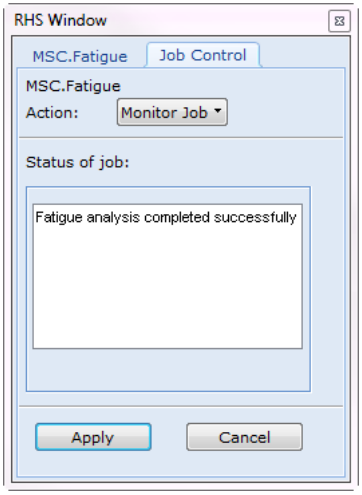
- The job will begin the save process by checking to see if an existing job of the same name exists. If it does, overwrite permission will be requested.

- All of the information requested in the main setup form and the subordinate solution parameter, materials, and loading information forms are written to a MSC Fatigue input file called `jobname.fin` as well as to the Patran database. The fin file includes the fatigue input parameters. The fin file is an ASCII text file whose lines consist of "Parameter = Value". Some portion of the fatigue input information is contained in the database and consists of the region application information (nodes or elements) for the material and surface finish/treatment combinations and, in some cases, the FE results. If any information is not complete, the user will be notified and the submission process will terminate.

This saved fatigue job setup can be loaded back into MSC Fatigue, if needed. Please see Read Saved Job, 36 for more information.

Monitor Job

When the action is *Monitor Job*, the current job status will be reported in the status box. This is achieved by examining the contents of the `jobname.sta` file which is updated at various points during the analysis. The `jobname.msg` file will contain a history of the job (i.e., all the messages generated while running the current MSC Fatigue session). Continue clicking the **Apply** button for an updated current report.



Following is a list of some of the possible messages and a brief explanation.

Job Execution Status Messages

When a job is submitted, it will pass through three to five phases. The user will be informed through the status option of the progress of the job. Both success and error messages are displayed. The following list summarizes some of the typical, normal operation messages which the user may experience. Not all the messages will be displayed since the status file is updated very quickly in some cases. In certain cases, the status file may not be available in which case a "Try again" message will appear. When execution is through MSC's Analysis Manager, these messages appear in the Analysis Manager message window.

Phase 1

```
fatigue job started
```

Phase 2 (if FE Results Access =fes)

```
FATTRANS" reading the neutral file...
FATTRANS" reading the.FIN file...
FATTRANS" reading the FE results...
FATTRANS" writing the.FES file...
FATTRANS" terminated normally
```

Phase 3

```
fatigue job running
```

Phase 4

```
fatigue complete
```

Error messages are also displayed via these status messages.

What To Do When a Job Stops

If the status message does not appear to be updating, it is possible that the job has halted due to an error. In many cases, that error message will be reported through the status facility. However, if it is not reported, you can investigate the problem by opening another window and examining the following file:

jobname.msg: This file will contain all the status messages for the job including any error messages.

Some hints on determining why a job has failed:

1. If the `jobname.fin` file and the `database` exist in your directory, try running the job interactively by typing:

   `fattrans jobname` `(if FE Results Access =fes)`

   then checking the message file.
2. If the `jobname.fes` file exists, run the **dtfat** program interactively and watch for error messages. Type **dtfat** at the system prompt.
3. If FE Results Access = PDA, run **dtfat** from the command line and watch for errors.

See Total Life and Crack Initiation (Ch. 6) for an explanation of these MSC Fatigue executables. See Command Line Usage, 40 for tips on running **dtfat** from the command line.

Error Messages

See Error Messages (App. 19) for a description of error messages and possible solutions.

Abort Job

If the action is set to *Abort*, the job will be aborted by selecting the Apply button. This is achieved by creating an empty `jobname.abo` file in the current directory. The MSC Fatigue modules periodically check for this file and when detected, will abort.

Likewise, a `jobname.abo` file can be created or the `jobname.sta` file renamed to `jobname.abo` from the operating system prompt and the same result will be achieved when a job is running.

If execution is via MSC's Analysis Manager, then the Analysis Manager will handle the abortion of the job. All files will automatically be cleaned up by the Analysis Manager.

Delete Job

If the action is set to *Delete*, the various files associated with the job will be deleted by selecting the **Apply** button. The files that will be deleted if encountered are:

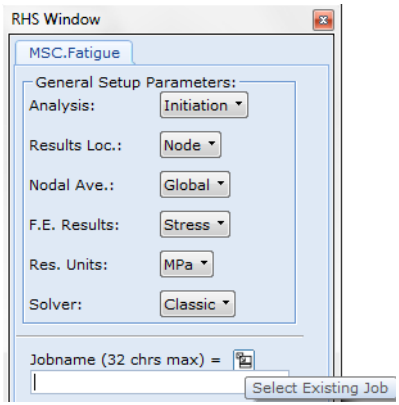
Filename	Description
***jobname*.fin**	Fatigue job parameter data.
***jobname*.fei or *jobname*.fes**	FE stress, if an intermediate file is created. Otherwise these files are not generated in the process.
***jobname*.config**	Data file containing a listing of the DTLib job setup. This is for information purposes only. The file is human-readable, but reflects the architecture of DTLib.
***jobname*.log**	A human-readable file containing detailed information about the fatigue analysis.
***jobname*.fer**	Binary fatigue results file. Typically, this file contains results for all Duty Cycle events and all plane angles in a critical plane analysis.
***jobname*.csv**	Human-readable fatigue results file. Typically, this file contains results for all Duty Cycle events and all plane angles in a critical plane analysis. This file is compatible with most spreadsheet programs.
***jobname*.fef**	Fatigue worst-case results file. If shell element results are included, jobname_Top.fef or jobname_Bottom.fef files may be produced.
***jobname*.tml**	Template file, giving a list of results column names included in the fef file. If shell element results are included, jobname_Top.tml or jobname_Bottom.tml files may be produced.
***jobname*.msg**	Message file.
***jobname*.sta**	Job status file.
***jobname*.vec**	Surface normals vector file.

Other files may also be deleted if associated with the particular job. Use this option with caution.

Read Saved Job

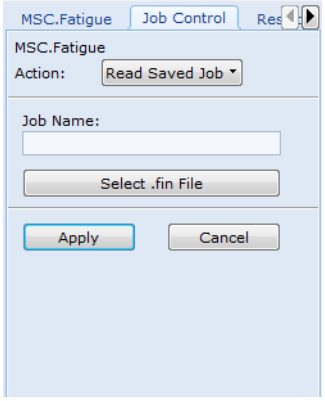
There are two different ways to load a saved job back into MSC Fatigue. If the job resides within the current database, it can be loaded by clicking on the small icon adjacent to the Jobname databox on the main form,

as shown below and then selecting the job from the resulting list of jobs. If the job does not exist in the current database, it can also be loaded from an existing fin file as described further below.



When the action is *Read Saved Job*, a databox and button will appear on the form. If the *select.fin file* button is pressed then a file select form is displayed with all the available jobs from the local directory. By selecting one of the existing jobs and clicking on the **OK** button, the Jobname databox is filled out. By clicking on the Apply button, the reading of the `jobname.fin` files will initiate. If these files are successfully read the widgets and parameters in the main MSC Fatigue setup form and the subordinate solution parameter, materials, and loading forms will be updated.

If the read is unsuccessful, some of the parameters certainly will not be complete. It is always beneficial to check and see if the parameters have been updated properly before attempting to submit the analysis.



Calculate Normals

This is an advanced feature of MSC Fatigue to allow for more control of the input stress or strain information to the fatigue analysis. Certain fatigue analysis options require 2-D FE results, such as Critical Plane or Biaxial Assessment. This feature allows the user to apply the equivalent of a "skin" on the free surface of a solid

element model, and surface resolve the stress tensor into 2-D tensor. Note that when this feature is used, only the nodes on the free surface of the solid model are used in the fatigue analysis.This feature is only applicable to a nodal based fatigue analysis.

Compute Stress Gradients

This is an advanced feature of MSC Fatigue to allow the inclusion of the normal stress adjacent to the surface of solid elements in the fatigue analysis. This is described in the Fatigue Quick Start Guide.

Fatigue Preferences

The Fatigue Preferences were previously set using Tools | MSC.Fatigue | Fatigue Pre-Processing | Set Fatigue Environment Variable... This was updated in the 2012 version, in order to enhance the understanding that the preferences can be set and saved for a subsequent session, just like any other Patran preference. The new method is to set the preferences using **Menu |Tools | MSC.Fatigue | Preferences...**

In order to save the preference settings from one session to another, a settings.pcl file should be created and located in the current working directory or the user home directory. The $p3_home/settings.pcl file can be copied to the user's preferred location for this purpose. Please be sure to remove the "Read Only" attribute from the file.

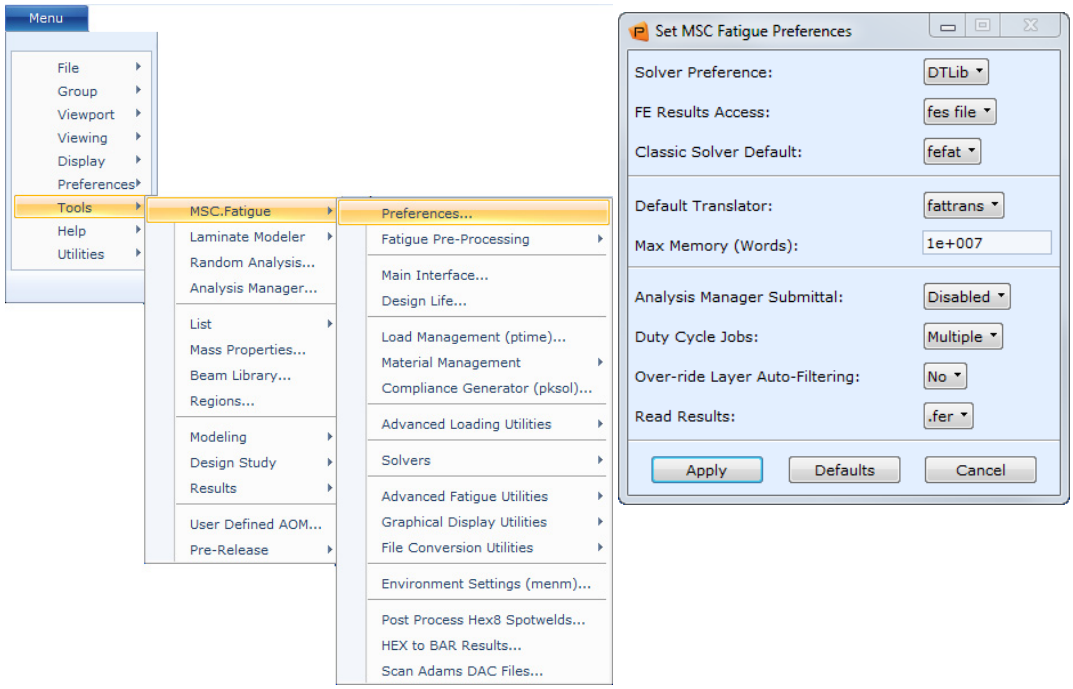


Figure 2-2 Fatigue Preferences Settings

The Fatigue Preference options are described in the table below.

Name	Options	Default	Description
Solver Preference	**Classic, DTLib**	**DTLib**	Determines whether the MSC Fatigue GUI is configured to set up and run jobs with the "Classic" nCode fatigue modules or use the new DTLib solver. Note that many new features are only available with the DTLib solver.
FE Results Access	**fes file, Direct Access**	**Direct Access**	Determines whether FE results and model data are written to the fes file or accessed directly (without having to save to intermediate file). Note that many new features will only be available using the Direct Access option.
Classic Solver Default	**fefat, mfatfe**	**mfatfe**	Determines which solver will be used for S-N and E-N analysis, when using Solver Preference = Classic. The mfatfe solver has newer technology than fefat.
Default Translator	**fattrans, pat3fat**	**fattrans**	Determines which translator is used to create fes file. Note that fattrans supports many newer features, which are not supported with pat3fat. Pat3fat is available to legacy purposes, and it may be useful for cases where fattrans fails.
Max Memory	N/A	**10 E7 words**	Used to limit the memory available to the fattrans translator
Analysis Manager Submittal	**Yes, No**	**No**	May be used to defeat fatigue job submittal with Analysis Manager (if installed)
Duty Cycle Jobs	**Multiple, One at a Time**	**Multiple**	Determines whether duty cycle jobs are allowed to submit multiple analyses or a job must finish before the next job is submitted. This option can be set to **One at a Time** if duty cycle submittal fails in **Multiple** mode. Applicable for the Classic Solver only.
Over-ride Layer Auto-Filtering	**Yes, No**	**No**	Can be set to **Yes** in the event that the Materials Info form fails to show all available layers for a given group
Read Results	**.fer, .fef**	**.fer**	This option determines which fatigue results file is imported from the **Read Results** button on the main form. The **.fer** file contains results on all layers and duty cycle events, and is preferred over the .fef file. The **.fef** file option is available for compatibility purposes.

Note that the previous methods using the p3epilog.pcl file or system environment variable for setting the preferences is still valid. However, the current session settings can be saved only via the settings.pcl file.

The fatigue preferences are set through a pcl command as follows (the default settings are shown here):

```
pref_env_set_string( "FAT_USE_FATTRANS", "YES" )
pref_env_set_real( "FAT_MAX_NUM_WORDS", 1.E+7 )
pref_env_set_string( "FAT_DISABLE_AM", "YES" )
pref_env_set_string( "FAT_DC_WAIT", "NO" )
pref_env_set_string( "FAT_USE_MFATFE", "NO" )
pref_env_set_string( "FAT_LAY_FILT_OVR", "NO" )
pref_env_set_string( "FAT_USE_DTLIB", "YES" )
pref_env_set_string( "FAT_FERESACCESS", "PDA" )
pref_env_set_string( "FAT_READ_RESULTS", "FER" )
```

Command Line Usage

The DTLib-based fatigue solver (dtfat) can be run from the command line, with several options available that are inaccessible from the MSC Fatigue UI. These options are especially useful for selecting specific entities for fatigue analysis and for creating special outputs such as stress time history, etc. The VIBOUT, CYCOUT, and TSOUT options are described in more detail in the following chapters.

The available command line options for dtfat are as given in the table below. The options are specified in a key-value pair relationship using a **/keyword=value** type of format.

Option Name	Allowable Values	Default Value	Description
/ENTIDS	**ID1,ID2,...**	no default values for the IDs	specify the IDs of particular nodes or elements to be used in the analysis; the default is to perform the fatigue analysis for all entities
/DUMPASCI	**Y, N**	**N**	create an ascii dump of the fes file
/VIBOUT	**Y, N**	**N**	output entity cycle list, response PSD, and damage histogram to a file; this option is best using in combination with the **/ENTIDS** option
/INP	jobname.fin, jobname.fes	no default; must be included*	specify the name of the fatigue job file
/CYCOUT	**Y, N**	**N**	output entity cycle list data to a file; this option is best using in combination with the **/ENTIDS** option
/TSOUT	**Y, N**	**N**	output entity stress time history; this option is best using in combination with the **/ENTIDS** option

Option Name	Allowable Values	Default Value	Description
/FERES	**PDA, FEI, FES**	no default; must be included*	specify the FE results access method, where: **PDA** Direct Access method used to access FE results and model data **FEI** An existing nCode-format fei file used to access FE results and model data **FES** fes-format file used to access FE results and model data
/OUTTXT	Y, N	N	output cycle list, time history, and other optional outputs as text-format
/DUMPJOB	Y, N	N	dump the job setup information to the nCode fej file (this is normally useful for testing purposes only)

Note that, since several different FE results access methods are supported, the command line options **/FERES** and **/INP** should always be included. For example:

```
$p3_home/bin/dtfat /FERES=PDA /INP=$jobname.fin
$p3_home/bin/dtfat /FERES=FES /INP=$jobname2.fin
```

*Note that even though the fes file contains both job setup and FE results data, the fin file is still typically used for job setup information in dtfat. This resolves fes file-related issues, such as too-short character string length. For compatibility purposes, the fes file can still be used for both FE results AND job setup data using the fes extension in the input file name as in the following example.

```
$p3_home/bin/dtfat /INP=$jobname.fes
$p3_home/bin/dtfat $jobname.fes
```

dtfat Optional Output

The DTLib-based solver has the capability to output stress/strain time history, cycle count information, and response psd; The type of output depends on solver type. This can be done by running the solver from the command line, and using the appropriate command line options. Not that this can result in a large amount of data, so it is recommended that this optional output is only performed for a few entities of interest. The following table describes the dtfat solver command line options and output type for each analysis type. Also described are the file formats available for each output type./

Table 2-3 Entity-Based Optional Outputs by Analysis Type

Analysis Type	Optional Output	Command Line Argument	Text File Format	Binary File Format
S-N	stress time history	**/TSOUT**=Y	csv	nCodeDACTimeSeries (.dac)
S-N	cycle list	**/CYCOUT**=Y	csv	S3MultiColumn (.s3m)
E-N	stress time history	**/TSOUT**=Y	csv	nCodeDACTimeSeries (.dac)
E-N	cycle list	**/CYCOUT**=Y	csv	S3MultiColumn (.s3m)
Seam Weld	stress time history	**/TSOUT**=Y	csv	nCodeDACTimeSeries (.dac)
Spot Weld	stress time history	**/TSOUT**=Y	csv	nCodeDACTimeSeries (.dac)
Vibration	response psd	**/VIBOUT**=Y	csv	S3MultiColumn (.s3m)
Vibration	rainflow histogram	**/VIBOUT**=Y	nCodeXmlHistogram (.xmh)	nCodeDACHistogram (.hst)
Vibration	damage histogram	**/VIBOUT**=Y	nCodeXmlHistogram (.xmh)	nCodeDACHistogram (.hst)

The following table describes the tools that can be used to view the files created by dtfat optional output commands.

Table 2-4 Applications Available to View Particular File Formats

File Format	Viewing Application
nCodeDACTimeSeries (.dac)	ptime, mqld, mmfd
nCodeDACHistogram (.hst)	mp3d
S3MultiColumn (.s3m)	currently unavailable
comma separated values (.csv)	Microsoft Excel, other spreadsheet programs, text editor
nCodeXmlHistogram (.xmh)	text editor

3 Using MSC Fatigue - The Classic Modules

- Introduction 44
- The Patran Environment 45
- Job Setup 47
- Job Control 85
- Postprocessing Results 100
- Other Modes of Job Setup 109

Introduction

A MSC Fatigue analysis is initiated in one of three ways. The most useful and practical way is either through its own graphical interface or directly through the Patran environment. The stand alone operation is described in Stand Alone Usage and the Patran environment is described in The Patran Environment, 45.

A MSC Fatigue job can also be setup outside of any type of graphical environment directly through the MSC Fatigue modules themselves, although this is limiting. See Other Modes of Job Setup, 109.

Regardless of how a fatigue analysis is set up there are always three basic inputs that must be specified before an analysis can proceed. Only setup for the basic analysis types (Total Life and Crack Initiation (Ch. 6) and Crack Growth (Ch. 8)) are explained in this chapter. For other more advanced fatigue analysis setup (Vibration Fatigue (Ch. 9), Weld Analysis (Ch. 10), Software Strain Gauges (Ch. 12), Multiaxial Fatigue (Ch. 7)), see the appropriate chapter.

Note:	Classic Fatigue forms have not been enhanced with the QT framework.

Preparing for a MSC Fatigue Analysis

The three basic pieces of information needed to compute a fatigue life estimation using MSC Fatigue are:

- **Materials Information:** This is information describing the cyclic fatigue properties of the component or material. This can be accomplished using MSC Fatigue's materials database manager, PFMAT. The materials data may already be found in the materials database, or it may be generated from the UTS of the material based on empirical formulations, or it may be obtained from materials tests or other references and input manually into the materials database. See Material Management (Ch. 4).
- **Loading Information:** Loads can be in the form of time histories, power spectral density functions (PSDFs) or cycle definitions (rainflow matrices). The loading is defined externally to the FE model in the case of static and frequency response FE analysis or else must be an integral part of the FE analysis for transient and random vibration analyses. External loading is defined using MSC Fatigue's time history database manager, PTIME. See Loading Management (Ch. 5).
- **Geometry Information:** For most fatigue analyses, this entails the finite element model and the stress or strain results. In the case of a crack growth analysis, a comprehensive library of compliance functions is provided in the module, PKSOL, to define the crack geometry. To obtain the stress/strain information, an appropriate finite element (FE) analysis must have been performed. Any fatigue analysis begins where a finite element analysis ends. Therefore, this guide does not go into any detail concerning model creation or finite element analysis itself.

The fatigue analysis can be thought of as a "five box trick" (as shown in Figure 3-1) where the first three are the inputs as described above, the fourth is the actual fatigue analysis, and the fifth is the interpretation and/or postprocessing of the results which could lead to a loop back to the first three inputs for sensitivity studies.

This is an important concept in MSC Fatigue in that it allows you to build a "fatigue model." Fatigue analysis is a logarithmic process, thus amplifying any errors or discrepancies of the three inputs through the analysis process. Sensitivity studies allow for easy and quick interpretation of resulting life predictions to see how sensitive your model is to variations in any of the inputs.

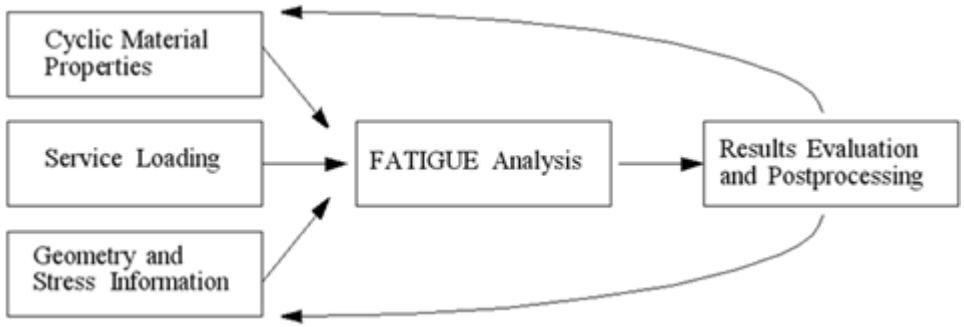
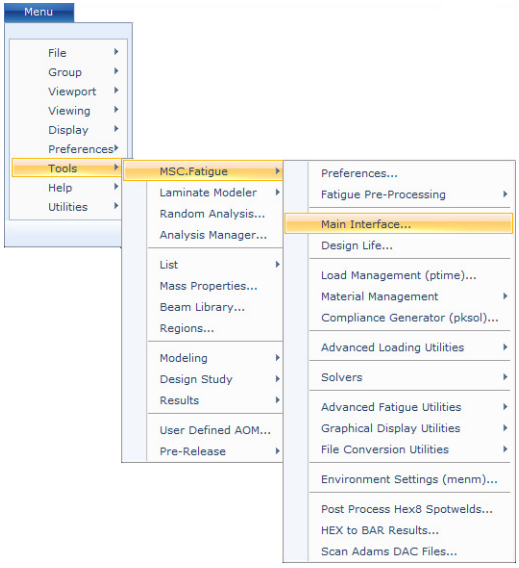


Figure 3-1 The MSC Fatigue “five-box trick”

The Patran Environment

MSC Fatigue is fully integrated into the Patran environment. It is assumed that the user has a working knowledge of Patran. The setup for a MSC Fatigue job is accessed through the Tools pulldown menu by selecting MSC Fatigue and then Main Interface from the Submenu. Other items are available from the MSC Fatigue pulldown but are described elsewhere in this guide.



The following operations are need to perform a successfully MSC Fatigue job from within Patran once a model has been created and the FE analysis completed:

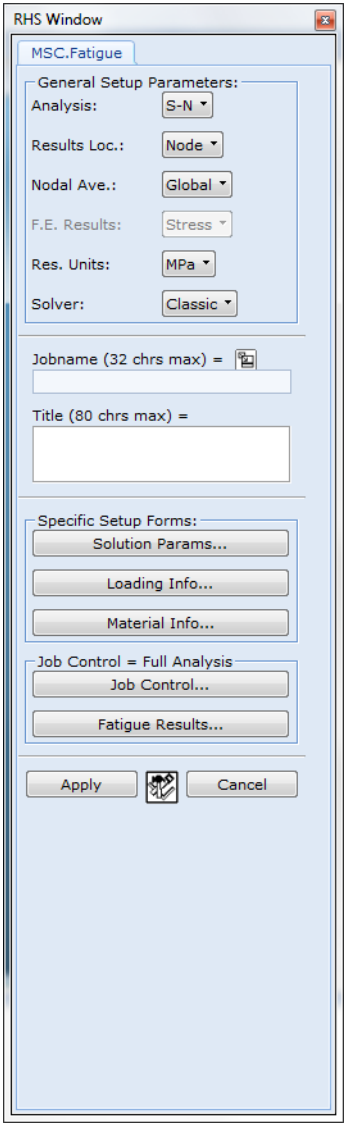
1. Open a database (under File menu). Import the FE model along with its results if necessary when starting from a new database. It is generally necessary that all model geometry and FE results be stored in the database.

2. Perform any model manipulations necessary such as creating groups, changing display styles or even viewing and manipulating the FE stress/strain results.
3. Set up the MSC Fatigue job and submit the job for analysis. This consists of defining the three inputs to the fatigue analysis (material, loading, and geometry information).
4. Import the fatigue analysis results and postprocess them for in-depth understanding.

It is assumed that the user has a working knowledge of how to perform the first two operations. The last two are described in Job Setup, 47, Job Control, 85, and Postprocessing Results, 100.

Job Setup

When the Analysis switch is selected from MSC Fatigue is selected from the Tools pull-down menu from Patran, the following form appears.



General Setup Parameters

The following table explains each of the options for the general setup parameters:

Parameter	Description
Analysis	Three basic fatigue analysis types are possible: *Crack Initiation*, *Crack Growth*, and *Total Life (S-N)*. Other types of analysis are available also and explained in their respective chapters. See Vibration Fatigue (Ch. 9), Multiaxial Fatigue (Ch. 7), Rotating Structures Analysis (Ch. 11), Software Strain Gauges (Ch. 12), and Weld Analysis (Ch. 10).
Results Location	This parameter tells MSC Fatigue whether to expect *Nodal* stress/strain results or *Elemental* centroid stress/strain results. This dictates whether the user is setting up a global multi-node or global multi-element fatigue analysis. Subsequent parameters, results file types, and results displays are dependent on whether nodal or elemental data is being considered. If nodal data is being considered, the resulting fatigue lives are reported at the nodes. Conversely, if elemental data is being considered, the fatigue lives are reported at the element centroids. Fatigue cracks invariably occur at free surfaces, and hence when a crack initiation method is used, node points results are usually required. The exception is when a shell model is used, element centroid results may be extrapolated to the top or bottom surface. This is useful when there may be some doubt as to the accuracy of the node point results due to extrapolation and/or nodal averaging practices. The spot weld analyzer uses forces and moments from both nodes and elements. The SEAM-weld analyzer takes stresses from both the top and bottom surface and needs both nodes and elements for this.
Nodal Averaging	Depending on how the finite element results are defined, nodal averaging of the stresses or strains may take place. If grid point stresses exist and are selected, no averaging will occur. However if the stresses or strains selected for the fatigue analysis are elemental based, such as results at integration points or elemental nodal values such that each element has a different value at the shared grid points, then nodal averaging will occur. This averaging is done on a global basis such that every contributing element surrounding a particular node will be used in the averaging. The exception to this is if Group is selected in which case only elements in the Current Group will be used in the averaging. For the SEAM-weld analyzer only the current group can be used and no choice will be given.

Parameter	Description
F.E. Results	For crack initiation, the fatigue analyzer may use either *Stress*, *Strain, or E-P Input* results from the finite element analysis. For crack growth and total life, only stress results can be used and no choice will be given. If Stress is selected, the intermediate `jobname.fes` file will contain nodal or element stresses for each load case or time step as opposed to strains. This selection should make no difference to the final results of a crack initiation calculation, as MSC Fatigue will always calculate the strains. The exception is when shell results are used. In this case, Stress should be selected because only 2D results are available and the absence of the out-of-plane strains will cause incorrect calculation of combined parameters. Another exception: when Strain results are selected, the analyzer requires finite element results in the form of six (6) components of strain, three (3) direct strains and three (3) *engineering* shear strains (i.e. two times the *tensor* shear strains). If FE strains are not available in this form, Stress results should be selected. Selecting E-P Input allows for elastic-plastic input when doing a crack imitation or a multiaxial CI analysis. The spotweld analyzer uses only forces and no choice will be given.
Results Units	This option menu is active for all analysis types utilizing stress results. The stress unit type must be identified for proper conversions within the fatigue analyzer. Available units are *MPa*, *Pascals*, *PSI*, *KSI*, *KG/M**2*. This parameter is set to None if the Tensor parameter is set to strain. The spot weld analyzer requires both cross sectional forces and moments from the CBAR elements representing the spot welds. For this reason, both the force and dimension units are required. Allowable options for forces & dimensions are: N and m, N and mm, lbf and in, and kip and in.
Jobname	In this databox, the user supplies a unique name by which to distinguish the fatigue job. All subsequent forms will key off of this jobname when performing certain tasks. If the jobname already exists when submitting the job, permission for overwrite will be requested. This jobname is limited to thirty-two (32) characters. Also, if a jobname is typed into this databox and the user presses the **<RETURN>** key the program will check for its existence and ask the user if he wishes to read in the old job parameters.
Title	A descriptive textual title can be supplied in this databox. The length is limited to eighty (80) characters.

Important:	The same answers should result whether stress or strain FE results are used in a crack initiation analysis. However, if the Young's modulus is different between the finite element analysis and the MSC Fatigue material being used, significant differences in fatigue results can occur when comparing between stresses and strains. For strain results, no conversion is necessary. Total life and crack growth jobs use stress tensors, so no conversion is required or allowed. Also, there could be a problem using strains with 2-D elements if any combined strain component is used which includes the Z-component strain. This is because the Z-component strain appears as zero from 2D elements in the `.fes` file which is not generally true. Absolute maximum principal, x and y component strain should be unaffected.

The MSC Fatigue Jobname

The MSC Fatigue jobname is used in all unique file names created during a MSC Fatigue run, and is used for retrieving previously executed MSC Fatigue jobs for editing, rerunning, and results display. The jobname is a character string containing a maximum of 32 characters with no spaces. On some systems, characters such as [, *, /, or : , are not allowed, and in fact only alphanumeric characters are recommended. The jobname related files generated during a MSC Fatigue run are shown in the table below.

Filename	Description
***jobname*.fin**	Job parameter file (ASCII).
***jobname*.fes**	MSC Fatigue Input file (Binary).
***jobname*.asc**	ASCII version of the *jobname*.fes file.
***jobname*.fpp**	MSC Fatigue intermediate results file (Binary).
***jobname*.msg/log**	MSC Fatigue message and log files (ASCII).
***jobname*.sta**	Job status file (ASCII).
***jobname*.fef**	Global multi-node/element results file (ASCII).
***jobname*.fos**	Factor of Safety results file(ASCII).
***jobname*.abo**	Job abort file (ASCII).
***jobname*.tcy**	Time ordered stress cycles file (Binary).
***jobname*.crg**	Crack growth results file (Binary).
***jobname*.vec**	Surface normals file (ASCII).
***jobname*.fpr**	Job currently active alert file (ASCII).
***jobname*nn.kfl/.kfm**	Stress concentration-Life XY data for specific node/element, nn (ASCII/Binary).
***jobname*nn.dcl/.dcm**	Design criterion-Life XY data (ASCII/Binary).
***jobname*nn.fal/.fam**	Scale factor-Life XY data (ASCII/Binary).

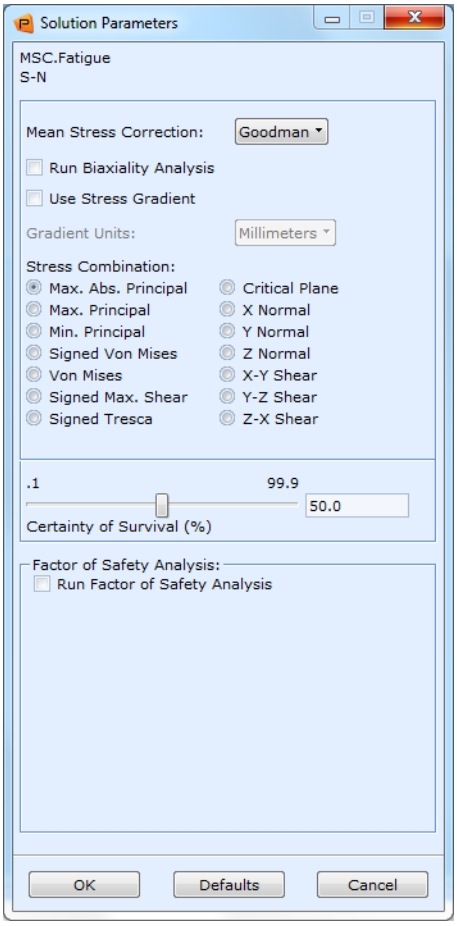
Filename	Description
***jobname*nn.rfl/.rfm**	Residual stress-Life XY data (ASCII/Binary).
***jobname*nn.cyh**	Rainflow cycle distribution at node/element (ASCII).
jobnamenn.dhh	Damage distribution at node/element (ASCII).
***.ksn**	K solution files (Binary).
***.dac, *.cyh**	Loading time history/rainflow matrix files (Binary).
***.xyd**	K solution XY data (ASCII).
***.tem**	Plotting format data (ASCII PCL file).
***_tmpl**	Results template files (ASCII).
***jobname_ts*/*ls*.nod**	Nodal results files from coordinate transforms (ASCII).
***.adb/.tdb/.mdb**	Time history and Materials database description files (ASCII/Binary/Binary).

Solution Parameters

Each analysis type has its own unique set of solution parameters. These are described in Total Life (S-N) Solution Parameters, 51, Crack Initiation Solution Parameters, 54, and Crack Growth Solution Parameters, 59.

Total Life (S-N) Solution Parameters

The following forms appear when invoking the Solution Parameters button on the main MSC Fatigue setup form for the S-N analysis type.



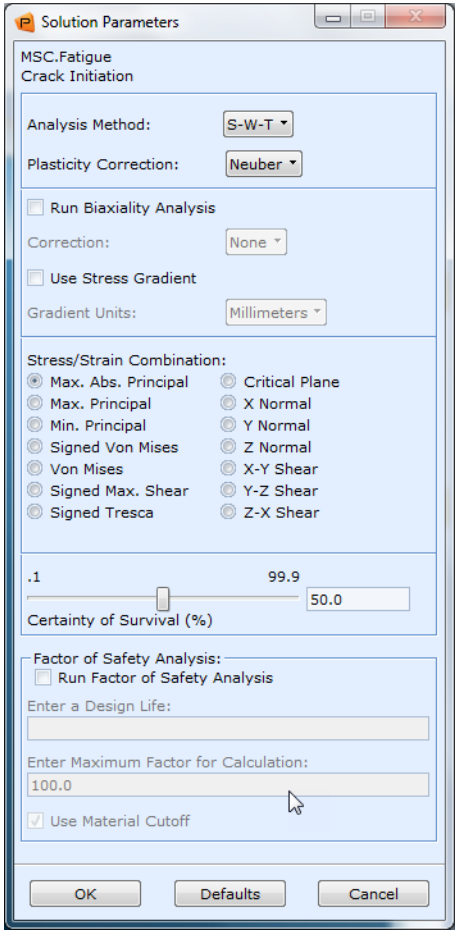
The following table describes each parameter in detail:

Parameter	Description
Mean Stress Correction	Acceptable values of mean stress are *Goodman, Gerber, Multiple Mean Curves*, or *None*. These mean stress correction methods are described in detail in Fatigue Theory (Ch. 16). Although one of the above must be selected, a sensitivity study allows the comparison of results using all of these correction methods. Goodman and Gerber are two methods of correcting S-N curves for mean tensile stress with Goodman being the most conservative. (For compressive mean stresses, both methods as applied in MSC Fatigue may be nonconservative. It may be better to chose None. Also BS 5400 pt 10 requires that no mean stress correction be made.)
Biaxiality Analysis	It is possible to request that a Biaxiality analysis be performed. This requires, first, that the stresses be aligned to the surface of the component. If the stresses from the FE analysis are not aligned to the surface, it is possible to accomplish this by selecting the Calculate Normals, 94 from the Job Control form before submitting the analysis. This option only makes sense when using nodal stress values or elemental stress values for shell elements. For the latter, the stresses must be aligned by the FE code as opposed to allowing MSC Fatigue to do it by requesting it to calculate normals. The biaxial parameters which are calculated inform the user as to the amount of multiaxiality present in the component due to the loading applied allowing for the user to determine the validity of the fatigue analysis. Please refer to Multiaxial Fatigue (Ch. 7) for further discussion of this feature.
Stress Combination	This option menu selects the stress parameter used in the fatigue analysis. The six multiaxial component stresses defined by the stress tensor are resolved into one uniaxial or combined value for fatigue calculations for each node for each time step. This is necessary since the fatigue damage models used in MSC Fatigue are based on theories which deal with uniaxial stress. These stress scalar combinations can be either one of these components, *X Normal*, *Y Normal*, *Z Normal*, *X-Y Shear*, *Y-Z Shear*, *Z-X Shear*, *Max. Abs. Principal*, *Max. Principal, Min. Principal, von Mises, Signed von Mises*, *Signed Max. Shear*, *Signed Tresca, or Critical Plane*. For Stress-Life, the Signed von Mises will be smaller than the Max. Abs. Principal when there is positive biaxiality and hence this selection would be less conservative. (Note also that some weld classes in BS 5400 pt10 require shear stress to be used, the Signed Tresca parameter.) The sign on the signed parameters is taken from the sign of the absolute maximum principal value. It is necessary to sign these stress parameters otherwise non-conservative fatigue life estimates will result!

Parameter	Description
Certainty of Survival	This defines the *Certainty of Survival* based on the scatter of the S-N curve. For example, to be 96% certain that the life will be achieved, set the slider bar at 96. This value is used to modify the S-N curve according to the standard error scatter parameter (SE). The design criterion parameter will be meaningless if the value of SE is 0. A Design Criterion value of 50 leaves the S-N curve unmodified.
Factor of Safety Analysis	This option will cause a type of Factor of Safety or over design analysis to be performed. It informs the user as to how much stress may be modified for optimization purposes based on fatigue life. This analysis is in addition to the normal fatigue calculations and must be requested. This analysis method can be very useful for those components which predict an infinite life, providing a measure of the risk of fatigue failure. The results of this analysis are stress factors for each location (node or element) for which the analysis has been performed. A value of one suggests that the specified life will be exactly attained whereas a factor less than one means the desired life will not be attained. Factors greater than one are, therefore, most desirable. Certain parameters must be supplied in order to proceed with this analysis. They comprise the remainder of the parameter descriptions in this table.
Options	A Factor of Safety analysis can be performed based on *Life Based* or *Stress Based*. If Life Based is selected, then three additional parameters are needed: the Design Life, the Maximum Factor, and whether to use the Material Cutoff value or not. For a Stress based analysis, one parameter is required: the Reference Stress.
Reference Stress	For a Stress based Factor of Safety analysis, enter the reference stress (stress fatigue limit) level at which the life is assumed to be infinite.
Design Life	For a Life based Factor of Safety analysis, enter the target design life.
Maximum Factor for Calculation	Enter a maximum factor (default is 100) to be used in the analysis. This number can be lowered to speed up the calculation if it is known that the maximum stress factor of interest will be less that the default.
Material Cutoff	This toggles the usage of the material cutoff value in the analysis.

Crack Initiation Solution Parameters

When the analysis type is set to Initiation (crack initiation or strain-life or local strain analysis), the form as shown appears.



The following table describes each parameter in detail:

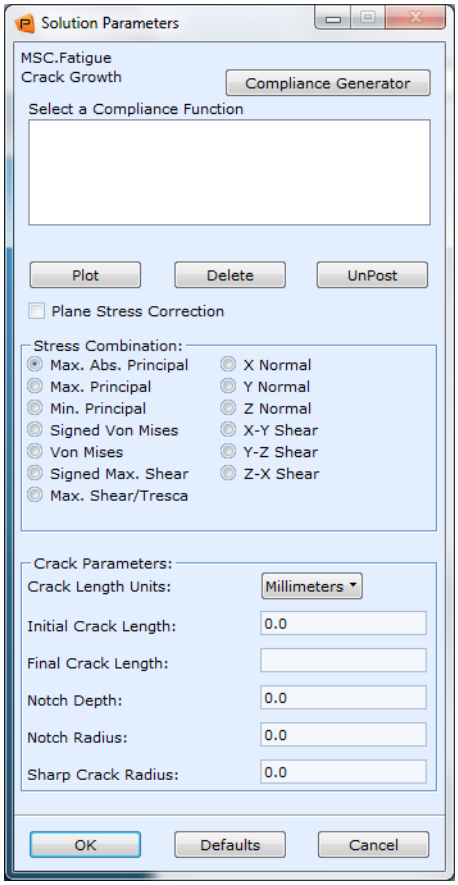
Parameter	Description
Analysis Method	Acceptable values of the Analysis Method (sometimes referred to as Mean Stress Correction) are *S-T-W (Smith-Topper-Watson)*, *Morrow* or *Strain-Life*. These methods are described in detail in Fatigue Theory (Ch. 16). Although one of the above must be selected, sensitivity study allows the comparison of results using all of these correction or analysis methods. Of the two methods, the Smith-Topper-Watson is most commonly used, especially with variable amplitude loadings. However, it cannot deal with wholly compressive cycles (for which it will predict zero damage) and when the loading is predominately compressive, the Morrow correction will be more conservative. The Morrow method may also be useful where mean tensile stresses are very high and the Smith-Topper-Watson method may give answers that are overly conservative.
Plasticity Correction	This option selects the method used to carry out the conversion from elastic to elastic-plastic stresses and strains. Acceptable values for this widget are: *Neuber, Mertens-Dittmann, Seeger-Beste*, or *None*. The default is Neuber. The other options, the Mertens-Dittmann and Seeger-Beste methods, are based on the Neuber method but include modifications which cause them to give better answers (and more conservative) results when geometries are unnotched, or plasticity is not highly localized. The latter two methods are very similar, with the Seeger-Beste method being more conservative. Both require an elastic strain concentration or shape factor which is a function of the shape of the cross section of the component and the type of loading. For instance, the shape factor for a rectangular beam in bending is 1.5. The shape factor for a notched beam could be estimated from the product of the shape factor for the unnotched geometry and the stress concentration factor of the notch. When the shape factor a_p tends toward infinity, both methods reduce to the Neuber method. These shape factors are input via the Materials Information form. See Elastic-Plastic Corrections, 1416 for an explanation of these various correction methods.
Biaxiality Analysis	It is possible to request a Biaxiality analysis to be performed. This requires, first, that the stresses be aligned to the surface of the component. If the stresses from the FE analysis are not aligned to the surface, it is possible to accomplish this by selecting the Calculate Normals, 94 from the Job Control form before submitting the analysis. This option only makes sense when using nodal stress values or elemental stress values for shell elements. For the latter, the stresses must be aligned by the FE code as opposed to allowing MSC Fatigue to do it by requesting it to calculate normals. The biaxial parameters which are calculated inform the user as to the amount of multiaxiality present in the component due to the loading applied and allows for the user to determine the validity of the fatigue analysis. Please refer to Multiaxial Fatigue (Ch. 7) for further discussion of this feature.

Parameter	Description
Biaxiality Correction	This option selects the method used to correct the treatment of material properties in the application of the Neuber method in order to take account of the biaxiality of the loading. Acceptable values include: *None, Material Parameter,* or *Hoffman-Seeger*. If None is selected, the software will carry out the Neuber method using the chosen Strain Combination and the uniaxial cyclic stress-strain curve. The Hoffmann-Seeger method uses the biaxiality ratio to convert the combined strain parameter to an equivalent strain (based on the von Mises Strain) before carrying out the Neuber correction and then recalculating the elastic-plastic stresses and strains. It is applicable to the Maximum Absolute Principal and the Signed Tresca strain combinations. In the case of the Signed von Mises, it reduces to the unmodified Neuber method. The Material Parameter modification (Ratio) method works by calculating a new cyclic stress-strain curve for each node or element on the basis of the mean biaxiality ratio. The new curve relates Maximum Absolute Principal stress and strain amplitudes. This method is therefore only applicable when using this strain combination. The latter two corrections require the mean biaxiality ratio. Therefore, the Biaxiality Analysis toggle must be set on to select these methods. They should only be used where surface resolved stresses are available. See Biaxiality Correction Options, 514 for more explanation of the different correction methods. These methods require that the loading be approximately proportional. The Neuber method in conjunction with the Maximum Absolute Principal strain is the method of choice if the mean biaxiality ratio is close to zero. Otherwise the biaxiality is better taken into account either by using the Neuber correction in conjunction with a yield criterion-based parameter such as Signed von Mises or Signed Tresca, or by using the Hoffmann-Seeger or the Material Parameter modification (Ratio) method.

Parameter	Description
Stress/Strain Combination	This option menu selects the stress or strain parameter used in the fatigue analysis. The six multiaxial stress/strain components defined by the stress or strain tensor are resolved into one uniaxial or combined value for fatigue calculations for each node for each time step. This is necessary since the fatigue damage models used in MSC Fatigue are based on theories which deal with uniaxial stress or strain. These stress or strain scalar combinations can be either one of these components, *X Normal*, *Y Normal*, *Z normal*, *X-Y Shear*, *Y-Z Shear*, *Z-X Shear*, *Max. Abs. Principal*, *Max. Principal, Min. Principal, Signed von Mises*, *von Mises, Signed Max. Shear, Signed Tresca or Critical Plane*. Note that the shear strain components are engineering shear strains (two times the tensor shear strains). When the stress state is uniaxial or the maximum shear stress is more than half the absolute maximum principal, the most appropriate selection is Max. Abs. Principal. In other circumstances, the Signed von Mises will be more conservative. The sign on the Signed von Mises and Signed Tresca is taken from the sign of the absolute maximum principal value. It is necessary to sign these stress/strain parameters otherwise non-conservative fatigue life estimates will result.
Certainty of Survival	This defines the *Certainty of Survival* based on the scatter of the e-N curve. For example, to be 96% certain that the life will be achieved, set the slider bar at 96. This value is used to modify the e-N curve according to the standard error scatter parameter (SE), so the design criterion parameter will be meaningless if the value of SE is 0. A Design Criterion value of 50 leaves the e-N curve unmodified.
Factor of Safety Analysis	This option will cause a type of Factor of Safety or over design analysis to be performed. It informs the user how much stress may be modified for optimization purposes based on fatigue life. This analysis is in addition to the normal fatigue calculations and must be requested. This analysis method can be very useful for those components which predict an infinite life providing a measure of the risk of fatigue failure. The results of this analysis are stress factors for each location (node or element) for which the analysis has been performed. A value of one suggests that the specified life will be exactly attained whereas a factor less than one means the desired life will not be attained. Factors greater than one are, therefore, most desirable. Certain parameters must be supplied in order to proceed with this analysis. They comprise the remainder of the parameter descriptions in this table.
Design Life	For a Life based Factor of Safety analysis enter the target design life.
Maximum Factor for Calculation	Enter a maximum factor (default is 100) to be used in the analysis. This number can be lowered to speed up the calculation if it is known that the maximum stress factor of interest will be less that the default.
Material Cutoff	This toggles the usage of the material cutoff value in the analysis.

Crack Growth Solution Parameters

This form appears when the Analysis type is set to Growth (Crack Growth).



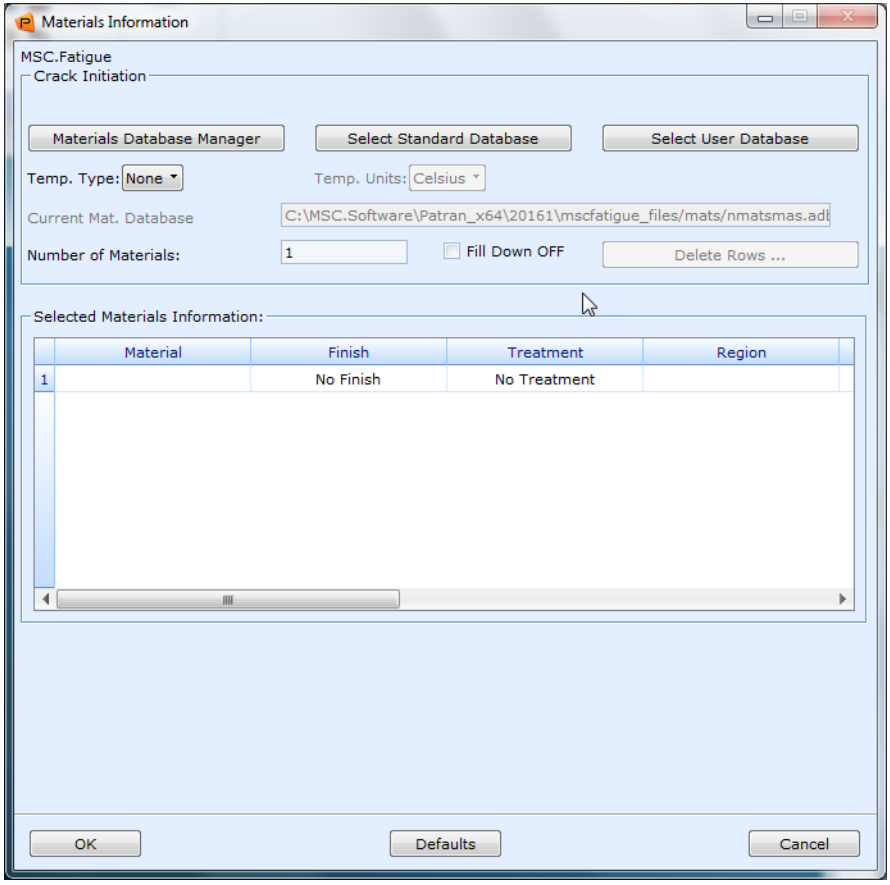
The following table describes the parameters in detail:

Parameter	Description
Compliance Function	Compliance Functions (denoted by Y or b parameters) which have been created by the MSC Fatigue module PKSOL will appear in this listbox. A crack growth analysis cannot proceed without a compliance function and must be selected. The compliance generator (PKSOL) can be invoked from this form by selecting the *Compliance Generator* button. After a compliance function has been created, it will appear in the listbox. The XY Plot application can also be used to plot the compliance function by using the *Plot* button. XY data values are used from a file called `function.xyd` and format information is retrieved from a file called `function.tem` where `function` is the name of the compliance function. Full operational instructions for PKSOL are provided in K Solution Library (PKSOL), 535. The compliance function will have a file extension of `function.ksn`.
Plane Stress Correction	To cause the crack growth code to correct for plane stress, turn this toggle ON.
Stress Combination	This option menu selects the stress parameter used in the fatigue analysis. The six multiaxial stress components defined by the stress tensor are resolved into one uniaxial or combined value for the crack growth calculation. This is necessary since the crack growth models used in MSC Fatigue are based on theories which deal with uniaxial stress. These stress or strain scalar combinations can be either one of these components, *Max. Abs. Principal, Max. Principal, Min. Principal, Signed von Mises, von Mises, Signed Max. Shear, Max. Shear/Tresca, X Normal, Y Normal, Z Normal, X-Y Shear, Y-Z Shear, or Z-X Shear*. When the stress state is uniaxial, or the maximum shear stress is more than half the absolute maximum principal, the most appropriate selection is Abs. Max. Principal. In other circumstances, the Signed von Mises will be more conservative. The sign on the Signed von Mises and Signed Max. Shear is taken from the sign of the absolute maximum principal value. It is necessary to sign these stress/strain parameters otherwise overly conservative fatigue life estimates will result.
Crack Length Units	Choose the crack length units that are convenient for defining the crack parameters below. Choices are *Millimeters*, *Meters*, *Inches*, *Milli inches*.
Initial Crack Length	Enter a crack size greater than or equal to 0.0 in the units specified above. If zero is entered, the actual starting crack size will be calculated from a consideration of the limits of fracture mechanics. See Fatigue Theory (Ch. 16) for a technical explanation.
Final Crack Length	Enter a crack length greater than the initial crack length in the units specified above. Note that the final crack length cannot exceed physical dimensions of the structure.

Parameter	Description
Notch Depth	It is possible to compensate for the effect of a notch (not necessarily represented by the K solution or the FEA analysis solution) analytically by filling in this and the next two databoxes. If it is not desirable to enter a notch depth, a notch radius, and/or a sharp crack radius, then enter zeros. If the Notch Depth is zero, then the other two parameters are not needed and are automatically set to zero in the analysis. Since zero values are not actually allowed, default values are used. These default values are explained in Crack Growth Prediction (PCRACK) (Ch. 8).
Notch Radius	This input defines the radius of the imposed notch which defaults to 0.000039 inches (0.001 mm) if the Notch Depth is zero or this parameter is set to zero.
Sharp Crack Radius	This input defines the root radius of a sharp notch or real fatigue crack which defaults to 0.000039 inches (0.001 mm) if the Notch Depth is zero or this parameter is set to zero.

Materials Information Form

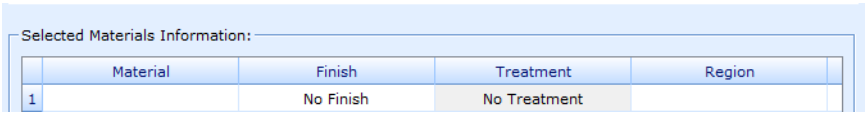
By selecting the Material Information button located on the main MSC Fatigue setup form, a Materials form will appear. This form differs for the various fatigue analysis types. Each analysis type and its form variations will be discussed separately. The form is divided into two basic parts: the material setup (spreadsheet) and the input selection area.



The area of the form below the spreadsheet will update itself depending on which cell of the spreadsheet is active.

From this area, the user selects the necessary input either from a listbox, an option menu, or a databox, all of which appear as the appropriate cell becomes active. When the appropriate input has been selected the adjacent cell automatically becomes active and the bottom of the form updates itself until the spreadsheet is completely filled out. The user may click on any cell at any time in order to modify the spreadsheet.

The input to the spreadsheet is different for each analysis type. The spreadsheet in the form on the previous page is representative of the Crack Initiation Material Parameters, 67 and the S-N Material Parameters, 64 analysis type. However, for the Crack Growth Material Parameters, 69 analysis type the spreadsheet appears as shown below. If the analysis type is changed on the Main Setup form, the Materials form will be closed automatically to ensure the proper input in the spreadsheet.



Hundreds of combinations of material data sets, surface finish and surface treatment may be defined for all analysis types except *Crack Growth* which can only have one (1). The default is one (1) material for any analysis type and the number of rows of the spreadsheet is dependent on the *Number of Materials* entered at the top of the form.

Important:	For the spreadsheet to update, the user MUST use the <RETURN> key after entering the number of materials or when entering any number into a databox.

Materials Form -- Definition of Buttons

Materials Database Manager

Also, at the top of the form, the user may access MSC Fatigue's materials *Database Manager* (PFMAT) by clicking on this button with the mouse. This option initiates a separate program in the MSC Fatigue system. The program may also be started from the operating system prompt by typing the symbol **pfmat**. The program operates interactively. The technical background for the materials data is described in Fatigue Theory (Ch. 16) and the detailed operation of PFMAT is described in Material Management (Ch. 4).

PFMAT accesses a central or a local materials database and is invoked for browsing or adding materials. The central database is protected and can only be read (unless the password is known and protections are set properly), whereas the local database may be edited freely. The local database is called `nmats.mdb` and may be seen in the working directory when a local copy of the database has been made. A file called `nmats.adb` will also reside in the local directory if a local database is created. This file is read by the MSC Fatigue PCL menus when referencing a material from the database. It is possible to also have a user-defined materials database. These databases must have the extension `.mdb` and `.adb` but the prefix may be any name. Both local and users defined material databases are created via the PFMAT module.

The default directory is always the central directory and shows up greyed out in the Current Mat Database box. When a material is added (normally it is not a good idea to touch the central database even if you have a password to access it), a local copy of the master database is made by copying the central database and is renamed to nmats.mdb & nmats.adb. Note that the user does not need to perform this 'copy', it is done automatically if the user tries to create or edit a material in the central database and enters a blank password. The contents of central are added to the local directory copy. The current Mat database box now shows the database it is using for the subsequent analysis.

It allows the addition of 2 materials in data set 1 and 2 at any one time. To address the addition of more than 2 materials, the user can unload the contents of data sets 1 and 2. Data set 1 and 2 only refer to what is currently loaded for display, not what is accessible during an analysis.

Select Standard Database

Scans the local directory for an nmats database and if one exists this is used. The message in the text box shows that the nmats directory is being used. If a local copy (nmats) does not exist the central directory is used and the message in the text box reflects this. This is the 'standard' method.

Select User Database

Allows the user to select a named database on any path accessible to the user. This would normally be the company's or the department's own preferred materials, either copied from MSC Fatigue supplied standard database, or their own test results.

In the case of the user created database, a password previously defined by the user at the creation stage, must be entered to add/remove or modify properties. Note that both a User database and a local (nmats) database can coexist in the run directory and selection is made by using the Select User Database button. However, it is recommended that the users create their own (company) named database and do not rely on the local nmats copy, for reasons of good data management.

S-N Material Parameters

The following tables explain each of the possible materials input of the spreadsheets for each of the fatigue analysis types.

Parameter	Description
Material	When this cell is selected, a listbox of available S-N curves appears at the bottom of the form. These are the available S-N curves that are defined in the MSC Fatigue material database. Selecting one of these curves from the listbox will fill the active cell with the material name and make the adjacent cell active. Although the cell and listbox refer to these curves as materials, they are actually S-N curves. All datasets stored in the MSC Fatigue materials database are referred to as materials. Datasets which have strain data (ε-N) associated with them also appear in the listbox since S-N curves can be synthesized from strain data. Care must be taken if both S-N and ε-N datasets have been defined in the same material entry. S-N datasets take precedence. There are two types of S-N curves, component and material. See Component vs. Material S-N Curves, 169 for more details.
Finish	The possible surface finish choices are *No Finish, Polished*, *Ground*, *Good Machined*, *Ave. Machined*, *Poor Machined*, *Hot Rolled*, *Forged*, *Cast*, *Water Corroded*, *Seawater Corr.*, and *User Defined*. Unless modified, the User Defined, Polished, and No Finish picks are the same. The No Finish simply treats the material properties 'as-is' with no modifying effects as does Polished. To modify the User Defined surface finish use the Loading Management (Ch. 5) module and load in the file `csuser.sur` delivered with the MSC Fatigue system in the `ptime` directory where it can then be modified. This file defines the relationship between the notch sensitivity and the UTS of the material for a particular surface finish. The edited `cfuser.sur` file must then be copied to the `mats` delivery directory before MSC Fatigue will recognize its presence. In certain instances, the surface treatment and surface finish menus may not be applicable. See Treatment below.
Treatment	A surface treatment may also be specified. This is a process which may be used to enhance the fatigue life as opposed to the surface finish which is normally a result of the manufacturing process. In certain instances, the surface treatment and surface finish menus may not be applicable. This is dependent on the type of material used. For instance, certain types of metals do not have data to support surface treatment and finish, and therefore are not allowed by the program. For these materials, a *Polished* finish and *No Surface Treatment* is assumed no matter what the user has set. Also certain surface finishes (*Water Corroded* and *Seawater Corr.*) do not allow surface treatment and therefore suppress the surface treatment menu assuming no treatment. The user will be informed of any inconsistencies at submit time. Available options are *No Treatment*, *Nitrided*, *Cold Rolled*, and *Shot Peened*. Component S-N curves ignore surface finish and treatment.

Parameter	Description
Region	When this cell is active a listbox will appear in which all existing groups will be listed. There is always one group called the default_group. Other groups must be created and/or modified using the Group facilities available from the main menu bar in either Patran. The groups selected for a fatigue analysis must have the nodes or elements of interest defined in them. For fatigue analyses which use more that one material dataset, care must be taken to ensure that there are no overlapping or duplicately defined nodes or elements. If multiple materials and surface finish/treatment combinations are used and there is an overlap in the defined regions, the last material combination will take precedence for that node or element. FE results at these nodes or elements will be used in the subsequent fatigue analysis. Only results for 2D and 3D elements are supported. For this reason, these groups should not reference entities of any other type. **Important Note**: Even though groups names allow spaces, for use with fatigue regions they cannot have any spaces in the name, either leading, trailing or anywhere in between.
Layer	Layer is generally associated with shell element surface (z1, z2). The z1/z2 nomenclature is the Nastran definition for bottom/top shell surface. The layers available are the only ones available in the model. If solids only, appears as un-layered. If "Composite" option is chosen, a model with multiple intermediate layers can have 1 – n layers.
Temperature	This cell is only available on the spreadsheet when the "Temp. Type" optionmenu is set to "Region". It allows the users to specify temperatures on the analysis groups.
Kf	A concentration factor can be specified for each material and surface finish/treatment combination. The default is 1.0 (no modification) and must be a number greater than zero.
Weld	If a fatigue analysis based on the British Standard BS5400 standard is being prepared, the S-N dataset name will be one of the weld classes, (e.g., CLASSC or CLASSF2). BS5400 also makes a further distinction between welded and non-welded details. Consequently, if an S-N data set name beginning with the word CLASS is supplied, an option menu will appear. If a weld is to be analyzed, the response should be *Yes*. Answering *No* implies that the actual detail is not a welded connection but is classified correctly by BS5400 under the S-N data set name supplied. (This can be confirmed by using the weld classification option in the materials database manager, PFMAT.) Weld S-N curves are considered component S-N curves. See Component vs. Material S-N Curves, 169 for more details.
Multiplier	A multiplier can be specified for each material and surface finish/treatment combination. The default is 1.0 (no modification). The multiplier and the offset below are useful when applying a correction over a region, e.g. for a residual stress or preload corresponding to a particular group that is not modeled or taken into account in the FE analysis.
Offset	An offset can be specified for each material and surface finish/treatment combination. The default is zero (no modification). See Multiplier.

Note:	For a **material**, the scaling is applied to the local response in the materials units. For an SN calculation, it will be applied in Stress, in the units used in the FE analysis, subject to the scaling factor if used. The offset is applied after the scaling factor.

Crack Initiation Material Parameters

Parameter	Description
Material	When this cell is selected a listbox of available material datasets appears at the bottom of the form. These are the available datasets containing strain-life (e-N) data that are defined in the MSC Fatigue material database. Selecting one of these from the databox will fill the active cell with the material name and make the adjacent cell active.
Finish	A poor surface finish will cause a significant reduction in the crack initiation life. For this reason, it is important to consider surface finish when carrying out such a life analysis. The possible choices are available in the resulting option menu. They are *No Finish, Polished*, *Ground*, *Good Machined*, *Ave. Machined*, *Poor Machined*, *Hot Rolled*, *Forged*, *Cast*, *Water Corroded*, *Seawater Corr.*, and *User Defined*. Unless modified, the User Defined, Polished, and No Finish picks are the same. The No Finish simply treats the material properties 'as-is' with no modifying effects as does Polished. To modify the User Defined surface finish, use the Loading Management (Ch. 5) module and load in the file `csuser.sur` delivered with the MSC Fatigue system where it can then be modified. This file defines the relationship between the notch sensitivity and the UTS of the material for a particular surface finish. The edited `cfuser.sur` file must then be copied to the `mats` delivery directory before MSC Fatigue will recognize its presence.
Treatment	A surface treatment may also be specified. This is a process which may be used to enhance the fatigue life as opposed to the surface finish which is normally a result of the manufacturing process. In certain instances, the surface treatment and surface finish menus may not be applicable. This is dependent on the type of material used. For instance, certain types of metals do not have data to support surface treatment and finish and therefore are not allowed by the program. For these materials a *Polished* finish and *No Surface Treatment* is assumed no matter what the user has set. Also, certain surface finishes do not allow surface treatment (*Water Corroded* and *Seawater Corr.*) and therefore suppress the surface treatment menu assuming no treatment. The user will be informed of these inconsistencies at submit time. Available options are *No Treatment*, *Nitrided*, *Cold Rolled*, and *Shot Peened*.

Parameter	Description
Region	When this cell is active a listbox will appear in which all existing groups will be listed. There is always one group called the default_group. Other groups must be created and/or modified using the Group facilities available from the main menu bar in either Patran or MSC Fatigue Pr & Post. The groups selected for a fatigue analysis must have the nodes or elements of interest defined in them. For fatigue analyses which use more that one material dataset, care must be taken to ensure that there are no overlapping or duplicately defined nodes or elements. The last time a node or element encountered in the analysis, its corresponding material dataset will be used. FE Results at these nodes or elements will be used in the subsequent fatigue analysis. **Important Note**: Even though groups names allow spaces, for use with fatigue regions they cannot have any spaces in the name, either leading, trailing or anywhere in between.
Layer	Layer is generally associated with shell element surface (z1, z2). The z1/z2 nomenclature is the Nastran definition for bottom/top shell surface. The layers available are the only ones available in the model. If solids only, appears as un-layered. If "Composite" option is chosen, a model with multiple intermediate layers can have 1 – n layers.
Temperature	This cell is only available on the spreadsheet when the "Temp. Type" optionmenu is set to "Region". It allows the users to specify temperatures on the analysis groups.
Kf	A concentration factor can be specified for each material and surface finish/treatment combination. The default is 1.0 (no modification) and must be a number greater than zero.
Shape Factor	The default for this parameter is infinity which implies a Neuber elastic-plastic correction. When selecting the Mertens-Dittmann or Seeger-Beste methods, any value greater than 1.0 may be defined. Only these methods use this parameter and setting the parameter to infinity reverts this method back to the traditional Neuber elastic-plastic correction. See the explanation on plasticity corrections in Crack Initiation Solution Parameters, 54.
Multiplier	A multiplier can be specified for each material and surface finish/treatment combination. The default is 1.0 (no modification). The multiplier and the offset below are useful when applying a correction over a region, e.g. for a residual stress or preload corresponding to a particular group that is not modeled or taken into account in the FE analysis.
Offset	An offset can be specified for each material and surface finish/treatment combination. The default is zero (no modification). See Multiplier above.

Note:	For a **material**, the scaling is applied to the local response in the materials units. For an EN calculation, this should be in strain -- and the units are always micro-strain. The offset is applied after the scaling factor. Please note that this is fixed, whatever the units are in the general setup. For example, a user may specify that they want an initiation life (also known as the strain-life method), based on Stress -- this is perfectly acceptable as MSC Fatigue converts the elastic stress into local elastic-plastic stress and strain during the Neuber and Rainflow calculations. The offset specified on the material form (to represent residual stress) MUST still be supplied in strain (micro-strain) and will be used on the elastic values before Neuber and Rainflow.

Crack Growth Material Parameters

Parameter	Description
Material	When this cell is selected, a listbox of available material datasets appears at the bottom of the form. These are the available datasets containing Linear Elastic Fracture Mechanics (LEFM) data that are defined in the MSC Fatigue material database. Selecting one of these from the databox will fill the active cell with the material name and make the adjacent cell active.
Environment	In the case of crack growth data, multiple data sets may be stored in the materials database representing different environments and so the relevant environment must also be selected. Hence, the environments being offered will depend on the materials dataset being used. In all cases the available environments will appear in the resulting listbox. User-defined corrosive environments may be specified using the data entry and environment modelling tools in Material Management (Ch. 4).
Region	Since the crack growth model uses a remote stress and corrects for geometry in the compliance function, it is necessary to define a node or element or set of nodes or elements in the nominal region only. The stresses should correspond to the stress that would have been computed if the crack or notch were not present. In the case of a set of nodes or elements, the stresses will be averaged for all nodes or elements to obtain a working nominal (or far field) stress. When this cell is active, a listbox will appear in which all existing Groups will be listed. There is always one group called the default_group. Other groups must be created and/or modified using the Group facilities available from the main menu bar in Patran The groups selected for a fatigue analysis must have the nodes or elements of interest defined in them. It is not recommended to use the default_group for this analysis type since it contains all the nodes and all the elements. **Important Note**: Even though groups names allow spaces, for use with fatigue regions they cannot have any spaces in the name, either leading, trailing or anywhere in between.

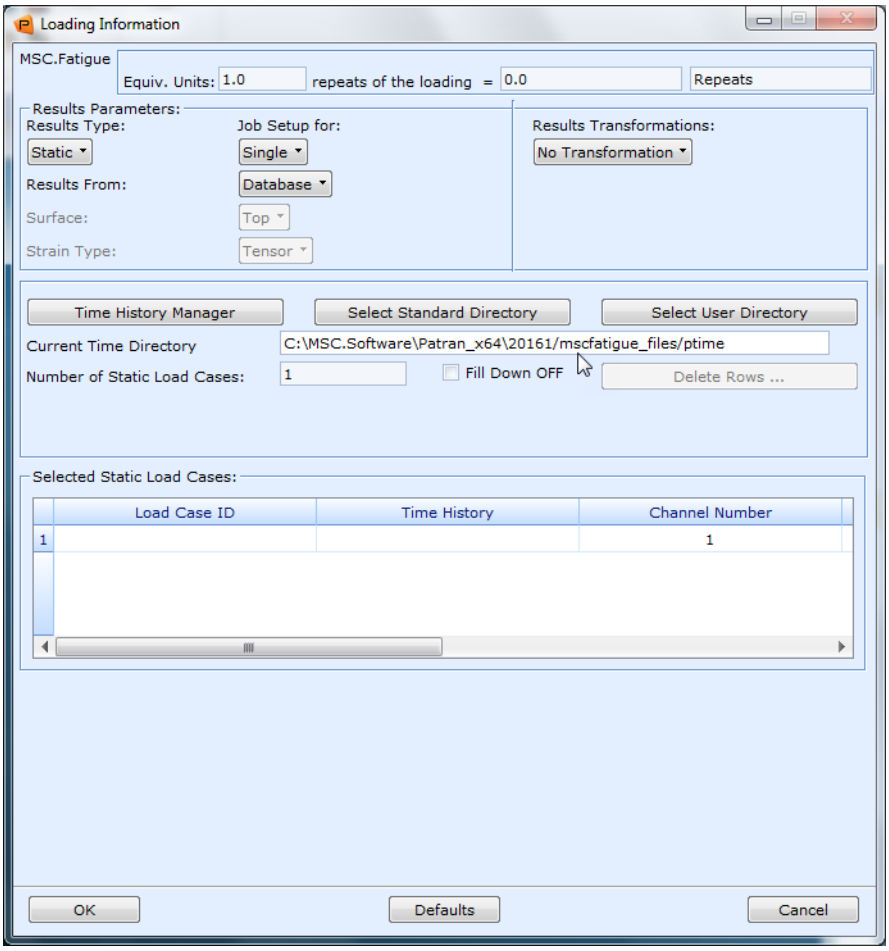
Parameter	Description
Layer	Layer is generally associated with shell element surface (z1, z2). The z1/z2 nomenclature is the Nastran definition for bottom/top shell surface. The layers available are the only ones available in the model. If solids only, appears as un-layered. If "Composite" option is chosen, a model with multiple intermediate layers can have 1 – n layers.
Multiplier	A multiplier can be specified for each material and surface finish/treatment combination. The default is 1.0 (no modification).
Offset	An offset can be specified for each material and surface finish/treatment combination. The default is zero (no modification).

Note: The offset is applied after the scaling factor to the time history file (.DAC) and the offset should be defined in the loading units.

Important: If a crack growth analysis is being performed using transient FE results, results from only the first node or element in the region (group) defined in the Materials Information form will be used in the analysis. Results from all nodes or elements in the defined region are averaged for crack growth analysis using static FE results.

Loading Information Form

By selecting the *Loading Information* button located on the main MSC Fatigue setup form, a Loading form will appear. This form applies to all of the basic fatigue analysis types. Each aspect of this form is discussed in detail in this section. The form is divided into three basic parts: general results parameters, loading setup (spreadsheet), and the input selection area.



Time History Database Manager

About a third of the way down the form, the user may access MSC Fatigue's time history Database Manager (PTIME) by clicking on the Time History Manager button with the mouse. This option initiates a separate program in the MSC Fatigue system. The program may also be started from the operating system prompt by typing the symbol **ptime**. The program operates interactively.

The detailed operation of PTIME is described in Loading Management (Ch. 5). PTIME manages a local database containing details of the loading time histories and rainflow matrices in the local directory. It enables the user to manipulate the time histories and matrices in order to prepare them for use during fatigue analysis. PTIME creates two files in the local user directory called `ptime.tdb` and `ptime.adb`. In most cases, these must exist for MSC Fatigue to operate successfully. The one exception to this is when the user uses finite element results from a *Transient* analysis as opposed to a *Static* analysis in which case no external time history or matrix data is necessary.

The two additional buttons at the same level on the form as the Time History Manager button allow for selection of the database directory. The Select Standard Directory button will select the local working directory as the location of the time history database (`ptime.adb`). The Select User Directory button will select a specified user directory where a time history `ptime.adb` may exist. Anytime a cell is selected under the Time History column of the spreadsheet, the spreadsheet that appears at the bottom of the form will be filled with the contents of the time history database from the selected directory. This enables users to keep libraries of time histories in separate locations and allows easy access to them without having to copy the needed time histories to the local working directory.

General Results Parameters

General results parameters that are set on the Loading Information form are described in the table below.

Parameter	Description
Results Type	There are two basic result types that MSC Fatigue can accept. These are Static Analysis Results, 78 and Transient Analysis Results, 81. The Loading Information form updates itself when this parameter is changed. The additional information needed when using one of these result types is described later in this section. For random vibration and frequency response analyses, see Vibration Fatigue (Ch. 9).
Job Setup For	This optionmenu has three possible values that are available depending on the Analysis type selected. The table below shows the available options for each analysis type: **Analysis TypeOptions Available** S-N - Static resultsSingle, Duty Cycle, Load Spectrum S-N - Transient resultsSingle, Duty Cycle E-N - Static resultsSingle, Duty Cycle, Load Spectrum E-N - Transient resultsSingle, Duty Cycle Growth (Static & Transient)Single Vibfat (Transfer Function &Single, Duty Cycle Direct PSD) WheelsNone Multiaxial CISingle, Duty Cycle (Static & Transient) Multiaxial FOSSingle (Static & Transient) SeamWeld (Static & Transient)Single, Duty Cycle Spotweld (Static & Transient)Single, Duty Cycle If the widget is set to "Single" then a single analysis is run. If this widget is set to "Duty Cycle" then a multiple analysis is run. Finally if the widget is set to "Load Spectrum" then a single analysis using the .spe and .lcs files is run.
Results From	The results can come from three different locations. The most common is directly from the Database Results, 75. Other options are directly from a Patran FEA Results, 76 file or from Patran External Results, 77 files (nodal or elemental).

Parameter	Description
Surface	Results are typically stored in the database in layers. For example, shell elements might have results at the top, middle, and bottom of the element and are reported as layers. Fatigue occurs at the surface generally; and therefore, the user is presented with the option of using FE results from the *Top* or *Bottom*. Top is the automatically default to be used. This parameter is unavailable and not used when the results are from MSC Patran external files.
Strain Type	Fatigue analyses based on FE strains must use engineering strain. Typically strains stored in the database are in tensor form. This means that in order to use them in a fatigue analysis, the shear strains must be multiplied by 2.0 to convert them to engineering strain. Unless it is known otherwise, strain quantities stored in the database are in tensor form. Strain from Patran FEA is already in engineering strain. Strain from external results files is unknown and it is incumbent upon the user to be aware that engineering strains are necessary for a proper fatigue analysis when requesting direct calculation from FE strains. Stresses do not have the same concern.
Results Transformations	This option is only available when the results are from the database. Also, this option is only applicable when doing a nodal-based fatigue calculation using FE results which are associated with elements. Often times elemental results from FE codes are output in the element coordinate systems. In order to properly calculate averaged nodal stresses from elemental FE results, it is necessary to transform them from their elemental systems to the basic coordinate system. The default is *No Transformation* in which the user must take responsibility that the elemental results are all in the same coordinate system, otherwise the choice is to *Transform to Basic* in which it is ensured that all results are in the basic coordinate system before any nodal averaging takes place.

Finite Element Results

Finite element results can be extracted and used in an MSC Fatigue analysis and can come from one of three distinct locations: directly from the Patran database, directly from an Patran FEA results file, or from external Patran results files. In all cases, only results for 2D and 3D elements are supported. 1D elements are not supported and their results should never be referenced.

An FE analysis may be carried out for a number of different purposes, and the modelling requirements depend on its intended use. For fatigue analysis, the results are very sensitive to the accuracy of the calculated stresses and strains in localized regions of a component. To achieve acceptable levels of accuracy, the following are essential requirements:

1. The **geometry** must be represented accurately.
2. Externally applied **loads and constraints** must also be represented accurately. Apparently insignificant changes to the way the loads and constraints are applied to the FE model can make surprisingly large changes to the deformation and hence the strains.

3. **Shell elements** must be used with care, and in particular, only where the structure is one which can reasonably be treated as a shell (i.e., where the thickness is small compared to significant geometric features).
4. It is important that elements are chosen with a view to generating **accurate grid point stresses** and strains as fatigue cracking usually starts at free surfaces and edges. In general, better results are likely to be achieved by using **higher order elements,** even if they are fewer in number. Use of higher order elements also permits better representation of geometric features.
5. Ideally, **the mesh should be refined** to a point where further refinement produces little change. The criterion used must be local stress and strain and not global stiffness. There is little to be gained by excessive refinement in non-critical areas; the sole requirement in these parts is that they transfer loads correctly to the critical areas.
6. Use of **triangular and wedge elements** should be minimized and care should be taken with aspect ratios. The effects of joins between elements of different types and shells of different thicknesses need to be carefully considered as these have the capacity to act as fictitious stress raisers.
7. Wherever possible, **verification of the FE calculated strains** should be made by comparing with strain gauge measurements.

Database Results

If the results are contained in the Patran database, set the *Results From* option menu to *Database*. For Static results, a listbox will appear each time the user selects the Load Case ID cell in the spreadsheet from which he can select an appropriate database results case. For Transient results, a similar listbox appears except that the user does not need to select the results. All results cases (time steps) that appear in the listbox will be used in the fatigue analysis.

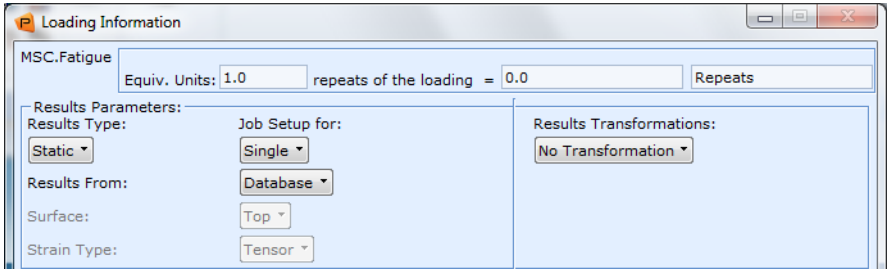


Figure 3-2 Database results

It is appropriate to review the manner in which results are stored in the database in order to avoid confusion as to what results type will be used in a fatigue analysis.

Results stored in the database can be associated with either the nodes or the elements of a model. When the results are associated with nodes each node will have six (6) component stresses or strains when considering tensor results.

Results associated with elements have element positions defined. This means that there will be multiple results for each element. These element positions can be the nodes, gauss points or the element centroid. If only one element position exists, then generally this means the results exist at the element centroid in which

case each element will have six (6) component stresses or strains. If more than one element position exists then there will be six (6) component stresses or strains for each element position for each element.

In addition, multiple layered results can exist for results associated with both nodes and elements. When multiple layers exist, it is necessary to select the *Surface* as either *Top* or *Bottom*. Top is the default if no Surface option is selected and the program encounters results with more than one layer.

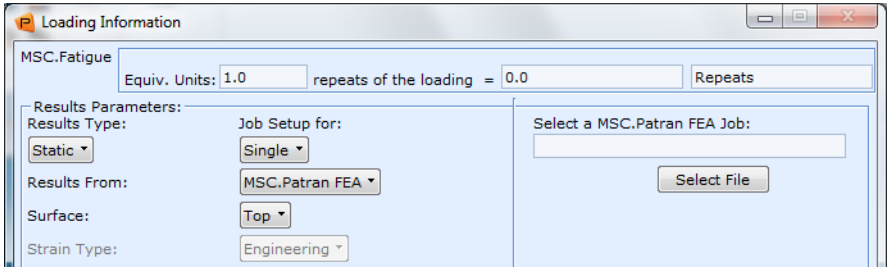
With all these in mind, four basic scenarios are possible:

Scenario	Description
Nodal fatigue analysis- results associated with nodes	This is the simplest case where a nodal-based fatigue calculation has been requested. If MSC Fatigue encounters stress or strain tensor results associated with nodes it will use them directly. No coordinate transformations are necessary and the results will be used 'as-is' from the database at each node.
Nodal fatigue analysis- results associated with elements	This is a secondary situation if the above situation does not exist. If MSC Fatigue encounters stress or strain tensor results associated with elements it will first determine how many element positions there are. If only one element position is encountered, the program will stop and suggest (via the `jobname.msg` file) that the user change the analysis to an element based fatigue analysis. If more that one element position is encountered the program will first transform the element based results to the basic coordinate system (if this option has been selected). Secondly, it will extrapolate the results from the element positions out to the nodes. Thirdly, it will average the results at the nodes for each element.
Elemental fatigue analysis- results associated with nodes	If this situation is encountered, the program stops and warns the user in the `jobname.msg` file to change the job to a nodal based fatigue analysis. The program was unable to find element based results in the database.
Element fatigue analysis- results associated with elements	This case has two scenarios possible. The first is if the results have only one element position. This is the simplest and preferable case. It means that the element results are at the element centroids and the results will be used directly with no changes. If there is more than one element position, the results at all positions are averaged to give a single tensor result for each element. This is the second condition. The first takes precedence if both of these results types are encountered.

Patran FEA Results

If the results are contained in an Patran FEA results file, set the *R*esults From option menu to Patran FEA. A databox and button will appear at the side. Either enter the name of the job that contains the results of interest or select the name from the listbox that comes up when the Select File button has been pressed. Results from

Patran FEA are stored in these files as nodal element results, or in other words, results for each element are reported at the nodes. MSC Fatigue averages the results at the nodes from each element. No consideration of element coordinate systems or other coordinate systems is given when doing the averaging. The results are used as contained in the results file. If it is necessary to consider element or other coordinate systems before averaging, the user will have to import the results into the database and use the transformation options available under the Database results option menu pick.

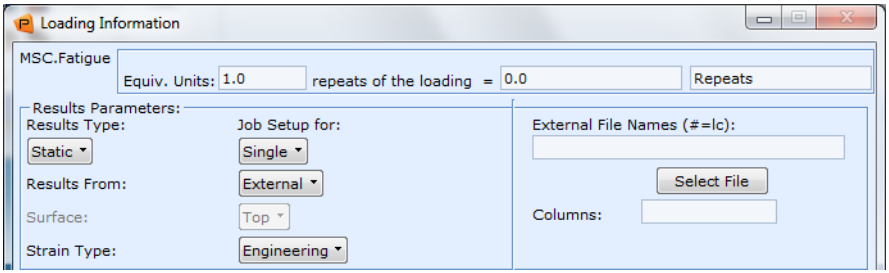


If shell elements are used, it is also necessary to specify the appropriate layer or *Surface* of results to use, either *Top* or *Bottom*.

Important:	It is highly recommended that when using shell elements from Patran FEA, a local coordinate frame be referenced to define the elemental x-coordinate direction. This ensures that all elemental stresses-strains are relative to the same coordinate frame to produce correct averaged nodal stresses and strains. This is only necessary when using Patran FEA nodal stress-strain results with shell elements.

External Results

If the results are in Patran external results files, set the Results From option menu to External. (These files are generated typically from old PATRAN 2.5 translators.) At the side of the form two databoxes will appear to allow specification of the external results files. In the top one, enter the generic filename for the results, with a # symbol where the load case ID should be placed. For example, if the load case results for load cases 1, 3 and 4 are stored in files `results1.nas`, `results3.nas` and `results4.nas`, respectively, enter the response `results#.nas`. The program will automatically insert the load case IDs into the filename. This is necessary even if there is only one load case. These load case IDs are determined from the Load Case ID column of the spreadsheet on this form.



In the bottom databox, identify the locations (*Columns*) of the component stress or strain results within the results files. The columns must be specified in the order X, Y, Z, XY, YZ, ZX, even if some of the columns may be zero. For example, if the six component stresses or strains are in columns 1 through 6, respectively, enter "`1,2,3,4,5,6`".

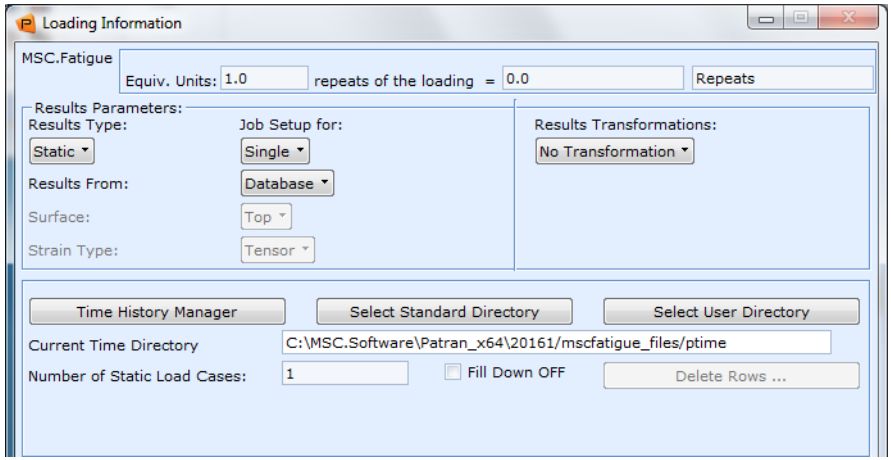
Important:	Using different elements in the same FE analysis could pose a problem if the stresses-strains for these different elements are not in the same columns. You may have to run separate fatigue jobs corresponding to the separate element types if this is the case. Also, if some components of stress-strain are known to be zero, make sure you reference null columns.

Results Types

MSC Fatigue accepts result from either a linear *Static* analysis or from a linear *Transient*/forced vibration analysis. The former requires that a definition of the variation in loading with time be defined by an external time history whereas the latter takes this into account automatically due to its nature.

Static Analysis Results

When performing a fatigue analysis using MSC Fatigue there is a distinction between load cases and separate fatigue analyses. A fatigue analysis can have associated with it up to 200 different static finite element load cases. These finite element load cases must have already been defined when setting up the model for an initial fatigue analysis. They MUST exist prior to setting up a fatigue analysis as well as their results.



The *Number of Static Load Cases* can be updated by changing the value in the corresponding databox. The user must use the **<RETURN>** key to make the change effective. The spreadsheet appearing below this will update itself to reflect the number of static load cases to be defined.

Again, the results by default are assumed to be in the database but can come from either Patran FEA results files or from Patran external results files also. The following table explains the rest of the input necessary to fully define the static load cases and their corresponding time history information. When a cell in the spreadsheet is activated the bottom of the form updates to allow for input. When the input has been accepted the next adjacent cell is selected and so forth until all the data has been completed.

Parameter	Description
Load Case ID	When this cell is activated a listbox or databox appears below the spreadsheet. When the results are from the *Database*, a listbox appears with a list of results contained in the database. At first this listbox will appear empty. To fill it with results contained in the database use the Getting and Filtering Database Results, 82 button. Select one of the results cases with the mouse to insert the ID into the cell. When the results are from Patran FEA or external files a databox is displayed where the load case identity must correspond to either the load case ID from Patran *FEA*, or to the load case ID to be inserted into the generic results file name when the code is *External*. In order for the spreadsheet to accept the Load Case ID, the user must use the <RETURN> key.

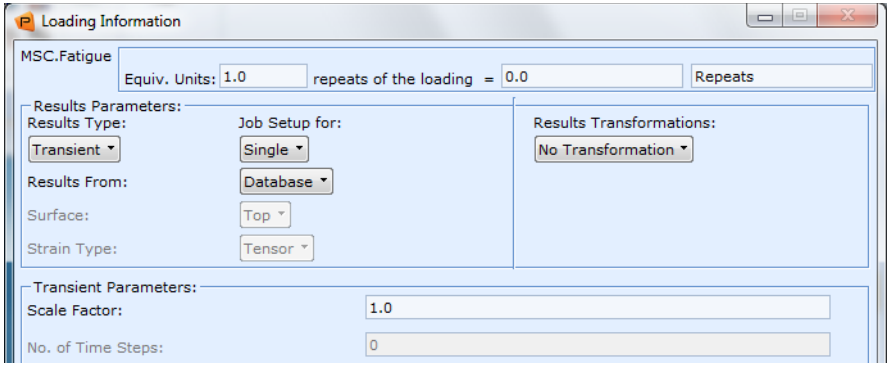
Parameter	Description
Time History	When this cell is activated another spreadsheet appears at the bottom of the form. The existing time histories and rainflow matrices available in the time history database appear here. Clicking on any portion of the row will input the time history name into the cell. The time history spreadsheet shows the name of the time histories with their corresponding load types, unit types, and maximum and minimum values. Only one time history is allowed per load case. Rainflow matrices can only be used for single load cases. The same time history cannot be used for two separate load cases even though the spreadsheet will allow it. (Use the Duplicate option in Loading Management (Ch. 5) to create an identical time history.) If more than one load case has been defined, a *STATIC Offset* entry will appear. A STATIC Offset load case means that the stresses/strains from this particular load case will be used as offsets to the stresses/strains used in the fatigue analysis from the other load case(s). This may represent a gravity or centrifugal loading, or a stress state arising from the manufacturing and assembly process. Also the type and units of the specified time histories must match those of the finite element load cases.
Load Magnitude	The magnitude of the FEA load must be supplied in the same units as those reported above for the time history. This magnitude is used as a scale factor to normalize the finite element stresses or strains to obtain the stress/strain distribution due to a unit loading. It is necessary to enter the total value of the loading applied in the FE load case. If multiple-point loads are applied across an edge (e.g., to simulate a distributed load), the time variation in this loading may be described using a single time history and therefore the sum of all loads should be entered. If a non-uniform distributed load is applied in the FEA, you should use a parameter proportional to the loading (e.g., displacement) and supply the value of this parameter resulting from the loading. This value will be used to normalize the stresses or strains computed in the FE analysis. The time history is then used to scale the stresses dynamically for each time increment. This calibration can be thought of as the following mathematical statement: $[\sigma_{ij}](t) = \left(\frac{[\sigma_{ij}]_{fea}}{P_{fea}}\right) * P[t]$ Similarly, you may use the stresses/strains directly from the finite element analysis by specifying unity as the FE load magnitude. If a rainflow matrix is used, the same basic procedure is used where the matrix range and mean axes are scaled by the applied FE stress and divided by the applied load. In addition, if the applied FE stress is negative, then the matrix cycled are mirrored around zero.
Scale Factor	A scale factor can be applied to the load or time history. The default is 1.0 (no scaling).
Offset	A scalar offset can be applied to the load or time history. The default is 0.0 (no offset). This is valuable to show uniform offset loading such as assembly or gravity loads.

Important:	If a nonlinear relationship between stress and loading exists, this can be compensated for in two different ways. First, the nonlinear relation could be built into the accompanying time history created in PTIME. The second is to use an actual time step FE analysis. For the former case, the Load Magnitude would be set to unity. The latter case would be set up under the Transient Analysis Results, 81 variation of this form.

The *Fill Down* toggle allows for easy and quick setup of multiple load cases. With the toggle set to OFF, the selected cell of the spread sheet simply moves from cell to cell in a horizontal manner until the end of the row is reached at which time it moves to the first cell of the next row. If the toggle is set to ON then, starting at the selected cell, if an input is selected the cells directly below in the column are automatically filled beginning at the selected input. For example, if there are five load cases and the Load Case ID cell is selected in row 2, when a Results Load Case is selected it will fill all rows starting at 2 to the end with the next four Results Load Cases. It will do the same with the other columns and their corresponding inputs of the spreadsheet.

Transient Analysis Results

For a Transient analysis, the form updates as shown.



For a Transient analysis, no spreadsheet appears and only three basic pieces of information are needed.

Parameter	Description
Scale Factor	If it is desirable to use the transient dynamic results 'as-is', then use the default value of 1.0. Otherwise, enter an appropriate scale factor. No load case data or time history loading is necessary.
No. of Time Steps	When using results directly from the database this parameter is not necessary and will be updated automatically when results are selected from the listbox. Otherwise, enter the number of time steps desired to use in the fatigue analysis. If the results are from Patran FEA, this defines the number of time steps to use from an existing Patran FEA results file. If the results are from Patran external files, this defines the number of external results files to read.
Results Time Steps	This listbox contains the time step results data that exist in the database. This listbox will be empty when the Loading Information form is first invoked or if the Results Type is changed. The user must use the Getting and Filtering Database Results, 82 button in order to fill the listbox. Once the listbox is filled with the appropriate time steps the form may be closed by clicking on the OK button. It is not necessary to select any of the time steps in the listbox. All time steps appearing will be used. If unwanted results appear in the listbox, it will be necessary to eliminate them from the list using the Filter form.

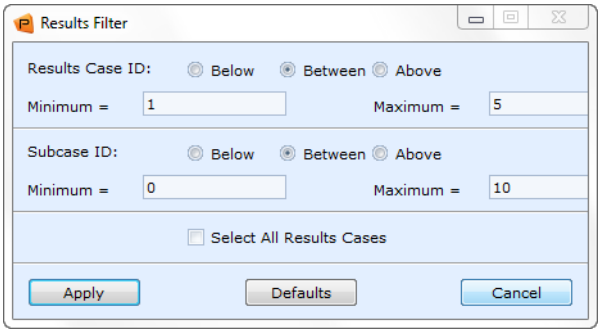
Important:	If a crack growth analysis is being performed using transient FE results, results from only the first node or element in the region (group) defined in the Materials Information form will be used in the analysis. Results from all nodes or elements in the defined region are averaged for crack growth analysis using static FE results.

Getting and Filtering Database Results

When the Loading Information form is first presented, the listbox containing the database result cases or time steps appears empty. It is necessary to invoke the Get/Filter Results form in order to fill the listbox with the relevant results.

Static Result Cases

For static result cases, the following form allows for result filtering and selection.



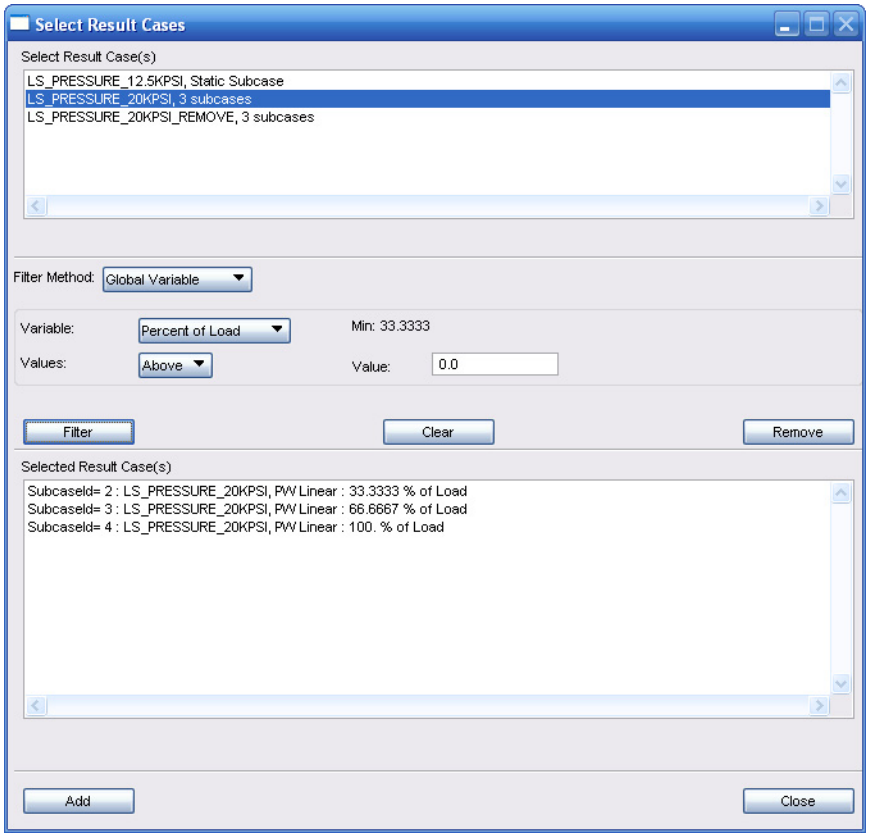
Results are stored in the database by Primary Results Case IDs and Subcase IDs. The results can be filtered using these IDs so that only the result of interest show in the listbox. To quickly fill the listbox without regard to which results cases show, simply click on the *Defaults* button or turn on the Select All Results Cases toggle. This will automatically determine the range of primary and secondary IDs. By clicking on Apply, the form will be put away and the filtering will take place filling the listbox. The Select All Results Cases toggle will cause all results cases to be displayed in the listbox and will ignore any filtering requests.

The other switches (Below, Above, and Between) will find all results with IDs less than or equal to or greater than or equal to the numbers appearing in the databoxes. If after filtering nothing appears in the listbox or the minimum and maximum number in the databoxes are zero after clicking the *Defaults* button, check to see that the results have been imported into the database.

Transient Result Cases

A different filtering form is used to filter and select transient result time steps.

For transient result cases, the following form allows for result filtering and selection.



This form is expandable (by dragging the corners or edges) to allow you to view the entire Result Case names and global variable if necessary.

The general use of this form to select the appropriate time steps for a fatigue analysis is as follows:

1. Select the Result Case. The top of the form lists all available Result Cases with the number of subcases associated with the Result Case. These subcases can be time steps, load steps, frequency steps, design cases, or static subcases.
2. Set the Filter Method and Criteria.
3. Click the Filter button. To select all subcases of a particular Result Case, simply click the Filter button after the first step. The default filtering should allow for selection of all subcases.
4. Click the Apply button to transfer the selected subcases (time steps) to the Loading Information form.

Because this form treats all Result Cases and their subcases in a general way, it is up to the user to ensure that the results selected are truly from linear static time step analysis.

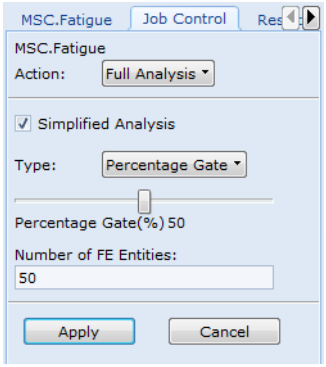
Table 3-1 Filter Methods

Method	Description
Global Variable	Any global variables associated with the selected Result Case will show up in the Variable option menu. Select the one you would like to filter with, change the criteria using the Values option menu and enter the value or range to filter by. Click the Filter button to complete the filter action. Click the Apply button at the bottom of the form to activate the filtered subcase selection.
String	Enter a string and use wild cards (the * character) to filter results. For example if you wanted all subcases with the string Time in it then you would use *Time* as the string with wild cards on each end of the word. Click the Apply button at the bottom of the form to activate the filtered subcase selection.
Subcase IDs	Subcases can be filtered on Subcase IDs by entering the appropriate IDs. To select separate IDs, separate them by spaces (1 3 5). To select a range use a colon between the numbers (1:5). To select by increments use two colons, for example: 1:10:2, which interpreted means select subcases 1 through 10 by twos. Or use any combination of spaces and colons between subcase IDs to select as many as you wish. Click the Apply button at the bottom of the form to activate the filtered subcase selection.
All	No filter method is selected. No options are available. Simply press filter and all subcases will be selected from whatever primary Result Case is selected. Click the Apply button at the bottom of the form to activate the filtered subcase selection.

Important:	Subcases from only one Result Case can be filtered and transferred to the Loading Information form.

Job Control

By selecting the *Job Control* button located on the main MSC Fatigue setup form, a Job Control form will appear. This form is generic for any of the fatigue analysis types. The form allows for job submission and monitoring, as well as other functions explained below. The form updates itself depending on the action required. In all cases, except when reading an old job setup, the action is linked to the jobname entered in the main MSC Fatigue setup form. This simple form appears as:



The actions that can be invoked from this form include submission of a Submit Full Analysis, 86 or Submit Partial Analysis, 89, Translate Only, 90, Save Job Only, 90 Monitor Job, 91, Abort Job, 93, Delete Job, 93, Read Saved Job, 94, Calculate Normals, 94, Interactive, 95 and Analysis Manager, 96. (The action that is invoked from this form uses the current jobname on the main setup form. Click on Apply to invoke the action.)

Submit Full Analysis

When the action is set to *Full Analysis* and the user clicks the Apply button the following occurs:

- The job begins the submission process by checking to see if an existing job of the same name exists. If it does, overwrite permission will be requested.
- All of the information requested in the main setup form and the subordinate solution parameter, materials, and loading information forms are written to a MSC Fatigue input file called `jobname.fin` and includes 90% of the fatigue input parameters. It is an ASCII unformatted file whose text lines consist of "Parameter = Value". The other 10% of the fatigue input information is retained in the database and consists of the region application information (nodes or elements) for the material and surface finish/treatment combinations and, in most cases, the FE results. If any information is not complete, the user will be notified and the submission process will terminate.
- Information is extracted from the database such as the region or group data and results via the PAT3FAT or FATTRANS translator. Patran is suspended while the translation is in progress. A `jobname.fes` file results from this translation. This is the fatigue analysis input file and is binary in nature. It can be translated to ASCII form and edited if desired.
- A Linux shell script , 383is invoked from which the actual fatigue analysis begins. The analysis goes through two or three basic steps. The fatigue input file is preprocessed via the FEFAT module. This consists of reading the `jobname.fes` file, superpositioning of load cases, rainflow cycle counting, among other things. The result of this operation is a file called `jobname.fpp` (`jobname.tcy` for crack growth analyses). (This submit script is actually a C program on Windows platforms.)
- The next phase consists of performing the actual fatigue calculations using the module FEFAT (or PCRACK for crack growth). The results of this operation is a file called `jobname.fef` (or `jobname.crg` for crack growth analyses).

- If additional calculations such as a Factor of Safety analysis have been requested, they are executed next.
- If MSC's Analysis Manager is installed and licensed, the job as described above will be submitted via the Analysis Manager as opposed to a Linux shell script (although the script is still executed). It is important that this module be configured properly for proper execution. See Analysis Manager, 96 for details.

When the job has been completed the results can be read into the database under the Results form. See Postprocessing Results, 100. For more information about the files created from a MSC Fatigue analysis and the actual operation of the MSC Fatigue modules, see Total Life and Crack Initiation (Ch. 6) or Crack Growth (Ch. 8). Also see The MSC Fatigue Jobname, 50.

The following files are generated during a Crack Initiation or Total Life (S-N) analysis:

Filename	Description
***jobname*.fin**	Fatigue job parameter data.
***jobname*.fes**	FE stress and fatigue input file.
***jobname*.fpp**	Fatigue preprocessing results file.
***jobname*.fef**	Fatigue results file.
***jobname*.fos**	Stress factor of safety results file.
***jobname*.msg**	Message file.
***jobname*.sta**	Job status file.
***jobname*nn.cyh**	Cycle distribution at node/element nn.
***jobname*nn.dhh**	Damage distribution at node/element nn.

The following files are generated during a Crack Growth analysis:

Filename	Description
***jobname*.fin**	Fatigue Input Data.
***jobname*.fes**	FE stress and fatigue input file.
***jobname*.tcy**	Crack growth analysis time history input.
***jobname*.crg**	Crack growth results file.
***jobname*.msg**	Message file.
***jobname*.sta**	Job status file.

Fast Analysis

A special option called Fast Analysis (FASTAN) is also available when performing a full analysis. This option is activated when the Simplified Analysis toggle is set to ON. The purpose of a Fast Analysis is to speed up the identification of the critical areas of a model. This option is especially useful for large models with

complex loading. The operation is done by extracting the peaks and valleys from the time histories (shortening the time histories but keeping the damage content) and then running a fatigue analysis which will identify the critical locations. These locations are then used in a full analysis with the full length time histories. This is the operation performed for multiple load cases.

If only a single load case is used, the peak valley extraction method is not used. Instead the time history is cycle counted and then the subsequent rainflow matrix is used for the remainder of the analysis. For a single load case, only the Simplified Analysis toggle needs to be turned on, but for multiple load cases, the settings of the other widgets on the Job Control form are taken into account. A description of all the parameters for multiple load cases follows:

Parameter	Description
Type	This specifies how the peak/valley extraction is to be performed. Three methods are available. The first is to specify a percentage gate of the total stress/strain range of the signal. The second is to specify the number of points to be retained in the time history. The third is to specify a percentage reduction of the total signal. Type is ignored for single load cases. Instead a rainflow matrix is created from the time history.
Percentage Gate	A percentage gate can be specified which is a percentage of the maximum stress/strain range of the time history. For example if the largest range is 1000MPa and the gate is set to 50 (percent), then any cycles encountered with ranges below 500 MPa will be ignored. This parameter is ignored for a single load case.
Number of Points	The total number of points to be retained in the time history can be specified. This number cannot be less than two nor can it be less than the number of points that a 99% gate would calculate. The program will automatically compensate if the user has specified an unacceptable number of points to retain. This parameter is ignored for a single load case.
Reduction Factor	This is the percent reduction factor by which to reduce the time history signals. For example if the signals have 1000 points, and you specify 50%, the signal will be reduced to 500 points. Again the same limits apply to this as apply when specifying the number of points to retain. This parameter is ignored for a single load case.
Number of FE Entities	Finally, you have the ability to specify how many of the most critical locations should be retained in the final analysis. If you specify 50 points, then the 50 points with the most damage from the analysis using the reduced time histories will be used in the full analysis with the full signals. Only these nodes will be reported back. (Although you may look at the results from the fast analysis which are called `jobname_short.*` and may be manually viewed and/or imported into the database.) This parameter is ignored for single load cases.

Important:	Using the Fast Analysis process can significantly speed up your analysis job (by orders of magnitude). However, you must be aware that by reducing the time histories, the subsequent analysis is only an approximation to help quickly identify the critical locations. Although the locations reported back have the correct life values, an improper gate or reduction factor could identify the incorrect locations, or miss other critical locations. Use with caution.

Submit Partial Analysis

When the action is set to *Partial Analysis* and the user selects the **Apply** button the following occurs:

- The job will begin the submission process by checking to see if an existing job of the same name exists. If it does, overwrite permission will be requested.
- All of the information requested in the main setup form and the subordinate solution parameter, materials, and loading information forms are written to a MSC Fatigue input file called `jobname.fin` and includes 90% of the fatigue input parameters. It is an ASCII unformatted file whose text lines consist of "Parameter = Value". The other 10% of the fatigue input information is retained in the database and consists of the region application information (nodes or elements) for the material and surface finish/treatment combinations and, in most cases, the FE results. If any information is not complete, the user will be notified and the submission process will terminate.
- Information is extracted from the database such as the region or group data and results via the PAT3FAT or FATTRANS translator. Patran is suspended while the translation is in progress. A `jobname.fes` file results from this translation. This is the fatigue analysis input file and is binary in nature. It can be translated to ASCII form and edited if desired.
- A Linux shell script , 383 is invoked from which the actual fatigue analysis begins. The analysis goes through two or three basic steps. The fatigue input file is preprocessed via the FEFAT module. This consists of reading the `jobname.fes` file, superpositioning of load cases, rainflow cycle counting, among other things. The result of this operation is a file called `jobname.fpp` (`jobname.tcy` for crack growth analyses).
- If MSC's Analysis Manager is installed and licensed, the job as described above will be submitted via the Analysis Manager as opposed to a Linux shell script (although the script is still executed). It is important that this module be configured properly for proper execution. See Analysis Manager, 96 for details.

At this point, the analysis stops and does not perform the full, global multi-location fatigue analysis or crack growth calculation. This might be convenient if it is desired to go directly to MSC Fatigue's Design Optimization or Crack Growth modules, knowing before hand where the critical locations are (see Design Optimization, 106) or for performing a Factor of Safety analysis at a known location.

For more information about the files created from a MSC Fatigue analysis and the actual operation of the MSC Fatigue modules, see Total Life and Crack Initiation (Ch. 6) or Crack Growth (Ch. 8). Also see The MSC Fatigue Jobname, 50.

Translate Only

When the action is set to *Translate Only* and the user selects the **Apply** button the following occurs:

- The job will begin the submission process by checking to see if an existing job of the same name exists. If it does, overwrite permission will be requested.
- All of the information requested in the main setup form and the subordinate solution parameter, materials, and loading information forms are written to a MSC Fatigue input file called `jobname.fin` and includes 90% of the fatigue input parameters. It is an ASCII unformatted file whose text lines consist of "Parameter = Value". The other 10% of the fatigue input information is retained in the database and consists of the region application information (nodes or elements) for the material and surface finish/treatment combinations and, in most cases, the FE results. If any information is not complete, the user will be notified and the submission process will terminate.
- Information is extracted from the database such as the region or group data and results via the PAT3FAT or FATTRANS translator. Patran is suspended while the translation is in progress. A `jobname.fes` file results from this translation. This is the fatigue analysis input file and is binary in nature. It can be translated to ASCII form and edited if desired.

At this point, the translation stops and no fatigue analysis or crack growth calculation are performed. This might be convenient if it is desired to edit the fatigue input deck and then continue on in an interactive mode. See Calculate Normals, 94 and Utilities, 360.

For more information about the files created from a MSC Fatigue analysis and the actual operation of the MSC Fatigue modules, see Total Life and Crack Initiation (Ch. 6) or Crack Growth (Ch. 8). Also see The MSC Fatigue Jobname, 50.

Save Job Only

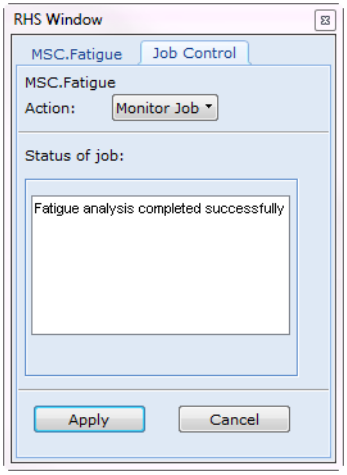
When the action is set to *Save Job Only* and the user selects the Apply button the following occurs:

- The job will begin the save process by checking to see if an existing job of the same name exists. If it does, overwrite permission will be requested.
- All of the information requested in the main setup form and the subordinate solution parameter, materials, and loading information forms are written to a MSC Fatigue input file called `jobname.fin` and includes 90% of the fatigue input parameters. It is an ASCII unformatted file whose text lines consist of "Parameter = Value". The other 10% of the fatigue input information is retained in the database and consists of the region application information (nodes or elements) for the material and surface finish/treatment combinations and, in most cases, the FE results. If any information is not complete, the user will be notified and the submission process will terminate.

This saved fatigue input file can be read back into the main MSC Fatigue setup form. See the Read Saved Job, 94 action below. For more information about the files created from a MSC Fatigue analysis and the actual operation of the MSC Fatigue modules, see Total Life and Crack Initiation (Ch. 6) or Crack Growth (Ch. 8). Also see The MSC Fatigue Jobname, 50.

Monitor Job

When the action is *Monitor Job*, the current job status will be reported in the status box. This is achieved by examining the contents of the `jobname.sta` file which is updated at various points during the analysis. The `jobname.msg` file will contain a history of the job (i.e., all the messages generated while running the current MSC Fatigue session). Continue clicking the **Apply** button for an updated current report.



Following is a list of some of the possible messages and a brief explanation.

Job Execution Status Messages

When a job is submitted, it will pass through three to five phases. The user will be informed through the status option of the progress of the job. Both success and error messages are displayed. The following list summarizes some of the typical, normal operation messages which the user may experience. Not all the messages will be displayed since the status file is updated very quickly in some cases. In certain cases, the status file may not be available in which case a "Try again" message will appear. When execution is through MSC's Analysis Manager, these messages appear in the Analysis Manager message window.

Phase 1

```
JOB jobname HAS BEEN SUBMITTED BUT HAS NOT STARTED EXECUTION
JOB HAS BEGUN EXECUTION
WRITING THE JOB (.FIN) FILE
```

Phase 2

```
PAT3FAT" reading the neutral file...
PAT3FAT" reading the.FIN file...
PAT3FAT" reading the FE results...
PAT3FAT" writing the.FES file...
PAT3FAT" terminated normally
```

Phase 3

```
Preprocessor loaded and running
Preprocessing n% complete
Writing intermediate data file
Preprocessing completed successfully
```

Phase 4

```
Fatigue analysis module loaded and running
Fatigue analysis n% complete
Fatigue analysis completed successfully
```

In addition there may be other messages giving status of other aspects of the job such as Factor of Safety analysis or Crack Growth analysis. Error messages are also displayed via these status messages.

What To Do When a Job Stops

If the status message does not appear to be updating, it is possible that the job has halted due to an error. In many cases, that error message will be reported through the status facility. However, if it is not reported, you can investigate the problem by opening another window and examining the following file:

jobname.msg: This file will contain all the status messages for the job including any error messages.

Some hints on determining why a job has failed:

1. If the `jobname.fin` file and the `database` exist in your directory, try running the job interactively by typing:

   `pat3fat jobname (if you were using the PAT3FAT translator)`

   `or`

   `fattrans jobname` `(if you were using the FATTRANS translator)`

   then checking the message file.
2. If the `jobname.fes` file exists, run the FEFAT program interactively and watch for error messages. Type **fefat** at the system prompt.
3. If the `jobname.fpp` or `jobname.tcy` files exist, run FEFAT or PCRACK interactively and watch for error messages. Type **fefat** or **pcrack** at the system prompt.

See Total Life and Crack Initiation (Ch. 6) or Crack Growth (Ch. 8) for an explanation of these MSC Fatigue executables. Also check the file called `batlog.lst` for any additional clues if none of the above helps.

Important:	If a job inadvertently quits, sometimes a `jobname.fpr` file is left in the directory. This file is created during submission to detect a running job so that inadvertent submissions while a job is in progress of the same jobname are detected. In some cases, it may be necessary to remove this file before resubmitting the job.

Error Messages

See Error Messages (App. 19) for a description of error messages and possible solutions.

Abort Job

If the action is set to *Abort*, the job will be aborted by selecting the Apply button. This is achieved by creating an empty `jobname.abo` file in the current directory. The MSC Fatigue modules periodically check for this file and when detected, will abort.

Likewise, a `jobname.abo` file can be created or the `jobname.sta` file renamed to `jobname.abo` from the operating system prompt and the same result will be achieved when a job is running.

If execution is via MSC's Analysis Manager, then the Analysis Manager will handle the abortion of the job. All files will automatically be cleaned up by the Analysis Manager.

Delete Job

If the action is set to *Delete*, the various files associated with the job will be deleted by selecting the **Apply** button. The files that will be deleted if encountered are:

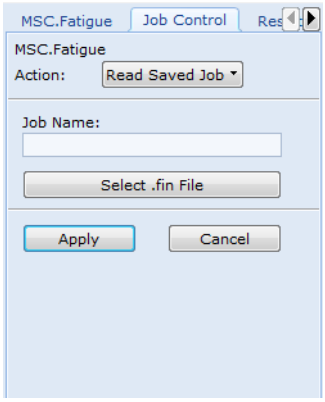
Filename	Description
***jobname*.fin**	Fatigue job parameter data.
***jobname*.fes**	FE stress and fatigue input file.
***jobname*.fpp**	Fatigue preprocessing results file.
***jobname*.fef**	Fatigue results file.
***jobname*.fos**	Stress factor or safety results file.
***jobname*.msg/log**	Message and log files.
***jobname*.sta**	Job status file.
***jobname*.tcy**	Crack growth analysis time history input file.
***jobname*.crg**	Crack growth results file.
***jobname*.abo**	Abort detection file.
***jobname*.rmn**	P/FATIGUE 2.5 results menu file. (obsolete)
***jobname*.vec**	Surface normals vector file.
***jobname*.*_tmpl**	Results template files.

Other files may also be deleted if associated with the particular job. Use this option with caution.

Read Saved Job

When the action is *Read Saved Job*, a databox and button will appear on the form. If the *select.fin file* button is pressed then a file select form is displayed with all the available jobs from the local directory. By selecting one of the existing jobs and clicking on the **OK** button, the Jobname databox is filled out. By clicking on the Apply button, the reading of the `jobname.fin` files will initiate. If these files are successfully read the widgets and parameters in the main MSC Fatigue setup form and the subordinate solution parameter, materials, and loading forms will be updated.

If the read is unsuccessful, some of the parameters certainly will not be complete. It is always beneficial to check and see if the parameters have been updated properly before attempting to submit the analysis.



Important:	If the name of the job is known beforehand, it is possible to type the name of the job in the Jobname databox on the main MSC Fatigue form and press the <RETURN> key to read a saved job or type in the fin filename directly into the databox on the Job Control form. If the job exists, permission to read will be asked.

Calculate Normals

This is an advanced feature of MSC Fatigue to allow for more control of the input stress or strain information to the fatigue analysis. This feature is only applicable to a nodal based fatigue analysis.

Fatigue crack initiation normally occurs at free surfaces. Equilibrium requirements dictate that the direct and shear stresses normal to a free surface are always zero. It follows that the surface normal is a principal stress direction, and that any non-zero principal stresses must lie in the plane of the surface. This state of plane stress means that any study of surface stress states can be reduced to a 2D problem, if the results are presented in a suitable coordinate system (e.g., one whose x and y axes lie in the plane of the surface). In addition, critical plane analyses require that stresses and strains be resolved onto planes intersecting the surface at angles of 90 and 45 degrees. This requires the surface normal to be referenced, and this is readily accomplished if the

results are in local coordinate systems such as those just described. The z-axes of these nodal coordinate frames must be normal to the surface of the component at each node.

The definition of a local coordinate system is dependent on the identification of a surface normal. A way to do this is to average the outward facing normals of element faces adjoining the nodes in question. This method may be applied, even if the node lies on an edge adjoining two facets of the component; the stress on external sharp corners should be uniaxial, and sharp internal corners generate theoretical elastic singularities where the approximate solutions generated by FE methods are meaningless anyway.

The outward surface normal defines the z-axis of the local coordinate system, and these surface normals are provided in the form of a set of direction cosines in the basic coordinate system. It then remains to define the local x- and y-axes. This is accomplished as follows:

1. If the direction cosines of the z-axis (i.e., the surface normal) are (0,0,1) i.e., k'=k, then the local coordinate system is the same as the global.
2. If the direction cosines of the z-axis are (0,0,-1) i.e., k'=-k, then the local x-axis is the same as the global x-axis and the y-axis is reversed, i.e., i'=i, j'=-j.
3. In any other case, the local x-axis is parallel to the cross product of the z- and z'-axes, with the y'-axis completing a right-handed set, i.e., parallel to the cross product of the z'- and x'-axes so that

$$i' = \frac{k \wedge k'}{|k \wedge k'|}; j' = k' \wedge i' \tag{3-1}$$

By clicking the **Apply** button with the action set to Calculate Normals, these surface normals are calculated and stored in a file called `jobname.vec` for each surface node of the model. The existence of this file during a job submission or when running the PAT3FAT translator will cause a transformation of nodal stress or strain values to these local coordinate system thus defining the surface state. Nodes that lie in the interior of the model are unaffected by this feature.

These averaged nodal outward normals are also graphically plotted for visualization and verification purposes. Press the *Remove Vectors* button to remove them. Once they have been removed they can only be replotted if the whole procedure is repeated.

In addition to this, after the translator PAT3FAT or FATTRANS has completed the translation, new nodal results files will exist by the names `jobname_lc#.nod` or `jobname_ts#.nod`. There will be one file for each load case or time step respectively. These files contain the new surface stress or strain states and can be imported into the database via the File Import form as Patran ".nod" files for visualization purposes. A template file to do this called `jobname.nod_tmpl` is also created.

Interactive

By clicking **Apply** with the Action set to *Interactive*, a separate MSC Fatigue module called FEFAT will be invoked. This module can be invoked at any time. However, to perform any fatigue analysis, the analysis steps must have been performed up through the translation stage. With the exception of the basic utilities in FEFAT, the existence of a `jobname.fes` file is a minimum requirement.

For operation of this MSC Fatigue module, see Total Life and Crack Initiation (Ch. 6).

Analysis Manager

The Analysis Manager is a client-server productivity tool for launching and managing analysis jobs on local and/or remote hosts. As a separately installed and licensed program, it is not part of MSC Fatigue. The Analysis Manager (AM) functionality will typically be accessed from the fatigue "Job Control" form. These functionalities include:

- Remote or local job submittal
- Automatic copying of files across network and back, even if job fails
- Interactive job status, elapsed and cpu time, % of cpu usage, % and total disk space usage
- Specification of maximum disk usage
- Customized submittal times
- Easy abortion of job
- Queue management to determine best location to run job
- Submission as another user if configured properly
- Mail notification on job completion

Note:	Patran and MSC Fatigue must be installed, licensed, and properly configured on both the client AND server machines.

Use of Analysis Manager with MSC Fatigue

The Analysis Manager (AM) can be used to submit, monitor, or abort MSC Fatigue jobs. This section describes how to perform these functions. Note that in order to access these functions, the Analysis Manager must be installed, licensed, and enabled. Please see the Analysis Manager documentation for a detailed description of the operation instructions.

Configuration of Analysis Manager for MSC Fatigue

For the Analysis Manager to be able to submit, monitor, and manage MSC Fatigue jobs, it must first be configured properly. A couple of settings are also required to the user's local machine. Please see the Analysis Manager documentation for more detailed installation and configuration instructions.

Setup on the Local User's Machine (Client):

Since the user's local machine is a client of the Analysis Manager, a couple of settings are used to properly configure the client. These are as follows:

- AM_HOME environment variable
- .p3mgrrc [optional-created after modifying and saving AM configuration through the AM GUI]

The AM_HOME environment variable is used to specify the location where Analysis Manager has been installed on the local machine. Note that the installation on the client machine should be exactly the same as on the server machine(s). AM_HOME can be set as in the following examples:

- setenv AM_HOME /msc/patran2004/p3manager_files (cshell)

- AM_HOME=/msc/patran2004/p3manager_files ; export AM_HOME (Bourne shell, kshell)

The .p3mgrrc is a configuration file that is saved any time the user changes and saves the AM configuration through the main AM GUI. This file is normally stored in the user's home directory. The .p3mgrrc is a text file, and may be viewed and edited by the user if needed. However, this is not normally required and should be done only after saving the original file.

Setup on the Analysis and Queue Manager Machines (Server):

Several files are used in the configuration of Analysis Manager on the server machine. These files are located in the AM installation directory, hereafter denoted as AM_HOME.

- $AM_HOME/default/conf/host.cfg
- $AM_HOME/default/conf/disk.cfg
- $AM_HOME/org.cfg [changes only needed for special situations]
- $P3_HOME/P3_TRANS.INI [Windows only-changes only needed for special situations]

Note:	$P3_HOME is the location on the server machine where Patran and MSC Fatigue_for_Patran has been installed.

The host configuration file (host.cfg) is normally set up by the system administrator. In order for changes in this file to take effect, the QueMgr and RmtMgr daemons (Linux) or services (Windows) must be re-started. (Please see the Analysis Manager documentation for more detailed instructions.)

Below is an example of the host.cfg file. Required parameters are denoted with bold letters. Note that a more extensively commented host.cfg file is given in the file "$AM_HOME/default/conf/host.cfg.sample".

```
#-----------------------------------------------------
# AnalysisManager host.cfg file
#-----------------------------------------------------
#
ADMIN: myname
#
QUE_TYPE: MSC
#
#-------------------------------------------------------------------
# A/M HOSTS Section
#-------------------------------------------------------------------
#
#A/M HostPhysical Host Type      EXE_Path    RC_Path
#-------------------------------------------------------------------
AM_HOSTS:
FAT_2005 mach.mycomp.com 20  /msc/patran2004/bin/FatigueExecute  NONE
#
#Physical HostClass    Max
#-------------------------------------------------------------------
PHYSICAL_HOSTS:
mach.mycomp.com SUNS          100
```

```
#
#TypeProg_name      App_name    MaxAppTask [ option args ]
#-------------------------------------------------------------------
APPLICATIONS:
20  FatMgr            MSC Fatigue  1000  -j  $JOBNAME  -h  $P3AMHOST  -d  $P3AMDIR
#-------------------------------------------------------------------
```

The following is a short description of the contents of the example host.cfg file shown above. Note that the following descriptions apply only to configuration for MSC Fatigue.

AM_HOSTS:

- A/M Host - a name that is used on the AM GUI's for a particular combination of analysis type and machine (any string)
- Physical Host - fully qualified name of the analysis host
- Type - AM application type (must use integer "20" for MSC Fatigue)
- EXE_Path - full path to the MSC Fatigue shell program FatigueExecute
- RC_Path - rc file name (not currently used)

PHYSICAL_HOSTS:

- Physical Host - fully qualified name of the analysis host
- Class - denotes machine type (legal names are: HP700, SUNS, RS6K, SGI5, WINNT, LX86)
- Max - Maximum allowable concurrent tasks for this host

APPLICATIONS:

- Type - AM application type (must use integer "20" for MSC Fatigue)
- Prog_name - application manager name (must use "FatMgr")
- App_name - application name (must use "MSC Fatigue")
- MaxAppTask - maximum number of this type of application allowed simultaneously

[option args] - arguments used by the application manager to start the fatigue analysis (muse use the exact string "-j $JOBNAME -h $P3AMHOST -d $P3AMDIR")

To submit an analysis job:

- From the "Job Control" form, click "Full Analysis, or "Partial Analysis." This will seamlessly start the job submittal process, and the Analysis Manager "Monitor" GUI will appear. From this form, the progress of the analysis can be monitored; if needed, the job can also be aborted from this form.
- From the "Job Control" form, click "Analysis Manager." This will cause the full AM GUI to appear. However this is only applicable for submitting an existing analysis job, i.e., one previously submitted as outlined above.

To monitor a job:

- From the "Job Control" form, click "Monitor Job." This cause the AM "Monitor" GUI to appear.

- As described above, the "Monitor" GUI will also appear automatically after a job has been submitted.

To abort a job:

- From the "Job Control" form, click "Abort Job." This cause the AM "Monitor" GUI to appear, and the job will be aborted. From this form, the disposition of the job can be seen as well as the output from the job.
- As described above, the "Monitor" GUI will also appear automatically after a job has been submitted; the job can be aborted from this form.

To disable Analysis Manager:

If the Analysis Manager is to be disabled for MSC Fatigue analysis, an environment variable can be set as follows. This will not affect Analysis Manager for other analysis types.

setenv FAT_DISABLE_AM 1 (cshell)

FAT_DISABLE_AM=1 ; export FAT_DISABLE_AM (Bourne shell, kshell)

For Windows, the environment variable can be set from the "SystemProperties/Advanced/Environment Variables/User Variables" form. For a "New" variable, set "Variable Name" = FAT_DISABLE_AM and "Variable Value" = 1.

Output from Analysis Manager:

Four files of special interest are created during a Fatigue job submitted using AM:

- jobname.stderr - contains text directed to standard error by programs running during the Fatigue analysis job on the analysis host
- jobname.stdout - contains text directed to standard output by programs running during the Fatigue analysis job on the analysis host
- jobname.mon - contains monitoring information from a job
- jobname.tml - this is the AM log file

Actions Performed During Job Submission:

When AM is used to submit an MSC Fatigue analysis job, the following actions are performed:

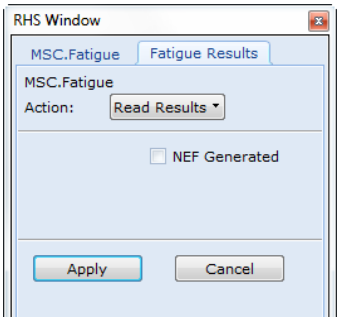
- The amf (.amf) file, which contains a list of files to be copied to the analysis host, is created
- A copy of the input (fin) file is created (jobname_bak.fin). The path information is stripped from the fin file.
- The solver input file (.fes) file, which contains job parameters, entity ids, stress results, etc., is created
- The solver input (.fes) file is converted to a neutral ASCII format (.asc) file
- All the above actions are performed on the local machine
- The Analysis Manager is started, and the Queue Manager is signaled; the Queue Manager process resides on the Queue Manager host

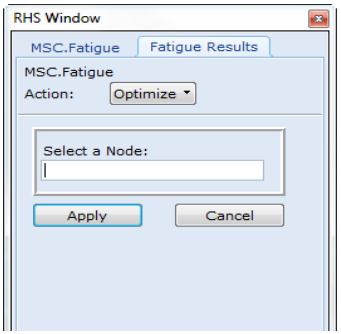
- The Queue Manager signals the appropriate Remote Manager process and copies the job file to the appropriate machine. The MSC Fatigue analysis process is started (FatMgr); this occurs on the analysis host
- The FatMgr process copies all needed files into temporary directory on the analysis host
- The pre-analysis process is started through a shell program (FatigueExecute); the neutral format analysis input file (.asc) and other binary files may be converted to a binary format compatible with the analysis host
- The analysis process is started through a shell program (FatigueSubmit); the fatigue solver is run
- Unessential files are removed from the temporary dir; the results files and other required files are copied back to the local machine, and the temporary directory is cleaned up
- The Analysis Manager signals the user that the analysis job is complete

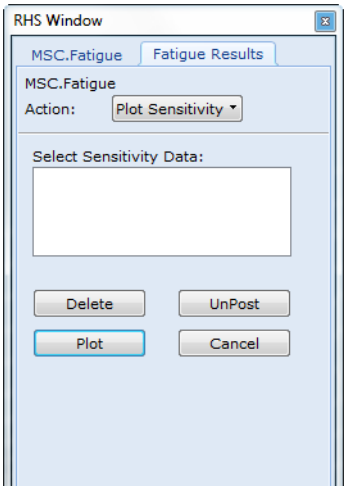
Note:	MSC Fatigue jobname may be used incorrectly by Analysis Manager. The jobname given when "Full Analysis" is clicked from the MSC Fatigue "Job Control" form may be over-ridden by the .p3mgrrc file contents. This generally occurs when the configuration settings are modified and saved from the AM main GUI. To overcome this problem, edit the .p3mgrrc file (normally located in the user home directory) and remove the line beginning with: "gen_submit[MSC Fatigue].gen_input_deck"

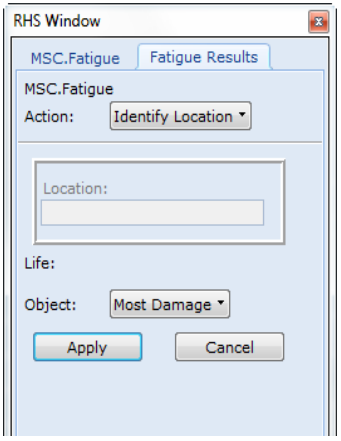
Postprocessing Results

By selecting the *Results* button, located on the main MSC Fatigue setup form, a Results form will appear. Each of the actions on the Results form is explained in detail in the following sections.









Read Results

With the Action set to *Read Results* on the Results form, the MSC Fatigue analysis results can be imported into the database. If the same database being used contains finite element results, then the database will now have fatigue results and finite element results. Each fatigue job is known as a Results Load Case under the Results application. The only things that differentiate them from a FEM load case are their results titles. This section will explain how to effectively postprocess fatigue results including:

- Displaying Fringe plots, writing reports, and X-Y plots (see Fringe Plots, Text Reports, XY Plots, 102).
- Setting color Spectrums (see Using Spectrums and Ranges, 105). When results are imported, a new spectrum is created for you called `fatigue_spectrum`. This spectrum is simply the `standard_spectrum` color scheme in reverse such that short fatigue lives can be displayed in red (hot) and long fatigue lives in blue (cold). This is opposite the way stress results are displayed with the `standard_spectrum`. It is your responsibility to set the spectrum to the one that you want.

- Displaying result Values. When results are imported a new range is also created for you in log increments called log_range. Some values such as Damage are Life are hard to visualize with a standard range of evenly spaced ranges. A log range helps spread the color bands out on results that may range from a small number to a very large (infinite life) number. Again you are responsible for selecting the range that best fits the displayed results.

See the Patran User's Manual for more detail on how to use the Results application and how to set up spectrums and ranges. Only the graphical display of fatigue results is explained in this section.

The Read Results option is not available for the Crack Growth analysis type.

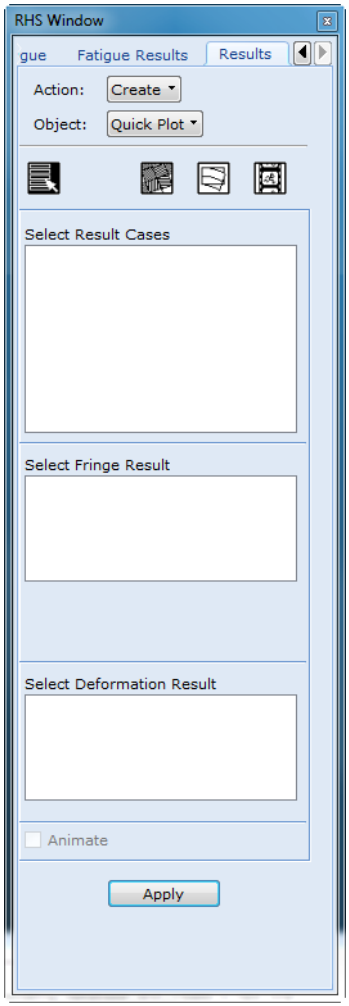
When a Fast Analysis has been executed and the results read back in, a group is created called `jobname_critloc` which can be used later in subsequent fatigue analyses. This is made possible by the existence of a `jobname.ent` file created during a Fast Analysis. This file contains a list of the nodes or elements of the most damaged locations.

Some things to note when importing results:

- No file name needs to be selected. Supplying a name is optional. If you do not select a file name, then the file that is imported is the `jobname.fef` file (and/or `jobname.fos` if present) if the *Solver* is set to **Classic** on the Main Fatigue Form. Otherwise it is the `jobname.fer` file. When the *Solver* is set to **DTLib**, either `jobname.fef` or `jobname.fer` files are valid. Which is imported is determined by the *MSC Fatigue Preference* setting called **Read Results** under *Tools* | *MSC Fatigue* | *Preferences.*
- If a file name is selected it will be imported regardless of the jobname set. You may select either `*.fef` or `*.fer` files (you may have to change the file filter on the file browser if the extension is not what you want).
- When a `*.fef` file is selected, the presence of a corresponding `*.tml` file will dictate the result quantities labels created for the results. If the `jobname.tml` file is not present, it will automatically create the result quantity labels.
- There is a toggle call *NEF Generated* that is available when selecting `*.fef` files. NEF stands for: *Nastran Embedded Fatigue*. `*.fef` files generated by NEF that are element nodal in nature should be imported with this toggle ON. `*.fef` files that are element centroidal based should not use this toggle, even if generated by NEF. MSC Fatigue only produces pure nodal or pure element centroidal result files (one set of results per entity, node or element). NEF produces `*.fef` files that have result at each node per element. If you import one of these files with the toggle OFF, then all the results at the nodes are mapped to the element centroid when they should be mapped to nodal locations. NEF also produces element centroidal `*.fef` files. You must know which is being imported when accessing results from NEF produce `*.fef` files in this manner.

Fringe Plots, Text Reports, XY Plots

To post process fatigue results bring up the Results form as shown below. This is done by selecting the Results toggle switch from the main menu bar in Patran.



Explanation of results that can be processed.

Parameter	Description
Damage	Damage is the reciprocal of the life.
Log of Damage	This is the Damage in log units. Each number should be interpreted as 10 raised to that power. Log results generally give better looking fringe plots in that the log units tend to linearize the results and spread them apart making it a bit easier to interpret.
Life in Repeats	Life is the reciprocal of damage. The event units are reported in the number of repeats of the time history or service loading.
Log of Life in Repeats	This is the Life in log units. Each number should be interpreted as 10 raised to that power. The event units are reported in the number of repeats of the time history or service loading. Log results generally give better looking fringe plots in that the log units tend to linearize the results and spread them apart making it a bit easier to interpret them.
Life in User Events	Life is the reciprocal of damage. The event units are reported as the user has defined them when creating the time history in PTIME.
Log of Life in User Events	This is the life in log units. Each number should be interpreted as 10 raised to that power. The event units are reported as the user has defined them when creating the time history in PTIME. Log results generally give better looking fringe plots in that the log units tend to linearize the results and spread them apart making it a bit easier to interpret them.
Safety Factor	This is the factor of safety calculated by the Factor of Safety analysis. This results will not appear if a Factor of Safety analysis has not been requested. Values of unity indicate that the design life is exactly meet. Numbers less than one should be investigated immediately.
Biaxiality Ratio Mean	This parameter is the average biaxiality ratio for every time step in the combined loading history. Biaxiality ratio is defined as the ratio of the minimum and maximum principal stresses at a location on the surface of a component. It can takes values between +/- 1 inclusive.
Biaxiality Ratio Standard Deviation	This parameter indicates the variation, or scatter in the biaxiality ratio. For low standard deviation values, uniaxial fatigue analysis will give reasonable answers. For large values, a multi-axial fatigue solution should be considered.
Most Popular Angle	This indicates the likely direction of crack initiation and growth. The angle is defined relative to the local coordinate set of the FE stresses.
Angle Spread	This is the difference between the largest maximum and smallest minimum angle encountered during the processing of the local time histories. Angle spreads less than 30 degrees indicate mainly uniaxial situations.
Maximum Principal Stress Range	This is the stress range of the largest and most damaging cycle encountered in processing the local time histories.

Parameter	Description
Gate in Stress Units	This is not a result that should be contoured. It is the stress level below which biaxiality is ignored regardless of what the principal angles are doing.
Location of Failure	For spot weld analysis this is the location of the failure - either at sheet one (1), spot weld center (2) or at sheet two (3).
Angle of Failure	This is the angle at which the spot weld failed.
Node ID of Failure	For spot weld analysis this is the node number at which the failure occurred.
Maximum Force Encountered	For spot weld analysis this is the maximum force encountered during the analysis at the particular failure location.

Using Spectrums and Ranges

In order to create a meaningful fringe plot of the fatigue results, it may be necessary to use a different color *Spectrum* and corresponding number *Range*. These can be accessed from either the Fringe Attributes or the Display Attributes icon on the *Results* form or from under the *Display* pull-down menu on the main form of the Patran window.

The Spectrum form is used for modifying, creating, and selecting color spectrums. The Range form is used for modifying, creating, and selecting results value ranges.

Two standard spectrums exist called `standard_spectrum` and `sequential_spectrum`. When fatigue results are read in an additional spectrum called `fatigue_spectrum` is created. When you make a fringe plot of fatigue results the `fatigue_spectrum` will be used.

The `fatigue_spectrum` contains the exact same number of colors as the `standard_spectrum` but in the reverse order. This is because short life results are associated with red as is high stress. Short life is a minimum number, whereas high-stress is a maximum number. Therefore, the spectrum is reversed as opposed to a spectrum for a stress contour plot.

For damage results (as opposed to life results), it will be desirable to change the spectrum back to the `standard_spectrum` to cause high damage to be associated with red.

Important:	It may be desirable with fatigue results to set a new spectrum with only as many colors as there are decades of values. Then set up some custom ranges with each color representing a decade, i.e, the lowest number to the first decade (10, 100, 1000,...), the number of decades in-between, and the highest decade to the highest value. This is left up to the user however. A new range called `log_range` is created for the user on import of results which can be accessed from the Ranges form For an explanation of the rest of this form and how to create your own spectrums and ranges, see Spectrums, 335 and Ranges (p. 335) in the *Patran Reference Manual*.

List Results

If the Action is set to *List Results* on the Results form, the following applies when the **Apply** button is clicked. If the analysis type is set to anything other than crack growth, the MSC Fatigue module PFPOST will be spawned. If the analysis type is crack growth, the MSC Fatigue module PCPOST will be spawned. A discussion of PFPOST can be found in Reviewing Results (PFPOST), 363. PCPOST is discussed in Reviewing Results (PCPOST), 571.

These two modules allow easy access to important tabular and graphical results such as the most damaged nodes or elements, biaxiality analysis parameters, factor of safety calculations, final growth curves, and various results file output utilities.

Re-Analyze

If the Action is set to *Re-Analyze* on the Results form, the following applies when the Apply button is invoked.

- When performing a crack initiation, a total life or crack growth analysis, the option executes the external MSC Fatigue module, FEFAT. This program may also be started from the operating system prompt by typing the symbol **fefat**. The program operates interactively in Motif or Mask mode, and selections are made according to the rules defined in Module Operations (Ch. 18).
- When FEFAT is invoked in this manner (as a reanalysis), it is assumed that a `jobname.fes` file exists and the user is placed directly into the preprocession option of the FEFAT program from which he has full interactive control over all job parameters at this stage of the analysis. If the `jobname.fes` file cannot be found, an error message will eventually result. From FEFAT, the user has the ability to change directories to find a appropriate input file.

The detailed operations of FEFAT in the reanalyze or preprocessing mode are described in Total Life and Crack Initiation (Ch. 6).

Design Optimization

If the Action is set to *Optimize* on the Results form, the following applies when the Apply button is invoked.

- When performing a crack initiation or total life analysis, the option executes the external MSC Fatigue module, FEFAT and places the user directly in the Design Optimization option. This program may also be started from the operating system prompt by typing the symbol **fefat**. The program operates interactively in Motif or Mask mode, and selections are made according to the rules defined in Module Operations (Ch. 18).
- When performing a crack growth analysis, the option executes the external MSC Fatigue module, PCRACK. This program may also be started from the operating system prompt by typing the symbol **pcrack**. The program operates interactively in motif or mask mode, and selections are made according to the rules defined in Module Operations (Ch. 18).

In all cases, these two programs will appear on the screen when invoked. However, if the appropriate results do not exist, they will not perform as desired. The detailed operations of FEFAT and PCRACK are described in Total Life and Crack Initiation (Ch. 6) and Crack Growth (Ch. 8).

Factor of Safety

If the Action is set to *Factor of Safety* on the Results form, the following applies when the **Apply** button is clicked.

- When performing a crack initiation or total life material analysis, the program executes the external MSC Fatigue module, FEFAT. This program may also be started from the operating system prompt by typing the symbol **fefat**. The program operates interactively in Motif or Mask mode, and selections are made according to the rules defined in Module Operations (Ch. 18). You are placed directly into the factor of safety option of FEFAT.
- When performing a total life component or crack growth analysis the program this option is dimmed and not selectable.

In all cases, this program will appear on the screen when invoked. However, if the appropriate results do not exist it will not perform as desired. The detailed operations of FEFAT's factor of safety option are described in Factor of Safety Analysis (Ch. 6).

Sensitivity Plots

If the Action is set to *Plot Sensitivity*, the Results form updates itself to show a listbox of available XY data from sensitivity analyses.

When one of these XY data files is selected and the Plot button is pressed, the XY Plot application will produce the plot graphically. The plots that are available are Stress Concentration vs. Life, Scale Factor vs. Life, Design Criterion vs. Life, and Residual Stress vs. Life. These files are called `jobnamenn.kfl`, `jobnamenn.fal`, `jobnamenn.dcl`, and `jobnamenn.rfl` respectively, where the `nn` represents the node or element number at which the sensitivity calculation applies. This data is created in the MSC Fatigue module FEFAT and must first be plotted directly from FEFAT before it can be plotted within the XY Plot application. See Design Optimization (Ch. 6) for a detailed description on how to create these plots.

Each time one of these plots is created from FEFAT, a `jobnamenn*.tem` file is created and is used to extract the plot attributes (such as the title, plot type, axis titles, etc.). Again the `nn` refers to the specific node or element of interest and the `*` refers to either `kfl`, `fal`, `dcl`, or `rfl`. This template file is actually a command file and can be read in as a session file from the File pulldown menu. This is transparent, however, when working from this form.

Use the **Delete**, **UnPost**, and **Plot** buttons to delete, unpost or plot the xy data, respectively. The difference between Delete and UnPost is that Delete removes all curve information and the plot viewport from the database, whereas UnPost simply hides the viewport. The information is retained in the database and can be recovered under the XY Plot application on the main menu.

Extract Time History

If the Action is set to *Extract Time History* on the Results form, the following applies when the **Apply** button is invoked.

When performing any analysis, the option executes the external MSC Fatigue module, FEFAT and places the user directly in the Extract Time History option. This program may also be started from the operating system

prompt by typing the symbol **fefat**. The program operates interactively in Motif or Mask mode, and selections are made according to the rules defined in Module Operations (Ch. 18).

The detailed operations of FEFAT and how to extract the combined stress or strain time histories are described in Total Life and Crack Initiation (Ch. 6).

Extract PSD

If the action is set to *Extract PSD* on the Results form, the following applies when the Apply button is invoked.

This option executes the external MSC Fatigue module, FEVIB and places the user directly in the Extract PSD option. This program may also be started from the operating system prompt by typing the symbol **fevib**. The program operates interactively in Motif or Mask mode, and selections are made according to the rules defined in Module Operations (Ch. 18).

The detailed operations of FEVIB and how to extract PSDs are described in Vibration Fatigue (Ch. 9).

Identify Location

If the Action is set to *Identify Location* on the Results form, the following is applicable when the Apply button is invoked.

This option is available for crack initiation and total life analyses. It provides a quick way to easily identify the critical location graphically on the model. The user has the choice of identifying the location with:

1. The Most Damage - reported in life (repeats to failure)
2. Worst Safety Factor
3. Most Biaxiality
4. Least Proportional Loading
5. Largest Mobility

See Read Results, 101 for an explanation of each of these quantities.

When the Apply button is pressed, an arrow pointing to the location and a circle will be drawn to indicate the node of interest. If the location is an element it will be highlighted. The corresponding value will be place on the graphics screen and also in the databox on the form.

If no results exist for the particular quantity, a message to this effect will be displayed. Keep in mind that the results must be imported into the database and the jobname corresponding to the results must be in the Jobname databox on the main form. Also, to find the correct results, the program keys off of the Analysis Type. So be sure that the Analysis Type is set correctly on the main form, else it will look for the wrong analysis type results. If the same results have been imported more than once, only the first will be detected. It is a good idea to delete previous results datasets from the database before importing new results if this feature is to be used, that is to say, results of the same type with the same jobname.

Other Modes of Job Setup

MSC Fatigue can be run in three different forms. The preferable manner is to run directly from within Patran where access to all the features of MSC Fatigue are available. However, another non-graphical mode of running MSC Fatigue is available.

MSC Fatigue requires three basic inputs for a successful analysis.

1. Cyclic material properties. MSC Fatigue provides a comprehensive materials database. This database is expandable allowing the user's own defined properties. A database manager is provided for viewing, editing, adding and manipulating materials data. See Material Management (Ch. 4) for detailed instructions.
2. Load information. MSC Fatigue also provides a time history database manager for managing, manipulating, adding and viewing time histories. It is necessary to define the load variation with time when using static stress results from finite element analyses. See Loading Management (Ch. 5) for detailed instruction. When using transient stress results a separate time history definition is not necessary.
3. Stress or strain results. This information can be provided by translating external results files from finite element analyses directly within MSC Fatigue or by simply supplying the stress component at a single location manually. This last method is described in Utilities, 360. The direct method is described here in this section.

To set up an MSC Fatigue job in this non-graphical mode, follow these basic steps assuming that materials data are available and loading time histories are also available if necessary:

1. Run the MSC Fatigue module FEFAT to produce a simple fatigue input file with stresses at a single location (see Utilities, 360), or run the MSC Fatigue module FEFTRN to translate finite element results into a MSC Fatigue input file. To run any of these modules at the system prompt simply type **fefat** or **feftrn** respectively. By either one of these methods you will end up with a MSC Fatigue input file called `jobname.fes`. FEFTRN operation is explained below.
2. Next run the MSC Fatigue module FEFAT to preprocess and run the actual fatigue analysis. The operation of this module is explained thoroughly in Total Life and Crack Initiation (Ch. 6). For Crack Growth analyses, do the same thing but with the crack growth analyzer called PCRACK which is described in detail in Crack Growth (Ch. 8).
3. For Total Life and Crack Initiation analyses, a results file will be produced which can be postprocessed by commercial postprocessing systems.

FEFTRN

This section documents the operation of FEFTRN, an MSC Fatigue module used to translate results from either an MSC Nastran results database (the `.xdb` file) or from a Universal file. To invoke the program simply type **feftrn** at the system prompt.

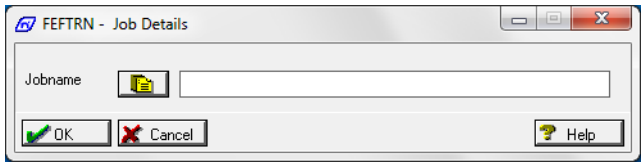
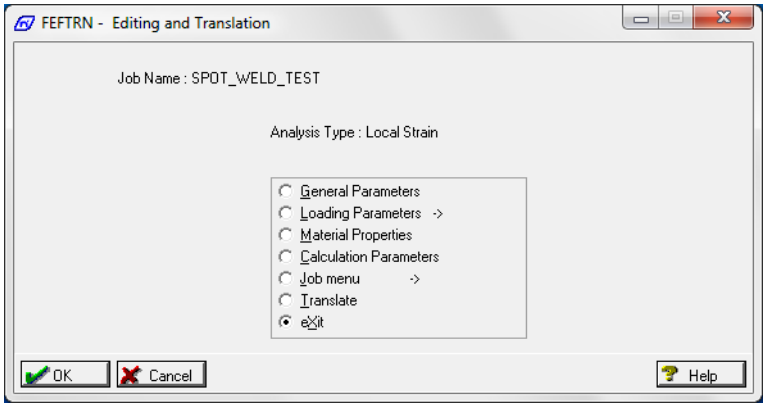


Figure 3-3 FEFTRN Utility Form

You will be presented with the above forms which first asks you for a jobname with a .fin file extension. You may select an existing one or create a new one. The `.fin` extension is not necessary to specify and will be added if not specified. The `jobname.fin` file is an ASCII fatigue job parameter file which is fully described in The Job Information File (jobname.fin), 369.

If an existing job file is selected the following main menu is displayed:



If a new job is specified, then you are led through a number of pages that ask you to set various parameters. Only one of these parameters is described in this section: FE Results Source. The other parameters are fully described in various sections in this chapter. General and Calculation Parameters are described in General Setup Parameters, 47, Loading Parameters in Loading Information Form, 70, and Material Properties in Materials Information Form, 61. Not all parameters are valid at all times. If a parameter is dimmed then it is not valid or necessary for the configuration that you currently have set up.

Once all parameters have been set properly you may press the Translate toggle to create a `jobname.fes` file for input to an MSC Fatigue analysis using FEFAT.

The only parameter that is not described elsewhere is the FE Results Source. You may read results from two sources: an `.xdb` file created by MSC Nastran with the PARAM, POST, 0 parameter in the input file, and from a SDRC Universal file. Simply set the FE Results Source to either MSC Nastran or SDRC Universal in the General Setup and then under General Loading specify the file name.

The MSC Nastran reader supports stress results at grid points (requested with a GPSTRESS case control card) and element centroidal stresses (requested with a STRESS case control card) for shell and solid elements for both static and transient analyses.

The Universal file reader only supports static analyses for stress and strain results. The Universal file must have stresses or strains on either nodes or elements only. Node at element is not supported. Supported datasets are 55, 56, and 2414. Also note that the Universal file format supported in from the older version of SDRC's IDEAS, not the new MasterSeries version. This type can only be imported using MSC Fatigue.

To create fatigue results in Universal file format as opposed to the standard Patran results file format, set the keyword FEFTYPE to the value UNIVERSAL using the MSC Fatigue environment module NENM See Modifying the MSC Fatigue Environment (MENM), 1463.

In both MSC Nastran and Universal file translations, all elements found for the defined load cases will be processed using the same group attributes, e.g. material, surface finish, Kt, etc. Multiple materials, surface finish and such are not supported.

4 Material Management

- Introduction to dtmat - The New Material Database Manager 114
- Supported Material Types 114
- Material Information Updates 116
- The dtmat User Interface 117
- Adding Material Data Sets 121
- Adding Multi-Curve Material Data Sets 128
- Introduction to PFMAT 137
- PFMAT Menu Options 139
- Component vs. Material S-N Curves 169
- Rules for Changing Young's Modulus 171
- PFMAT in BATCH Mode 173
- PFMAT Material Listing 177
- Grey Cast Iron 190
- Materials with Temperature Dependent Properties 192
- Materials with Multiple Mean S-N Curves 195

Introduction to dtmat - The New Material Database Manager

Dtmat (dtmat) is the new database manager that exploits the new shared library DTLib. By using this database manager, users of MSC Fatigue have access to all the necessary data in one central location, thus ensuring the highest-quality design procedures for durability analysis. Dtmat supports material databases stored in the **MXD** and **MDB** formats. It comes with a default materials database **iceflow_standard.mxd** that includes fatigue properties for many commonly used materials. The location of the central **MXD** database is $p3_home/mscdtfat_files/mats. Note that the default material database provided with the dtmat is different from previously used database **nmatsmas.mdb**. However, the Classic **MDB** database can also be viewed/edited using dtmat.

Dtmat provides you with the functionalities to:

- Open a specific named database.
- Load a material data set.
- Add, delete, or modify a material data set.
- Export or import a dataset.
- View material properties graph.
- Query available materials of a particular data set type, i.e. those which can be used with a particular type of analysis.

Supported Material Types

The newer architecture of the **MXD** material database allows for an expanding library of material models, which is not compatible with the Classic modules or with PFMAT. The following material data sets are supported in dtmat:

- Standard S-N material
- Standard E-N material
- nCode SN Mean Stress Curves S-N material
- nCode SNR Ratio Curves S-N material
- nCode SN Haigh Curves S-N material
- nCode EN Mean Stress Curves E-N material
- nCode EN RRatio Curves E-N material
- nCode SN Seam Weld material
- nCode Spot Weld material
- nCode Bastenaire material
- nCode LEFM Austen material
- nCode SN Temperature Curves
- nCode E-N Temperature Curves

Data Set Name	Application Type	Notes
nCodeSNMatData	Standard SN Data Set	
nCodeENMatData	Standard EN Data Set	
nCodeSNMeanStressCurve	S-N Analysis Engine	Three different multi-curve data set types used for non temperature SN analysis. The first stores curves of stress versus life for constant mean stress, the second stores curves of stress versus life for constant r-ratio, and the last stores stress versus mean stress for constant life (also known as Haigh diagram). These should have multiple child datasets of type: Constant mean stress curve data set, or Constant r-ratio curve data set, or Constant life curve data set.
nCodeSNRRatioCurve	S-N Analysis Engine	
nCodeSNHaighCurve	S-N Analysis Engine	
nCodeDangVanMatData	Dang Van Safety Factor Analysis	
nCodeSpotWeldMatData	Spot Weld Analysis	
nCodeSNSeamWeldMatData	Seam Weld Analysis	It should have 2 child datasets for flex and stiff conditions of set type name nCodeSNSeamWeldProps nCodeBastenaireMatData S-N analysis engine
nCodeENTemperatureCurve	EN High Temperature Analysis	It should have multiple child datasets at different temperatures of set type name nCodeENTemperatureCurve
nCodeSNTemperatureCurve	SN High Temperature Analysis	It should have multiple child datasets at different temperatures of set type name nCodeSNTemperatureCurve

Material Information Updates

A few entries are available in the Materials Information spreadsheet for the DTLib-based solver that may not be available for the Classic Modules.All available entries in the Material Information spreadsheet are described in the table below. Note that these items are not available for Seam Weld and Spot Weld.

Table 4-1 Material Information

Table Entry	Option	Description	
Material		Material Name	**No Default**
Finish		Surface finish and treatment can have a significant effect on fatigue behavior. Rough surface finishes, e.g., due to machining marks, will in general reduce the fatigue strength, whereas surface treatments are often applied to increase the fatigue strength.	**Polished**
	Polished		
	Ground		
	Machined		
	Poor Machined		
	Rolled		
	Cast		
	Enter K-Roughness	The value of the K-Roughness property is used directly	
	Enter Surface Roughness	The value of the Surface Roughness property is used to calculate the surface roughness factor	
Roughness	0 <= **Value**	Value of K-Roughness or Surface Roughness; applicable only for Finish options "Enter K-Roughness" and "Enter Surface Roughness"	**0.**
K-Treatment	0 <= **Value**	User surface treatment factor	**0.**
Region		Material Group to which material properties apply	**No Default**
Layer		Layer is generally associated with shell element surface (z1, z2). The z1/z2 nomenclature is the Nastran definition for bottom/top shell surface. The layers available are the only ones available in the model. If solids only, appears as un-layered. If **Composite** option is chosen, a model with multiple intermediate layers can have 1 – n layers.	**No Default**
Kf		User surface factor	**1.0**

Table 4-1 Material Information

Table Entry	Option	Description	
Shape Factor		This property is used by the EN analysis engine only, and then only when the **Seeger-Heuler** plastic limit load correction is invoked.	**No Default**
Weld			**N/A**
Multiplier		The scale factor to apply to this material	**1.0**
Offset		A stress offset for this material	**0.0**

The dtmat User Interface

Dtmat can be accessed either from the command line (`$p3_home/bin/dtmat`) or from the Materials form by clicking the "Material Database Manager" button; this presumes that the fatigue Solver type is set to "DTLib".

From the two types of databases supported, only MXD material database type is useful in the spot weld and seam weld analysises with DTlib.

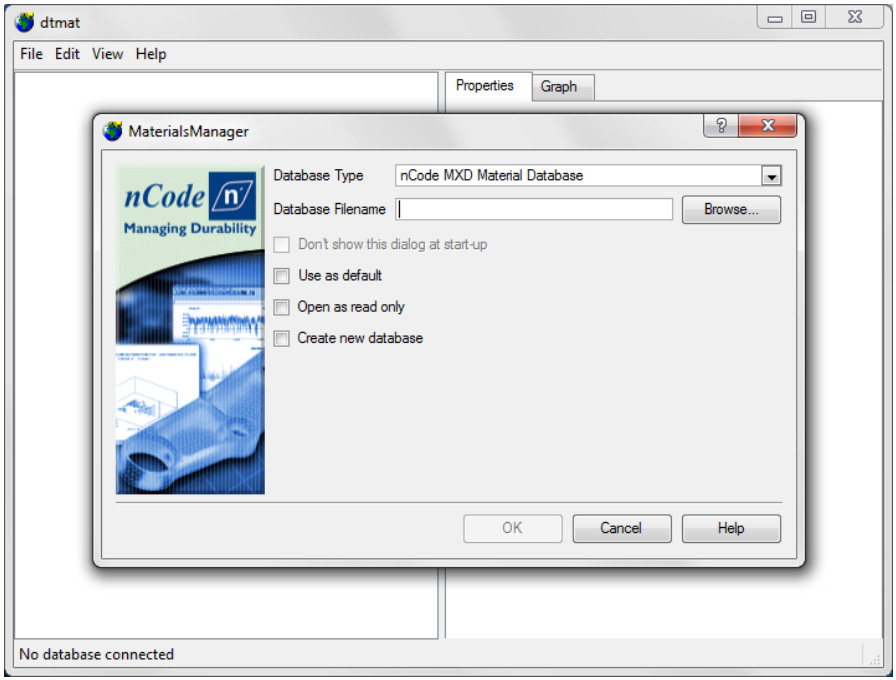


Figure 4-1 Dtmat Startup Form

On the dtmat start up form shown above, you can either select an existing database or create a new database. By default, the `$p3_home/mscdtfat_files/mats/iceflow_standard.mxd` is selected. The materials database in use, may be the central or a local database. If you are using a central database then it is advisable to copy it on local machine before modifying it, or the modification will affect all users of the central database.

After the database is selected, click OK to open the dtmat database browser form. Here you can create, edit, and plot materials data.

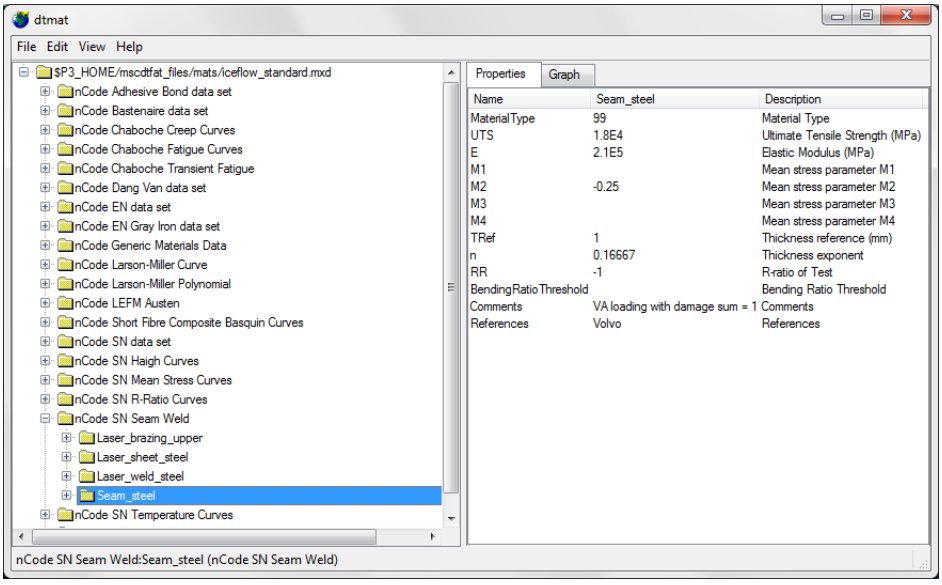


Figure 4-2 Dtmat Database Browser

This form can be divided into two sub sections:

- Left Navigation Pane or Tree view
- Materials Properties Window

Tree View or the Left Navigation Pane: It provides hierarchical tree view of the material data sets in the database. You can select database or data sets here, and then add, delete, and rename the data sets by using Edit menu. To select multiple data sets hold CTRL key and click on the data sets.

Materials Properties Window: This windows contains two tabs, Properties and Graph, and displays either the properties value or the properties graphs respectively.

Opening a Database

In this guide, the default database 'iceflow_standard.mxd' is used to explain the spot weld and seam weld problems.

To open an existing database:

1. Open the dtmat startup form.
2. Select the type of database from the options:
   - nCode Material database (this is the Classic mdb material database)
   - nCode MXD Material database

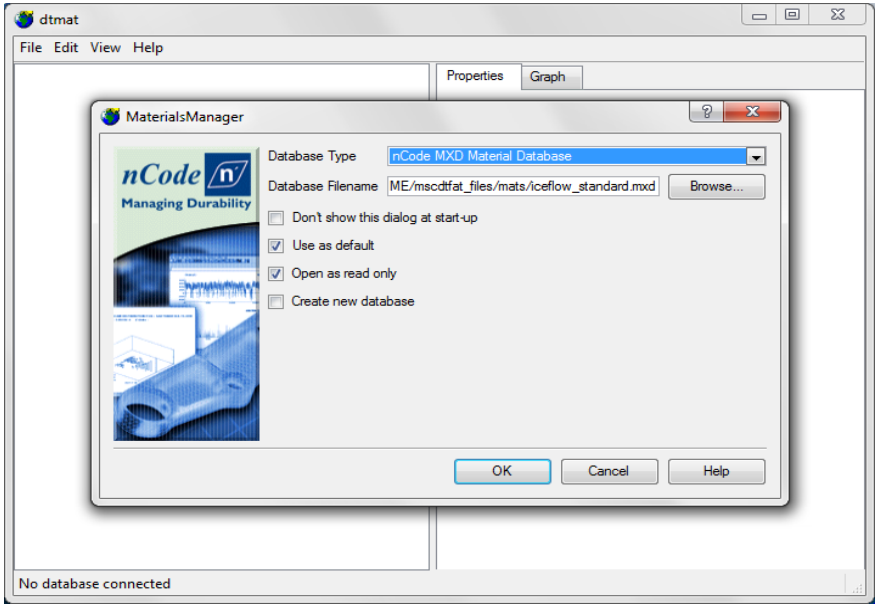
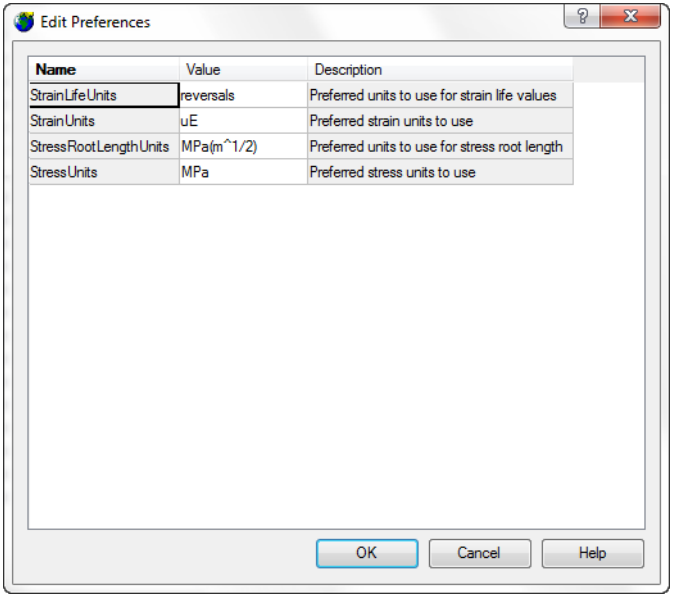


Figure 4-3 Database Selection

3. Browse or type the name of the database.
4. Select **Use as default** if you want to set the database as default database.
5. Select **Open as read only** if you do not want to make any modification in the database.
6. Click **OK**.

dtmat Units Preferences

If you prefer to view and enter stress and strain data in a set of units other that the default "MPa" units, a different set of units can be selected by clicking the "Edit" pull-down on the main form and selecting "Units Preferences". The following form appears.



The set of units that can be chosen are listed in the $p3_home/mscdtfat_files/nsys/unitconv.sys and unitspell.sys. You may also extend the unit types available by adding entries to the unitconv_custom.sys and unitspell_custom.sys files.

Creating a New Database

Provide the type and name and of the database to create a new database, and then add material data sets in it. Detailed instructions for creating a database are given below.

To create a new database:

1. Open the dtmat startup form.
2. Select the type of database from the options:
   - nCode Material database
   - nCode MXD Material database
3. Type a name for the database to be created.

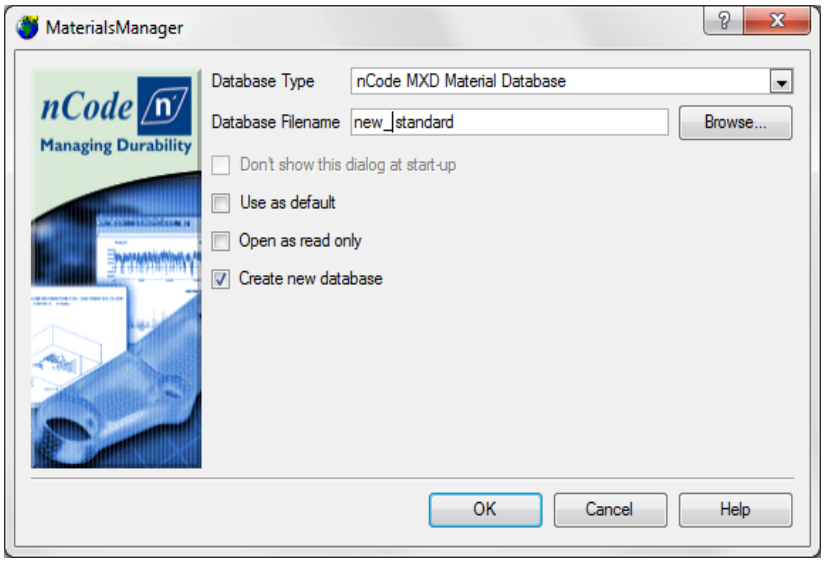


Figure 4-4 Create New Database

4. Select **Create new database**.
5. Click **OK.**

   The new database will be created.

Adding Material Data Sets

You can add a material data set in a newly created or an existing database. First, open the database in dtmat database browser, and then add material data set following the instructions given below. Note that the new dataset will not be usable, unless all pertinent information is supplied.

To add a data set in the database:

1. Open the database.
2. Click **Add Data** from the Edit menu.

   OR

   In the tree view, right-click on the database name, and then select **Add Data**.

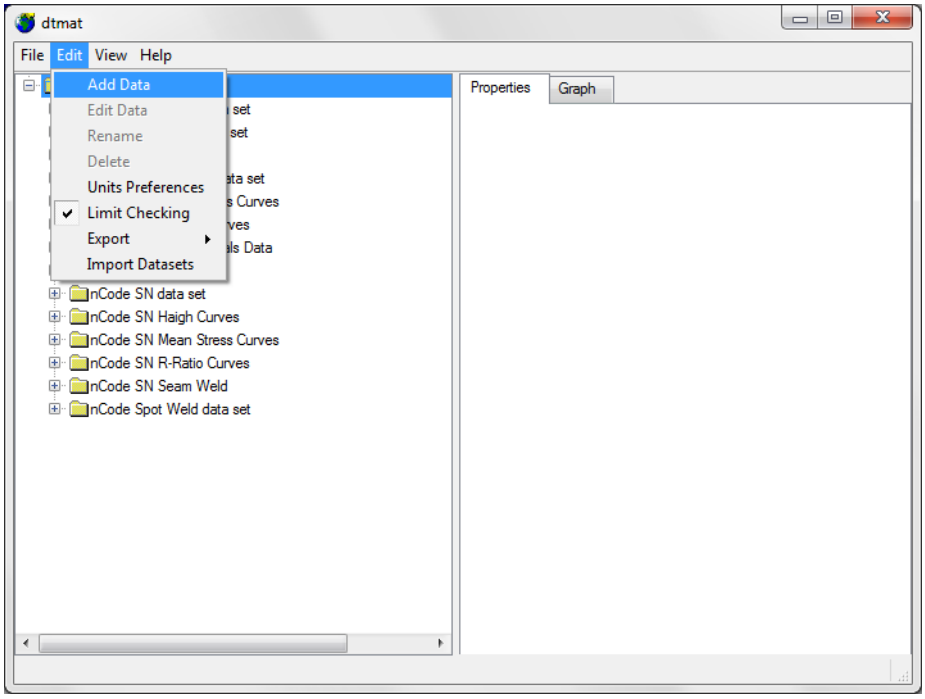


Figure 4-5 Edit Menu

3. Enter the type and name of the data set in the respective boxes on the 'Add data' form.

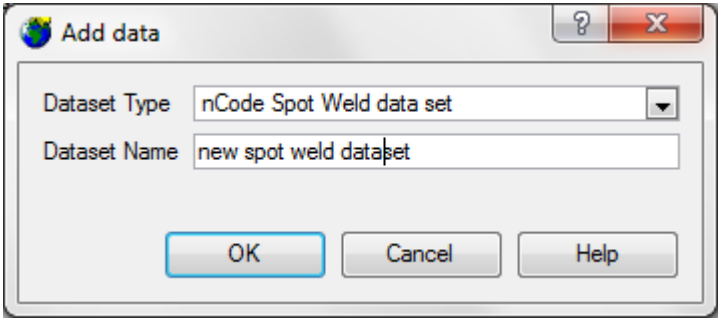


Figure 4-6 Add Data Form

4. Click **OK** to open the Editing form.

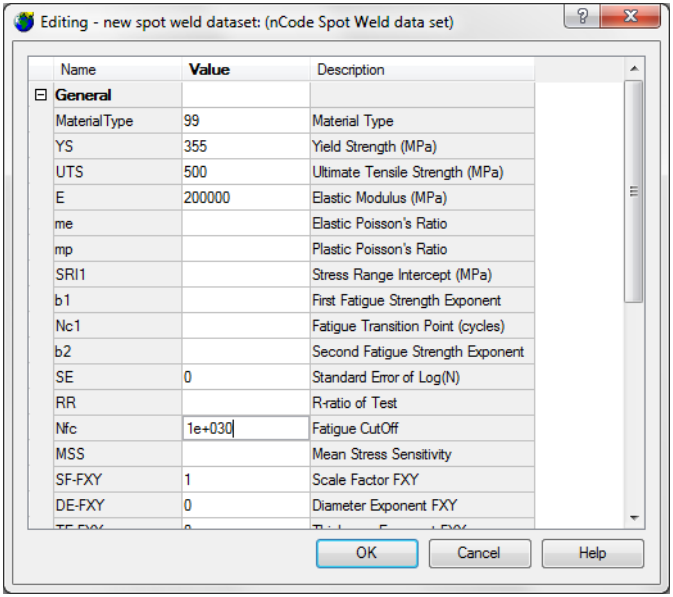


Figure 4-7 Edit Properties Form

5. Type the desired values of the various data set properties and click **OK** to close the form.

Working with Data Sets

You can edit data set properties, rename, and/or delete a data set by following simple instructions given below.

To edit a data set:

1. Open the **dtmat** database browser.
2. Select the data set to edit.
3. Click **Edit Data** from the Edit menu.

   OR

   In the tree view right-click on the data set to edit, and then select **Edit Data.**

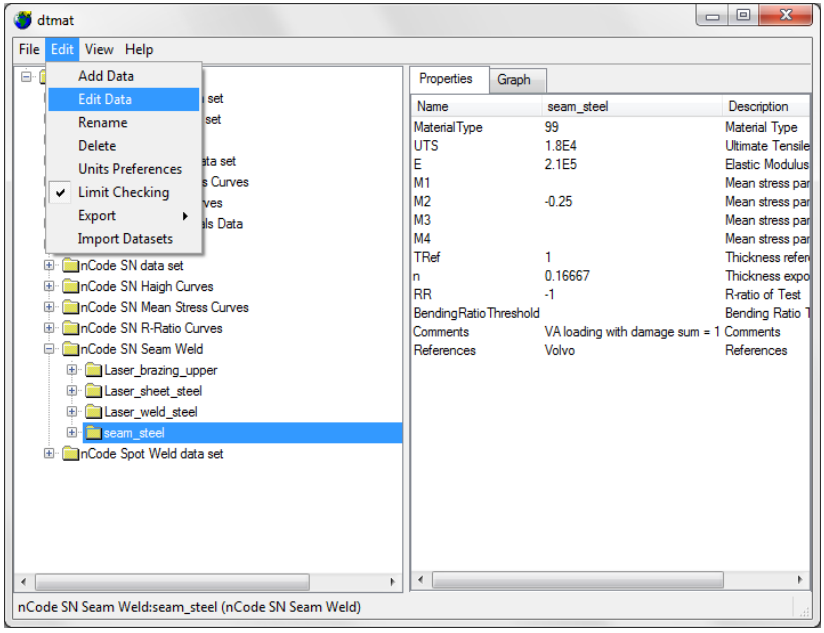


Figure 4-8 Edit Data Sets

4. Edit the properties of the selected data set on the Editing form, and click **OK** to save the changes.

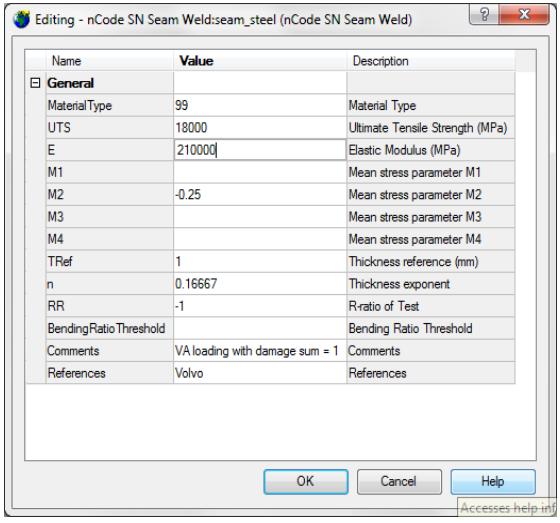


Figure 4-9 Edit Properties Form

To delete a data set:

1. Select the data set to delete, in the tree view.
2. Click **Delete** from the Edit menu.

   OR

   In the tree view right-click on the data set to edit, and then select **Delete**.

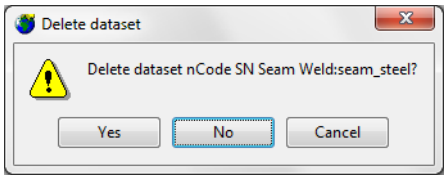


Figure 4-10 Delete Confirmation

3. A confirmation form will appear, click **Yes** to confirm the delete operation.

To rename a data set:

1. Select the data set to rename, in the tree view.
2. Click **Edit | Rename**.

   OR

   In the tree view right-click on the data set, and select **Rename**.

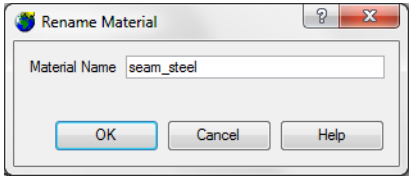


Figure 4-11 Rename Material Form

3. Type a valid name for the database and click **OK**.

Viewing Material Properties Graphs

The material properties graph displays a curve of the material properties, for the selected data set. There can be more than one type of curve for a data set, where each curve plots different properties.

To view the material properties graphs:

1. Open the dtmat database browser.

2. Select the desired datasets to view the material properties graph. You can select multiple data sets by holding the CTRL key.

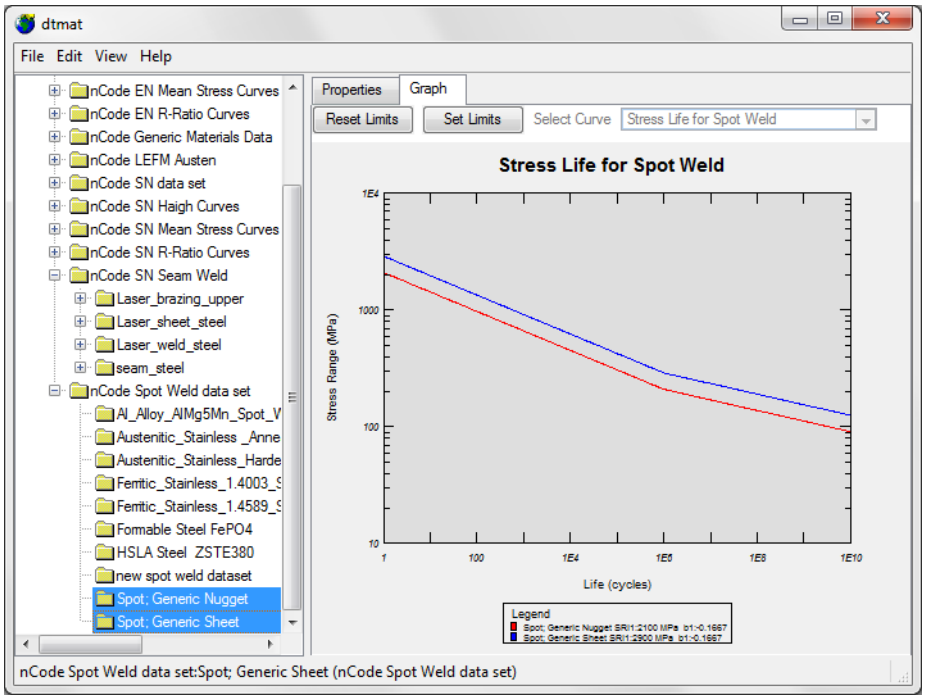


Figure 4-12 Properties Graph

3. Select the **Graph** tab in the material properties window. The material properties graphs will be displayed. There can be various types of graphs, depending on the material dataset selected.

Exporting / Importing Data Sets

Dtmat provides you with the functionality to export data sets and definitions, or import data sets in a database. Data sets are imported from or exported to, an XML file while data definition are saved in the XDEF file format. The exported data sets are stored in human readable XML format.

To export data sets:

1. Open a database in the dtmat database browser.
2. Select the data sets to export.
3. Click **Edit | Export** or right click, and select one from the following options:
   - **Export Selected Datasets,** if you want to export only selected data sets.
   - **Export Selected Datasets and Definitions**, if you want to export the data definition along with the data sets. For each data definition a separate XDEF file is created.

- **Export All Definitions**, if you want to export all data definitions that are in the current database. A separate file is created for each data definition in the selected folder.

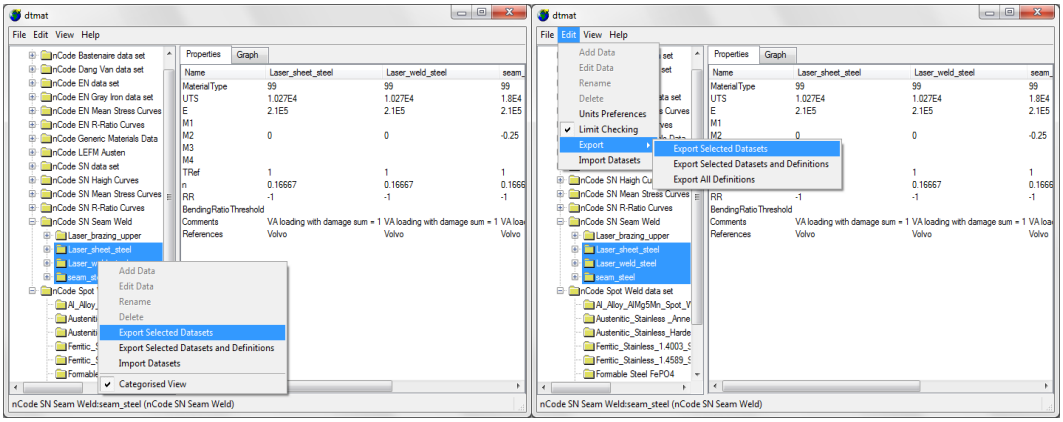


Figure 4-13 Export Data Sets

4. Provide the name and location of the file, in which you want to export the data sets, and click **Save**.

To Import data sets:

1. Open a database in the dtmat database browser.
2. Click **Edit | Import Datasets** or right click and select **Import Datasets** to open the Import form.

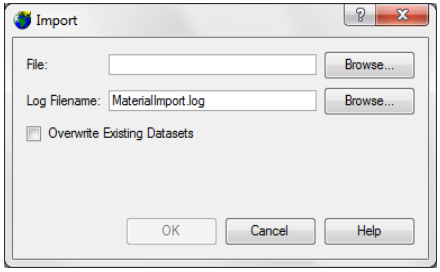


Figure 4-14 Import Data Sets

3. Browse and select the XML file, from which you want to import the data sets in the database.
4. Type the name of the log file in the **Log Filename** box. This log file stores the event log and can be useful if an error occurs.

   By default, a log file named 'MaterialImport.log' is created in your current working directory.
5. Select **Overwrite Existing Datasets**, only if you want to replace the existing data sets with the data sets to be imported.

6. Click **OK**.

Adding Multi-Curve Material Data Sets

The MXD material database has a number of material dataset types that can be used to model fatigue material behavior subject to given mean stress levels. One such material dataset type is the Haigh curve constant life dataset.The Haigh curve stress-life data to be used in this particular example is given below, as estimated from the S-N data for Manten material given in the nCodeDT v11 User Guide; note that stress is given as amplitude (Smax – Smin)/2 rather than stress range

7. In the tree view, right-click on the *database name*, and then select **Add Data**
8. As *Dataset Type*, select **nCode SN Haigh Curves**
9. As *Dataset Name*, for example **Manten HC**
10. On the *Editing* form, type in the properties as shown in the figure below; in this case set stress type as **Amplitude**

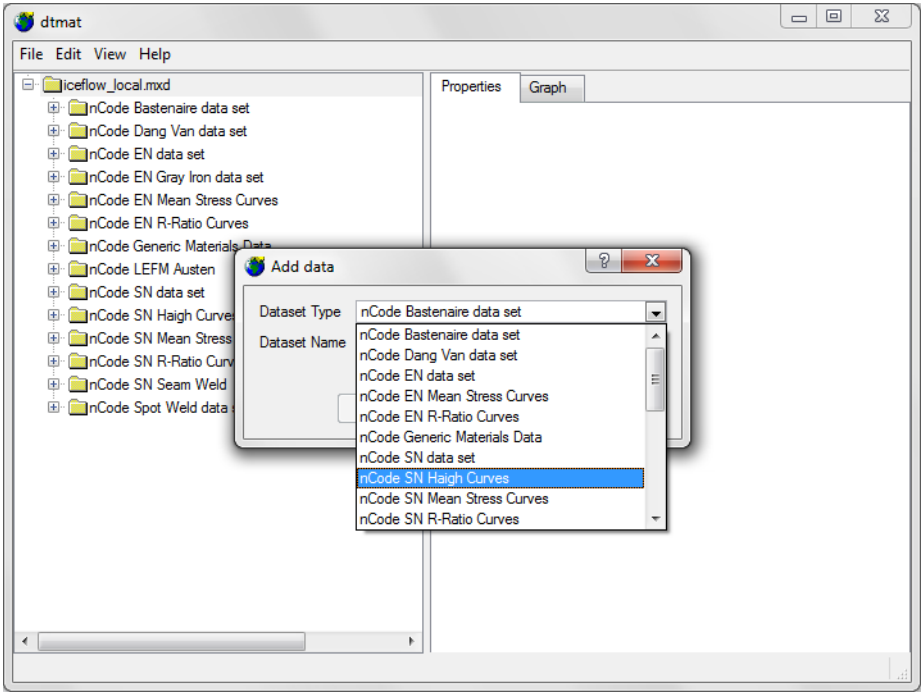


Figure 4-15 Creating the Haigh Curve S-N curve data set

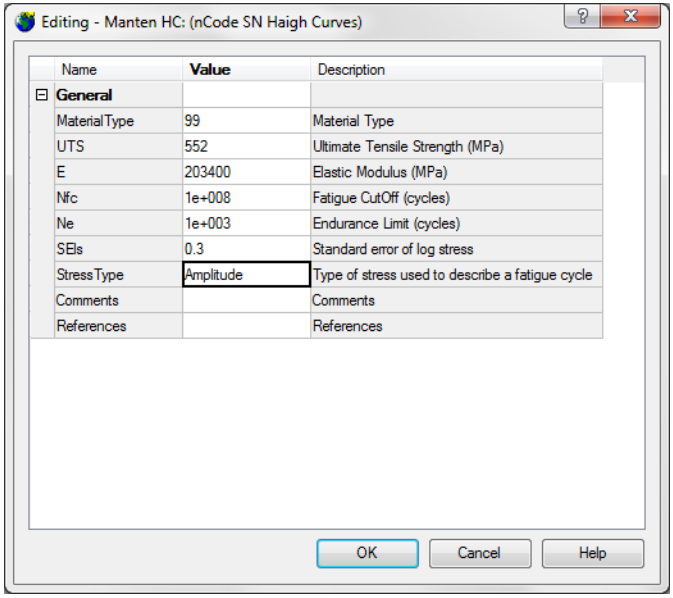


Figure 4-16 Adding the General S-N data

11. Click **Ok** to save the data
12. Add the mean curve data, which requires a data set for each mean stress; in this particular case for lives of **10e4**, **10e5**, **10e6**, **10e7** respectively as shown below
13. In the tree view, the final data set will look like the figure below:

S-N Haigh Curve Data example

Life = 1e4

Stress Ampl.	Mean Stress
800	-70.85
600	84.86
400	240.58
200	396.29

Life = 1e5

Stress Ampl.	Mean Stress
600	-188.11
400	58.59
200	305.30
100	428.65

Life = 1e6

Stress Ampl.	Mean Stress
400	-229.59
200	161.20
100	356.60
50	454.30

Life = 1e7

Stress Ampl.	Mean Stress
250	-223.28
175	9.30
100	241.89
50	396.94

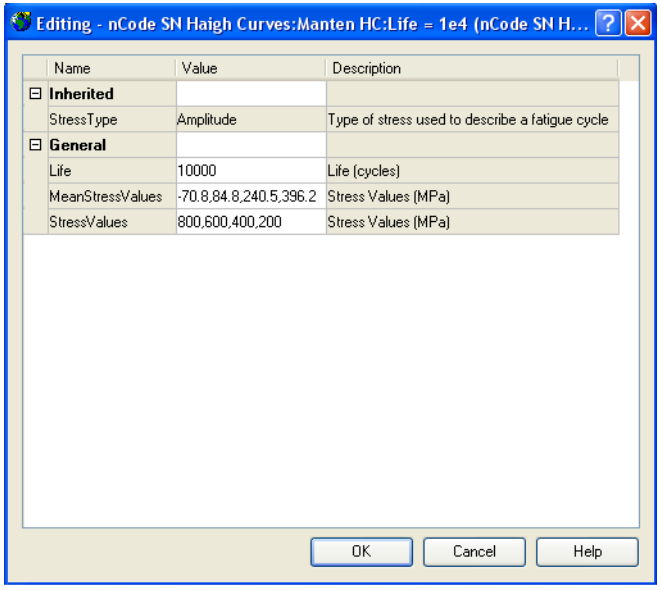


Figure 4-17 Adding the Individual Constant Life S-N data

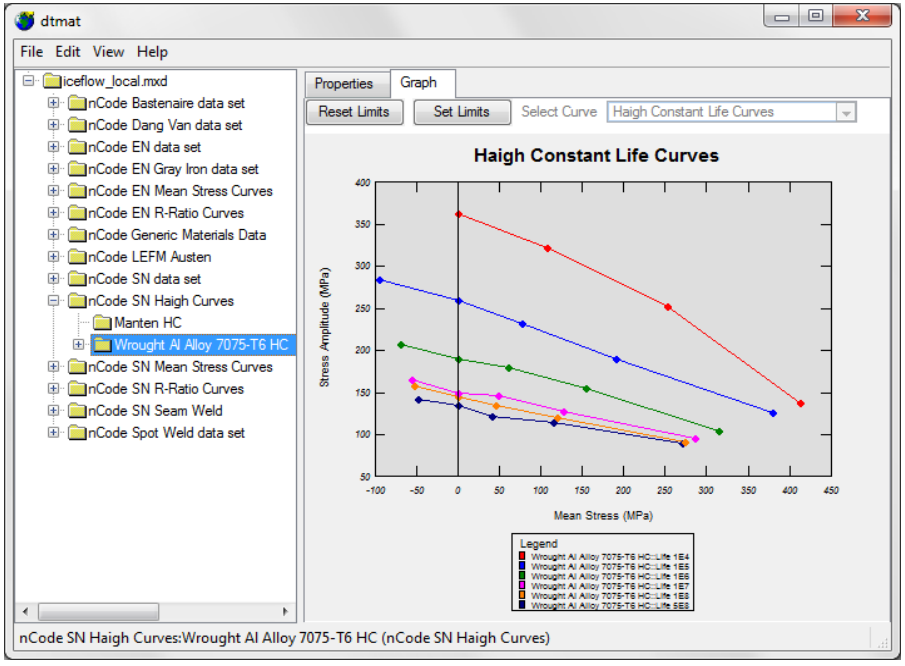


Figure 4-18 New Manten HR Constant Life S-N curve

Creation of a new nCode SN Mean Stress Curves S-N material

As an example, a multi-mean stress curve S-N dataset is constructed using data. Another example of a multi-curve dataset is the set of S-N curves for a given set of mean stresses as taken from the nCodeDT v11 User Guide. Another example of a multi-curve dataset is the set of S-N curves for a given set of mean stresses. This dataset is called the Mean Stress Curves S-N material dataset. As an example, a dataset is constructed using material properties from the nCodeDT v11 User Guide. Note that stress is given as range (Smax – Smin) rather than stress amplitude for this particular example.

1. In the tree view, right-click on the *database name*, and then select **Add Data**
2. As *Dataset Type*, select **nCode SN Mean Stress Curves** data set
3. As *Dataset Name*, for example **Manten MM**
4. On the *Editing* form, type in the properties as shown below

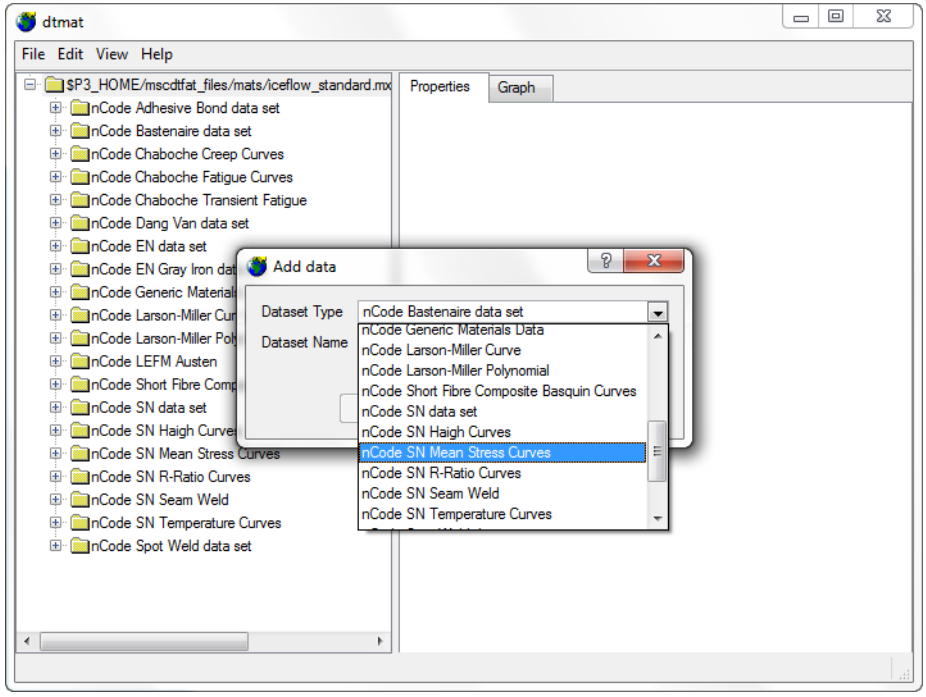


Figure 4-19 Create the Mean Stress Curve material

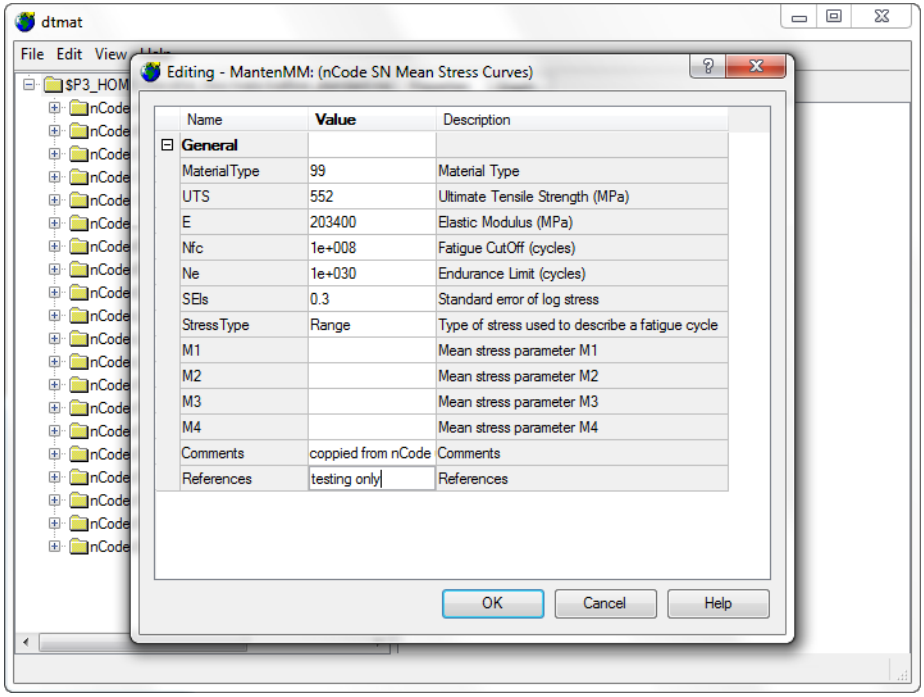


Figure 4-20 Enter the general Data Set properties

5. Click **Ok** to save the data
6. Add the mean curve data, which requires a data set for each mean stress; in this particular example for mean stresses of **-50**, **0**, and **100** MPa respectively as shown below

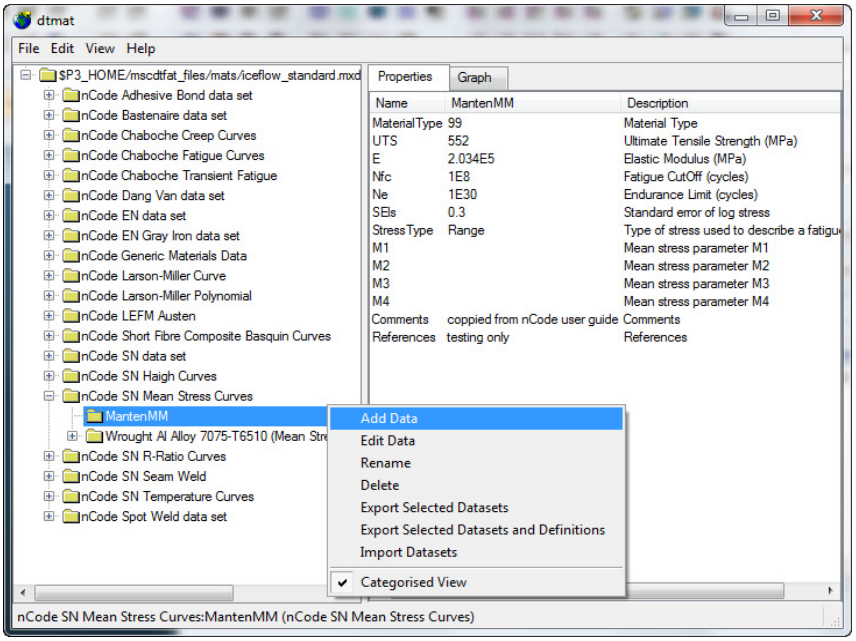


Figure 4-21 Create the individual mean curve data sets

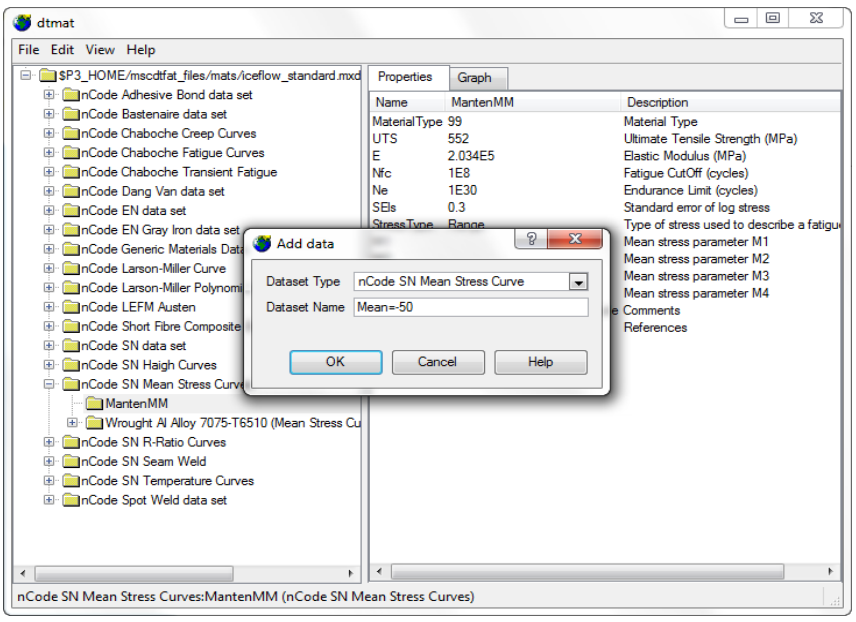


Figure 4-22 Name the individual mean curve data sets

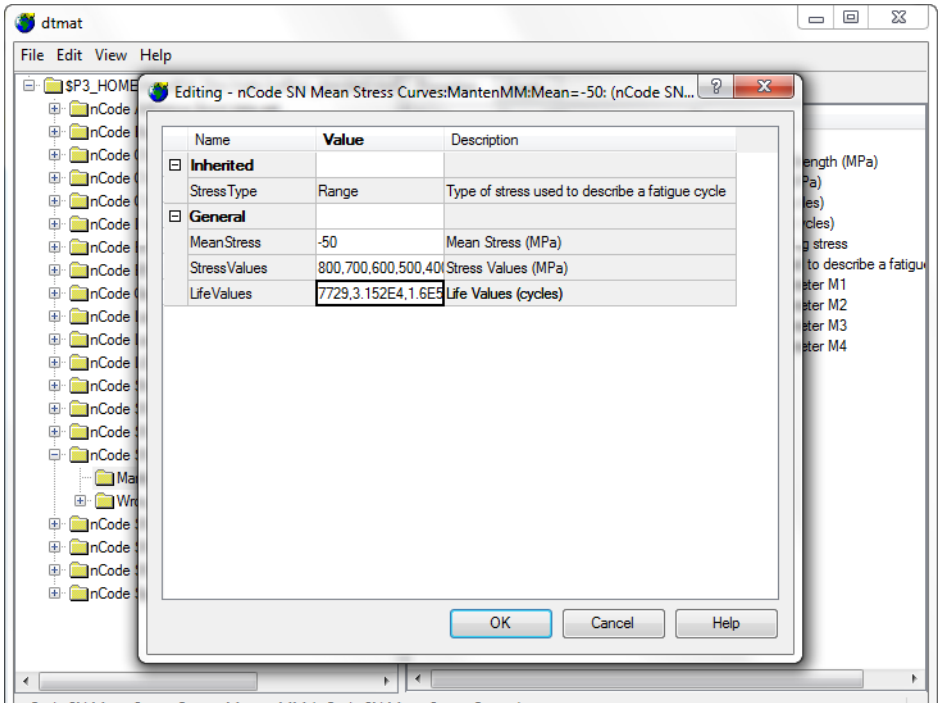


Figure 4-23 Add S-N curve data for a given mean stress

7. In the tree view, the final data set will look like the figure below

Multi-mean curve data for an example steel MANTEN MM:

Property	Value
MaterialType	99
UTS	552
E	203400
Ne	1e8
Nfc	1e30
SEls	0.3
StressType	Range

Curve Data – mean stress -50

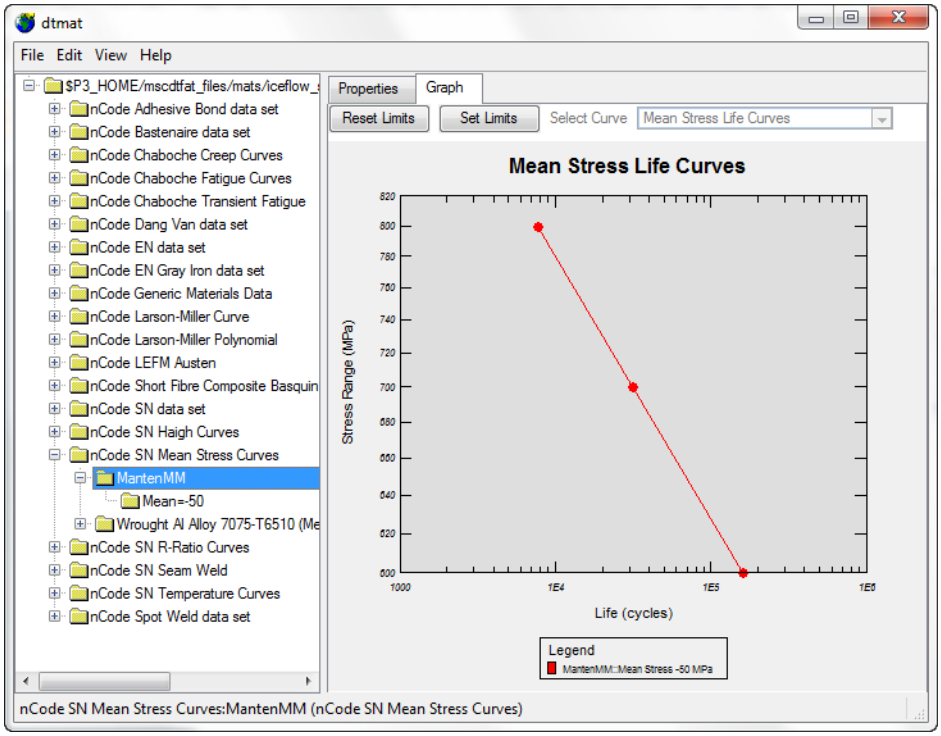
Stress Range	Life
800	7729
700	31520
600	1.6E5
500	1.08E6
400	1.14E7
300	5.12E8
200	1.74E12

Curve Data – mean stress 0

Stress Range	Life
800	3102
700	12651
600	64098
500	4.369E5
400	4.576E6
300	9.45E7
200	3.05E11
100	3.3E17

Curve Data – mean stress 100

Stress Range	Life
800	52
700	1542
600	7819
500	53289
400	558140
300	1.15E7
200	5.546E9
100	6.03E15



New Manten MM multi-mean stress S-N curve

Introduction to PFMAT

PFMAT is a materials database manager which has been designed for use with MSC Fatigue, a fatigue life estimation system. The materials database to which it interfaces has a structure which supports all the materials data (listed below) associated with the three methods of fatigue life estimation:

- Monotonic data
- Stress-life data (including component and spot weld S-N curves)
- Strain-life data
- Cyclic stress-strain data
- Multi-environment crack growth data

By using this database, all users of MSC Fatigue have access to all the necessary data in one central location, thus ensuring the highest-quality design procedures for durability analysis.

PFMAT may be accessed from the Analysis form from the form displayed by selecting **Tools | MSC Fatigue | Main Interface** in Patran by opening the Materials Information form and clicking the Database Manager button. It can also run by typing the symbol **pfmat** at the system prompt.

Once initiated, a set of screen displays which may be manipulated using the keyboard and mouse are presented to the user. A description of the way to use the screen displays is given in Module Operations (Ch. 18) for the Motif and Mask drivers.

When first invoked with the motif driver PFMAT appears displaying two forms.

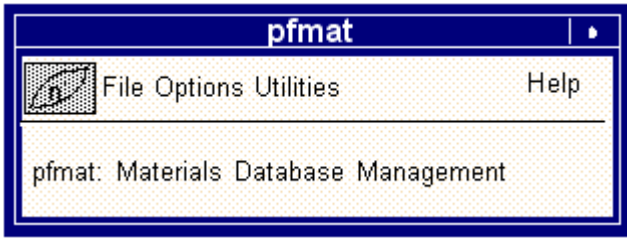


Figure 4-24 PFMAT Utility Form

The top, small form is a generic form and allows for general program control. This is discussed in detail in Module Operations (App. 18) for the Motif driver.

The menu structure for PFMAT is shown in Figure 4-25. The top menu contains options to additional submenus, such as the Graphical display option. This multi-menu layering has been necessary to ensure the legibility of each menu.

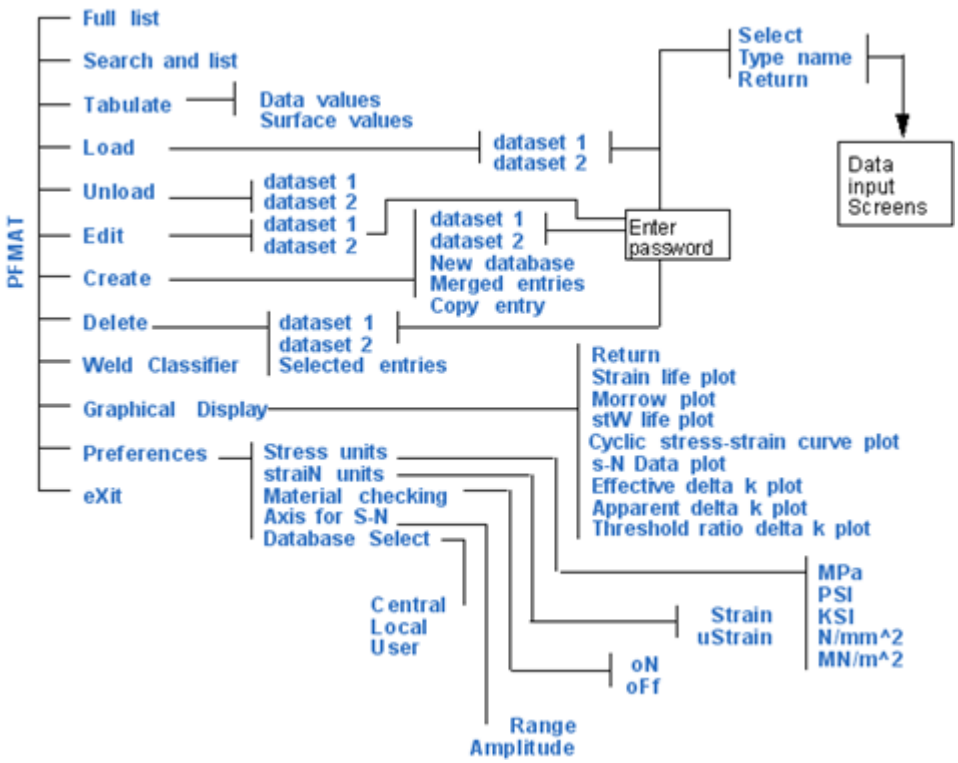


Figure 4-25 The PFMAT Menu Structure

PFMAT Menu Options

The main options and associated text on the PFMAT start-up form is shown in Figure 4-26.

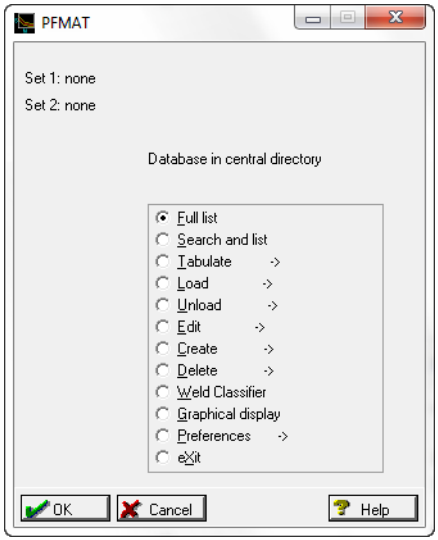


Figure 4-26 PFMAT Start-up Screen Display

The menu options are shown as a vertical list. To select one of the options, move the cursor over the option you wish to select and click the mouse button.

The materials database in use may be the central or a local one. The actual database in use is shown above the menu. The materials datasets that have been loaded are indicated toward the top of the start-up form (see Figure 4-26). One or two materials may be loaded into the manager at any one time.

Important:	To use the central database when a local one exists, the local database must be deleted or renamed from `nmats.mdb` to another name.

Full List Option

The Full list option will list on the screen an index of all the materials dataset names in the materials database together with details of the datasets available for each material, the Young's modulus, and the ultimate strength of the material. An example of the listing is shown in Figure 4-27.

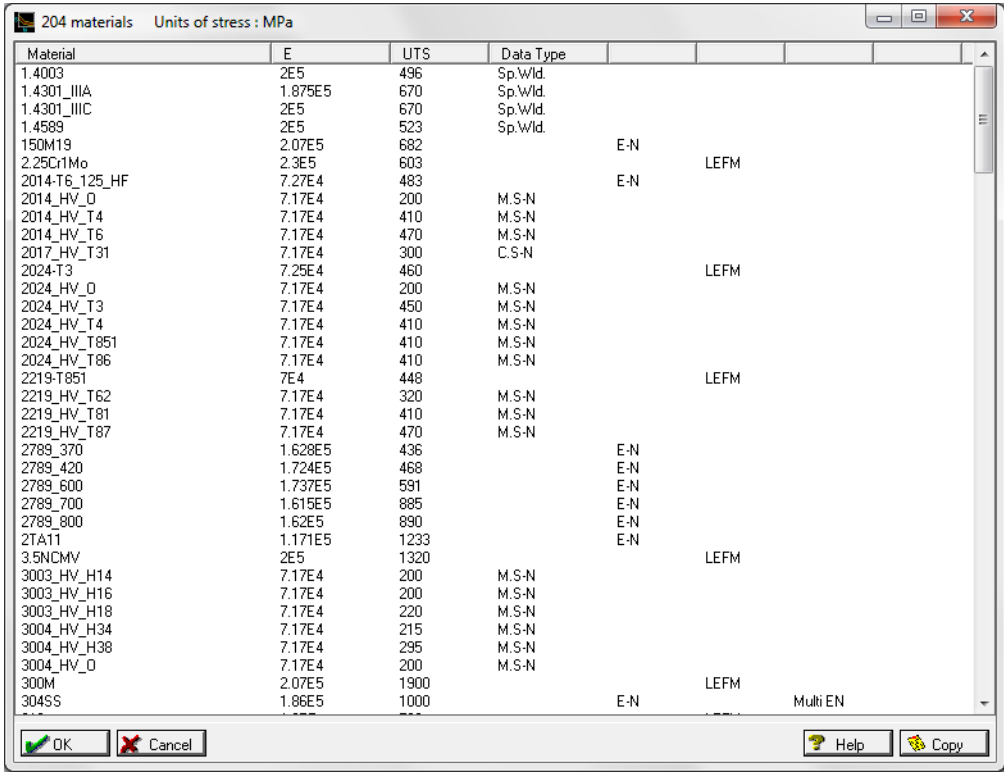


Figure 4-27 Example of the Database Materials Listing

Various columns of information are shown and the data available. For example, in the example screen shown in Figure 4-27, material dataset 2.25Cr1Mo only contains data for crack growth (LEFM) calculations, Young's Modulus (E), and UTS. (E-N = Strain-life, M.S-N = Material stress-life, C.S-N = Component stress life, Sp.Wld. = Spot Weld S-N.)

Depending on the number of entries in the database, the listing may run over a number of pages.

Search and List Option

Search and list is a powerful search facility which allows the user to find materials datasets which meet certain specific requirements such as a given ultimate strength (UTS) or cyclic hardening coefficient (K'). Up to 10 parameters may be used in any one search out of a total list of 34 parameters covering monotonic (10 parameters), cyclic/strain-life (10 parameters), stress-life (7 parameters) and LEFM (8 parameters) datasets.

Once the Search and list option has been chosen, a form is presented as shown in Figure 4-28.

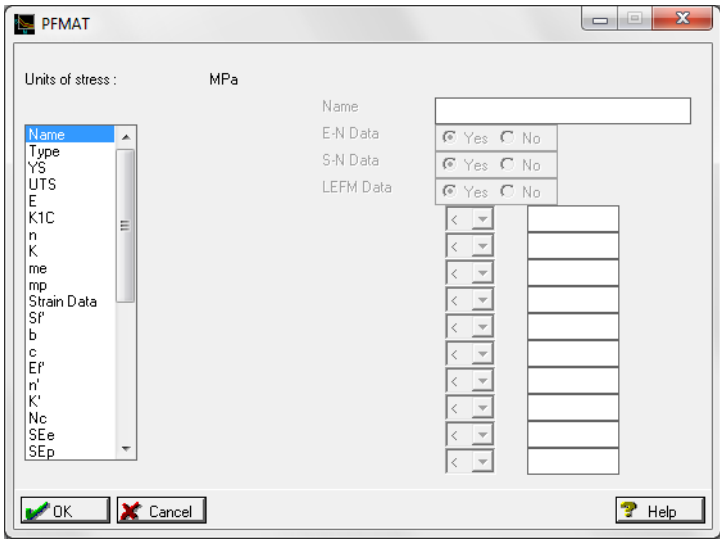


Figure 4-28 The Selective Listing Setup Screen Display

To select a field for use in the search, the following procedure should be used.

1. Select a parameter from the list box at the left with the cursor using the mouse button.
2. A databox will then become enabled with the selected parameter which is located toward the right of the screen. If the search field chosen is a numeric parameter, choose the logical parameter by selecting it from the option menu next to where the selected parameter appeared (<,<=,>=,>,=).
3. For numeric parameters, it is also necessary to define the value of the search parameter (e.g., 500 for UTS = 500 MPa). Type this into the databox next to the logicals option menu.
4. If the search field is non-numeric (e.g., Strain data), then it is necessary to select either YES or NO once the selection has been made.
5. Continue to setup the search parameters until all the values required have been defined (in Steps 1-4). A maximum of 10 search parameters may be defined for any one search. To commence the search, click the **OK** button.

Once a search has been completed, a new window will present the results of the search as shown below in Figure 4-29. PFMAT will have filtered out all those materials that did not meet the search criteria.

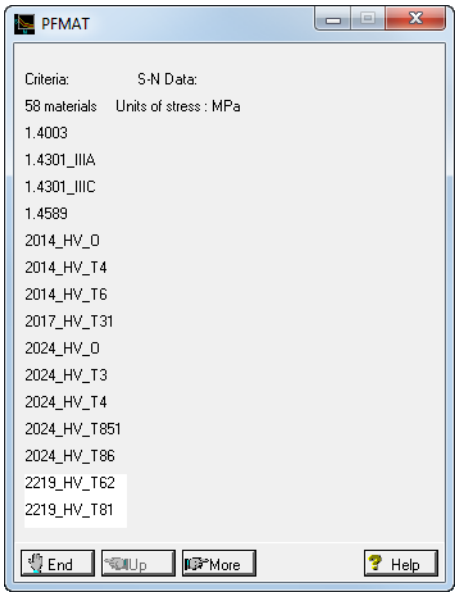


Figure 4-29 A Materials Search Report

Important:	Hint for alternative/international/material names — if alternative names have been defined (e.g., Werstoff number, DIN or BS designations), these will be reported using the search on name option. This search field searches all 8 name fields in the database for each entry.

Tabulate Options

The Tabulate option lists details of datasets 1 and 2.

When selected, it gives the option of choosing Data Values or Surface (finish correction factor) values as shown is Figure 4-30.

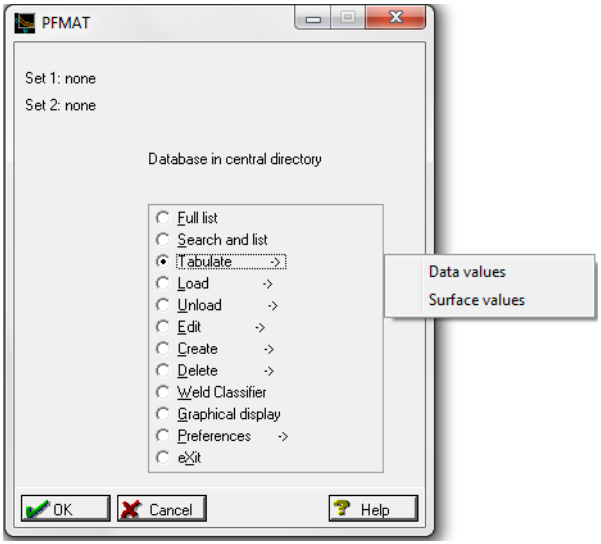


Figure 4-30 Tabulate Options

For example, Figure 4-31 shows that if SAE1006_85A_HR and SAE1045_390_QT were resident as datasets 1 and 2, the Tabulate option would display the following information about them.

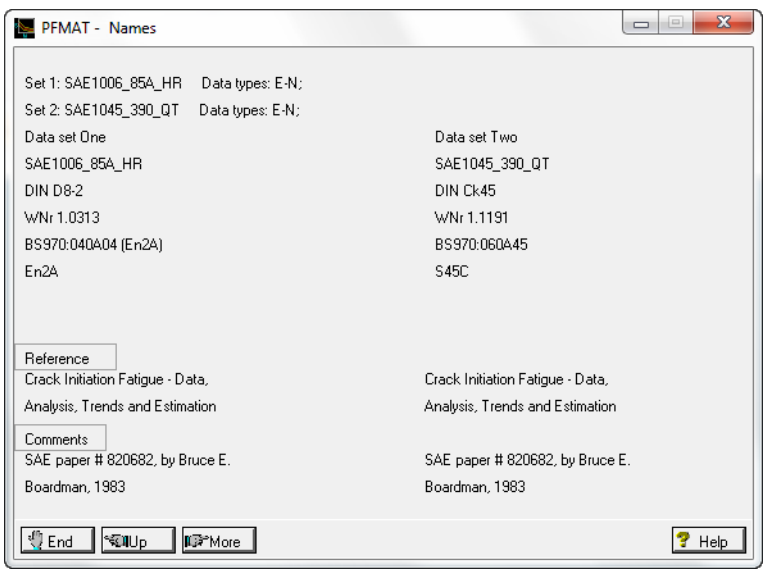


Figure 4-31 .Materials Information Listing, page 1

The above figure is only the first page of several. Subsequent pages list information on:

Page 1	:Dataset name(s), References and Comments.
Page 2	:Monotonic data.
Page 3	:Strain life.
Page 4	:Stress-life data.
Page 5,6,etc	:LEFM datasets for different environments.

See Figure 4-32 and Figure 4-33 for examples of page 2 and 3. The pages of listed data will appear differently depending on the type of data stored for the material(s) loaded. Similar pages will be displayed for pages 4, 5, 6, etc.

For example, the other screen pages look like this:

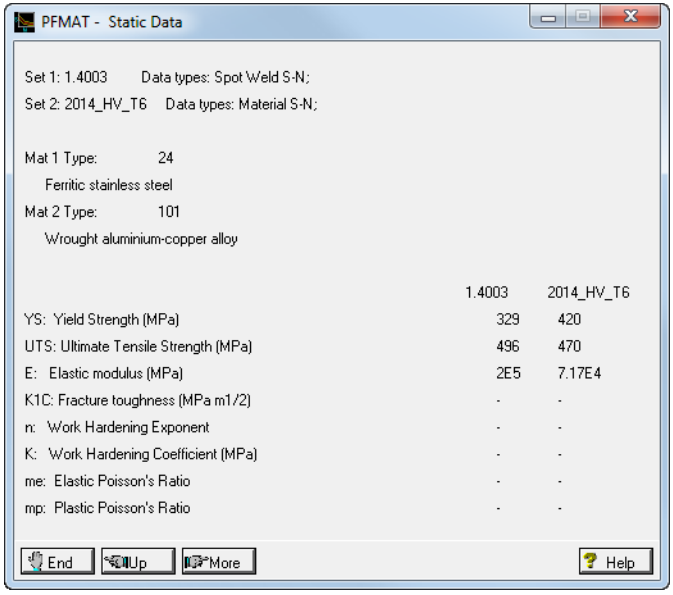


Figure 4-32 Materials Information Listing, page 2

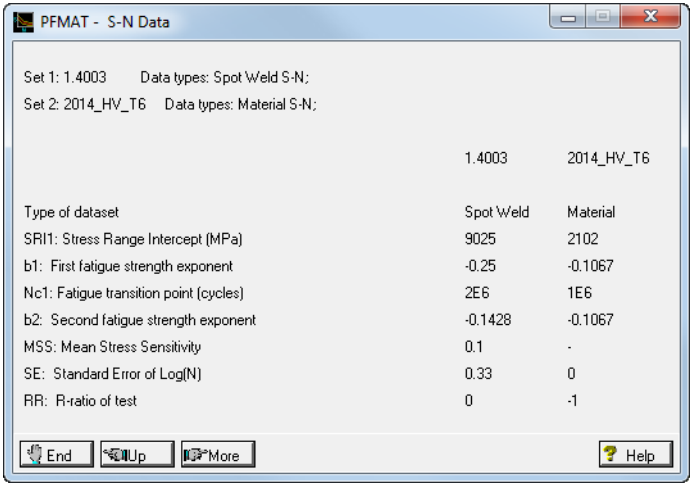


Figure 4-33 Materials Information Listing, page 3

If the Surface Values option is chosen, then a list of surface finish correction factors is displayed. Figure 4-34 shows an example of this type of table.

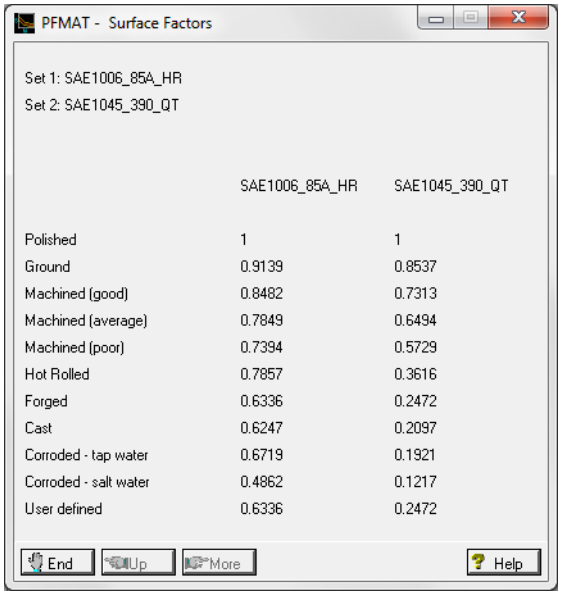


Figure 4-34 List of Surface Correction Factors

The finish of a component can have a significant effect on the fatigue life since a rough surface provides stress concentrations which will promote crack initiation. To calculate the stress concentration due to a range of surfaces, the ultimate strength (UTS) of the material must be known. The UTS is obtained from the materials database and the factors computed for each available surface, resulting in a screen display similar to the one shown in Figure 4-34.

An additional surface finish may be added to the current list providing the user can define the relationship between UTS and endurance limit. A user-defined surface finish can be supplied by modifying the file `cfuser.sur` which is delivered with the MSC Fatigue in the `mats` directory. This file can be edited by loading it into MSC Fatigue's Time History Database Manager, PTIME. The unedited file consists of a straight line at unity. Using PTIME's editing capabilities the user can modify this file to his needs. See Loading Management (Ch. 5) for details on how to use PTIME.

Load Option

Selection of this option allows you to load a material into Set 1 and/or Set 2, or replace an existing material for the purpose of editing, tabulating, or graphing. When first selected, the default dataset is Set 1. You may change it to Set 2 by moving the selection with the cursor or arrow keys or by using the hot keys, **1** or **2** for Set 1 or 2, respectively. See Figure 4-35.

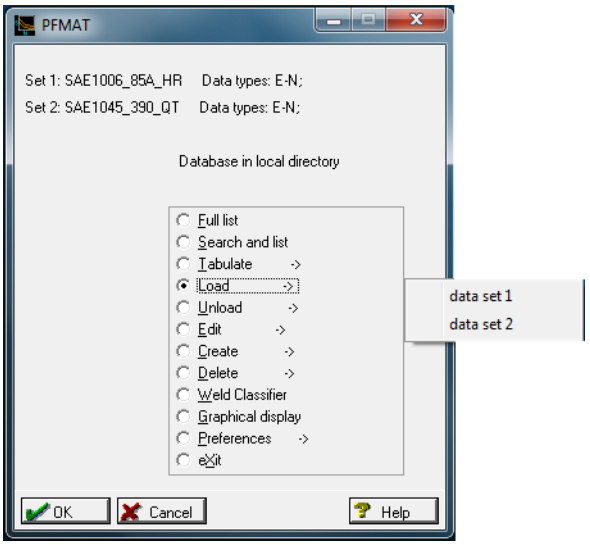


Figure 4-35 The Load Option

When the dataset is chosen, PFMAT lists the contents of the database on a screen such as that shown in Figure 4-36.

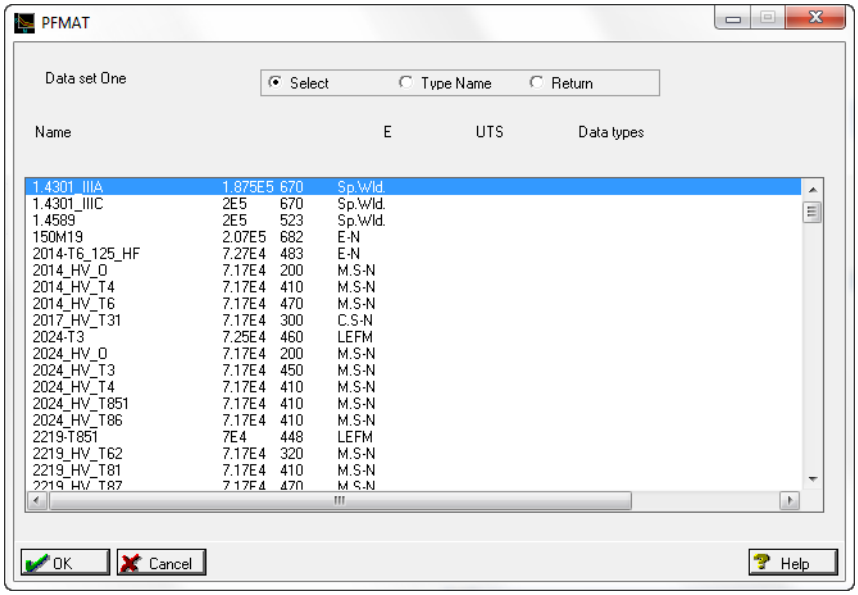


Figure 4-36 A Listing of the Available Materials

To load a material from the list box, simply select one with the cursor. Alternatively, a material name can be typed by selecting the Type Name toggle at the top of the form. This will bring up a databox in which the user can type the name or part of the name of a material, which if found, will be loaded. The Return toggle will return the user to the main PFMAT form.

Note that the scroll bar at the right of the list is a visual cue telling you that (in this case) more selections are available. Moving the scroll bar to the side of the list box will enable viewing of materials not visible.

Important:	A word of caution: if a name fragment is entered (such as SAE) and if several database entries exist with that fragment, then PFMAT will load the first one encountered into the named set. Type the minimum number of characters that uniquely define an entry.

Unload Option

This option has the opposite effect to the Load option described above. Any material resident in datasets 1 or 2 can be unloaded and ready for replacement with a new entry. This option is very easy to use; simply select set 1 or set 2 from the popup menu. The selected set will be emptied.

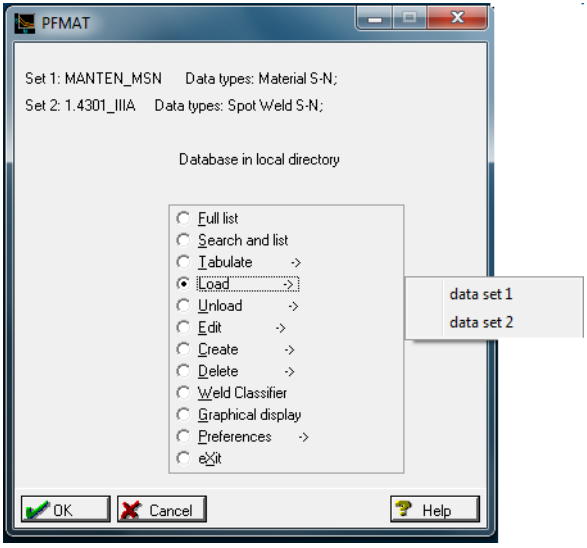


Figure 4-37 The Unload Option

Edit Option

Edit allows the entry in either set 1 or set 2 to be changed (edited). Note that the central database should not normally be edited. The edit option only allows editing the central database if the password is known. The password will be requested on a screen such as that shown in Figure 4-38.

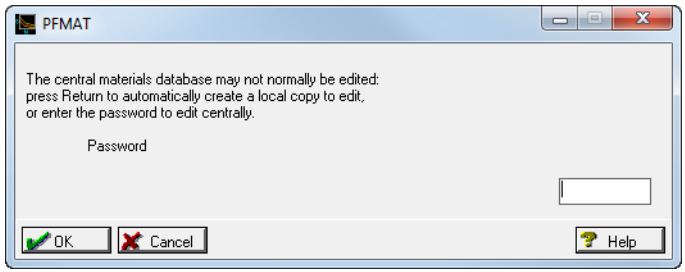


Figure 4-38 Password Screen

When the materials database is shipped, the password is set to PFATIGUE. However, the system manager may have changed this password at installation time. If no authorization to enter data in the central materials database has been given, it is still possible to make a local copy of the database and edit the latter. The system manager will normally know the password and he should be consulted if data needs to be entered in the central database. The password can only be changed if the database manager, PFMAT, is used to access the central database while working in the directory where the central database resides. File and directory protections will have to be set properly also if access is to be given outside of the central directory where the database resides.

Important:	If a local database exists, then PFMAT will automatically start editing the local database, and the above password stage will be skipped.

The first edit screen is shown in Figure 4-39 (it is the same screen if editing an entry in the central or local database).

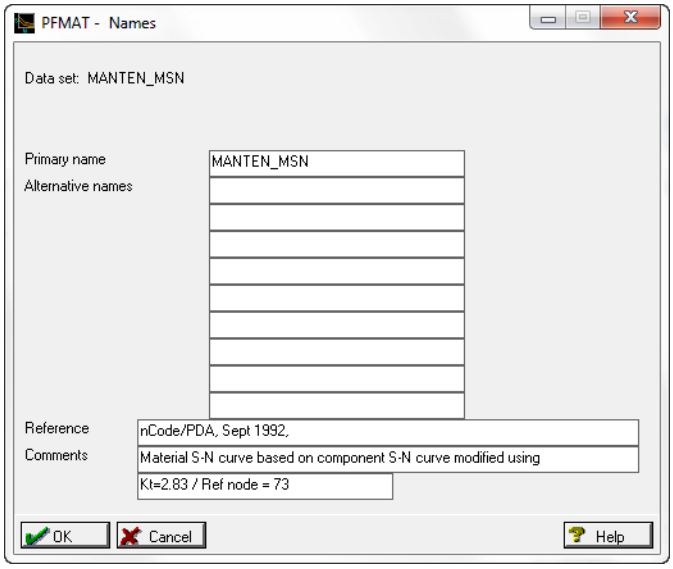


Figure 4-39 Edit Database Screen, page 1

Edit the name, references, or comment, and click the **OK** button to accept the changes. Successive pages devoted to information on general physical properties, E-N data, S-N data, and fracture mechanics data will be presented after this page.

Sample screens are shown in Figure 4-40, Figure 4-41, Figure 4-42, in the order that they appear. Clicking the Cancel button during any of any edit session will page back up to the previous edit screen.

Important:	Related specifications are based on chemical composition ONLY; mechanical properties should be confirmed before considering alternative grade designations.

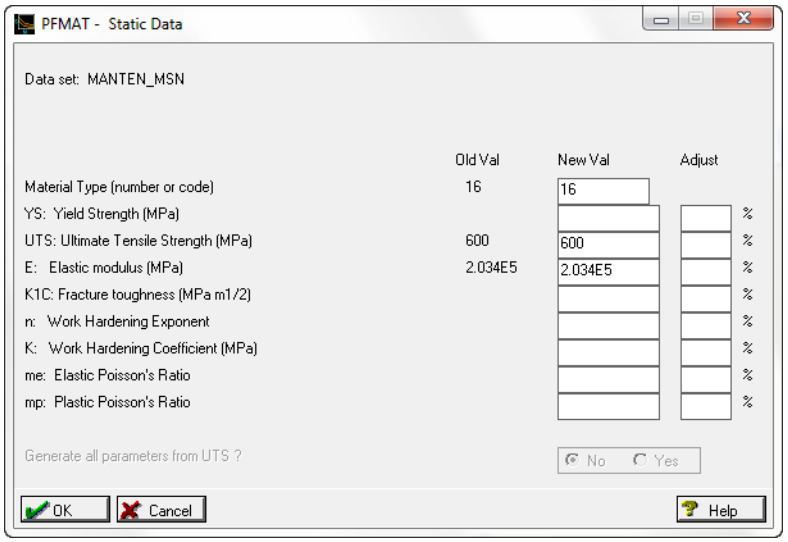


Figure 4-40 Edit Database Screen, page 2

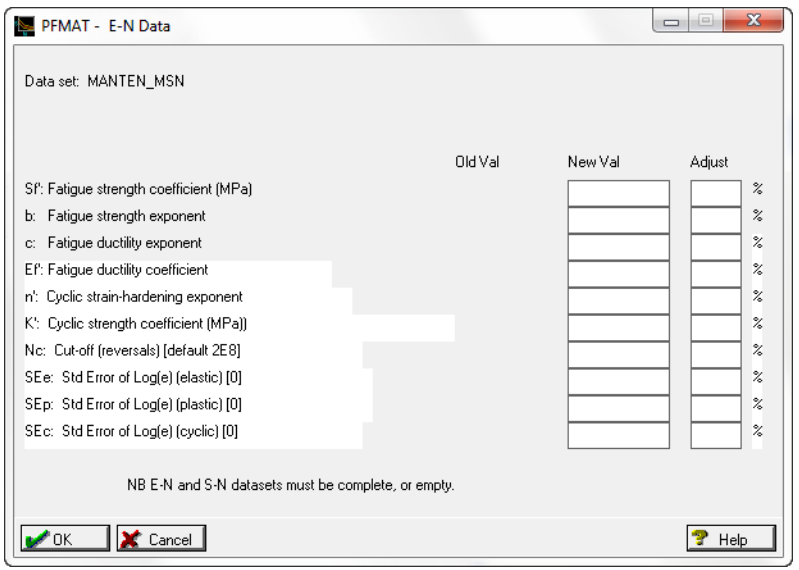


Figure 4-41 Edit Database Screen, page 3

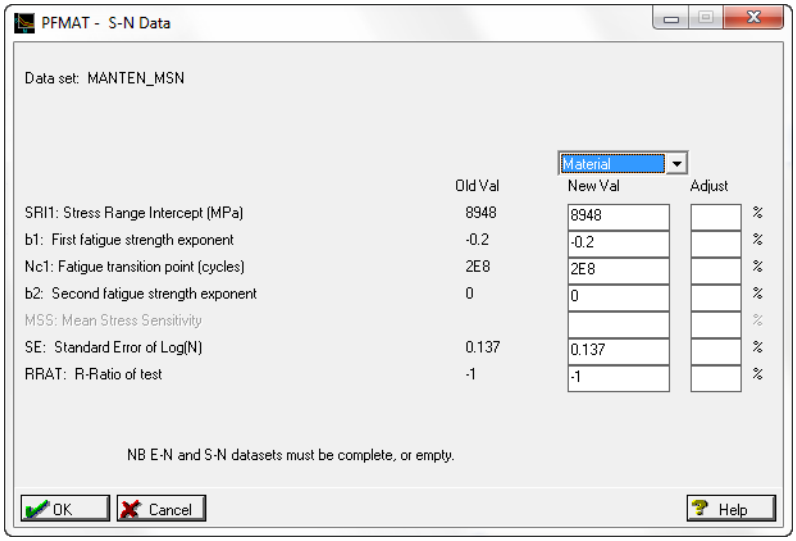


Figure 4-42 Edit Database Screen, page 4

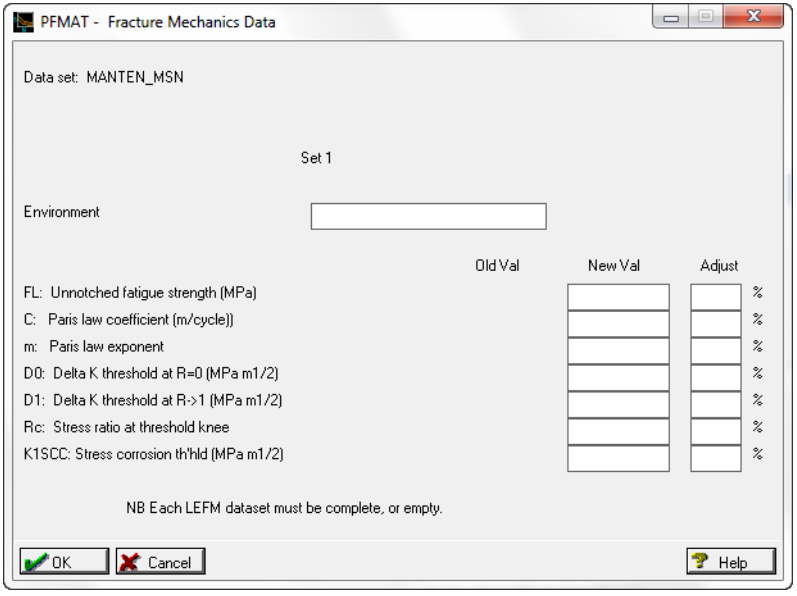


Figure 4-43 Edit Database Screen, page 5

Note that every field has a help window associated with it. For example, if guidance is needed about the units for work hardening coefficient on the generic data page, then move the highlight bar over that field and press the F1 key. A pop-up help window will appear.

Also, data values may be edited by applying a percentage change. The change is automatically and immediately updated on the screen.

Important:	Each data value has a valid range. If the user wishes to input values which are outside of these ranges, he must turn material checking off. See the Preferences Option, 167 option in this chapter. The valid ranges can be determined by placing the cursor or highlight bar over the relevant data value and invoking help with the F1 key.

Create Option

Create allows a new material to be entered into the attached database. If this option is chosen, than either dataset 1 or 2 must be nominated to receive the new material. If the chosen set already contains an entry, then it will be overwritten. If the name chosen to represent the new entry already exists in the database, an option is given to load it directly.

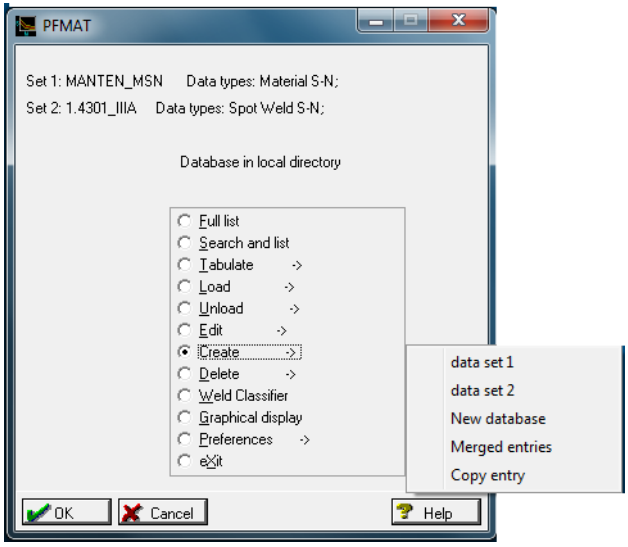


Figure 4-44 The Create Option

Under this option, it is also possible to create a New Database, 158 or Merged Entries, 159 into the attached database from another database or simply Copy Entry, 160 to a new name.

One very useful option is that of generating material properties from UTS values based on empirically derived formulas. This helps create useful (although not exact) material properties to get started on a fatigue analysis when cyclic materials data are not available. You must know the UTS, Young's Modulus and enter the appropriate metal type code on the static page. See Figure 4-46 and Table 4-2 for the listing of valid metal type codes.

When loading new materials information into the database, various input screens will be presented. The create screens are illustrated in Figure 4-45, Figure 4-46, Figure 4-47, Figure 4-48, and Figure 4-49.

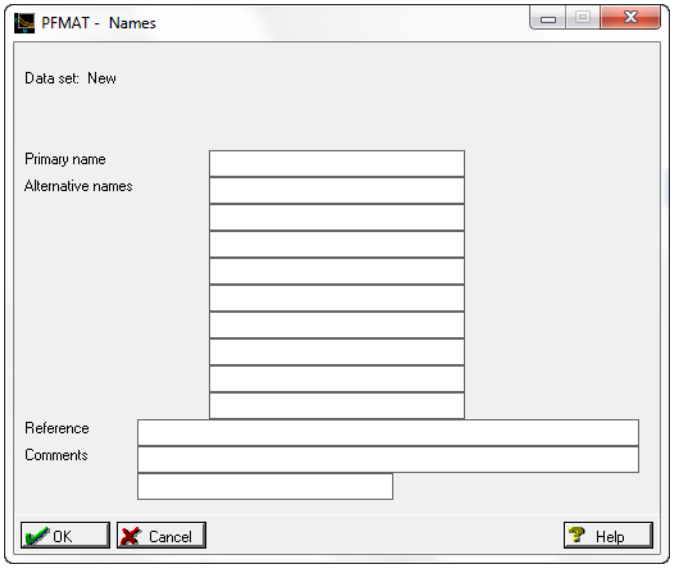


Figure 4-45 Create Material Screen, page 1

When some or all of the fields have been filled, the new entry is appended to the database. Again, if no local database exists in the working directory, a password must be supplied to modify the central database. If no password is given, a local database will be created. For more information about modifying the central database and the password, see the Edit Option, 149 description.

Each data value has valid numerical ranges that are deemed acceptable. If numbers outside of these ranges are input, they will be rejected. To see the valid ranges, place the cursor in the appropriate databox and use the help option by pressing the F1 key. If the user wishes to input numbers outside of the appropriate range, turn Material checking off. See the Preferences Option, 167 for more information.

Materials data may be loaded automatically from other databases using the ASCII batch operation of this create option. This is described later in this chapter, PFMAT in BATCH Mode, 173. It can also be generated automatically from only the UTS value as explained previously.

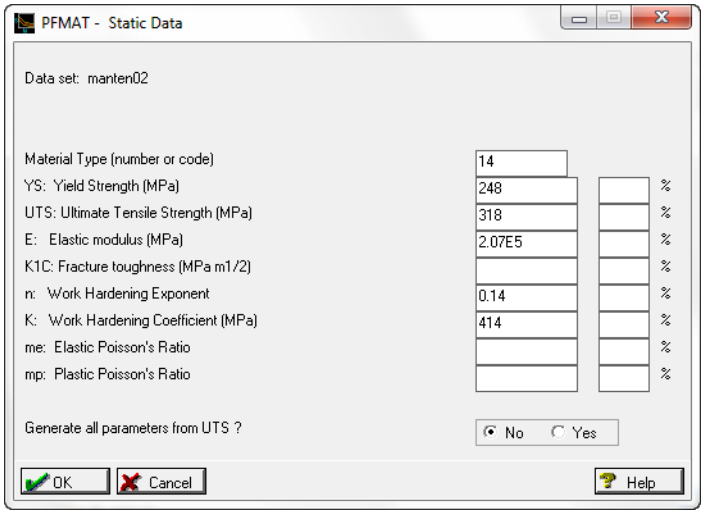


Figure 4-46 Create Material Screen, page 2

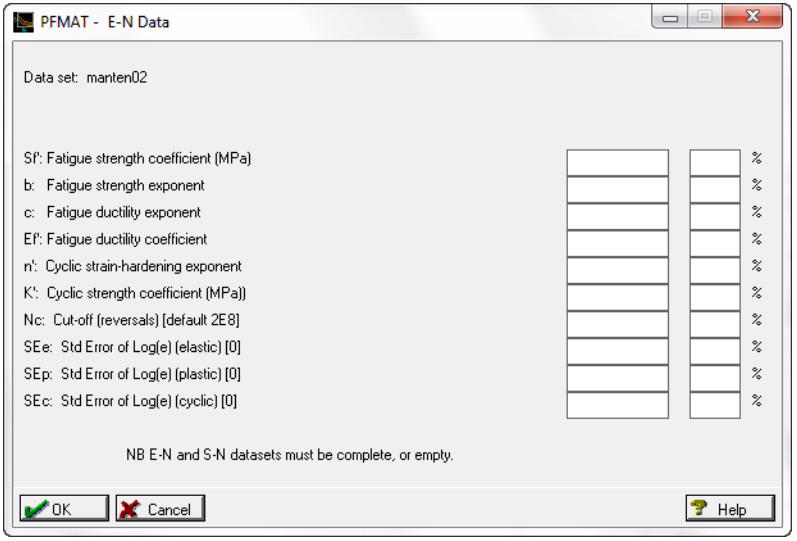


Figure 4-47 Create Material Screen, page 3

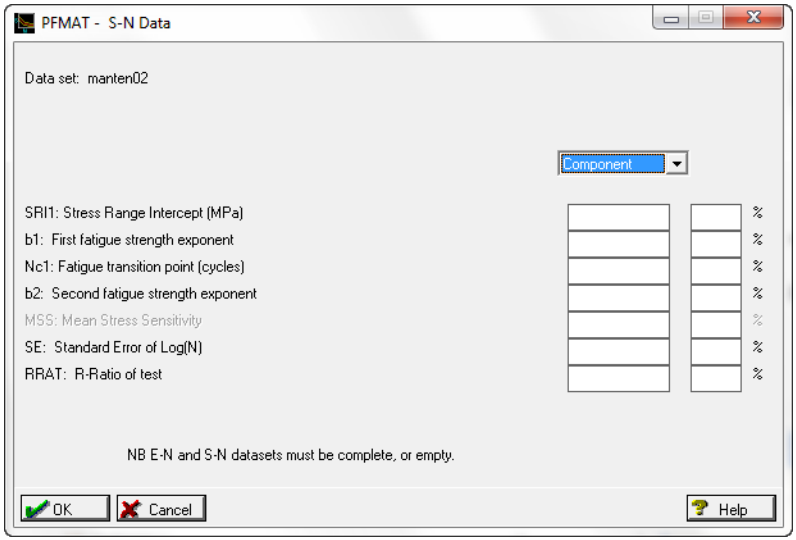


Figure 4-48 Create Material Screen, page 4

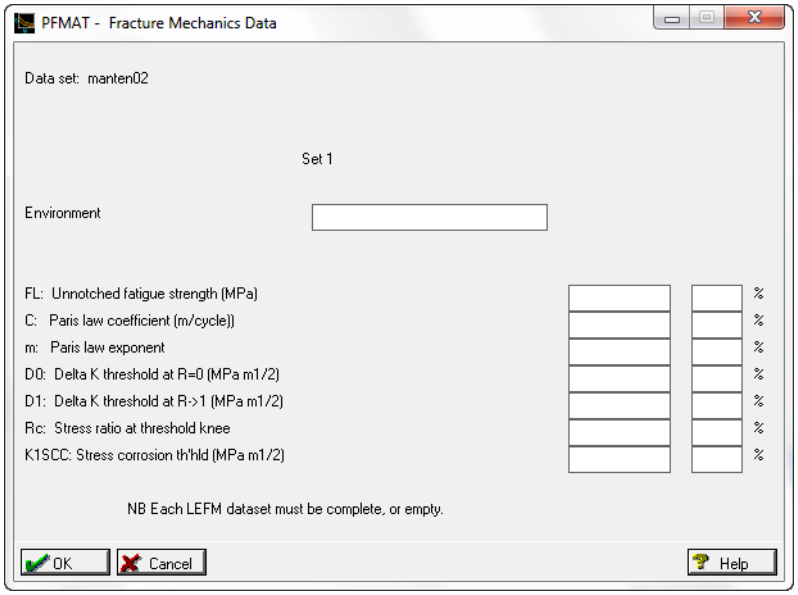


Figure 4-49 Create Material Screen, page 5

New Database

This feature allows for creation of a new, empty database. (This is necessary when batch loading materials into a local database.) A form will appear into which the following information will be requested.

1. The new database name
2. A character database ID
3. A password

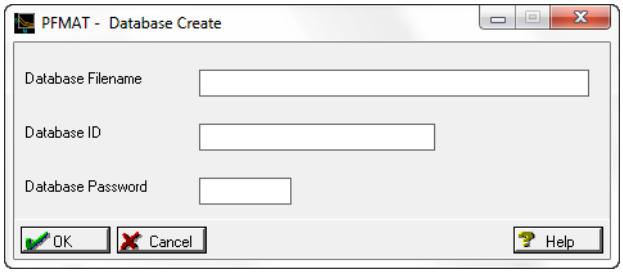


Figure 4-50 The New Database Option

All information on this form is required. After clicking the **OK** button, a new database file will be created in the local directory called `database.mdb` where `database` is the name given in the first field. The main PFMAT form will reflect the fact that this new database is now attached.

Important: If, after creating this database, it is desired to use it with MSC Fatigue in a fatigue analysis, it must be renamed to `nmats.mdb`.

Another form will appear after the database has been created requesting to set the database preference on start up of PFMAT. This is more fully described in the Preferences Option, 167 description.

Merged Entries

This feature allows merging of databases or database entries to make one database from two. A form will appear requesting the name of a database. The materials associated with this database will be appended to the attached database. An option exists for overwriting existing records of the same name if duplicates are encountered.



Figure 4-51 The Merged Entries Option

It is possible to merge only materials with names starting with a particular character string using the Material Filter databox. For example, if all materials beginning with the string SAE are desired, then type SAE into the databox. When specifying a material filter, an additional form will appear at each material encountered with the option of merging it (YES), skipping it (NO), turning the prompt off and accepting all materials encountered (ALL), or stopping the merge (QUIT).

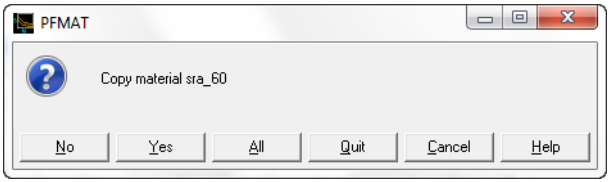


Figure 4-52 Confirmation of Merged Entries

Copy Entry

This option simply asks for you to select an existing material entry in the database. It will then ask you for a new name or names and then duplicate the rest of the materials information and create a new material entry.

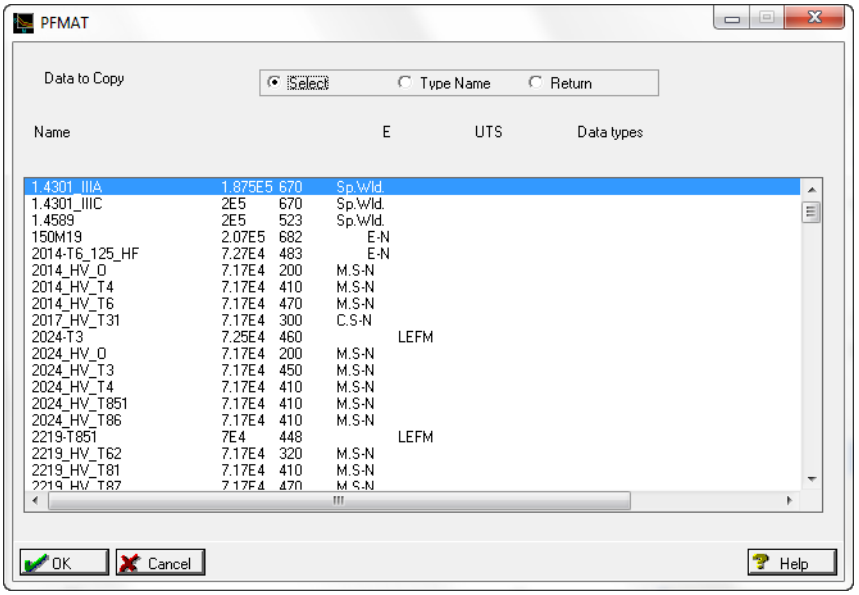


Figure 4-53 Copy Entry Screen

Delete Option

This option allows for deletion of a material from the database. The material to delete is the one currently selected (material dataset one or two). To delete a different material from one currently loaded in the database manager, it is first necessary to load the material using the Load Option, 147, or use the selected *Selected Entries* option.

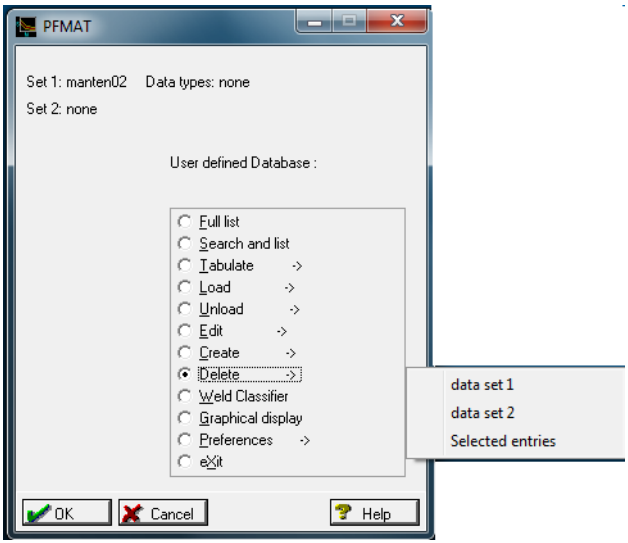


Figure 4-54 The Delete Option

Note that PFMAT must receive the user's permission before permanently deleting data. Also, if deleting a material from the central database, the password is necessary. See the Edit Option, 149 for information about the password.

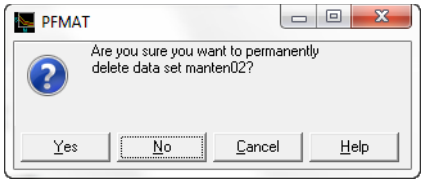


Figure 4-55 Delete Confirmation Screen

Selected Entries

This feature allows for deletion of more than one material entry from the attached database at one time. Again, a character string may be entered to specify materials which start with that particular sequence. As with merging records, confirmation is requested to delete the material (Yes), skip it with no delete (No), turn the prompt off and delete all records encountered (All), or stop the delete entirely (Quit).

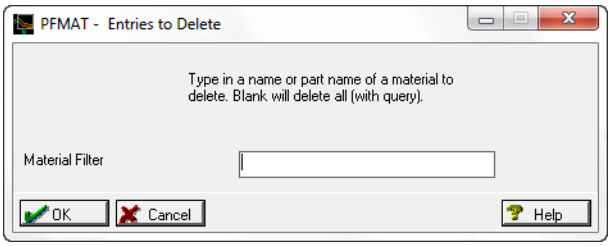


Figure 4-56 Delete Selected Material Entries

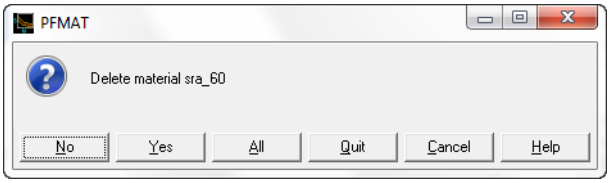


Figure 4-57 Delete Confirmation Screen

Weld Classifier Option

MSC Fatigue uses a fatigue analysis model for welds which has been thoroughly tested and approved by the British Standards Institute (BSI). The standard which defines the procedures for weld durability analysis is BS7608, Part 10 (Ref. 1). In this standard, weld types are defined according to a set of rules. These rules have been encoded in this menu option in order to identify the weld type for the weld under analysis. The Weld classification main menu is presented in Figure 4-58 with a brief description of the three main paths through the program.

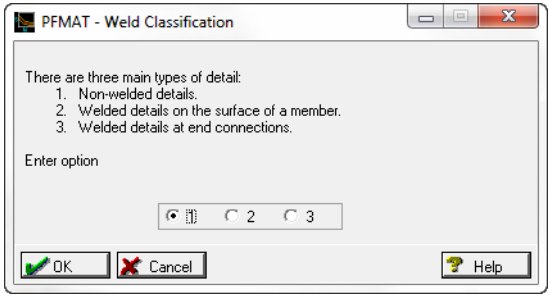


Figure 4-58 Weld Classifier Main Menu

From this point, you may select one of the three options and follow a particular path through the weld classification questions.

The major paths are summarized on the flowchart in Figure 4-59.

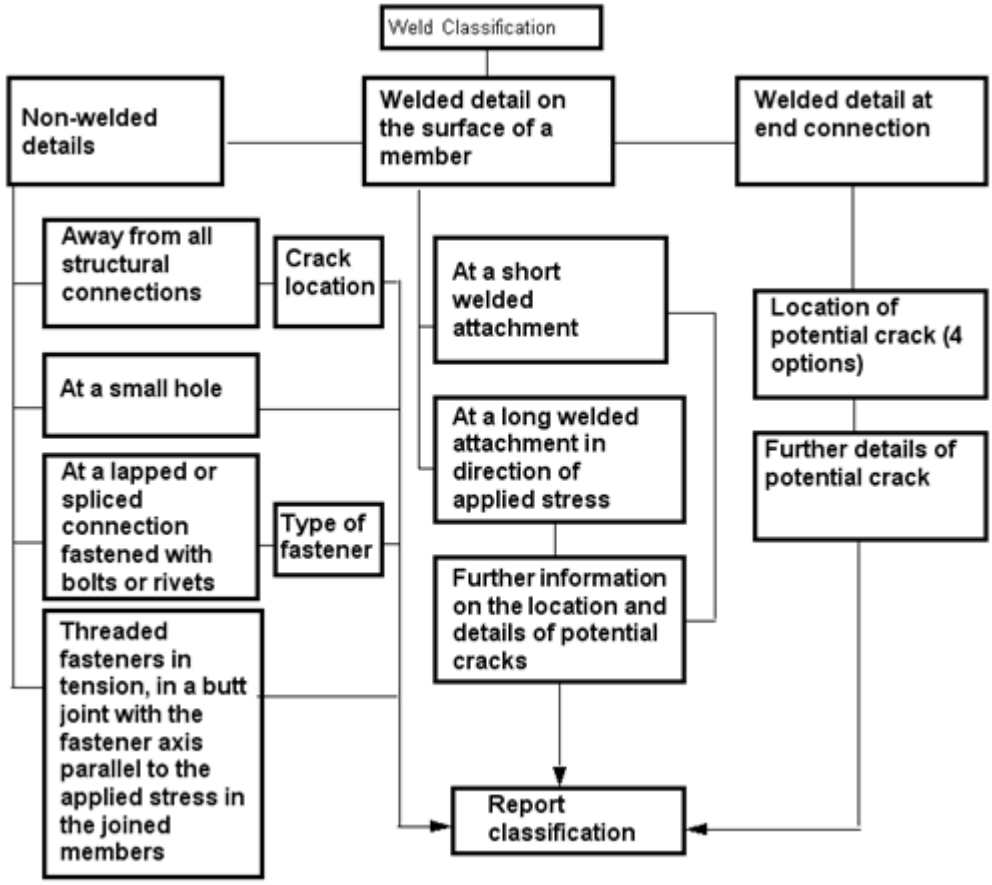


Figure 4-59 Partial Weld Classifier Flow Diagram

The following notes are made with respect to classifying welds.

1. For option 2 (small hole) of Non-welded details, the diameter must not be greater than three times the plate thickness. The hole may contain a bolt for the attachment of a minor fixture. If the edges of the plate are cut, this must be from machining, grinding, planing, or flame cutting by a controlled procedure.
2. Option 4 of Non-welded details applies to bolt threads conforming to BS3692 or BS4395, and rod threads conforming to BS3643 part 2.
3. Option 1 of Welded details on the surface of a member includes long attachments welded by intermittent longitudinal fillet welds, even if the individual welds are less than 150mm long.
4. For options 2 and 3 of Welded details at end connections, “members” includes built-up members, rolled steel members, and rolled steel plates. Option 2 also applies if a third transverse member is present between the two members being joined — see BS7608 (Ref. 1).

Depending on the options selected, prompts for a series of questions defining the details of the weld will be given. At the end of the consultation, PFMAT will report a weld classification similar to Figure 4-60.

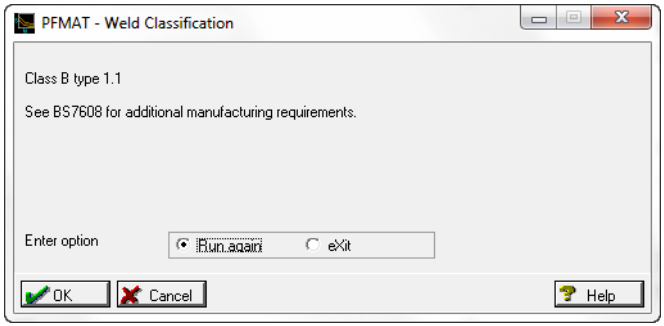


Figure 4-60 Typical Weld Classification Example

Graphical Display Option

The graphical display submenu leads to a dynamic menu of graph types. The possible options are shown in Figure 4-61.

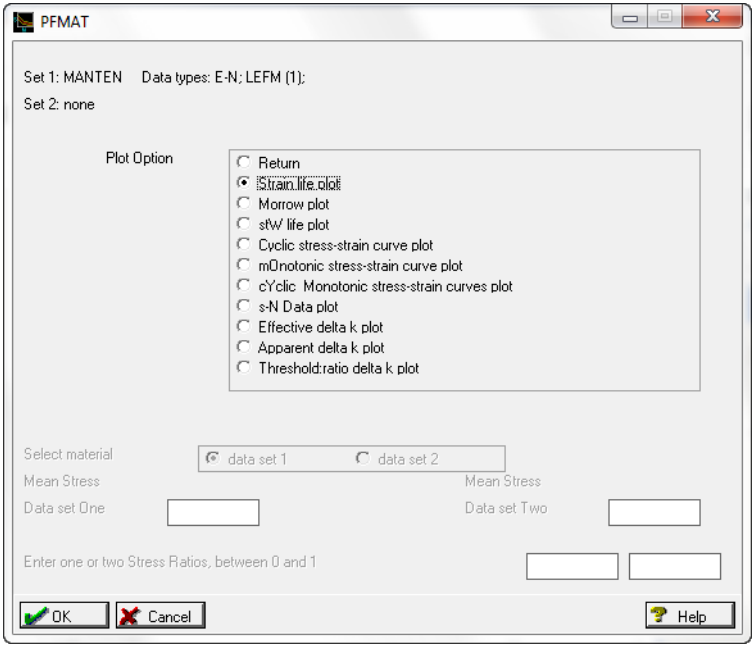


Figure 4-61 The Graphical Display Option

Important:	The Graphical display submenu may change in detail according to the data present in the dataset.

To select a particular plot for one or two materials, first load the material dataset(s) and then select the Graphical displays menu option. Pick one of the plots by moving the cursor over the required plot description and press the mouse button. In some cases, additional information will be required. If this is necessary, enter the data in the databoxes which will be enabled when the appropriate plot type is requested. Then use the **OK** button to display the plot.

The plot will be displayed together with some pull down options menus at the top of the window. Not all options will be present on all graphical display menus since some of the options are data-type specific.

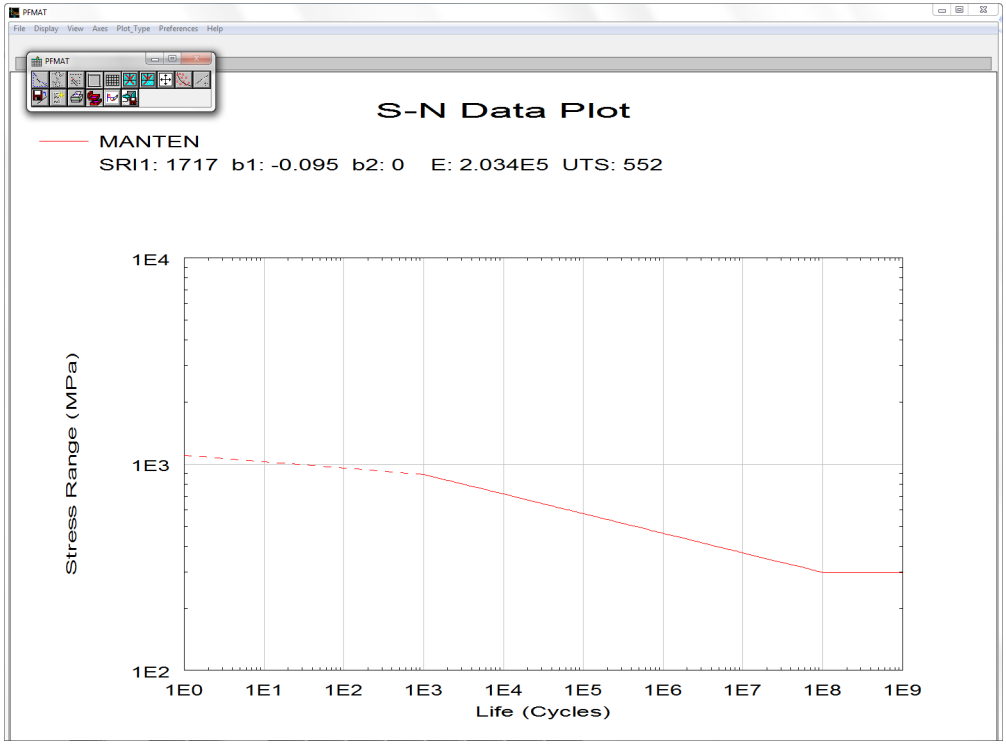


Figure 4-62 Menu Options on the Graphical Display Screen

Many of the preferences and other options available through these pull-down menus are generic for graphical operations throughout the MSC Fatigue system and are not discussed here. The operation of this graphical user interface is described in Module Operations (App. 18), as well as the commands that may be entered in the Command databox. The functions of the pull-down menus which are specific to PFMAT are described below:

Option	Description
Scatter Curve	This option plots the +/-3 standard deviation lines for the S-N curve.
Elastic Line	This option adds the linear elastic to the cyclic and/or monotonic stress-strain curves (i.e., the line that defines Young's Modulus).
E-P Lines	This option adds the elastic and plastic strain-life response lines to the strain-life curve.
Fatigue Limit	This option adds the fatigue limit line to the stress-life or strain-life curves.
Remove Lines	This option removes the elastic and plastic strain response lines, elastic line, and the fatigue limit line and scatter curve from the plot.
X-Window	This option allows definition of the width of the plot in terms of the x-coordinates by entering two X positions using the keyboard. The input is prompted for in the Command databox. The mouse can also be used to window around a certain location by positioning the cursor and the RIGHT button depressed to mark each window edge. After the second edge is selected, the plot will be redrawn automatically. The first mouse click is the lower left corner.
Y-Window	This option allows definition of the height of the plot in terms of the y-coordinates by entering two Y positions using the keyboard. The input is prompted for in the Command databox. The mouse can also be used to window around a certain location by positioning the cursor and the RIGHT button depressed to mark each window edge. After the second edge is selected, the plot will be redrawn automatically. The first mouse click is the lower left corner.
Full Plot	This option redraws the whole plot from the minimum x and y values to the maximum x and y values.

Important:	You can obtain coordinate locations on any of the graphs by placing the mouse cursor over the graph and depressing the LEFT mouse button. The coordinates are reported in the upper left corner of the window below the pull-down menus.

The following keyboard letters perform the functions described below:

Option	Description
V	Coordinates: place the cursor anywhere over the graph and depress the V key. The coordinates of the graph will be reported in the upper left corner of the window.
P	Plot: the P key will replot the graph.
W	Window: place the cursor anywhere in the graph to define the lower left corner of a new window and depress the W key. Move the cursor to the upper right corner and depress the W key again. The plot will automatically update to define the new window.

Preferences Option

This option gives the chance to select the units of stress or strain that all records will use, to set material checking on or off, to change the S-N curve axis, or change the working database. Each of these options has its own options from which to choose from.

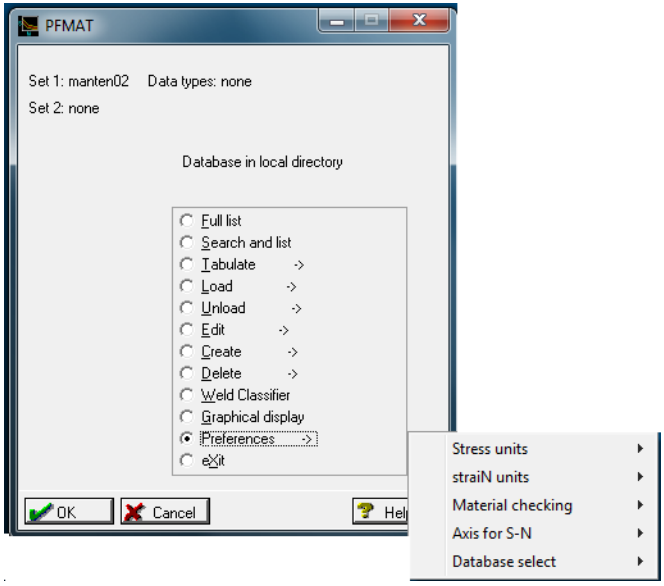


Figure 4-63 The Preferences Options

Stress units available are: **MPa, PSI, KSI, N/** mm^2 **, MN/** mm^2 (strain or microstrain). Note that when different units are used the database is not actually changed, they are passed through a conversion factor table stored in PFMAT and then displayed on screen.

Through the Preferences option there is the ability to turn material range checking on or off. When editing or creating new materials PFMAT, check to see that certain numerical values are within acceptable ranges. If not, PFMAT will not allow entry unless Material checking is set to OFF.

The axis of the S-N curve can be modified to report either the stress range or the stress amplitude.

The currently attached or working database may be changed to another database through the Database select option. The three choices are either the central database delivered with the MSC Fatigue system, a local copy of the central database, or a user created database. If the user-created database is specified, a window will appear allowing selection of the user's database.

All of these preferences can be saved locally or globally.

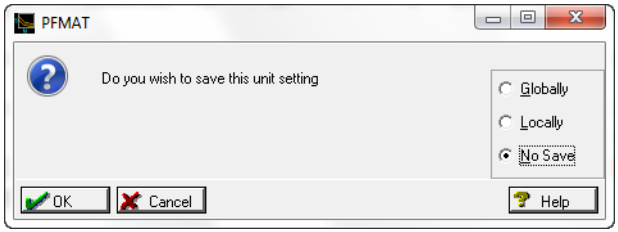


Figure 4-64 Saving the Users Preferences

If saved globally, then each time PFMAT is invoked, these preferences will be retained regardless of the directory in which the program is running. If saved locally, then the settings will only be retained in the local directory in which PFMAT is running. This is done via a `envi.usr` file. There is also a "no save" option which will only remember the setting during the current session. Once PFMAT is exited, the preference will be set back to the previous setting. The user is prompted for this save option each time the preferences are changed.

Important:	To set the preference globally, it may be necessary to set the appropriate protection on the `envi.sys` file delivered with the MSC Fatigue system. Check with the system manager.

Exit Option

The Exit option will exit the materials database manager, PFMAT, and take the user back one level to the MSC Fatigue main menu or return the user to the operating system if PFMAT is being run in stand-alone mode. At this stage, a new ASCII database index file (`pfmat.adb`) will be produced for use by the MSC Fatigue setup forms. This file is saved in the local directory only if there is a local database. Otherwise the file is retained centrally.

Component vs. Material S-N Curves

Material (*Local*) S-N Curves

The S-N curve relates elastic stress, S, to the number of cycles, N, required to cause failure. Typically, nominal stresses are calculated from equations, derived from elastic theory, which relate applied loading force and geometry to stress. Figure together with equations (4-1), (4-2) and (4-3) illustrate such relationships for a circular bar subjected to axial, torsional and bending loads respectively.



Figure 4-65 Axial, Torsional, and Bending Loads on Beam

$$S_{\text{Axial}} = \frac{4P}{\pi d^2} \tag{4-1}$$

$$S_{\text{Torsion}} = \frac{8T}{\pi d^3} \tag{4-2}$$

$$S_{\text{Bend}} = \frac{32M}{\pi d^3} \tag{4-3}$$

The S-N curves generated when the stresses calculated from the above equations are plotted against observed cycles to failure are often referred to a *material* or local elastic S-N curves since they represent the relationship between nominal stresses at the point of failure in a material to life. Such curves can be used for detecting failure locations and estimating lives across an entire finite element model for which appropriate elastic stresses have been calculated. All else being equal, the failure location(s) will correspond to regions of the model exhibiting the highest stresses. Furthermore, the distribution of expected lives can be usefully represented by a contour plot of life.

Component (*Global*) S-N Curves

Where stress-life data have been generated by testing complete components or pseudo-components rather than smooth polished bars of material, the resulting S-N curves are often referred to as *component* S-N curves. These curves can be used to estimate how long the component as a whole will last under cyclic loading with the failure location being defined by the component itself during the cyclic testing process. The component S-N approach is very useful in situations where an accurate description of local stress, either elastic or elastic-plastic, is difficult to achieve such as in the case of welded constructions or composite materials.

Unlike the case of a material S-N curve, where the maximum elastic stress at the failure location is plotted against life, the stress plotted on the stress-axis of a component S-N curve is any nominal value conveniently measured during the fatigue test; the failure location usually being remote from the measurement position.

This situation is illustrated in Figure 4-66 which depicts a through thickness fillet weld joining two plates of varying thickness.

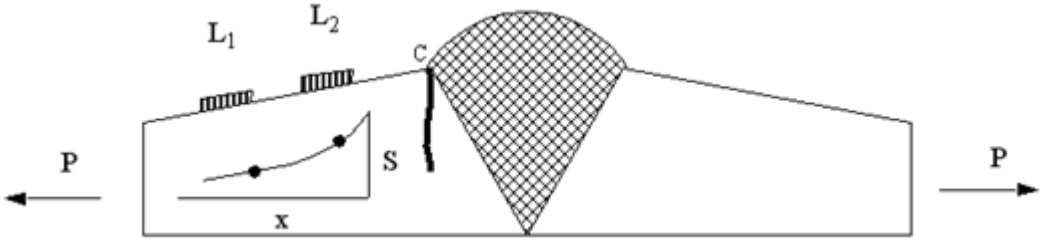


Figure 4-66 Fillet Weld Between Two Plates of Varying Thickness

As a result of the cyclic load, P, the component fails a position C. The nominal stress distribution is detailed and the explicit values of two stresses, S_1 and S_2, at locations L_1 and L_2 respectively, given. If a number of tests were carried out with this component then location specific S-N curves, similar to those given in Figure 4-67, would result. Note that the stresses S_1 and S_2 used to generate these curves are estimated from strain gauges placed at locations L_1 and L_2. It is apparent that the S-N curve is now a function of the location at which the nominal stress was defined, i.e. position L_1 or L_2, and that differing stresses S_1 and S_2 will result in the same life, N, for the component with failure occurring at position C.

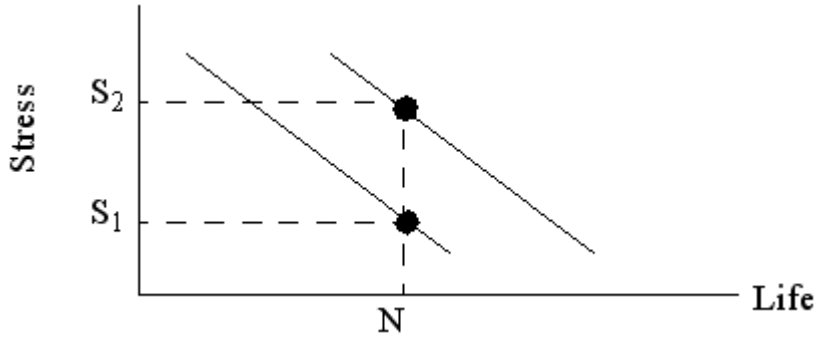


Figure 4-67 Component S-N Curves for Locations L1 and L2

It should be apparent that the role of finite element analysis in the estimation of the fatigue life of a component when using the component S-N approach is not to predict the failure location, C, for example in Figure 4-67, but rather to calculate the magnitude of the appropriate nominal stress, S_1 or S_2 above, used to define the component S-N curve.

A special case of the component S-N curve is the spot-weld S-N curve. Here the stress is the structural stress calculated using the method described in more detail in Spot Weld Analysis Theory, 828.

Reference node/element

The reference node(s) or elements(s) are those which are chosen to represent an estimate of the nominal stress appropriate to any particular component S-N curve, i.e. those giving rise to stresses S_1 and S_2 at locations L_1 and L_2 respectively in Figure 4-67 above. Individual or groups of nodes or elements may be selected as being representative.

Rules for Changing Young's Modulus

It is possible to select a new material for use in the fatigue analysis in either FEFAT or PCRACK to explore the effect on life. However, the user must be careful to ensure that the stress or strain values calculated from the FE analysis still apply. For example, if a steel is changed to an aluminum alloy, the value of E will drop by about a factor of three. This means that the FE results which were calculated for steel are no longer appropriate and need modification to allow for the new Young's Modulus.

There are two scenarios that need to be considered in order to decide which correction method should be used. These are where firstly the load (P) is the constant and secondly where the displacement (U) is fixed. In a test machine, these would be described respectively as "load" and "displacement" controlled tests. The type of loading scenario is easily worked out by considering the loading applied in the FE analysis.

The general equation for linear static analysis is:

$$P = KU \tag{4-4}$$

Where P = force, K = structural stiffness, and U = displacement.

If we rewrite the structural stiffness as a function of geometry and the Young's Modulus then the force becomes:

$$P = f(w)EU \tag{4-5}$$

(i) For load controlled situations,

$$U \propto \frac{P}{E} \tag{4-6}$$

and since the displacements are proportional to the strains,

$$\varepsilon \propto \frac{P}{E}, \sigma \propto E\frac{P}{E} \tag{4-7}$$

If the Young's Modulus (E) changes, then the stresses remain unchanged but the strains are factored by the ratio of E/E' where E' is the new value of Young's Modulus, i.e.,

$$\varepsilon' \propto \left(\frac{E}{E'}\right)\frac{P}{E} \tag{4-8}$$

(ii) For displacement controlled situations,

U is defined and hence the strains are prescribed, and so from $E=\frac{\sigma}{\varepsilon}$.

$$\sigma = \frac{E}{c} \tag{4-9}$$

If the Young's Modulus (E) changes, then the strains remain unchanged but the stresses are factored by the ratio E'/E where E' is the new value of Young's Modulus, i.e,

$$\sigma' \propto \left(\frac{E'}{E}\right) Ec \tag{4-10}$$

Implementation in MSC Fatigue

For a more complete assessment of the effects of a change in the Young's Modulus, a full reanalysis of the FE model will be required. However, a first approximation may be made using FEFAT or PCRACK. To do this properly, the user will need to factor the strains or stresses by the appropriate ratio of Young's Moduli. This scale factor may be applied using the Scale Factor option in the main FEFAT or PCRACK analysis menus. For example, in a load controlled situation if E changes from 210,000 MPa to 70,000 MPa, the stresses will remain the same and the strains will increase by a factor of 3.0. This correction must be applied carefully using the following rules:

1. Total Life Analysis

   Since the analysis is only ever dealing with stresses, the following applies:

load controlled	No change in the stresses and no factor needs to be applied. Simply select the new material.
displacement controlled	Select the new material and apply a factor E'/E using the Scale Factor option.

2. Crack Initiation Analysis

   Since the analysis uses both stresses and strains, the following applies:

load controlled	No change in the stresses, select new materials and apply the factor E'/E using Scale Factor option.
displacement controlled	No Scale Factor required by the user since strains must be kept constant and the new value of E will automatically adjust the stresses.

3. Crack Growth Analysis

   Since the analysis is only ever dealing with stresses, the following applies:

load controlled	No change in the stresses and no factor needs to be applied. Simply select the new material.
displacement controlled	Select the new material and apply a factor E/E' using the Scale Factor option.

4. Spot Weld Analysis.

The analysis uses forces and moments from the FE model. If these are likely to be modified the FE analysis should be rerun. The forces and moments are likely to change if the modulus of the whole structure is not changed, if the component is in displacement or strain control, if the sheet thicknesses are also changed. Otherwise there is no problem.

PFMAT in BATCH Mode

The whole of PFMAT cannot be operated in batch, however, to automate the creation of the materials database, the Create option may be operated from batch. This will be useful for those who have existing material databases in ASCII format who wish to transfer these to MSC Fatigue.

The format for entering materials data in batch is slightly different than other batch operations of MSC Fatigue programs described throughout the guide. The batch file command line points to a file which contains the list of keywords, material parameters, and values required to describe the particular materials. The command below references the ASCII file `mymat.mat` as an example.

```
pfmat @mymat.mat
```

Note that there should be NO space between the @ and the file name.

The format of the lines in this ASCII file is:

```
/<keyword>=<value>
```

These keywords are described below. There are certain keywords which are mandatory and others which are optional. In addition, there are groups of keywords which must either all be included or all excluded. These are indicated. Also, only the capitalized portions of the keyword need be input. An example is given for each keyword and an example input file is shown at the end of this section. Note that no spaces are allowed between the equal signs.

Keyword	Desription
/OPTion=	Batch operation is restricted to the Create and eXit options of PFMAT. This keyword is mandatory and must appear as the first and last lines in the file. (/OPT=CREATE, /OPT=EXIT)
/INDB=	This is mandatory. It means, put the material IN the DAtabase. (/INDB=YES)
/MATNO=	This specifies whether loading material data into material dataset number 1 or 2. This keyword is optional. (/MATNO=1)
/PASSword=	If the user wishes to load the data into the central database, he must supply the password. This keyword is optional and a local database must be present if it is not supplied. (/PASS=PFATIGUE)
/PRImary=	This is the primary name the user is giving to the material he is loading. This keyword is mandatory. (/PRI=MYMAT)

Keyword	Desription
/SECondary	This is the secondary name the user wishes to give to the material he is loading. He may specify up to 9 secondary or alternative names by using this keyword repeated times. This keyword is optional. (/SEC=MEGALLOY_1_2_3)
/REFerence=	It is possible to specify a reference for the material. This is a character string up to 64 characters long. This keyword is optional. (/REF=FROM SOME HANDBOOK)
/COMment	It is possible to also specify two comment lines using this option. The first is a character string up to 64 characters in length. The second is a character string up to 32 characters in length. To specify two comment lines, use this keyword twice. This keyword is optional. (/COM=HIGH STRENGTH STEEL)

The following keywords define the generic properties of the material. Of these, all are optional except /TYPE, /UTS, and /E1 which are required.

Keyword	Description
/TYPE=	This is the material type code. This keyword is mandatory. However, it can be left blank. It is suggested that either 0 or 99 is used if it is not known to which family of materials it should be grouped. A listing of all current material type codes. (/TYPE=99) is given in Table 4-2
/YS=	This defines the yield strength of the material in stress units. (/YS=324)
/UTS=	This defines the ultimate tensile strength of the material in stress units. (/UTS=552)
/E1=	This defines the Young's modulus of the material in stress units. (/E1=2.034E5)
/K1C=	This defines the fracture toughness of the material in units of MPa*m$^{1/2}$ or KSI*inch$^{1/2}$. (/K1C=121)
/N1=	This defines the work hardening exponent. (/N1=0.21)
/K2=	This defines the work hardening coefficient in units of stress. (/K2=965)
/ME=	This defines the elastic Poisson's ratio. (/ME=0.3)
/MP=	This defines the plastic Poisson's ratio. (/MP=0.5)

The next set of keywords define the strain-life (E-N) dataset. Either all must be present or none when defining strain-life data.

Keyword	Description
/ESF=	This defines the fatigue strength coefficient in units of stress. (/ESF=917)
/EB=	This defines the fatigue strength exponent. (/EB=-0.095)
/EC=	This defines the fatigue ductility coefficient. (/EC=-0.47)
/EF=	This defines the fatigue ductility exponent. (/EF=0.26)
/EN1=	This defines the cyclic strain hardening exponent. (/EN1=0.19)
/EK=	This defines the cyclic strength coefficient in stress unit. (/EK=1103)
/ENC=	This defines the cut-off in reversal. The default is 2E8 reversals. (/ENC=2E8)
/EESE=	This defines the standard error of log (N) (Elastic). The default is zero. (/EESE=0)
/EPSE=	This defines the standard error of log(N) (Plastic). The default is zero. (/EPSE=0)
/ECSE=	This defines the standard error of log(N) (Cyclic). The default is zero. (/ECSE=0)

The next set of keywords define the stress-life (S-N) dataset. Either all must be present or none when defining stress-life data.

Keyword	Description
/SNT=	This defines the S-N type, either Material, Component, or Spot Weld. (/SNT=M)
/SRI=	This defines the stress range intercept in stress units. (/SRI=8948)
/B1	This defines the first fatigue strength exponent. (/B1=-0.2)
/NC1=	This defines the fatigue transition point in cycles. (/NC1=2E8)
/B2	This defines the second fatigue strength exponent. (/B2=-0.2)
/SFL=	This defines the stress range fatigue limit in stress units. (/SFL=10)
/SMSS=	This defines the mean stress sensitivity factor used with Spot Weld. (/SMSS=0.1)
/SE=	This defines the standard error of log (N). (/SE=0.137)
/RRAT=	This defines the R-Ratio of test. (/RRAT=-1)

The next set of keywords define the fracture mechanics (LEFM) dataset. Either all must be present or none when defining fracture mechanics data. This group of keywords may be repeated for as many environments as are being defined.

Keyword	Description
/ENV=	This is the name of the environment under consideration. It is a character string up to 20 characters in length. (/ENV=AIR)
/FL=	This defines the unnotched fatigue strength in stress units. (/FL=226)
/C	This defines the Paris law coefficient in units of meters/cycle or inches/cycle. (/C=3E-12)
/M=	This defines the Paris law exponent. (/M=3.43)
/D0	This defines the delta K threshold at R=0 in units of MPa*$m^{1/2}$ or KSI*$inch^{1/2}$. (/D0=8)
/D1=	This defines the delta K threshold at R=1 in units of MPa*$m^{1/2}$ or KSI*$inch^{1/2}$. (/D1=2)
/RC=	This defines the stress ratio at threshold knee. (/RC=0.75)

Example:

The following batch section shows how a full set of materials data can be entered from a batch environment.

The command below points to the ASCII file `example.mat`.

```
pfmat @example.mat
```

This ASCII file contains the following keywords, materials parameters and values to specify the material MYMAT, and is listed below. First, however, the following notes should be made:

1. It is necessary to create a local database, either by manually editing or creating a material but not giving the password or by using the utility under the Create option to initialize a new empty database. The batch load will fail if there is not a local database already created.
2. The units used must be those expected by PFMAT. Check before the batch load that the units preference has been set properly and is what is expected. See the Preferences Option, 167 for details.
3. If entering data values outside of the acceptable, preset ranges of PFMAT, Material checking must be set to OFF. Check before batch load operation that this is set properly. See the Preferences Option, 167 for details.
4. If there are problems loading or the data loaded is not what was expected, check the `pfatigue.prt` files for a description of the batch load operation.

The contents of `example.mat` are:

```
/OPT=CREATE
/INDB=YES
/PASS=
/PRI=MYMAT
/SEC=first alternative
/SEC=second alternative
/REF=Data from somewhere
/COM=Example for manual
/TYPE=99
```

```
/UTS=552
/E1=2.034E5
/YS=324
/K1C=121
/N1=0.21
/K2=965
/ME=.3
/MP=.5
/ESF=917
/EB=-0.095
/EC=-0.47
/EF=.26
/EN1=.19
/EK=1103
/ENC=2E8
/EESE=0
/EPSE=0
/ECSE=0
/SNT=M
/SRI=1102
/B1=0
/NC1=1E2
/B2=-.0602
/SFL=0
/SE=0
/RRAT=-1
/ENV=AIR
/FL=226
/C=3E-12
/M=3.43
/D0=8
/D1=2
/RC=.75
/K1S=121
/ENV=WATER
/FL=200
/C=2E-12
/M=2.43
/D0=7
/D1=3
/RC=.85
/K1SCC=100
/OPT=EX
```

PFMAT Material Listing

Table 4-2 shows PFMAT material classes. Table 4-3 lists all materials that are delivered with the MSC Fatigue system and available datasets. Table 4-4 lists all materials delivered with the MSC Fatigue system and any alternative names by which they may be known.

Table 4-2 Material Type Numbers and Descriptions

0	Type undefined
1	Flake cast iron (FCI)
2	Ferritic cast iron with compacted graphite (FCICG)
3	Pearlitic cast iron with compacted graphite (PCICG)
4	Bainitic cast iron with compacted graphite (BCICG)
5	Ferritic cast iron with spheroidal graphite (FCISG)
6	Ferrite/pearlite cast iron with spheroidal graphite (FPCISG)
7	Pearlitic cast iron with spheroidal graphite (PCISG)
8	Bainitic cast iron with spheroidal graphite (BCISG)
9	Cast steel with less than 0.2% carbon (CSL2C)
10	Normalized cast steel with 0.2-0.4% carbon (NCS24C)
11	Quenched & tempered cast steel with 0.2-0.4% carbon (QTCS24)
12	Normalized cast steel with 0.4-0.7% carbon (NCS47)
13	Plain carbon wrought steel with < 0.2% carbon (PCWS)
14	Hot rolled/normalized plain carbon wrought steel, 0.2-0.4% carbon (HNPCWS24)
15	Quenched & tempered cast steel with 0.4-0.7% carbon (QTCS47)
16	Quenched & tempered plain carbon wrought steel, 0.2-0.4% carbon (QTPCWS24)
17	Hot rolled/normalized plain carbon wrought steel, 0.4-0.7% carbon (HNPCWS47)
18	Quenched & tempered plain carbon wrought steel, 0.4-0.7% carbon (QTPCWS47)
19	Normalized low alloy wrought steel (NLAWS)
20	Quenched & tempered low alloy wrought steel (QTHSLAWS)
21	Normalized Ni/Cr/Mo wrought steel (NNCMWS)
22	Quenched & tempered Ni/Cr/Mo wrought steel (QTNCMWS)
23	Austenitic stainless steel (ASS)
24	Ferritic stainless steel (FSS)
25	Martensitic stainless steel (MSS)
26	Annealed plain carbon wrought steel, 0.2-0.4% carbon (APCWS24)
27	Annealed plain carbon wrought steel, 0.4-0.7% carbon (APCWS47)
28	Normalized carbon/manganese steel (MCMS)
29	Quenched and tempered carbon/manganese steel (QTCMS)
30	Hardened chromium steel (HCS)
31	Quenched and tempered chromium steel (QTCS)

Table 4-2 Material Type Numbers and Descriptions

99	Steel of unknown heat treatment (STEEL)
100	Wrought aluminium (WA)
101	Wrought aluminium-copper alloy (WACA)
102	Wrought aluminium-manganese alloy (WAMNA)
103	Wrought aluminium-magnesium alloy (WAMGA)
104	Wrought aluminium-magnesium-silicon alloy (WAMGSA)
105	Wrought aluminium-zinc alloy (WAZA)
106	Cast aluminium alloy (CAA)
107	Wrought complex special purpose aluminum alloys (WCSPAA)
200	Wrought copper (WCU)
201	Wrought brass (WBR)
202	Wrought aluminium bronze (WABR)
203	Cupronickel (CUPNI)
204	Nickel silver (NIAG)
205	Wrought phosphor bronze (WPHBR)
206	Wrought copper beryllium (WCUBE)
207	Cast copper alloys (CCUA)
300	Titanium alloy (TA)
400	Wrought magnesium alloys (WMGA)
401	Cast magnesium alloys (CMGA)
500	Fusible alloys, solders (FUSSOL)
600	Cast zinc alloys (CZINCA)
700	Wrought nickel alloys (WNIA)
701	Cast nickel alloys (CNIA)
800	Precious metals (PRECMET)
900	Clad materials (CLADMAT)
1000	Thermoplastics (THERPLAS)
1001	Thermosetting plastics (TSETPLAS)

Table 4-3 MSC Fatigue Material Listing (MPa)

Name		E	UTS	Data types		
150M19		2.07E5	682		E-N	
2.25Cr1Mo		2.3E5	603			LEFM
2014-T6_125_HF		7.27E4	483		E-N	
2014_HV_0		7.17E4	200	M.S-N		
2014_HV_T4		7.17E4	410	M.S-N		
2014_HV_T6		7.17E4	470	M.S-N		
2017_HV_T31		7.17E4	300	C.S-N		
2024-T3		7.25E4	460			LEFM
2024_HV_O		7.17E4	200	M.S-N		
2024_HV_T3		7.17E4	450	M.S-N		
2024_HV_T4		7.17E4	410	M.S-N		
2024_HV_T851		7.17E4	410	M.S-N		
2024_HV_T86		7.17E4	410	M.S-N		
2219-T851		7E4	448	LEFM		
2219_HV_T62		7.17E4	320	M.S-N		
2219_HV_T81		7.17E4	410	M.S-N		
2219_HV_T87		7.17E4	470	M.S-N		
2789_370		1.628E5	436		E-N	
2789_420		1.724E5	468		E-N	
2789_600		1.737E5	591		E-N	
2789_700		1.615E5	885		E-N	
2789_800		1.62E5	890		E-N	
2TA11		1.171E5	1233		E-N	
3.5NCMV		2E5	1320			LEFM
3003_HV_H14		7.17E4	200	M.S-N		
3003_HV_H16		7.17E4	200	M.S-N		
3003_HV_H18		7.17E4	220	M.S-N		
3004_HV_H34		7.17E4	215	M.S-N		
3004_HV_H38		7.17E4	295	M.S-N		
3004_HV_O		7.17E4	200	M.S-N		
300M		2.07E5	1900			LEFM

Table 4-3 MSC Fatigue Material Listing (MPa)

Name		E	UTS	Data types		
316		1.9E5	590			LEFM
349S52		1.9E5	991		E-N	
352S52		1.735E5	1027		E-N	
5052-H32		6.96E4	231		E-N	
5052_HV_H34		7.17E4	215	M.S-N		
5052_HV_H38		7.17E4	295	M.S-N		
5052_HV_O		7.17E4	200	M.S-N		
5056_HV_CON		7.17E4	260	C.S-N		
5083_114_CF		6.9E4	414			
5083_87_CF		6.9E4	385		E-N	
526M60		2.02E5	939		E-N	
5454_NONE_CF		6.9E4	334		E-N	
605M30		2E5	705		E-N	
605M36		2.07E5	835		E-N	
6061-T6 80 HF		7.27E4	340		E-N	
6061-T6_NONE_CF		6.9E4	389		E-N	
6061-T6_NONE_SHEET		6.96E4	314		E-N	
6061_HV_O		7.17E4	150	M.S-N		
6061_HV_T4		7.17E4	215	M.S-N		
6061_HV_T6		7.17E4	305	M.S-N		
7075-T6		7.09E4	558			LEFM
7075_HV_O		7.17E4	220	M.S-N		
7075_HV_T6		7.17E4	570	M.S-N		
709M40		2.1E5	781		E-N	
7175-T73_NONE_HF		7.13E4	524		E-N	
722M24		2.05E5	976		E-N	
817M40		2E5	1277		E-N	LEFM
826M31		2E5	1209		E-N	
835M30		2E5	1550			LEFM
835M30_V		1.943E5	1034		E-N	
A533B		2E5	552			LEFM
AISI1012		2E5	333		E-N	

Table 4-3 MSC Fatigue Material Listing (MPa)

Name		E	UTS	Data types		
AISI1020		2E5	416		E-N	
AISI_4340		2E5	1700			LEFM
alphafe		2.1E5	1700		E-N	LEFM
ASTMA536		1.447E5	480		E-N	
B40PK		2E5	394		E-N	
B40PO		2E5	438		E-N	
B50XF		2E5	486		E-N	
B50XK-CR		2E5	461		E-N	
B50XK-HR		2E5	450		E-N	
B55XF		2E5	488		E-N	
B60RO		2E5	503		E-N	
B80RK		2E5	610		E-N	
B80XF		2E5	645		E-N	
Beryllium		2.894E5	323		E-N	
bs1452-260		1.253E5	277		E-N	
BS376_Nickel		2.068E5	366		E-N	
BS4360-43A		2.07E5	486		E-N	
BS4360-43C		2.07E5	478		E-N	
BS4360-43D		2.07E5	490		E-N	
BS4360-50D		1.914E5	480		E-N	LEFM
classB		2.07E5	500	C.S-N		
classC		2.07E5	500	C.S-N		
classD		2.07E5	500	C.S-N		
classE		2.07E5	500	C.S-N		
classF		2.07E5	500	C.S-N		
classF2		2.07E5	500	C.S-N		
classG		2.07E5	500	C.S-N		
classW		2.07E5	500	C.S-N		
Cold_rolled_sheet		2E5	303		E-N	
Copper		1.136E5	206		E-N	
DP1		2E5	659		E-N	
DP2		2E5	753		E-N	

Table 4-3 MSC Fatigue Material Listing (MPa)

Name		E	UTS	Data types		
EIBSG1400		1.75E5	1407		E-N	
EICG315		1.51E5	315		E-N	
EICG400		1.5E5	404		E-N	
EICG493		1.63E5	493		E-N	
EN24V		1.902E5	1047		E-N	
EZ33A_HV_T5		4.4E4	140	M.S-N		
FeE255TM		2E5	475		E-N	
FeE37D		2E5	388		E-N	
FeE420TM		2E5	490		E-N	
FeE52D		2E5	550		E-N	
HSLA4		2E5	486		E-N	
HT-30		7.1E4	355		E-N	
HY130		2E5	1010			LEFM
HY80		2E5	735			LEFM
HYBRID_CASTIRON		1.51E5	296		E-N	
hypress20		2E5	445		E-N	
hypress23		2E5	437		E-N	
hypress26		2E5	523		E-N	
hypress29-4		2E5	544		E-N	
hypress29-8		2E5	539		E-N	
IMI685		1.2E5	955			LEFM
INC718		2.041E5	1304		E-N	
MANTEN		2.034E5	552		E-N	LEFM
MANTEN_MSN		2.034E5	600	M.S-N		
MANTEN_SN		2.034E5	600	C.S-N		
Mild_Steel		2E5	462		E-N	
Nitro		2E5	483		E-N	
Nitro-sa		2E5	648		E-N	
Rephos		2E5	421		E-N	
RQC100		2.034E5	863		E-N	LEFM
RQC100_MSN		2.034E5	800	M.S-N		
RQC100_SN		2.034E5	800	C.S-N		

Table 4-3 MSC Fatigue Material Listing (MPa)

Name		E	UTS	Data types		
RQT501		2E5	590		E-N	LEFM
RQT701		2E5	825		E-N	LEFM
RR58		7.5E4	450			LEFM
SAE1006_85A_HR		2.07E5	318		E-N	
SAE1006_85B_HR		2.07E5	318		E-N	
SAE1006_85_HR		2.07E5	318		E-N	
SAE1008_91_HR		2.07E5	363		E-N	
SAE1015_80_NORM		2.07E5	415		E-N	
SAE1018_106_HR		2.07E5	354		E-N	
SAE1018_118_QT		2.07E5	496		E-N	
SAE1018_209_QT		2.07E5	696		E-N	
SAE1020_107_HR		2.07E5	441		E-N	
SAE1020_108_ANLD		2.07E5	392		E-N	
SAE1030_128A_HR		2.07E5	454		E-N	
SAE1030_128_HR		2.07E5	454		E-N	
SAE1035_169_CON		2.1E5	550	M.S-N		
SAE1045_225_ANLD		2.07E5	751		E-N	
SAE1045_390_QT		2.07E5	1343		E-N	
SAE1045_450_QT		2.07E5	1584		E-N	
SAE1045_500_QT		2.07E5	1956		E-N	
SAE1045_595_QT		2.07E5	2239		E-N	
SAE1045_705_QT		2.07E5	2067		E-N	
SAE1045_HV_HR		2.07E5	671		E-N	
SAE1050_189_CON		2.1E5	637	M.S-N		
SAE1055_251_CON		2.1E5	860	M.S-N		
SAE1080_371_QT		2.07E5	1298		E-N	
SAE1080_410_QT		2.07E5	1432		E-N	
SAE1080_421_AUST		2.07E5	1349		E-N	
SAE1315_155_CON		2.1E5	530	M.S-N		
SAE1522_289_HR		2.07E5	1005		E-N	
SAE1522_304_HR		2.07E5	1088		E-N	
SAE1541_362_QT		2.07E5	1200		E-N	

Table 4-3 MSC Fatigue Material Listing (MPa)

Name		E	UTS	Data types		
SAE1561_234_HR		2.07E5	836		E-N	
SAE2310_138_CON		2.1E5	480	M.S-N		
SAE2335_217_CON		2.1E5	745	M.S-N		
SAE30304		1.87E5	670			LEFM
SAE4130_259_QT		2.07E5	895		E-N	
SAE4130_267_CON		2.1E5	912	M.S-N		
SAE4130_366_QT		2.07E5	1426		E-N	
SAE4142_380_QT		2.07E5	1412		E-N	
SAE4142_400_QT		2.07E5	1550		E-N	
SAE4142_450A_QT		2.07E5	1929		E-N	
SAE4142_450_QT		2.07E5	1757		E-N	
SAE4142_475A_QT		2.07E5	2032		E-N	
SAE4142_475_QT		2.07E5	1929		E-N	
SAE4142_560_QT		2.07E5	2239		E-N	
SAE4142_670_QT		2.07E5	2446		E-N	
SAE4340_242_HR		2.07E5	826		E-N	
SAE4340_350A_QT		2.07E5	1171		E-N	
SAE4340_350B_QT		2.07E5	1171		E-N	
SAE4340_350C_QT		2.07E5	1240		E-N	
SAE4340_409_QT		2.07E5	1467		E-N	
SAE5160_434_QT		2.07E5	1584		E-N	
SAE52100_517_H		2.07E5	2011		E-N	
SAE8630_254_NORM		2.07E5	785		E-N	
SAE8640_361_QT		2.07E5	1373		E-N	
SAE9262_260_NORM		2.07E5	923		E-N	
SAE9262_271_QT		2.07E5	999		E-N	
sra_60		2E5	531		E-N	
sra_70		2E5	570		E-N	
st00		2.1E5	347		E-N	
Ti-6Al-4V		1.2E5	986			LEFM
unsg10200		2E5	393		E-N	
1.4003		2E5	496	Sp.Wld		

Table 4-3 MSC Fatigue Material Listing (MPa)

Name		E	UTS	Data types		
1.4301_IIIA		1.875E5	670	Sp.Wld		
1.4301_IIIC		2E5	670	Sp.Wld		
1.4589		2E5	523	Sp.Wld		
AlMg5Mn		7E4	300	Sp.Wld		
FePo4		2E5	313	Sp.Wld		
spot_nugget_generic		2.1E5	500	Sp.Wld		
spot_sheet_generic		2.1E5	500	Sp.Wld		
ZSTE380		2E5	484	Sp.Wld.		

Table 4-4 MSC Fatigue Material Alternative Names

Table 4-5 Steels

SAE (USA)	DIN(German)	W.Nr.(German)	British
SAE1006_85A_HR	D8-2	1.0313	040A04,En2A
SAE1006_85B_HR	D8-2	1.0313	040A04,En2A
SAE1006_85_HR	D8-2	1.0313	040A04,En2A
SAE1008_91_HR	St13	1.0333	050A04
SAE1015_80_NORM	C15	1.0401	050A15
SAE1018_106_HR	-	-	080A17
SAE1018_118_QT	-	-	080A17
SAE1018_209_QT	-	-	080A17
SAE1020_107_HR	C22	1.0402	070M20,En3
SAE1020_108_ANLD	C22	1.0402	070M20,En3
SAE1030_128A_HR	-	-	080A30,En5B
SAE1030_128_HR	-	-	080A30,En5B
SAE1035_169_CON	Cm35	1.1180	060A35
SAE1045_225_ANLD	Ck45	1.1191	060A45
SAE1045_390_QT	Ck45	1.1191	060A45
SAE1045_450_QT	Ck45	1.1191	060A45
SAE1045_500_QT	Ck45	1.1191	060A45
SAE1045_595_QT	Ck45	1.1191	060A45
SAE1045_705_QT	Ck45	1.1191	060A45

Table 4-4 MSC Fatigue Material Alternative Names

SAE1045_HV_HR	Ck45	1.1191	060A45
SAE1050_189_CON	C53	1.210	060A52
SAE1055_251_CON	C55	1.0535	070M55,En9
SAE1080_371_QT	-	-	060A81
SAE1080_410_QT	-	-	060A81
SAE1080_421_AUST	-	-	060A81
SAE1315_155_CON	-	-	-
SAE1522_289_HR	20Mn5	1.1133	120M19
SAE1522_304_HR	20Mn5	1.1133	120M19
SAE1541_362_QT	36Mn5	1.1167	150M36,En15B
SAE1561_234_HR	-	-	-
SAE2310_138_CON	-	-	708A30
SAE2335_217_CON	-	-	-
SAE30304	X5CrNi18_9	1.4301	304S16,En58E
SAE4130_259_QT	-	-	708A30
SAE4130_366_QT	-	-	708A30
SAE4130_267_CON	-	-	-
SAE4142_380_QT	42CrMo4	1.7225	708A42,En19C
SAE4120_400_QT	42CrMo4	1.7225	708A42,En19C
SAE4142_450A_QT	42CrMo4	1.7225	708A42,En19C
SAE4142_450_QT	42CrMo4	1.7225	708A42,En19C
SAE4142_475A_QT	42CrMo4	1.7225	708A42,En19C
SAE4142_475_QT	42CrMo4	1.7225	708A42,En19C
SAE4142_560_QT	42CrMo4	1.7225	708A42,En19C
SAE4340_HV_NONE	-	-	-
SAE4340_242_HR	40CrNiMo6	1.6565	817M40,En24
SAE4340_350A_QT	40CrNiMo6	1.6565	817M40,En24
SAE4340_350B_QT	40CrNiMo6	1.6565	817M40,En24
SAE4340_350C_QT	40CrNiMo6	1.6565	817M40,En24
SAE4340_409_QT	40CrNiMo6	1.6565	817M40,En24
AISI4340M_HV-NONE	-	-	-
SAE5160_434_QT	55Cr3	1.7176	527A60,En48
SAE52100_517_H	100Cr6	1.3505	539A99,En31

Table 4-4 MSC Fatigue Material Alternative Names

SAE8630_254_NORM	30NiCrMo2_2	1.6545	-
SAE8640_361_QT	40NiCrMo2_2	1.6546	-
SAE9262_260_NORM	60SiCr7	1.7108	-
SAE9262_271_QT	60SiCr7	1.7108	-
-	-	-	835M30,En30B
-	St52	-	BS4360-50D
ASTM A542 Class 2,3	2.25Cr1Mo	-	-
-	3.5NiCrMoV	-	-
316_S/S	-	-	-
A553B	-	-	-
HY80	-	-	-
HY130	-	-	-
MANTEN	-	-	-
MANTEN_SN	-	-	-
NAMTEN_MSN	-	-	-
RQC100	-	-	-
RQC100_SN	-	-	-
RQC100_MSN	-	-	-
RQT501	-	-	-
RQT701	-	-	-
Aluminum Alloys and Other Light Alloys			
SAE	**DIN**	**W.Nr.**	**British**
2025_T3	-	-	-
2219_T851	-	-	-
5056_HV_CON	Al_Mg5	3.3555	-
2014-T6_125_HF	-	-	-
6061-T6_80_HF	-	-	-
5052-H32	-	-	-
6061-T6_NONE_SHEET	-	-	-
6061-T6-NONE_CF	-	-	-
2014_HV_O	Al_Cu4_Si_Mg	3.1255	-
2014_HV_T4	Al_Cu4_Si_Mg	3.1255	-

Table 4-4 MSC Fatigue Material Alternative Names

2014_HV_T6	Al_Cu4_Si_Mg	3.1255	-
2017_HV_T31	Al_Cu4_Si_Mg	3.1355	-
2024_HV_O	Al_Cu_Mg2	3.1355	-
2024_HV_T3	Al_Cu_Mg2	3.1355	-
2024_HV_T851	Al_Cu_Mg2	3.1355	-
2024_HV_T4	Al_Cu_Mg2	3.1355	-
2024_HV_T86	Al_Cu_Mg2	3.1355	-
2219_HV_T87	-	-	-
2219_HV_T81	-	-	-
2219_HV_T62	-	-	-
3003_HV_H18	-	-	-
3003_HV_H16	-	-	-
3003_HV_H14	-	-	-
3004_HV_H38	-	-	-
3004_HV_H34	-	-	-
3004_HV_O	-	-	-
5052_HV_H38	-	-	-
5052_HV_H34	-	-	-
5052_HV_O	-	-	-
5083_114_CF	-	-	-
5083_87_CF	-	-	-
5454_NONE_CF	-	-	-
6061_HV_O	-	-	-
6061_HV_T4	-	-	-
6061_HV_T6	-	-	-
7075_HV_O	Al_Zn_Mg_Cu1.5	-	-
7075_HV_T6	Al_Zn_Mg_Cu1.5	-	-
7175-T73_NONE_HF	-	-	-
HT30	-	-	-
RR58	-	-	-
EZ33A_HV_T5	-	-	-
TI-6Al-4V	-	-	-
IMI685	-	-	-

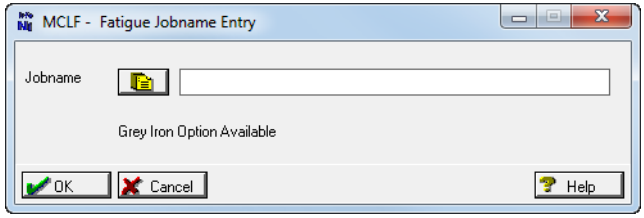
Table 4-4 MSC Fatigue Material Alternative Names

Weld Geometries			
BSB5400:CLASS B			
BSB5400:CLASS C			
BSB5400:CLASS D			
BSB5400:CLASS E			
BSB5400:CLASS F			
BSB5400:CLASS F2			
BSB5400:CLASS G			
BSB5400:CLASS W			

Important: More materials are added from time-to-time. Use the Search and List Option, 141 with the letters "D", "W", and "B" as entries for the Name keyword to see a listing of all materials with DIN, W.Nr. or British standard names.

Grey Cast Iron

The single location strain-life (e-N) module, mclf supports fatigue life and damage anlysis for Grey Cast Irons.



The material properties for Grey Cast Irons must be stored in an ASCII file with extension .GIM in the local or central directory. The file must adhere to the following format.

<keyword> = value

NT, Tensile strain hardening exponent

KT, Tensile strain hardening coefficient (Note: this is not to be confused with stress concentration factor)

MT, Tensile secant slope

NC, Compressive strain hardening exponent

KC, Compressive strain hardening coefficient

MC, Compressive secant slope

MU, Unloading secant slope

E0, Tangent modulus at zero

KS, Damage curve coefficient

NS, Damage curve exponent

CUTOFF, Damage curve cutoff (Reversals)

SCATTER, Standard error of log life

A sample .GIM file (gray.gim) is available in the central materials directory

Characterization of Material Properties for grey cast iron

A series of fully reversed strain controlled fatigue tests are carried out at a number of different strain amplitudes. The procedure is the same as for conventional tests. Stress/strain hysteresis loops are captured at predefined rates. (Typically 2n , and also when load drops by specific amounts, 5% increments say). The life at each strain amplitude is noted and the parameters defined above are determined.

The Tensile Cyclic Stress Strain Curve

The tensile stress strain curve is defined as

$$\varepsilon_t = \frac{\sigma}{E_0 + m_t \sigma} + \left(\frac{\sigma}{K_t}\right)^{1/n_t}$$

The total strain amplitude is the sum of secant strain, ε_s and the residual non-elastic strain, ε_r i.e.

$\varepsilon_t = \varepsilon_s + \varepsilon_r$

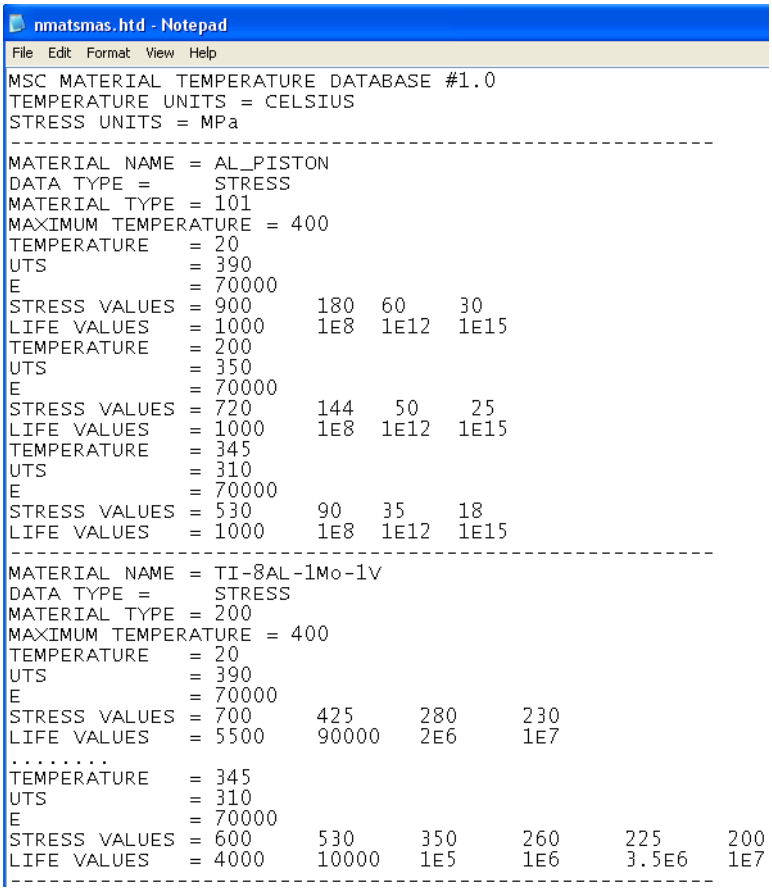
The unloading modulus Eu is related to the maximum tensile stress as

$$E_u = M_u \sigma_{max} + E_0$$

For each strain amplitude, the unloading modulus is plotted against peak tensile stress. M_u and intercept E_0 are obtained by linear regression.

Materials with Temperature Dependent Properties

MSC Fatigue can calculate fatigue damage and life for materials with temperature dependent properties, using an ASCII materials database (nmats.htd). Users can edit and add their own materials data, by copying nmats.htd from the installation directory to their own run directories. The following printout from the ASCII database shows the properties of one material named AL_Piston.



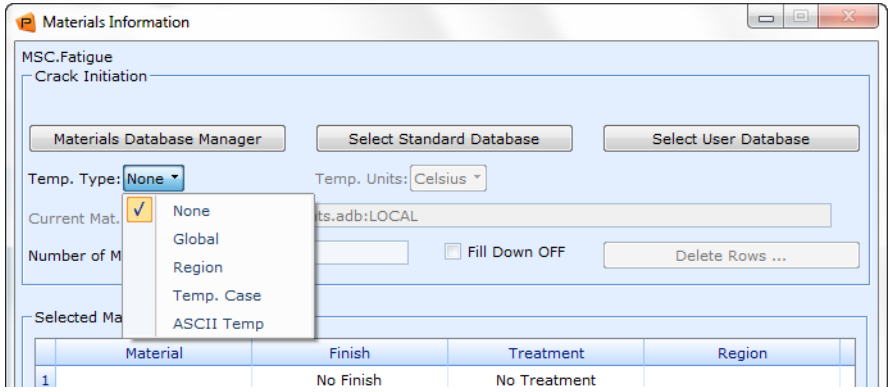
Temperatures may be assigned to the analysis globally, by group (region) or extracted from a Thermal Case in the Patran database.

The temperatures are steady state (i.e. constant with time) and do not include time varying or creep effects, but may vary spatially across the FE model. Currently, temperature corrected fatigue analysis may only be

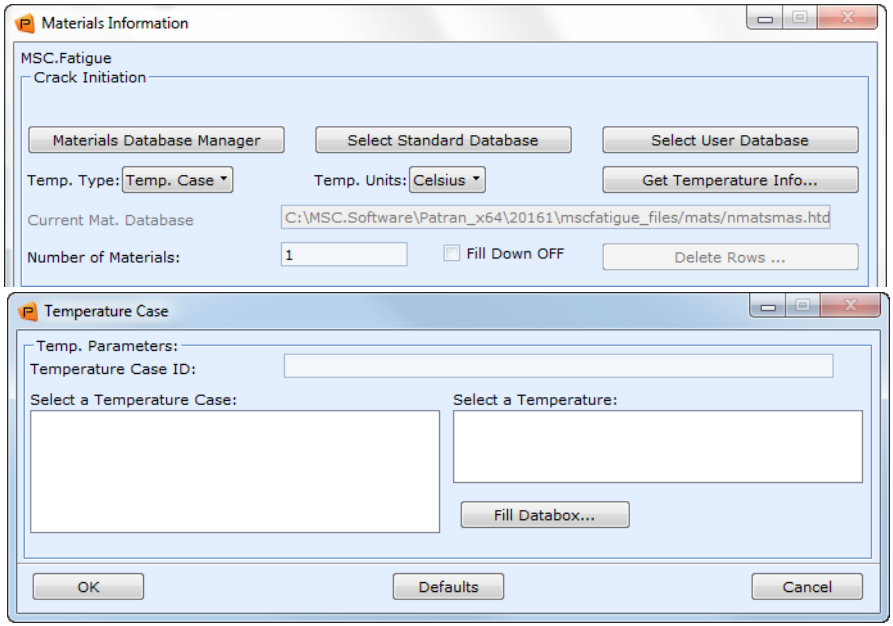
performed with S-N (Stress Life) and E-N (Strain Life) analysis only. Optimization and Fast Analysis options are not currently available.

Material Information

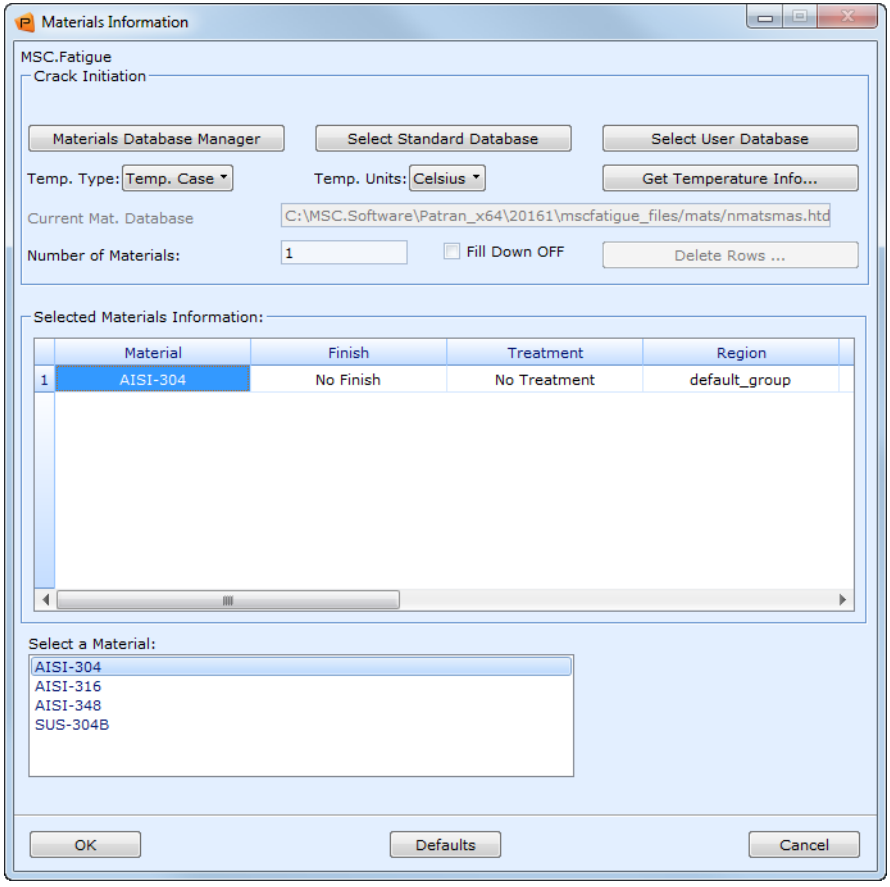
Temperature assignments to the analysis entities are made on the **Material Information** form. There are 4 methods: **Global**, **Region**, **Temp. Case**, or **ASCII Temp** to assign temperatures to the analysis group(s) from the **Temp Type** pull down menu. **Global** assigns the same temperature to all the analysis groups. **Region** allows the user to specify temperatures on the analysis groups on the material form. **Temp. Case** extracts the temperatures from a Thermal analysis case from the Patran database. **ASCII Temp** requests that the user assign a default global temperature and then specify the name of the **TDS** file to use in overwriting the temperature for certain nodes. For **Global**, **Region**, **Temp. Case**, or **ASCII Temp**, an ASCII materials database is used (**nmats.htd**) that is copied form the installation directory to the user's run directory, to facilitate editing and addition of material data. You can click on the **Materials Database Manger** button to view the **.htd** file with a text editor. Just like other analysis types, the list of available materials will be displayed in the list box on the material form. The regular Material database is displayed via **PFMAT** if the default **Temp. Type** of **None** is selected.



If the **Temp. Case** is selected from the **Temp. Type** pull-down menu, a **Get Temperature Case** button appers. Clicking on the **Get Temperature Case** button brings the **Temperature Case** selection form for selecting a temperature case.



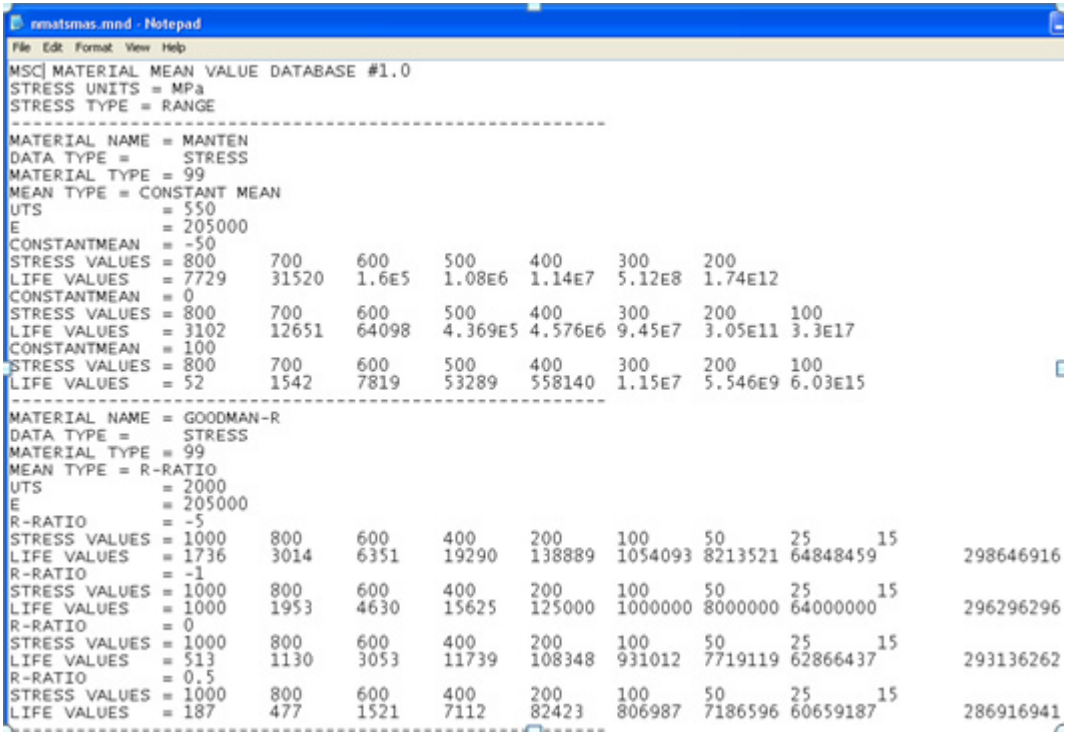
After selecting the temperature case, Users can select a material by clicking the **Material** cell on the **Material Information** form.



Materials with Multiple Mean S-N Curves

Multiple mean stress curve analysis uses empirical data to account for mean stress effects rather than analytical methods such as Gerber and Goodman. MSC Fatigue can calculate fatigue damage and life for materials with multiple mean stress S-Ncurves, using an ASCII materials database (nmats.mnd). Users can edit and add

their own materials data by copying nmats.mnd from the installation directory to their own run directories. The following printout from the ASCII database shows the properties of two materials:



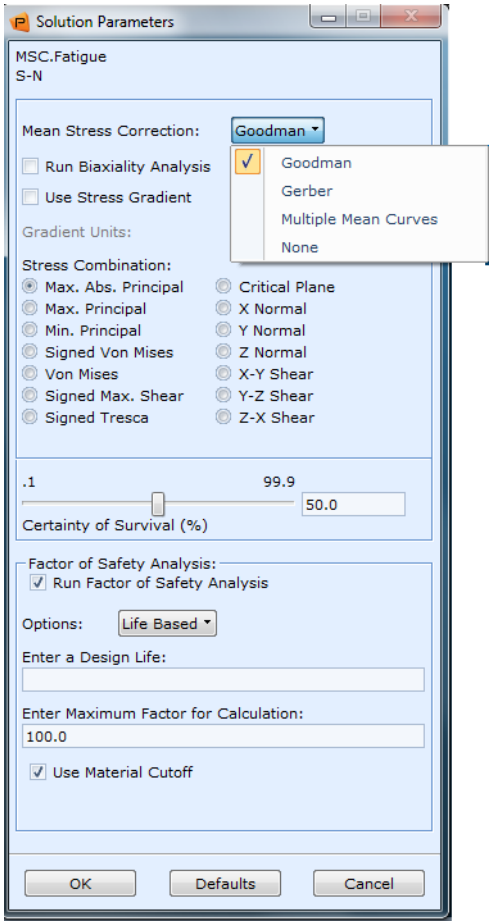
Multiple mean stress curve analysis is for S-N analysis only. Temperature corrections, certainty of survival, Optimization and Fast Analysis are not available for Multiple Mean Stress Curve S-N analysis.

Mean Stress Correction

Set the **Mean Stress Correction** option menu in the **Solution Parameters** to **Multiple Mean Curves.**

Solution Parameters

.



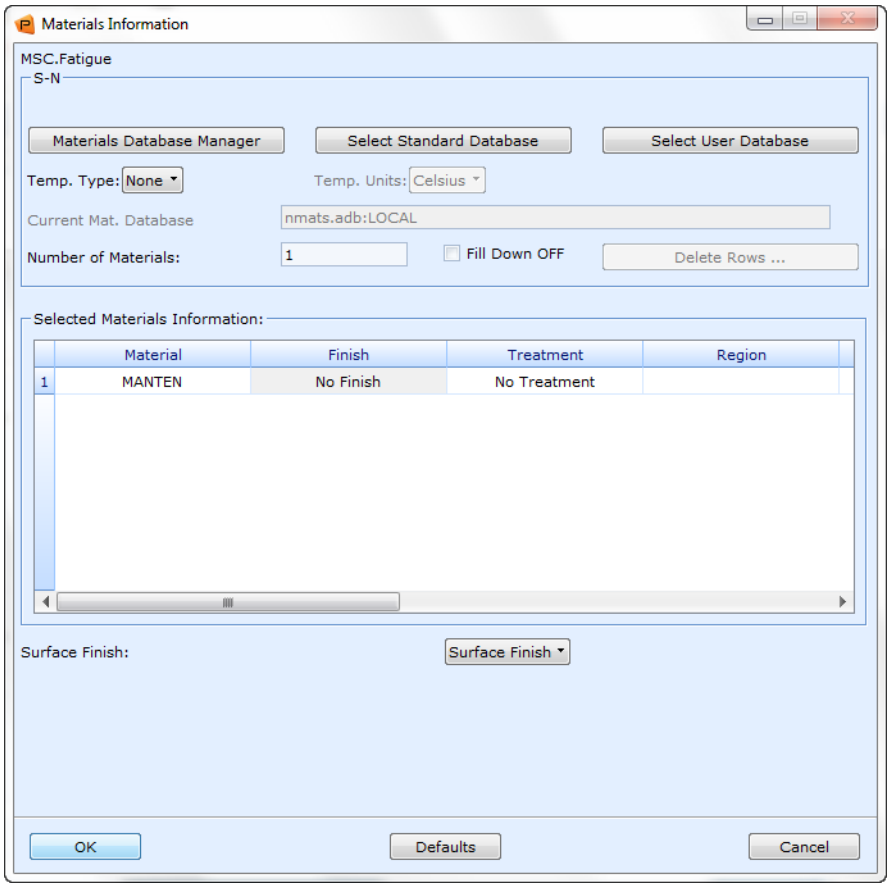
Material Information

The Material Information form is identical to the standard S-N analysis, except:

An ASCII materials database is used (extension .mnd) instead of the standard Materials database (.mdb).

Selecting the Materials Database Manager button brings up a text editor instead of PFMAT.

Clicking on the **Materials Databse Manager** brings the file containing the **Multiple Mean Stress** Curve data.


Materials Information
MSC.Fatigue
S-N
Materials Database Manager
Select Standard Database
Select User Database
Temp. Type: None
Temp. Units: Celsius
Current Mat. Database
nmats.adb:LOCAL
Number of Materials:
1
Fill Down OFF
Delete Rows ...
Selected Materials Information:
Material
Finish
Treatment
Region
1
MANTEN
No Finish
No Treatment
Surface Finish:
Surface Finish
OK
Defaults
Cancel

5 Loading Management

- Automatic Time History Recognition 200
- Multi-Channel Time History Support 201
- PSD And Cross-Spectra Automation Support 202
- Introduction to PTIME 208
- PTIME Menu Options 213
- Chapter 5: Multi-File Display (MMFD)
- Peak-Valley Extraction (MPVXMUL) 254
- Auto Spectral Density (MASD) 260
- PTIME Central Database Listing 273
- DAC File Format Description 274
- Loading and Units 283
- Aerospace Spectrum File Support 286

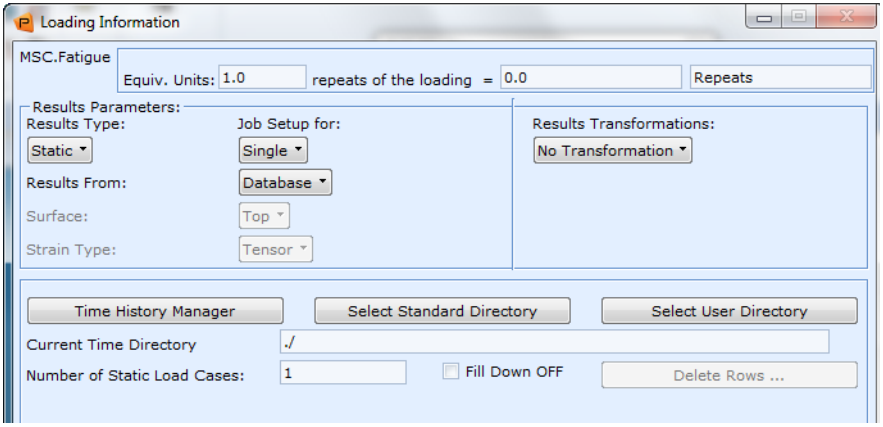
Automatic Time History Recognition

Previously, a process had to be followed in which time history files needed to be loaded into PTIME in order to be recognized by MSC Fatigue. This file management system is still in place when using the **Classic** solver option. However, when using the **DTLib** solver option, the user need only have the required time history files in his working directory or specify a particular directory for time history files. MSC Fatigue automatically scans what is considered as the **Current Time History Directory** for time history files, and it makes these available to the user interface.

When the MSC Fatigue Loading Information form is first opened, a background process is initiated in which:

- A program **tsinfo** is called, which scans the current time history directory and lists the single and multi-channel time history files residing in that directory. For a new MSC Fatigue session, the current time history directory is considered initially as the current working directory. The time history listing is loaded to a file named **tsinfo.txd**, which is created in the currently scanned directory. The list of time history files in tsinfo.txd is the made available on the Loading Information form.
- If no time history files are located in the current working directory, the user home directory is searched. If time history files are located there, this becomes the current time history directory and the list of time history files is made available on the Loading Information form.
- If no time histories are located in the user home directory, the $p3_home/mscfatigue_files/ptime directory becomes the current time history directory since there are time histories located in the directory. The list of time history files is made available on the Loading Information form.
- The current time history directory is listed on the Loading Information form as shown below, as indicated in the **Current Time Directory** text box.
- The above process may be instantaneous or take a few seconds, depending on the number of time history files in the directory being scanned.

The user may also specify a particular directory to use as the current time history directory by clicking the **Select User Directory** button on the Loading Information form, and then navigate to the desired location. If new time history files are copied to the user directory, the directory should be selected again so the directory can be re-scanned.



The user may perform the scanning process at any time by clicking the **Select Standard Directory** button on the Loading Information form. This may be necessary if the user copies new time history files to the current time history directory. Clicking this button causes the process mentioned above to be performed, and the newly copied files are thus recognized.

Although the above process mentions time history files, any PSD or similar loading files are also recognized and made available on the Loading Information form.

Multi-Channel Time History Support

MSC Fatigue previously supported the MTS RPC format in a roundabout fashion, which required conversion to multiple single-channel dac files. MSC Fatigue provides direct support for some of the most popular multi-channel formats (RPC and s3t). This support is only available for the DTLib-based fatigue solver.

The new support has the following features:

- Auto-recognition and interrogation of time history files.

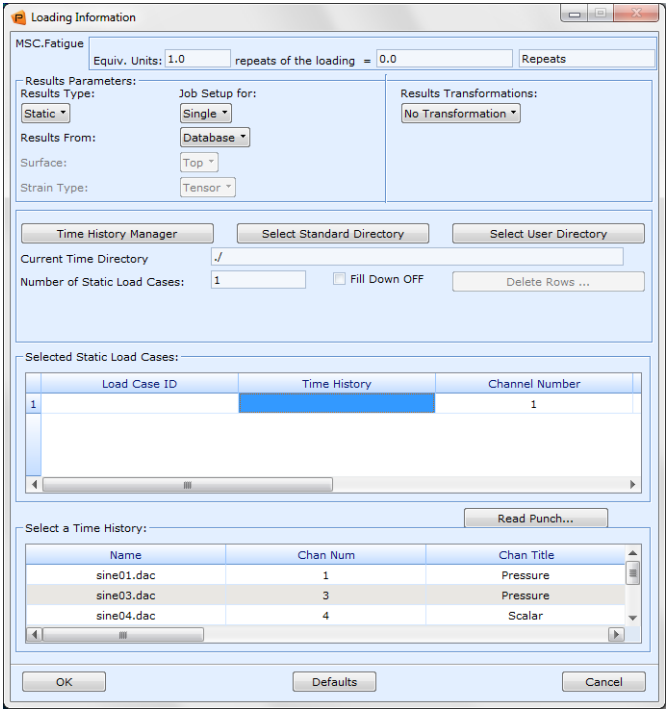
  Time history information is automatically obtained and loaded to MSC Fatigue. There no need to load time history files via ptime. The user working directory is first scanned for supported files, and then if none are found the user home directory or mscfatigue_files/ptime time histories are used.

- Channel ID visible in the MSC Fatigue Loading form.

  For single channel files such as nCode dac, channel ID is 1. Other information such as channel title, units, max, min are also visible on the Loading form.

- Time history information stored in tsfiles.txd.

  Since the ptime cannot recognize multi-channel files, time history information is stored in this new XML format file. The file is not readable by ptime. Multi-channel files cannot be used directly with the Classic fatigue solvers.



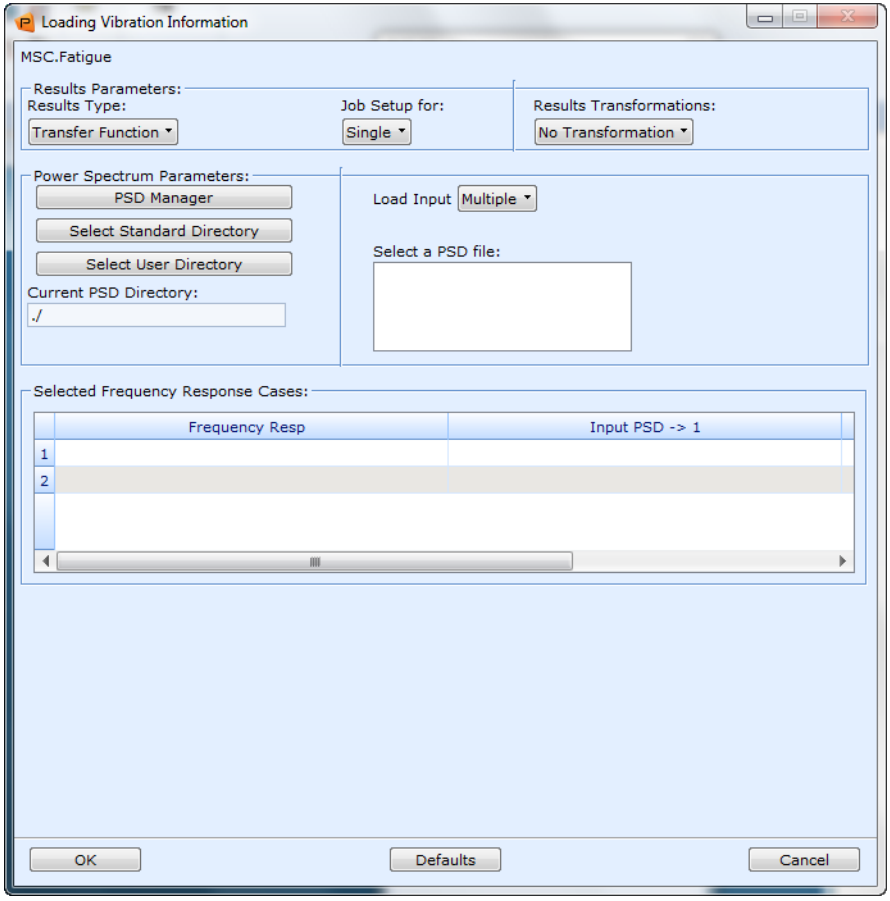
Even though dac's are single-channel files, on the Loading Info form they appear as multi-channel with a post-fix appearing as channel number. Sorting order is alphabetical by file base name and then numerical by post-fix rather than alphabetical-only.

Naming single channel dac time history files as name01, name02, etc., makes the set of time history files appear as a single multi-channel time history. In such a case the post-fix numeral is taken as a channel ID. This can be very advantageous when a large number of time history files is to be associated with a large number of load cases. By naming each time history with a numeral associated with a load case, the **Fill Down** capability can allow the user to associate the time histories with all load cases in a single click.

Multi-channel time history files appear as file name and then channel number for each channel in the time history. By setting the **Fill-Down** button to **On**, all channels from the currently selected channel on may easily be associated with the currently selected load case and those following.

PSD And Cross-Spectra Automation Support

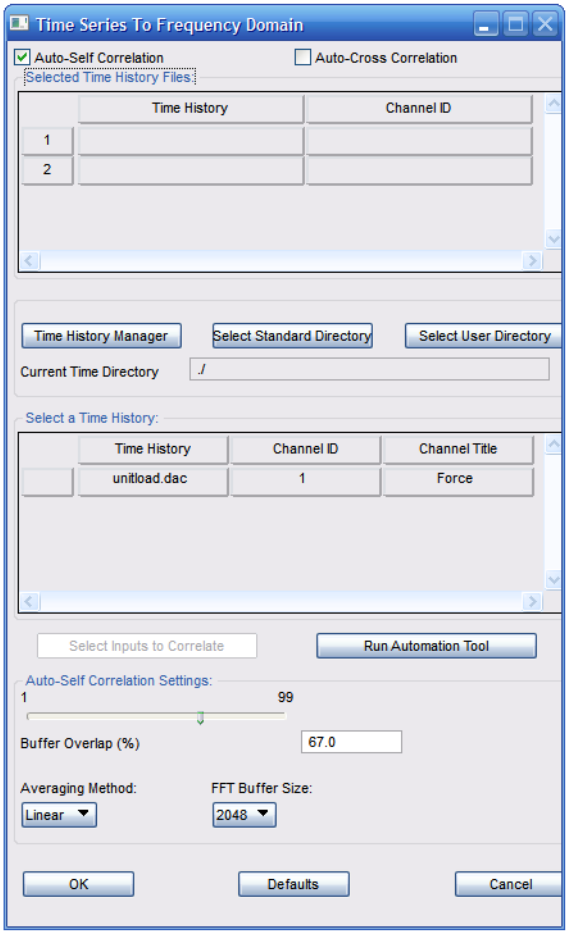
The automation support was mainly intended for use with Multi-Input vibration fatigue, but can be useful for single PSD vibration as well. The tools automatically create PSDs from selected time history files and Cross-Spectra from pairs of selected time history files. The automation also loads the resulting file names to a matrix file for analysis. Access to the automation tools forms is only available by setting Analysis = Vibration on the main form and Loading Method = Multiple PSD on the Solution Parameters form.



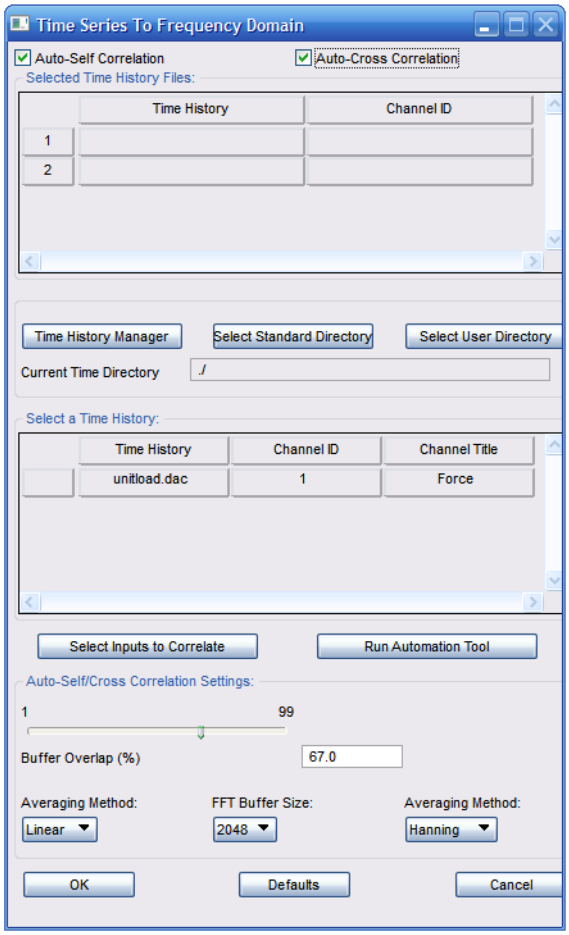
To make the automation tools available for use, set the **Matrix Input** option on the Loading Information form to **From Matrix File**. Then click the **Auto-Create Matrix File** button. This opens the Time Series to Frequency Domain form, also know as the automation form.

On the automation form, there are only as many rows as there are on the main Loading Information form. On this form, the time histories selected can automatically be converted to PSD files. This is the default action.

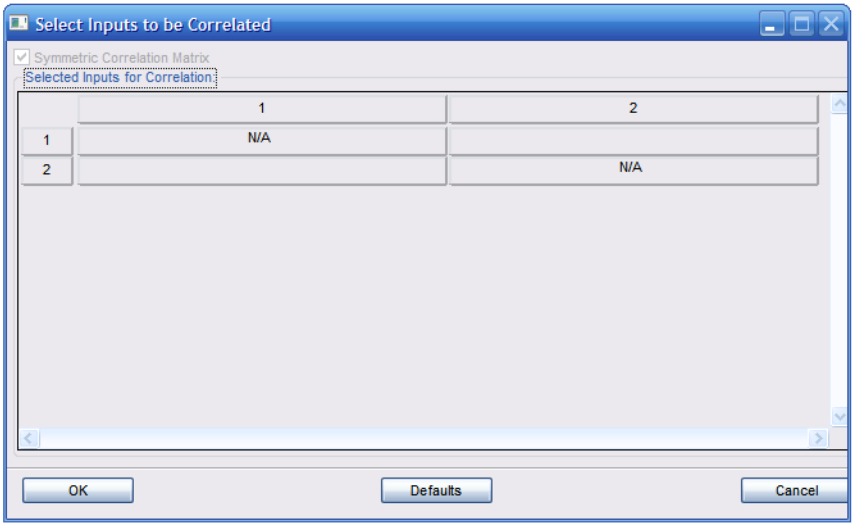
Note that MSC Fatigue automatically checks the **Current Time Directory** for the appropriate file type and makes them available for selection. If any time history files are available in the current working directory, this becomes the **Current Time Directory**. If some other directory contains the required time histories, select it using **Select User Directory** and selecting any time history in that directory. Note however, that the automation process uses the current working directory and all resulting files are created in the current working directory. For example, note that if the **Current Time Directory** is set to `$p3_home/mscfatigue_files/examples`, the selected time histories are copied to the current working directory for use in the automation process.



If cross-correlation on some or all the time histories is required, this is also supported. By clicking the **Auto-Cross Correlation** check-box at the top of the form, a button is activated named **Select Inputs To Correlate**.



Clicking this button brings up the **Select Inputs to Correlate** form. This form is shown below.



Each row and column is associated with a time history in the **Selected Time History Files** spreadsheet on the previous form. To specify that inputs 1 and 2 are correlated, click in the row 1/column 2 cell in the spreadsheet which will result in an "**X**" marker being placed in that cell. To remove a marker, click in the spreadsheet cell again.

In order to save computation time, it is assumed that the correlation matrix is symmetric and the cross-correlation of input "**i**" to input "**j**" is the same as for input "**j**" to input "**i.**" For this reason, only the upper-diagonal of the spreadsheet is selectable. The **Symmetric Correlation Matrix** check box is currently not selectable and is always set to **True**. Click **OK** to save the data, and close the form.

The final step in the automation process is preformed by clicking the **Run Automation Tool** button, and waiting for the process to complete. The progress can be viewed in the Patran output window. Click **OK** to close the form. For example, if a fully correlated two input job is setup, the following list of files is automatically created:

ts01.psd

ts01_ts02.csv

ts01_ts02.plt

ts02.psd

jobname.mpl

The plots of the psd files can be viewed using *mqld* or *mmfd.* Note that the cross-correlation process results in a multi-column (csv) file, which contains both magnitude and phase in order to account for the phase

relationship between the points in the time histories. The results of the cross-correlation process can be viewed graphically using *mwnplot* by selecting any of the plt files. These plotting tools can accessed from the top-level Tools menu, from the ribbon, or run from the command line.

To view the resulting psd files with *mmfd*: 1) click the folder button on the *mmfd* main form, 2) set the filter to show the psd files by entering "*.psd" in the "File name" box, and hit return, 3) select all the psd files, and click **Open**, 4) click **OK** on the main form to create the plot.

The results of the cross-correlation for time histories, for example ts01.dac and ts02.dac can be viewed using **mwnplot** for the file ts01_ts02.plt. Since the module **mwnplot** acts on an existing plt file, the axes cannot be changed. The first two figures in the plot show the results of the auto-correlation function for time histories 1 and 2. The last four figures show the amplitude, gain, phase, and coherence of the cross-correlation function. The amplitude and phase vs frequency data are used in the cross-spectra file, ts01_ts02.csv.

On closing of the automation tools form, a matrix file (mpl) is created. This file contains a mapping of the PSD and cross-spectra files to each vibration load case. The matrix file is an text-based XML format file. It is possible to create this file in a text editor, but the correct format must be followed exactly.

The mpl file can also be created by setting the **Matrix Input** method on the **Loading Information** form to **From Spreadsheet**, and then entering the PSD and cross-spectra file names directory on the spreadsheet. When the **Loading Information** form is closed, the matrix file is automatically created.

As mentioned, the matrix file is an XML format file, consisting of **LoadCase** and **OffDiagonalLoad** elements. A **LoadCase** element must be present for each frequency response results case used in the analysis. These represent the diagonal of the loading matrix. The type of file associated with the **LoadCase** element must be of type **Histogram**. The file may be single or multi-channel.

If the loads are uncorrelated, the **OffDiagonalLoads** section is unpopulated. Otherwise, it is populated with names of the files representing the cross-correlation between the loading associated with **FELoadCase i** (i.e, the **RowLoadCase**) and the off-axis loading (i.e., **ColumnLoadCase**) j. Note that the cross-correlation automation process is carried out for each possible loading interaction, and the correlation matrix is not assumed to be symmetric. The type of file associated with the **OffDiagonalLoad** element must be **Multicolumn**. The file must contain magnitude and phase vs frequency information. An example of the first few lines of this type of file is shown below.

```
<?xml version='1.0'?>

<MultiPSDDefinition Version='1'>

<Loadcases>

<LoadCase Chan='1' Name='ts' Testname='ts01.psd' FELoadCase='1'
LoadingType='Histogram'/>

<LoadCase Chan='2' Name='ts' Testname='ts02.psd' FELoadCase='2'
LoadingType='Histogram'/>

</Loadcases>

<OffDiagonalLoads>

<OffDiagonalLoad Chan='2' Name='ts01_ts02' Testname='ts01_ts02.csv' RowLoadCase='1'
ColLoadCase='2'/>

<OffDiagonalLoad Chan='2' Name='ts01_ts02' Testname='ts01_ts02.csv' RowLoadCase='2'
ColLoadCase='1'/>
```

```
</OffDiagonalLoads>
</MultiPSDDefinition>
```

Load the automation tools results in the *Loading Info* spreadsheet. Click the *Select a Matrix File* button, and then select the mpl file **jobname.mpl**.

Click **OK** to close *Loading Info...* form

The Classic modules **masd**, **mfra**, and **mdta** are used in the automation process. In producing the PSD files, **masd** is used for each time history using the settings available on the **Auto Self-Correlation Settings** page. The cross-correlation automation is performed using *mfra* with the settings "Auto Cross-Correlation Settings" page. The visibility for the automation settings can be changed by toggling the "Auto-Cross Correlation" check box on and off. For this example, use the default settings.

Currently, **masd** and **mfra** only recognize dac files, and multi-channel files must be converted to dac format. Note that rpc format files can be converted to dac using the classic module **mremdac**.

Use the default settings for the **Buffer Overlap**, **Averaging Method**, etc. These settings are used in the background processes used to compute the PSD and cross-spectra files. The actual command used during the automation tool run is posted in the Patran command window.

Introduction to PTIME

PTIME is a loading (time series, histogram, PSDF) database manager which has been designed to enable the MSC Fatigue user to manipulate and manage time history and other data file types. PTIME provides a secure system for recording information about the loading. By using PTIME, MSC Fatigue jobs may be archived and restored at a later date with no loss of information. This is essential for the successful re-analysis of the component.

The time history and other loading type files themselves are not loaded into the database, but are resident in the local directory together with the `ptime.tdb` file which contains the associated database data for each loading file. The loading information are held in binary files with default file type `filename.dac` (`.cyh` or `.psd` for matrix files or PSDFs respectively). See for DAC File Format Description, 274.

PTIME may be accessed from the MSC Fatigue main menu under the Loading Information form by depressing the Database Manager button on the top of the form. It can also be run in batch mode by typing ptime in a command window as described on page 162.

Once initiated, PTIME will present a set of screen displays which may be manipulated using the keyboard and mouse. A description of the way to use the screen displays is given in Module Operations (Ch. 18) for the Motif and Mask drivers as well as some of the generic capabilities of these drivers. The Motif driver will be referred to throughout this chapter.

When first invoked, PTIME appears displaying two forms when in the motif driver.

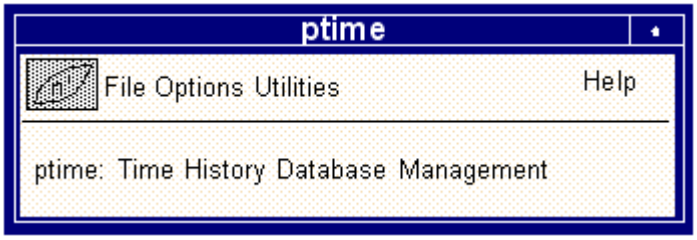


Figure 5-1 PTIME Utility Form

The top, small form is a generic form and allows for general program control. This is discussed in detail in Module Operations (Ch. 18), using the Motif driver.

The menu structure for PTIME is shown in Figure 5-2. The top menu contains options to further submenus such as the Change option. This multi-menu layering has been necessary to ensure the legibility and usability of each menu.

If the database is being created for the first time, the first menu to be presented will automatically be the Add submenu.

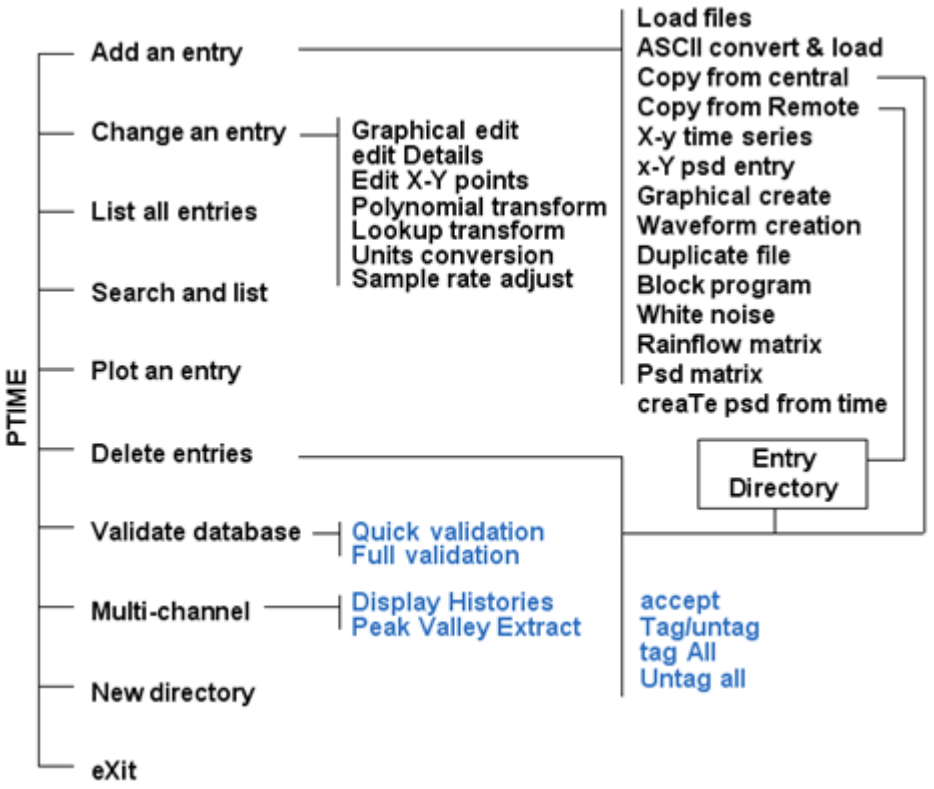


Figure 5-2 The PTIME Menu Structure

Using PTIME to Create Loading Inputs

The first step in setting up the loading inputs is to determine what loading inputs are required. These differ for each FE analysis type. These requirements are set out below.

FEA Type	Description
Linear Static	The variation of load with time for every linear static analysis load case must be defined for the MSC Fatigue analysis. The only exception is where the FE analysis models a non time varying stress state such as residual stresses from forming or preload stress. These non-varying loads are referred to as STATIC load cases. The association of the time histories with the individual FE load cases and their subsequent superposition in the case of multiple load cases give rise to the response time history necessary to feed the fatigue analysis.
Transient	The time history for transient FEA is defined at the FE job setup stage and used in the FEA itself. Consequently, the response (stress or strain) time history is embedded in the FE results and no other external time history is required beyond the one used in the FE analysis itself. The use of PTIME is not necessary for these types of FE analyses.
Frequency Response	Also known as steady state dynamics. In a similar way to the linear static analysis, this analysis requires a description of the load input in the form of power spectral density function (PSDF). In the case of multiple loading inputs, the definitions of the cross correlation between inputs using cross spectral density functions (CSDF) is allowed. The relationship between load cases and loading PSDFs/CSDFs is defined using an n x n matrix where n is the number of load inputs.
Random Vibration	Also known as Random Response. Again in a similar way to linear transient, no input PSDFs are required for this analysis type since they are defined in the FE analysis itself and the response PSDFs are created directly from the FE analysis. The use of PTIME is not necessary for these types of FE analyses.

All the functions to add an entry to the PTIME databank or to create a binary loading description file which is then registered in the databank are selected from the "Add an entry…" form. The specific functions required for each analysis type are detailed below.

Linear Static Analysis

The time history data may be in a number of forms. These are described in the table below together with the tools/action path to import or create the required PTIME entries.

Time History Format	PTIME Tool
Standard Time History	Copy it from the central databank (**Copy from central**) and apply a calibration factor, setting the load type and units to match the FEA loading.
Constant Amplitude	Create using **Waveform creation** or **Block program** option with 1 cycle in 1 block.
Block loading	Create using **Block program** option defining the number of cycles and their size for each block.
`.dac` Files	Load the file - `.dac` means the file must be in the MSC Fatigue single parameter binary format.
ASCII File- Value and Time Pairs	Use the **ASCII convert + load** option. The **Sample Rate** (in samples/second or Hz) can be any real number. Linear interpolation is used to calculate intermediate values. Finally set **Equally Spaced Data** to X-y pairs. If the data is arranged in columns and there are 2 columns (a time column and a value column) then **Take all Values** should be set to Yes; if not select No and refer to the documentation for more information.
ASCII File - Fixed Time Increment	Use the **ASCII convert + load option**. The **Sample Rate** (in samples/second or Hz) can be any real number. Linear interpolation is used to calculate intermediate values. Finally set **Equally Spaced Data** to Y-values only. If the data is arranged in columns and there is just 1 column containing the values, then **Take all Values** should be set to Yes, if not select No and refer to the documentation for more information.
List of Numbers	Either: Create an ASCII file using a text editor and use **ASCII convert + load** option, or Use the **X-y time series** option to enter the values interactively.
Rainflow Matrix Input	In this case, there must be just 1 load case and 1 FEA result. Load the rainflow matrix (must be format `.cyh` file) or create a Rainflow matrix from a time history using the **rainflow Matrix** option.

Transient Analysis

No PTIME action is required.

Frequency Response Analysis

The PSDF and CSDF data may be in a number of forms. These are described in the table below together with the tools/action path to import or create the required PTIME entries.

Loading Format	PTIME Tool
Standard Loading PSDF	Copy it from the central databank (**Copy from central**). Apply a calibration factor and set the load type and units to match the FEA loading using the **Change an entry** option. Note that the Units of the PSD are $(Load)^2/Hz$ - set the **Load Type** to the correct value of Load, e.g. g^2/Hz implies Load Type = g
ASCII File - Values and Frequency Pairs	Use the **ASCII convert + load option**. The **Frequency Rate** (points/Hz) can be any real number. Linear interpolation is used to calculate intermediate values. Finally set **Equally Spaced Data** to X-y pairs. If the data is arranged in columns and there are 2 columns (a frequency column and a value column) then **Take all Values** should be set to Yes. If not, select No and refer to ASCII Convert + Load, 217.
ASCII File - Fixed Frequency increment	Use the **ASCII convert + load option**. The **Frequency Rate** (points/Hz) can be any real number. Linear interpolation is used to calculate intermediate values. Finally set **Equally Spaced Data** to Y-values only. If the data is arranged in columns and there is just 1 column containing the values, then **Take all Values** should be set to Yes. If not, select No and refer to the documentation for more information.
List of PSDF/CSDF - Frequency Numbers	Either: Create an ASCII file using a text editor and use **ASCII convert + load** option, or Use the **x-Y psd entry** option to enter the values interactively in a spreadsheet.
ASCII File - PSDF Matrix	The PSDF Matrix defines the correlation between PSDFs, CSDFs and load cases. The ASCII file must be formatted in **n** columns by **n** rows where **n** is the number of load cases and is defined on line one. The file type for the ASCII file must be .pmx. The names of the PSDFs must be defined on the leading diagonal, e.g. `test101.psd`. The cross correlation file names must be defined in the relevant cross matrix positions. If no cross term is required, a zero (0) or the word NONE must be entered. To import an ASCII PSDF Matrix into the PTIME databank, ensure that all PSDF and CSDF files have been defined in the PTIME databank. Select the **ASCII convert + load** option. Choose the .pmx file, and set **Data Type** to psd Matrix. Set the **Load Type** and **Units** in the same way as for loading PSDFs.
Interactive Creation	Select the **Psd matrix** option. Define the **Load Type** and **Units** in the same way as for loading PSDFs. Set the Matrix Size; normally equal to the number of load cases. Input the PSDF and CSDF file names into the spreadsheet. These file names must be already defined in the PTIME databank.

Random Vibration Analysis

No PTIME action is required or necessary.

PTIME Menu Options

The main menu with associated text on the PTIME start-up form is shown in Figure 5-3. When first starting PTIME, only the Add submenu will appear until a time history has been created or loaded. From that point forward, when starting PTIME, the main menu will appear.

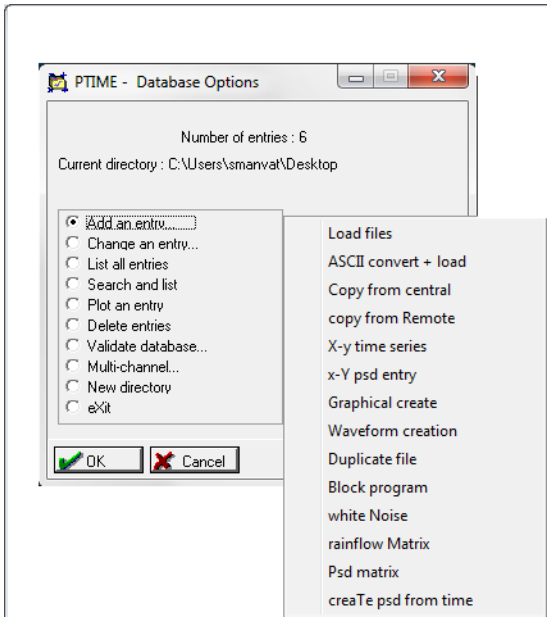


Figure 5-3 PTIME Main Menu

The menu options are shown as a vertical bar. To select one of the options, move the cursor over the desired option and simply depress the mouse button. The following subsections describe the PTIME main menu options.

Add an Entry Option

If this Main menu option is chosen, an additional submenu appears next to the main menu, as shown in Figure 5-4.

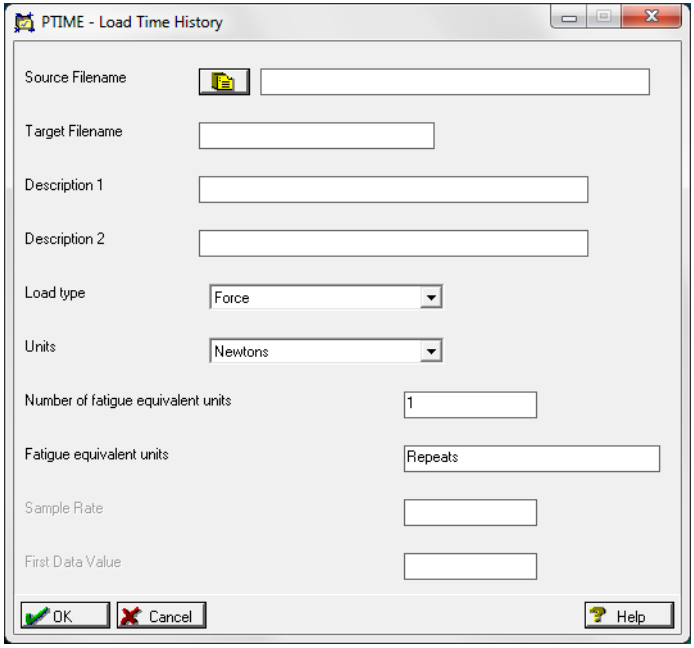


Figure 5-4 PTIMEAdd a Time History Menu

The **Add an entry** options are explained over the following pages.

Load Files

This option provides the ability to load time histories into the database and set the descriptive loading type and loading units information. To do this, a form will be presented with the screen display shown below in Figure 5-5. This screen or portions of this screen will be presented whenever creating a time history dependent on the method of creation. Please refer back to this Figure 5-5 often throughout this chapter.

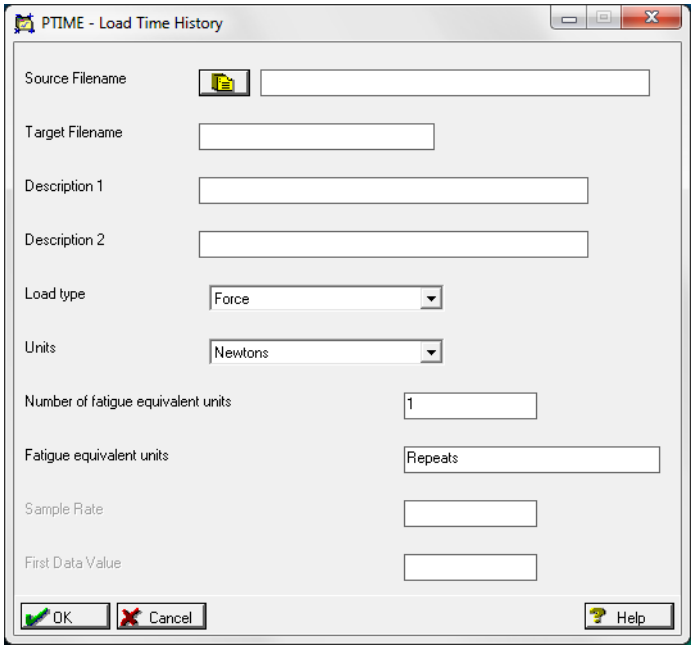


Figure 5-5 The Load Option

Note that only those units appropriate to the load type are available in the **Units** field.

Field	Description
Source File Name	This is the name of the file which contains the loading. This file must be a MSC Fatigue standard binary loading file which has a default file type of `filename.dac`. See DAC File Format Description, 274 for a full description of this file. The complete directory path must be supplied if the file is not already resident in the local directory. Use the List button to show all available `filename.dac` files and select a file from the list. You will have to change the filter to load other files such as `filename.psd` or `filename.cyh`.
Target File Name	This is the file name under which the file specified above will be stored in the local directory. The default file name is the same as the one specified in the Source File name field (excluding the directory path and the file extension).

Field	Description
Description 1 and 2	These two fields are provided to associate other information with the loading. In particular, a description of the direction and position of the transducer could be defined together with details of the source from which the time history was obtained (e.g., test details, job identity, etc.). Description 1 is presented together with the file name in the database index listing. It is required to use this first field to store essential information which may be necessary to inform you of the identity of the time history.
Load type/Units	The Load type and Units fields are closely associated. Both are toggle operated. They allow for selection of the load type and units to be used by the `filename.dac` file chosen in Source File name (above). The load types and units available by default are shown in the Table 5-1.
Equivalent units	In order to report fatigue life in units which are appropriate for the component or structure being analyzed, it is required to enter both the number of units and unit type which describes the duration of the loading. These units are independent of the time or frequency base of the loading. For example, it may be the case that the loading time history represents 27.3 kilometers, so the Fatigue equivalent units are kilometers and the Number of fatigue equivalent units is 27.3. Other examples are: 2 flights, 11 pressure cycles, 3.5 days, 1 repeat. The default equivalent unit is 1 Repeat of the time history.

If a required load type and/or unit is not on the list above, then the list can be extended. The list can be edited and is held in a MSC Fatigue system directory. For full details, see Loading and Units, 283.

Table 5-1 Table of Load Types and Units

Load Type	Units Available
Force	Newtons, kNewtons, lbs force, Tons force, Tonnes force
Pressure	Pascals, MPa, PSI, KSI, TSI, Kgf/m^2
Temperature	Degress Kelvin, Degress Celsius, Degress Fahrenheit
Acceleration	m/s^2, $feet/s^2$, g
Displacement	m, mm, inches, milliinches, microinches
Rotational Velocity	rph, rad/s, rad/min, rps, rpm
Velocity	kph, ips, mph, m/s, mm/s
Uncalibrated	undefined
Scalar	%, levels, none
Moment	Newton-meter (Nm), Ft-lbs, Nmm

ASCII Convert + Load

This option allows for conversion of an ASCII file to a file with default `filename.dac` file extension used by MSC Fatigue. Its first screen is shown in Figure 5-6.

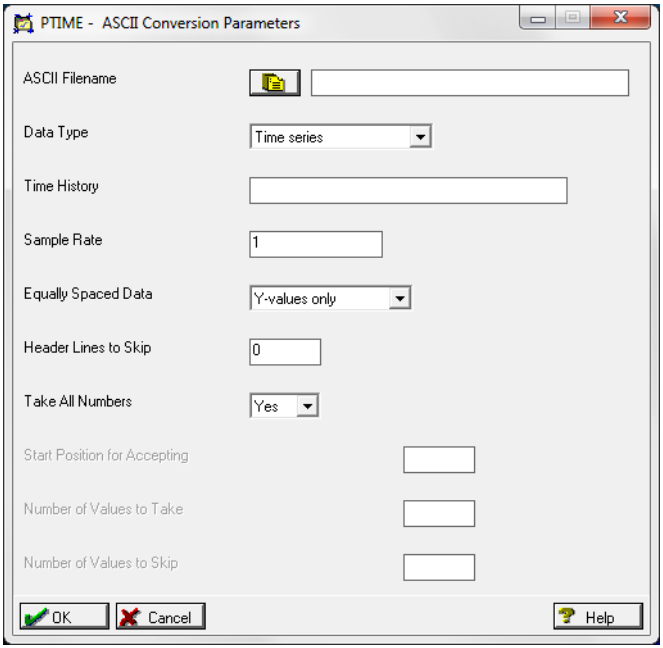


Figure 5-6 ASCII Convert & Load Option

The fields are as follows. Fields are active or inactive depending on the options chosen.

Field	Description
ASCII File Name	Enter the name of the ASCII file to be converted to `filename.dac` (time history) file format. Use the List button in top bar to list all available ASCII files. The files must be called `filename.asc`. Note that the Filter sub-option allows for specification of alternative directories for the ASCII file. Any file name may be entered. However, if a file suffix is not specified, then a `filename.asc` is assumed.
Data Type	If this is a time dependent signal, keep the default of **Time series**. If it is a PSDF then set it to either **Power spectrum**, or **psd X-y data**. The only difference between these two is that the first allows more options, just as in **Time series** to import data. The **psd X-y data** entry assumes the numbers are in Frequency/Value pairs. The **psd Matrix** option will import a matrix definition of the PSDF/CFDFs from a `filename.pmx` file. This is for management only.
Time Series/ **Power Spectrum/** **PSD Matrix**	Enter the name of the loading here. A `filename.dac` or `filename.psd` file will be created from the ASCII file named above. The default file name is the same as whatever was entered in the ASCII File name field. A copy of the original file is created but with a `.dac` or `.psd` file extension rather than a `.asc` extension. The original file is not changed. For PSD Matrix, only an entry is made in the database for management purposes only.
Sample rate	This is the number of values per X-axis increment (usually time). So, for 1 value very 0.1 seconds, the answer to this question is 10 (i.e.: 10 values per second or 10Hz). The number of values per second is only important in fatigue crack growth calculations since the crack growth rate is sensitive to the loading rate in corrosive environments. The equivalent units facility allows the user to define time units that are more appropriate to the actual analysis being carried out and so the sample rate could be set to 1 if the total time for the file is unimportant. If, however, the entire signal does not import properly, it may be that the sample rate needs to be higher for more resolution of the actual signal.
Equally Spaced Data	Two types of data can be converted: 1. Data that consists of Y-values only which (it is assumed) are spaced equally apart in time or frequency. The sample rate is stated or implied elsewhere. Even if time values are present they may well be omitted provided that the correct sample rate is entered. 2. Where the file contains paired time or frequency and data values (X-y pairs), and the values are NOT equally spaced, then the data is assumed to be formatted with a time or frequency followed by a value sampled at that time or frequency. X values must be in ascending order.
Header Lines	If the ASCII file contains non-relevant header lines then PTIME should be told how many header lines exist. It will then ignore that number of lines.

Field	Description
All Numbers	If it is desirable to convert all values in the ASCII file then select Yes. Otherwise say No. Selecting No enables skipping some data values in the ASCII file. This is extremely useful if, for example, converting a multi-channel file such that only channel 5 and 8 are extracted. As an example, say each vertical column represents a channel of data, and extraction of only data from channel 4 is desired. To do so first select Take All Numbers = No. Three additional fields will then be displayed.
Start Position	Setting this will start reading at the column specified. So for the example, reading only channel (column) 4, set this to 4.
Number of Values to Take	In the example, this is set to 1, meaning; take 1 value then skip 7 values, then take another value, etc., until the end of the file is encountered. To take all of columns 4 and 5, this would be set at 2 and Number of Values to Skip would be set at 6. When all the above fields have been filled and the OK button is pressed, another screen will appear asking information on the details about the file that will be created such as description, fatigue equivalent units, etc. See the Load Files, 214 option above for a detailed description of this screen.
Number of Values to Skip	This causes the program to skip n number of values from the starting position. So for example, set this to 7 to skip all other values except those in column 4. (There must be an even number of columns for this to work properly since values are counted from left to right and then continued on the next row.)

Batch Operation

PTIME can be run in batch mode for easy import of ASCII files.

A list of PTIME's batch keywords:

/OPT	Main menu option to add time history or exit. /OPT=A or /OPT=X
/AOPT	Option name for invoking ASCII import. /AOPT=A
/INP	File name of the ASCII file to convert (.ASC) /INP = AERO
/DTTYP	Data type to load: Time series, Power spectrum, psd X-y data, psd Matrix. /DTTYP=T
/OUT	File name of the output signal file (.DAC) /OUT=AERO
/SRATE	Sample rate for the output file /SRATE = 2
/EQSP	Whether the data is equally spaced or not, Y-values only, X-y pairs. /EQSP = Y
/HEAD	Number of header lines to skip /HEAD =4
/ALL	Whether to take all the numbers in the file /ALL = N
/STA	The start position for accepting /STA = 10

/SKIP	How many values to skip /SKIP = 4
/TAKE	How many values to take /TAKE = 1
/DESC1	Description 1 of details. /DESC1=Description1
/DESC2	Description 2 of details. /DESC2=Description2
/LTYP	Load type in details area. /LTYP=Force
/UTYP	Unit type in details area. /UTYP=Newtons
/SOU	Source filename (.dac, ,psd, or .cyh). /SOU=filename .dac
/TAR	Target filename (can be any name). /TAR=newfilename .dac
/NUMEQU	Number of equivalent units in details section. /NUMEQU=5
/EQUNI	Equivalent units in details area. /EQUNI=hours

Example:

As an example, you can invoke PTIME with the following options all on the same line to load two time histories at the same time. Keyword not used will use default values.

```
ptime /opt=a/aopt=ascii/inp=test/out=load1/eqsp=y/all=y
/opt=a/aopt=ascii/inp=test2/out=load2/eqsp=y/all=n/sta=1
/numtake=4/skip=5/opt=x
```

Or you can put each keyword and value in a file (called `commands.cmd` for this example) on separate lines and then invoke PTIME to process the keywords with

```
ptime @commands.cmd
```

To ensure proper execution, check the `batlog.lst` file.

Copy from Central (Database)

MSC Fatigue is delivered with a set of standard loading time histories which are used in a number of fields such as the aerospace or automotive industries. It is possible to copy one or more of these standard time histories into the local directory for use in a particular durability analysis. Enter the name of the file if it is known, or use the List button to obtain a listing of the database entries contained in the central area. See Figure 5-7.

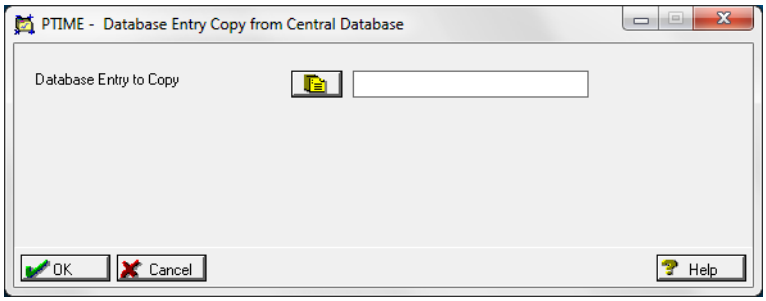


Figure 5-7 Central Database Copy Option

It is also possible to store your own here as a data management tool. Simply move all of you own .dac, .psd, or .cyh files into the ptime directory delivered with MSC Fatigue and run PTIME from this directory and load all the files. They will then be available for others to copy from the central database.

Copy from Remote (Database)

In order for users to share loading time histories without corrupting the remote database or time histories, the copy from Remote database option has been provided. A database must exist in the remote directory in order for this option to work. When this menu option is chosen, the following is presented with the question:

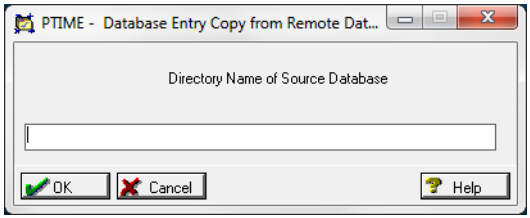


Figure 5-8 Copy from Remote Option

The full path and name of the directory must be defined. For example:

```
//node1/top_level_directory/sub_directory/
```

Important:	Care must be taken to put the last delimiter before the start of the filename (e.g., \ or /).

Once the directory has been defined and the **OK** button pressed, PTIME will read the database in the specified directory and present the user with a form as in Figure 5-7. The operation of this menu is identical to that described for the Copy from Central (Database), 220 option. This option can be used as a tool for loading data management.

X-Y Time Series

This option allows for creation of a simple time series file from scratch. Before being allowed to enter data to define the time history, the generic information screen similar to that in Figure 5-5 (presented previously) in order to provide some descriptive data associated with the time history. The functionality and fields of this screen are identical to those described in the Load files option above.

When the new file details have been specified, click OK to accept the details. A new window appears to allow for data entry on a screen such shown on Figure 5-9 below. Click **OK** when finished.

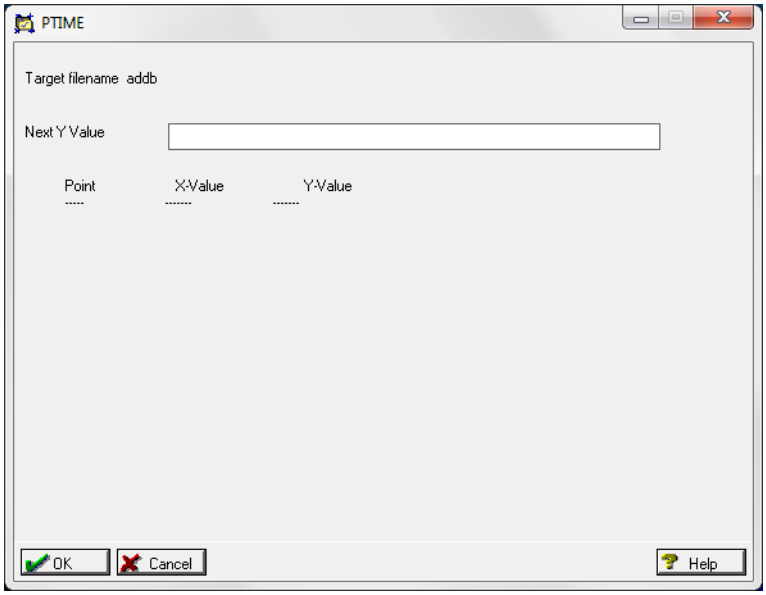


Figure 5-9 The X-Y Data Points Screen

Always end input of the Y values with a blank line. Clicking OK without doing so will result in the last point being discarded.

Important:	Only Y-values can be input in this mode. The X-values are automatically incremented by unity (starting at zero) each time a new Y-value is added. To input X-Y data pairs to more precisely specify the time increment, use the ASCII Convert + Load, 217 option.

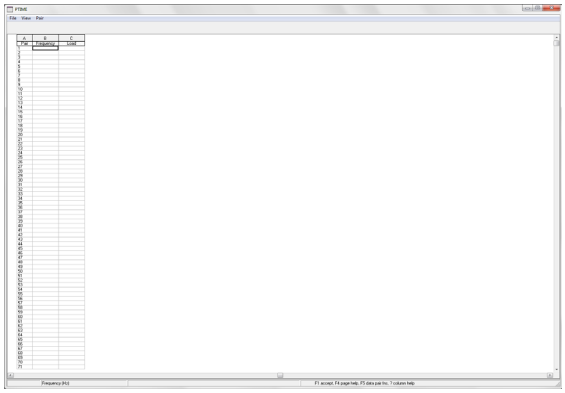
X-Y PSD Entry

This option is similar to the time series X-Y entry, except now you are using frequency/value pairs.

When the new file details have been specified, click **OK** to accept the details. A spreadsheet appears to allow for data entry.

A	B	C
Pair	**Frequency**	**Load**
1	0	0
2	1	0
3	10	100
4	20	0
5		

A	B	C
6		
7		
8		



Data entry is accomplished very easily by clicking on a cell. Start typing the value. A form will appear allowing entry. You can specify the pair by separating the frequency and value by a space or comma. Clicking the OK button or using the ENTER key will scroll to the next row automatically. You can simply continue to type the values in without every having to remove your hands from the keyboard.

If you need to make a correction, click on the cell and enter the data value. If you enter one value, only that value will be overwritten. If you enter two values separated by a comma or space, both the frequency and value will be updated.

The View pull-down menu on the spreadsheet form allows you to scroll to the portion of the spreadsheet that you want.

You can insert or delete rows of frequency/value pairs by pressing the Pair button at the top of the form. Or press the F5 key. The currently selected row will be deleted or a new row inserted above.

When you are satisfied with the input, press the F1 key or choose OK from the File pulldown menu. A new `filename.psd` entry will be created.

Graphical Create

The Graphical Create option allows for generation of a `filename.dac` file via a graphical editor. A program called MGED is spawned for this purpose.

When Graphical Create is selected, again the database entry definition screen is presented first. The functionality and fields of this screen are similar to those described in the Load files option described earlier (see Figure 5-5). However, there is an additional field to complete: the Sample rate. The sample rate is the number of readings or data points per second, and is normally expressed in Hertz (Hz).

When all of the fields have been completed (or their defaults accepted), press the OK button to go to the graphical editor. The graphical editor screen is shown below in Figure 5-10.

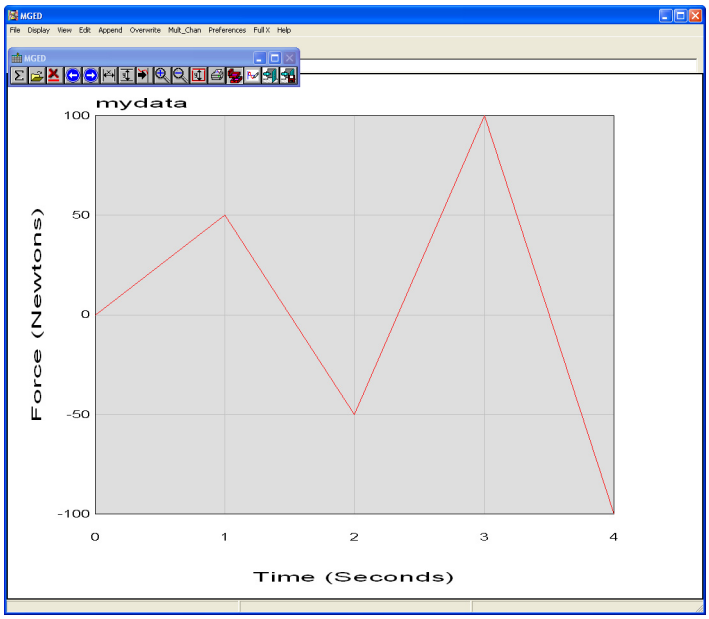


Figure 5-10 The Graphical Editor Screen

Many of the preferences and other options available through the pull-down menus at the top of the window are generic for graphical operations throughout the MSC Fatigue system and are not discussed here. The operation of this graphical user interface is described in Module Operations (Ch. 18) as well as the commands that may be entered in the Command databox. The functions of the pull-down menus which are specific to PTIME and the graphical editor are described below.

The main pull-down menus specific to PTIME are the Display, Edit, Append, Overwrite, and Move menus. The Display menu is very simple and is explained in the table below.

The Edit menu contains several different operations, including deleting and inserting sections of signal, rescaling and copying sections of signal to and from files.

The Append menu allows functions and waves to be generated at the end of the signal. Note that in append mode for functions which only require the end point setting, and no question answering, the option does not need to be repeatedly selected if several of the same functions are to be appended to one another. A series of end points can be selected and the function will be generated between each of them.

The Overwrite menu allows functions and waves to be overwritten anywhere in the signal. The Overwrite submenu is the same as the Append menu but the functions and waves may overwrite sections in the middle of the file. The Overwrite submenu will allow functions to overwrite parts of the signal anywhere, so, unlike the Append option, a start point and an end point must be defined.

The View menu controls the moving of the display window over the signal. This allows for display of various sections of the signal.

Option	Description
DISPLAY	
Replot	This simply replots the data and associated plot. The same thing can be accomplished by pressing the P key.
Stats	This displays some signal information, including the number of points, total time, sample rate and coordinates of the last point.
EDIT	
File Insert	This function allows the data from an external file to be inserted into the signal at any point. The program requests the start point to be selected, and then prompts for the name of the file to insert, this may not be one of the files already being edited. The inserted file must have the same sample rate as the main file, but may be any length. Any signal after the start point is shuffled forward to make room for the inserted file. The file must be of type `filename.dac`.
File Extract	This function allows a section of signal to be written out to a file. The section is not deleted from the main signal. The program requests the start and end points to be set and then prompts for the name of the file. The name given must not be the name of an existing file. This creates a file of type `filename.dac`.
Truncate	This option allows the signal to be truncated. The program requests the new signal end point to be set. All signal beyond that point will be deleted.
Rescale	This function allows a section of signal to be rescaled. The program requests the start and end points to be selected and then prompts for a scaling factor and an offset. Care should be taken when using this function with signals that have large mean offsets, since the offset is applied after the multiplication factor, and very large numbers could be produced.
Drift	This function allows drift in a signal to be corrected. The program requests the start and end points to be selected, and then prompts for the offset to be applied at each end. A linear correction is applied between the start and end points based on the offsets defined.
Insert	This function allows a space to be opened in the signal. The program requests the start point to be entered, and then prompts for the number of seconds of data to insert. Any signal after the start point is shuffled forward in time to make space for the new data which is initialized to zero.
Delete	This function deletes a section of signal. The program requests the start and end points to be selected. Any signal beyond the end point is shuffled forward to the start point, effectively deleting the signal between the two points.
APPEND/OVERWRITE	
Ramp	This function creates a straight line between the start and end points.
Half Sin	This function creates a half sine wave between the start and end points, starting with a phase of zero degrees.

Option	Description
Step 1	This function creates a 'rise then along' step. This means that all points after the start point will be at the same level as the end point.
Step 2	This function creates an 'along then rise' step. This means that the signal will be horizontal from the start point until the point before the end point.
Exp 1	This function produces an exponential curve which tends to infinity in the x direction.
Exp 2	This function produces an exponential curve which tends to infinity in the y direction.
VIEW	
Centre	This option allows a point to be selected to be the center of the screen. The graph is then redrawn with this point in the center.
Page Left	This option moves the display window left by one window width.
Page Right	This option moves the display window right by one window width.
Full X	This option displays the full signal in the x direction.
Full Y	This option displays the full signal in the y direction.
Goto End	This option will display the last part of the signal in the first 20% of the time window.
Goto Start	This option will display the first part of the signal.
Window X	This option allows an x-axis window to be set. The program will prompt for the start and end points of the window. This allows sections of particular interest to be zoomed in the x direction.
Window Y	This option allows a y-axis window to be set. The program will prompt for the start and end points of the window. This allows sections of particular interest to be 'zoomed' in the y direction.
Zoom In	This option zooms in by contracting the time window by 5 times.
Zoom Out	This option zooms out by expanding the time window by 5 times.
Local Full	This option adjusts the y-axis limits to fill the current display with the data shown.
MISCELLANEOUS	
Cancel	This option will cancel any option which is currently part way through its operation.
Restore	This function restores a section of signal to its original form. The program will prompt for the user to set the start and end points, after which the data that existed when the program was started will be restored within the selected section. Note that sections beyond the end of the original signal cannot be restored, and also the signal cannot be restored if the input file is a new file.
K/UP	In many operations, a prompt for start and/or end points will be displayed. These points can be picked by the mouse or cursor. However, it may be easier to actually specify these values by keyboard input. To do this use, the **K** key. It will then be possible to type in the positions in the Command databox. To switch back to full mouse operation in the middle of the option, type **UP** at the prompt for the point selection.

Option	Description
P	If the **P** key is pressed at any time, the current option is terminated and the whole screen is redrawn.
W	If the **W** key is pressed at any time with the cursor over a menu option, help is displayed for that option.
V	If the **V** key is pressed at any time with the cursor over a menu option, that menu option is invoked.

Waveform Creation

This option initiates the waveform generation option which provides facilities for creating square, triangular, or sine waves with an option to carry out multiple wave summation to create multi-frequency/phase periodic waveforms.

The first stage in the waveform creation process is to define the database information concerning the time history about to be created. See Load Files, 214 option described earlier for an explanation of the fields.

Having provided the general information, it is then necessary to define the waveform creation parameters. These data are shown in Figure 5-11.

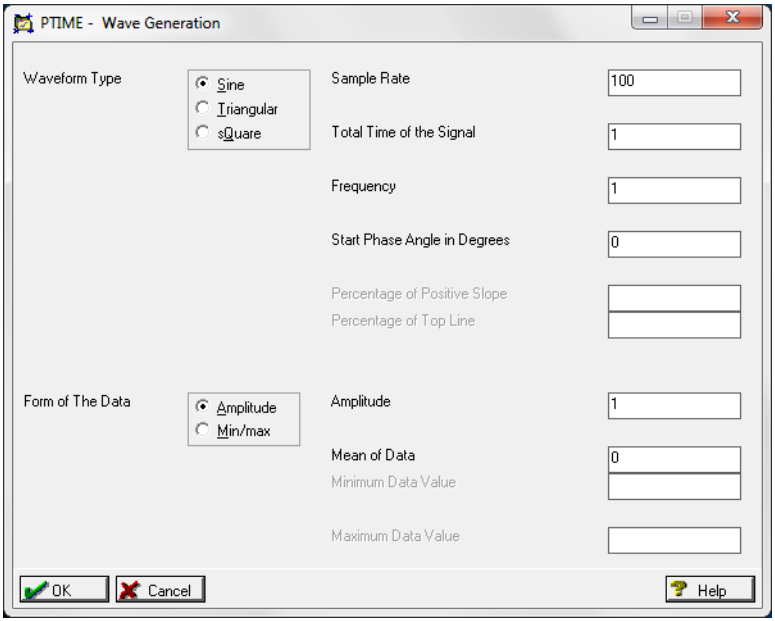


Figure 5-11 Waveform Generation Setup Screen

Each field is explained in the table below.

Field	Description
Waveform Type	Three types of waveforms are possible: *Sine*, *Triangular*, or *Square*.
Sample Rate	The sample rate is the number of points in the signal per second of signal. It is important to remember that an inadequate number of samples will distort the output signal. Shannon's sampling theorem says that at least two samples are required to define uniquely the highest frequency in the signal (i.e., the sample rate must be at least twice the maximum frequency in the signal). However, fatigue is an amplitude dependent process and so it is necessary to consider the number of samples to define amplitudes correctly. In practice, it has been found that 10 times the maximum frequency in the signal will define the amplitudes to within 5% accuracy for Sine Waves. For other waveforms, the relationship is worse due to the sharpness of the peaks. It is advisable to check the signal visually after creation using the graphical display option.
Total Time	This defines the total time of the signal (i.e., 500 seconds).
Frequency	This defines the frequency of the signal (i.e., 10 Hz or 10 cycles/second). This is related to the Sample Rate described above.

Field	Description
Phase Angle in Degrees	A sine wave has an initial phase of 0 degrees and a cosine wave has one of 90 degrees. The phase may take any value between 0 to 360 degrees.
Percentage of Positive Slope	This option is only enabled if a Triangular wave is being created. This defines the percentage of each wave that has positive slope. Fifty percent defines a perfectly symmetric triangular peak. If a sharp drop at the end of the peak or a sharp increase at the beginning of the peak is desired this number can be modified to accomplish this.
Percentage of Top Line	This option is only enabled if a Square wave is being created. This defines the amount of the top of the square wave that makes up the entire signal.
Form of the Data	*Amplitude/Mean* or *Minimum/Maximum* values are allowable to define the form of the wave. With Min/Max the two input lines to the side of the Form of the Data will change to Minimum Data Value and Maximum Data Value.
Amplitude	The amplitude of the signal is the height of the peak above the mean level. The magnitude of the signal must be in the units defined on the previous setup page. The phase of the signal is defined in degrees and the convention is as used to describe sine/cosine waves. The amplitude of the signal is the height of the peak above the mean level. The magnitude of the signal must be in the units defined on the previous setup page. The phase of the signal is defined in degrees and the convention is as used to describe sine/cosine waves.
Mean of Data	The mean is the DC offset of the signal from zero.
Minimum of Data	This is the minimum Y-value of the signal in the units defined on the previous setup page.
Maximum of Data	This is the maximum Y-value of the signal in the units defined on the previous setup page.

Having created the first waveform, an option to add further frequencies to the wave by choosing the Summation option is presented. The comments on sample rates above must be taken into account also in the Summation option. Sample rates must be the same when summing waveforms. The screen shown in Figure 5-12 is displayed after the first wave creation.

If Summation is selected, the screen shown in Figure 5-12 is presented in order to add to the previous wave. When done, select the Finish option.

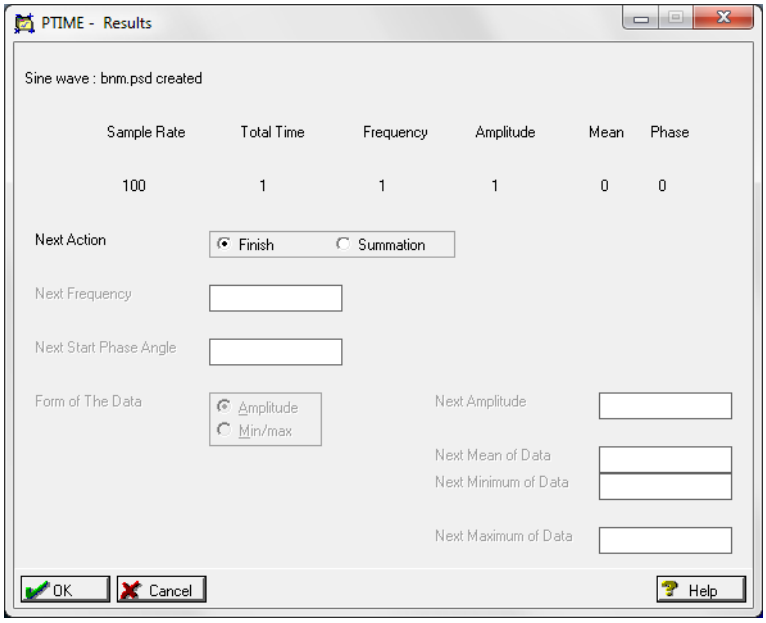


Figure 5-12 Waveform Summation Screen

Duplicate File

To use this option, first indicate which loading file is to be duplicated. Type the name if it is known, or use the List function to select a file by highlighting the file to be duplicated. PSDF matrix files cannot be duplicated.

Once the correct file is selected, the duplicate screen appears as shown in Figure 5-13. The fields in this screen are intuitive and are described more fully in the Load Files, 214 option. A new, unique name must be supplied and at least the first description field must be filled. Click the **OK** button to accept the name and descriptions. The name and descriptions of the time history being duplicated also appear at the top of the screen.

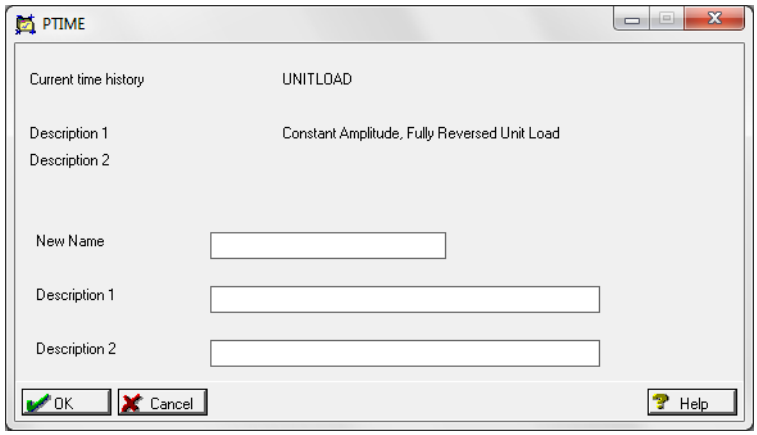


Figure 5-13 The Duplicate File Screen

Block Program

Time histories can be easily created by specifying a number of constant amplitude block with possible varying means values between them. After selecting this option and specifying a file name, description and other descriptive data for the time history, the following form appears. You may specify the constant amplitude blocks either by inputting their amplitudes or the maximum and minimum values.

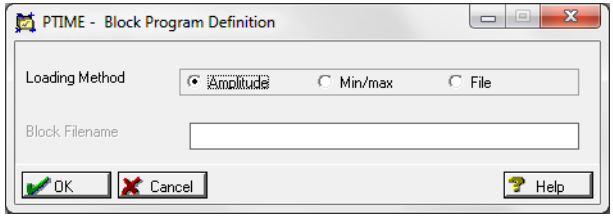


Figure 5-14 Block Form Creation Type

The next form that appears allows for specification of the amplitude/mean or maximum/minimum values. The amplitude/mean form is shown here. You may enter as many blocks as you wish. When a block is defined, click the **OK** button. The Block Number will increment one. You may view the blocks already defined by pressing the List button. Press the Cancel button when all blocks have been defined or leave the Block Number databox blank when pressing OK.

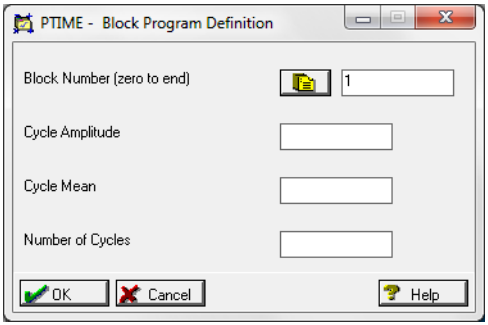
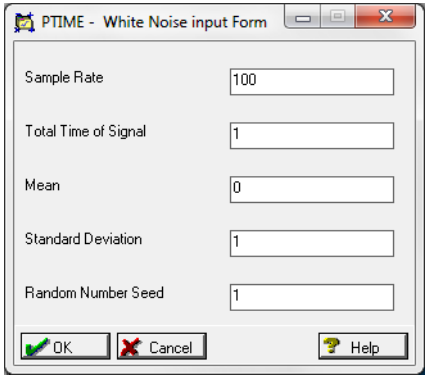


Figure 5-15 Block Program Creation Form

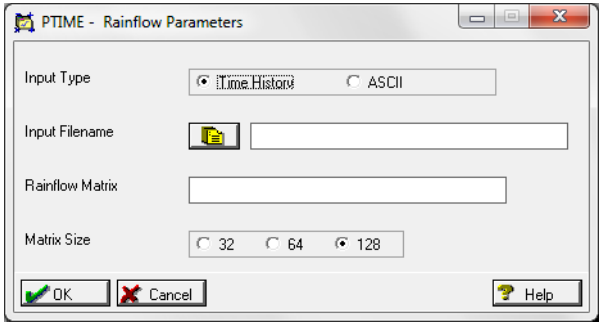
White Noise

If white noise is chosen, aside from the standard input, the user must supply a random number seed from which a pseudo-random number will be generated (the resultant wave sequence will, in fact, repeat after a very large number of cycles). It has been proven that the best seed is an odd number. Different seeds produce different pseudo-random sequences.



Rainflow Matrix

The Rainflow Matrix option allows you to create a matrix file in the form of a histogram. You may select an existing time history file and rainflow cycle count it or you may read in the data from an ASCII file.



Input the existing time history file or the ASCII file in the Input File name databox. Give it a new name or keep the same name in the Rainflow Matrix databox. A .cyh extension will be added to the name. For matrices created from Time Histories, you may select the matrix size (bin size). The default is 128 which is most accurate.

The ASCII file is defined by keyword/value pairs to specify the matrix size, axes limits and data type. The data then follows and can be specified either by bin location + number of cycles or as a range mean pair + number of cycles.

Lines beginning with the # character are comment lines and are ignored.

The first line in the file must be #V6.0

Mandatory keywords are:

```
BINS=<value> where <value> is 32, 64 or 128
RANGE_MIN=<value> where value is minimum range of matrix.
MEAN_MIN=<value> where value is minimum mean value of matrix.
```

The size of the matrix axes can be specified either by

```
RANGE_SIZE=<value> where <value> is the range bin size
MEAN_SIZE=<value> where <value> is the mean bin size
```

or

```
RANGE_MAX=<value> where <value> is the range maximum.
MEAN_MAX=<value> where <value> is the mean maximum.
```

The data is then entered, starting with one of the following lines:

```
BIN_DATA or
RANGE_MEAN_DATA
```

For BIN_DATA, the data is specified as follows, one entry per line.

```
<range bin> <mean bin> <number of cycles>
```

For RANGE_MEAN_DATA,

```
<range value> <mean value> <number of cycles>
```

An example of BIN_DATA follows:

```
#V6.0
# Example using BIN_DATA
```

```
BINS=32
MEAN_MIN=-1.1
MEAN_MAX=1.1
RANGE_MIN=0
RANGE_MAX=2.1
BIN_DATA:
31 15  5
16  9 10
16 23 10
 8  2 20
 8 32 20
An example of RANGE_MEAN_DATA follows:
#V6.0
# Example using RANGE_MEAN_DATA
BINS=32
MEAN_MIN=-1.1
MEAN_MAX=1.1
RANGE_MIN=0
RANGE_MAX=2.1
RANGE_MEAN_DATA:
2 0 5
1 0.5 10
1 -0.5 10
0.5 1 20
0.5 -1 20
END_DATA
```

PSDF Matrix. This is a simple utility to allow you to create a file that contains the relationship between PSDFs and CSDFs and the corresponding FE load cases from a frequency response (FRF) analysis. This is a necessary step when dealing with multiple load inputs from FRF analysis.

After entering a name and giving a description and the number of loads, a spreadsheet is presented. The names of the loading PSDFs should be entered in the diagonal terms and the names of the loading CSDFs should be entered in the off-diagonal terms. Click on a cell and start typing the name of the PSDF/CSDF as it has been defined in the database. You do not need to include the `.psd` extension. The program will automatically pick this up. The loading PSDFs and CSDFs MUST have already been defined and exist in the PTIME database.

Make sure you enter the load PSDFs in the same order that the loading will be defined from the FE run, i.e. load case 1 corresponds to PSDF in row/column 11, load case 2 corresponds to PSDF in row/column 22, etc. If there are no cross terms (loads are completely independent of one another), then enter either 0 or NONE in the off-diagonal terms.

You may make as many changes as you wish while you are in this spreadsheet. Once you are satisfied, use the F1 key or OK from the File pulldown to accept the matrix. Once the matrix is defined it CANNOT be changed. You will have to delete it and re-enter it; or you can edit the `filename.pmx` file with an editor. Alternatively you can create the matrix file entirely in an external editor and then load it into PTIME. See ASCII Convert + Load, 217.

PSDF matrix file format is shown below:

Filename: `xxxxxx.pmx`
Number of load cases=`n`
`n` rows by `n` columns

Example 1: `5load.pmx`
(5 load case matrix created manually using a text editor, no cross correlation)

```
5
test101.psd NONE NONE NONE NONE
NONE test102.psd NONE NONE NONE
NONE NONE test103.psd NONE NONE
NONE NONE NONE test104.psd NONE
NONE NONE NONE NONE test105.psd
```

Example 2: `3load.pmx`
(3 load case matrix created with PTIME, some cross correlation terms)

```
3
test105.psd test12.csd test13.csd
test12.csd test105.psd NONE
test13.csd NONE test105.psd
```

Create PSDF from Time History

The Auto Spectral Density program, MASD, is spawned from this option. It allows the conversion of a time domain signal into a frequency domain signal or power spectral density function (PSDF). It performs a frequency analysis on a single parameter input file, a `.dac` file for example. MASD produces an output file that indicates the frequency content of the input file. This module is described in Auto Spectral Density (MASD), 260.

Change an Entry Option

If this Main menu option is chosen, an additional submenu appears next to the main menu, as shown in Figure 5-16.

From this submenu, the user can change both the textual information associated with the database entries and the actual time history data values. It also provides options to change the units system of the data, adjust the sample rate of the time histories, and perform recalibration using polynomial transformations or a lookup table.

These options are explained over the next few pages.

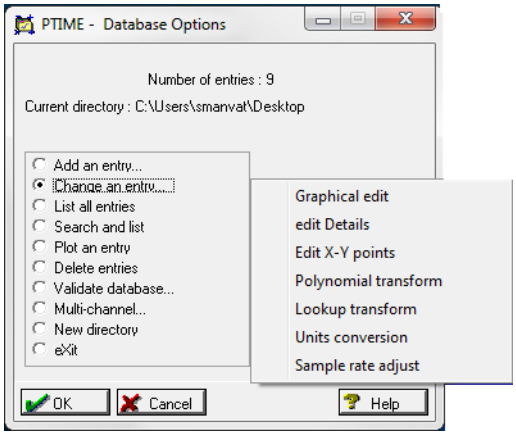


Figure 5-16 Change Time History Submenu

Graphical Edit

The Graphical Edit option of Change a time history invokes a specialized editor which has a mouse-driven menu user interface. This editor has been designed to work with time history data, providing facilities for appending or overwriting data values using:

- Ramp functions
- Step functions
- Various waveforms
- Point-by-point definition
- Time segment extraction/insertion and deletion
- Drift correction and spike removal

When the option is invoked, the screen display will be presented as shown in Figure 5-17 below.

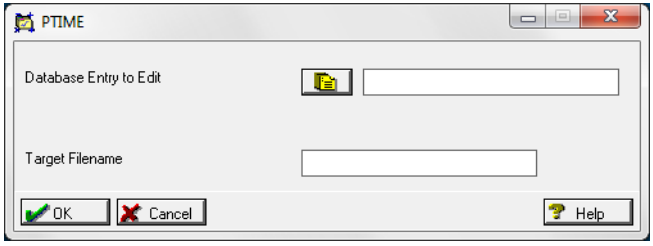


Figure 5-17 The First Screen of Graphical Edit Option

This screen enables specification of the file to edit, and specification of its post-edit name. The same name may be specified. PTIME will prompt for verification to overwrite the existing time history.

The graphical screen will be presented. The operation of this graphical user interface has been described earlier under the Graphical Create, 223 option. Please refer to this section.

Edit Details

The form in Figure 5-17 is presented to the user to select a file to modify and to specify a new name if necessary. The same name may be specified and overwrite permission will be requested.

A screen, as in Figure 5-18, is presented to allow modification of the loading details. The fields presented on the screen for this option are described under the Load Files, 214 option.

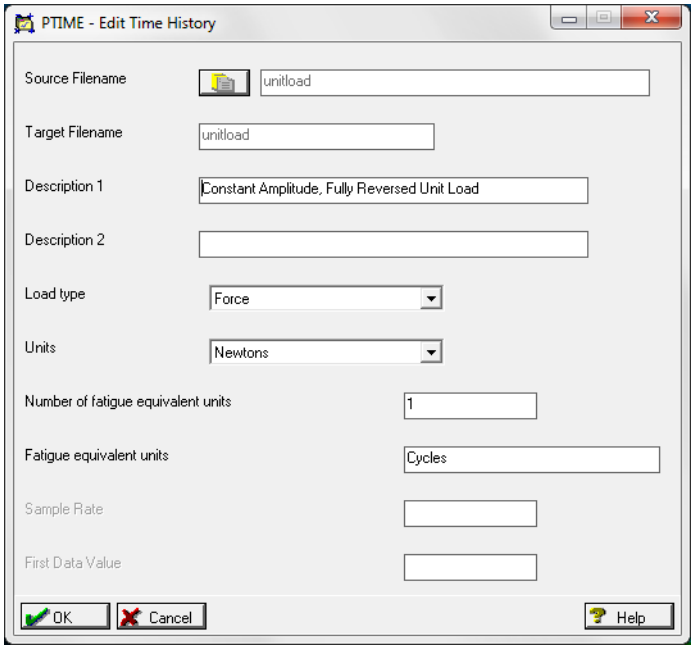


Figure 5-18 The Edit Details Option

Important:	It is possible to change the load type and units of the loading history on this page to values which do not apply to the loading itself! This option should be used with caution since the user may be lulled into thinking that there is a change in the actual data values as well as the descriptive data. There is NO change to the actual loading file.

Edit X-Y Points

The third option from the Change submenu is Edit X-Y points. This function allows editing the `filename.dac` file or any other type with data points. The first screen prompts for the file to be edited. By default, the target file name is the same as the input file name. If this is the case, then the new file will overwrite the old and overwrite permission will be requested. If preservation of the original file is desired, then enter a different name in the target file name field (Figure 5-17).

When a file is accepted the screen in Figure 5-19 appears.

In the Next Point databox, the user may type in the point number to be edited. Also, if all the points are not visible on the screen, type the point number and press the TAB key. This will display the current point in the middle of the list with a few points above and below. To replace that value with another, simply type that number in the New Y Value field. When the user presses the TAB key, the new value is accepted and the screen is updated to reflect the change. Use the TAB key successively to move from the Next Point field to the New Y-Value field.

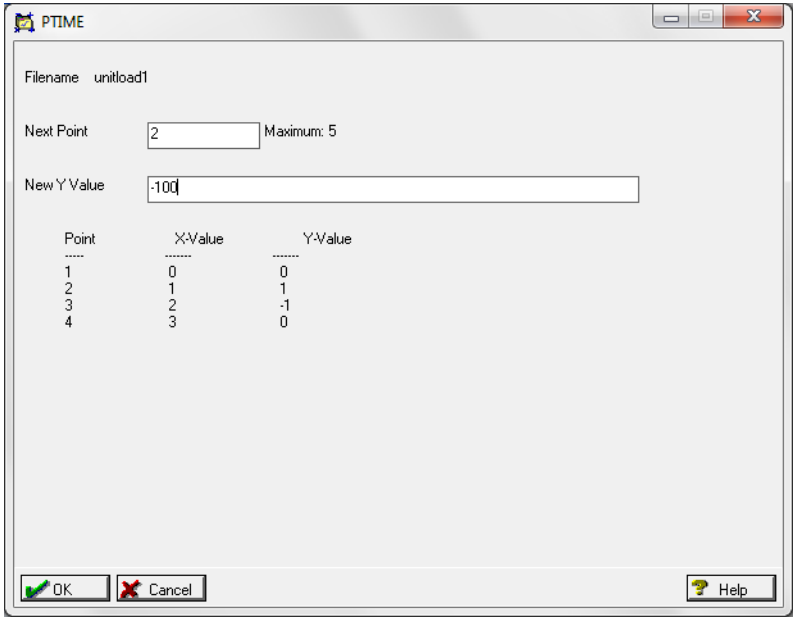


Figure 5-19 The X-Y Edit Screen with List of X-Y Points

Polynomial Transform Option

This option provides for a method to carry out a polynomial transformation or conversion of the loading data in a data file. The form of the polynomial is as follows:

$$Y' = A + BY + CY^2 + DY^3 \quad (5\text{-}1)$$

Y is the value at time X

Y' is the new value of Y at time X

The first screen prompts for the file to be edited. By default, the target file name is the same as the input file name. If this is the case, then the new file will overwrite the old, and overwrite permission will be requested. If preservation of the original file is desired, then enter a different name in the target file name field (Figure 5-17). The screen display shown in Figure 5-20 is then presented, where B is set to unity and all other multipliers are set to zero. These default settings ensure that the time history will remain unchanged if the polynomial transformation is accidentally initiated.

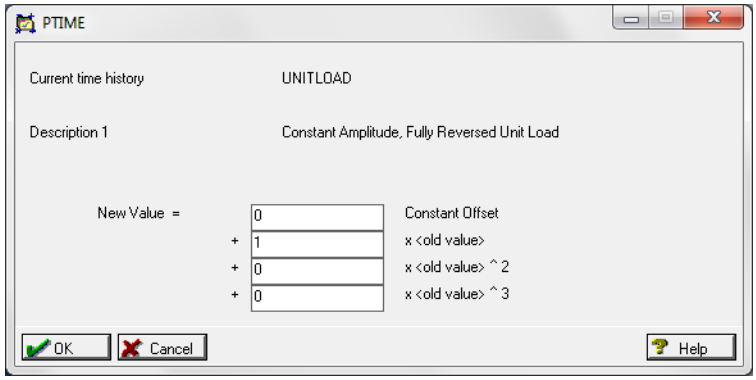


Figure 5-20 The Polynomial Transformation Setup Screen Display

Once the parameters A, B, C, and D have been defined as necessary to represent the polynomial to be used in the transformation, click the **OK** button. The user is then presented with the Edit Details form so that the details of the file may be changed if required. This is explained in the Edit Details, 237 option.

Lookup Transform Option

The Lookup transformation takes a user-defined relationship between the input Y values and the output Y values.

The first screen prompts for the file to be edited. By default, the target file name is the same as the input file name. If this is the case, then the new file will overwrite the old and overwrite permission will be requested. If preservation of the original file is desired, then enter a different name in the target file name field (Figure 5-17).

The relationship is defined in an ASCII file, an example of which is shown below in Table 5-2 Linear interpolation is used between data points

Table 5-2 An Example of a Lookup Table in Two Valid Formats

Table A	Table B	
-1000,-4000	-1000	-4000
-100,-300	-100	-300
-10,-20	-10	-20
0,0	0	0
10,20	10	20
100,300	100	300
1000,4000	1000	4000

Important: It is important that the lookup table covers the full range of the loading to be manipulated (i.e., from the maximum to the minimum Y value).

To specify the ASCII file, the screen display shown in Figure 5-21 is displayed.

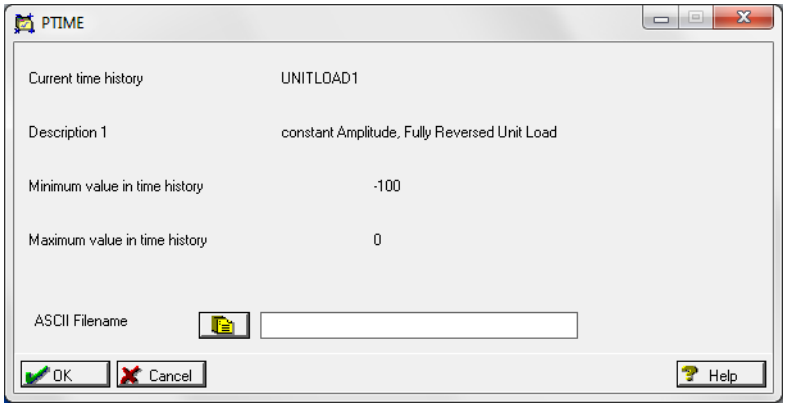


Figure 5-21 The Lookup Table Transformation Setup Screen

If the lookup table does not span the Y range of the loading, the user will be informed and asked to supply an alternative lookup table file name.

As with the Polynomial Transformation option, the user is then presented with the Edit Details form so that the details of the file may be changed if required. This is explained in the Edit Details, 237 option.

Units Conversion Option

The Units conversion option has been provided to allow switching of the units of any loading into any other units of the same type (e.g., Pressure in Pa, MPa, psi, ksi, etc.). If the units supported by MSC Fatigue do not

include the desired unit, it is possible to extend the units list to include a new unit providing the relationship to S.I. units is known. However, the units may only be extended by the system manager. See also Loading and Units, 283.

The first screen prompts for the file to be edited. By default, the target file name is the same as the input file name. If this is the case, then the new file will overwrite the old and overwrite permission will be requested. If preservation of the original file is desired, then enter a different name in the target file name field (Figure 5-17).

To specify the new units, PTIME will then display a screen similar to Figure 5-22.

Use the mouse to select the new units from the listbox. Note how PTIME automatically presents a list of appropriate units. For example, it does not offer inches as a potential unit of force. See the Edit Details, 237 options if the expected units do not appear in the listbox because the load type is set incorrectly.

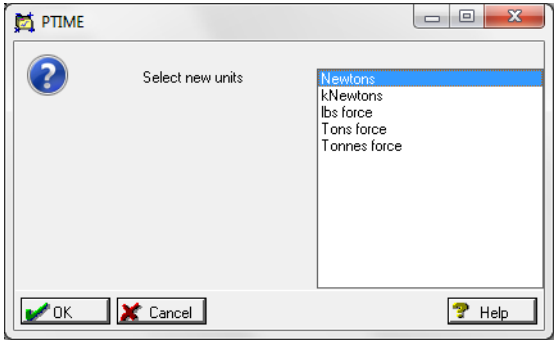


Figure 5-22 The Units Conversion Screen

Sample Rate Adjust Option

The Sample rate adjust option allows for increasing or decreasing the sample rate of the loading files. This is sometimes necessary since loadings which have come from different sources may be used simultaneously in a fatigue analysis. Rather than adjust the sample rate online for every node being analyzed in a fatigue analysis, it is necessary to match the sample rates prior to the fatigue analysis.

A new sample rate which is either higher or lower than the original sample rate may be chosen. New data values that lie between existing ones will be obtained using interpolation. Two forms of interpolation are provided.

Method	Description
Linear Interpolation	This is quick to compute, but less accurate than spline fit when the waveform contains curves.
Spline Fit	Cubic splines are more accurate for curved waveforms, but they do take more computational power to solve.

The first screen prompts for the file to be edited. By default, the target file name is the same as the input file name. If this is the case, then the new file will overwrite the old and overwrite permission will be requested. If preservation of the original file is desired, then enter a different name in the target file name field (Figure 5-17).

The following screen appears after a file has been selected as shown in Figure 5-23.

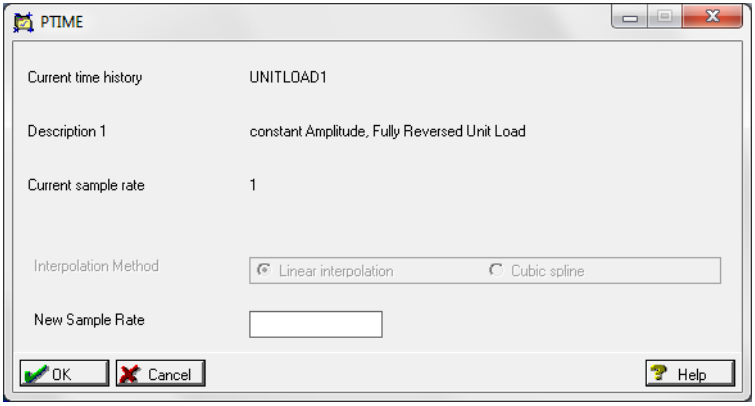


Figure 5-23 The Sample Rate Adjust Option

List All Entries Option

This is a very simple Main menu function that is very easy to operate. It lists (on screen) all the `filename.dac` that are loaded into the PTIME database manager. It also list the PSDFs, CSDFs, Rainflow matrices (histograms) and PSDF matrix files that are loaded into the database.

A typical listing is shown in Figure 5-24.

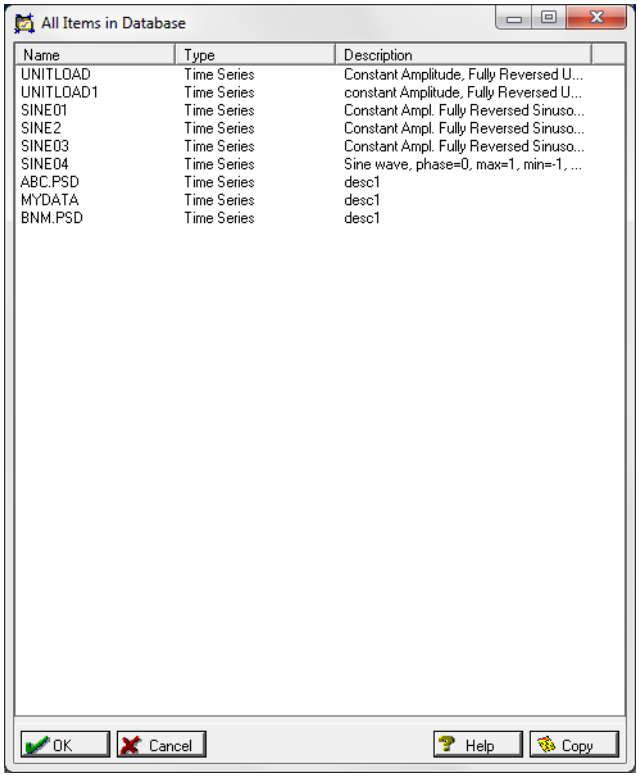


Figure 5-24 A List of Time History Entries

Search and List Option

This is a Main menu option that allows for searching of the available loading (`filename.dac`) files and extracts technical data from them. The following items may be used as search criteria.

- The name of a file.
- A generic file name.
- A text string from the first line of its description.
- A type of load and its units.

For example, it is possible to extract the details of only those files that have a load type of force, expressed in Newtons. Alternatively, it is possible to search for and list all those files with `test` in their names (`test1, test2, test3`, etc.), or for the specific file name of `test3`.

The amount of information extracted from the files can be just a Brief description (name, description, load type, units, creation date, modification date), or a Full description (all the above plus details of the number of data points, min/max points, etc.).

The output destination can be to the Screen or to a list File. An ASCII file is stored on disk with the name `pfatigue.prt`. This file can be printed out later.

On selecting the Search and list option, the screen display shown in Figure 5-25 will be presented.

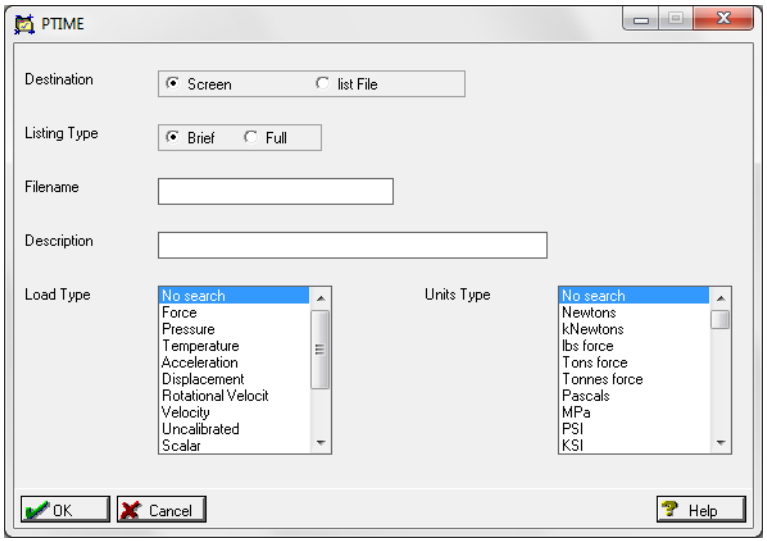


Figure 5-25 Listing/Search Option Screen Display

The Search and List option embodies two functions, as follows:

- Listing the information for a set of database entries.
- Searching for database entries with specific characteristics.

Searching

The search function is only activated if an incomplete file name is specified, or no file name at all. The search facility is independent of the text case (i.e., **a** or **A** may be used). The following fields are active during the search.

Field	Valid Entries
Description data (both lines)	Any text string
File Name	Any text string
Load Type	Only supported load types valid
Units Type	Only supported units types

Once all search parameters required by you have been defined, the search may be initiated using the **<F1>** (accept) key. When a match is found, the details of the matched database entry will be displayed on the screen

or sent to the session file according to your selection in the same way as the listing with no search option. The Full option is also available for the search, producing the same result as for a listing.

Listing

The listing may be viewed on the screen or sent to an ASCII file called `pfatigue.prt` file by choosing the options Screen or list File, respectively.

For example, a Brief listing of a file called `pdata02.dac` will produce the information similar to that shown below:

```
Filename:PDATA02
Description:Actuator flap acceleration
Load type:Acceleration Units type:g
Entry date:19:46 21-JUN-92Last mod. date :20:04 21-JUN-92
```

A Full listing includes the statistics of the actual loading data as shown below in this specific example together with the information for the Brief listing:

```
Number of fatigue equivalent units:12
Fatigue equivalent units:flights
Number of data values:497000
Sample rate (Hz):540
Minimum in series:-9.283Maximum in history:9.333
Point no. of minimum:1410Point no. of maximum:352131
Mean:-0.4845Standard deviation:4.013
R.M.S.:0.1822
```

Plot an Entry Option

This is the Main menu option that causes the loading graphical display system to be spawned (MQLD for time histories, MTPD for PSDFs, and MP3D for rainflow). The main form for this option is shown in Figure 5-26.

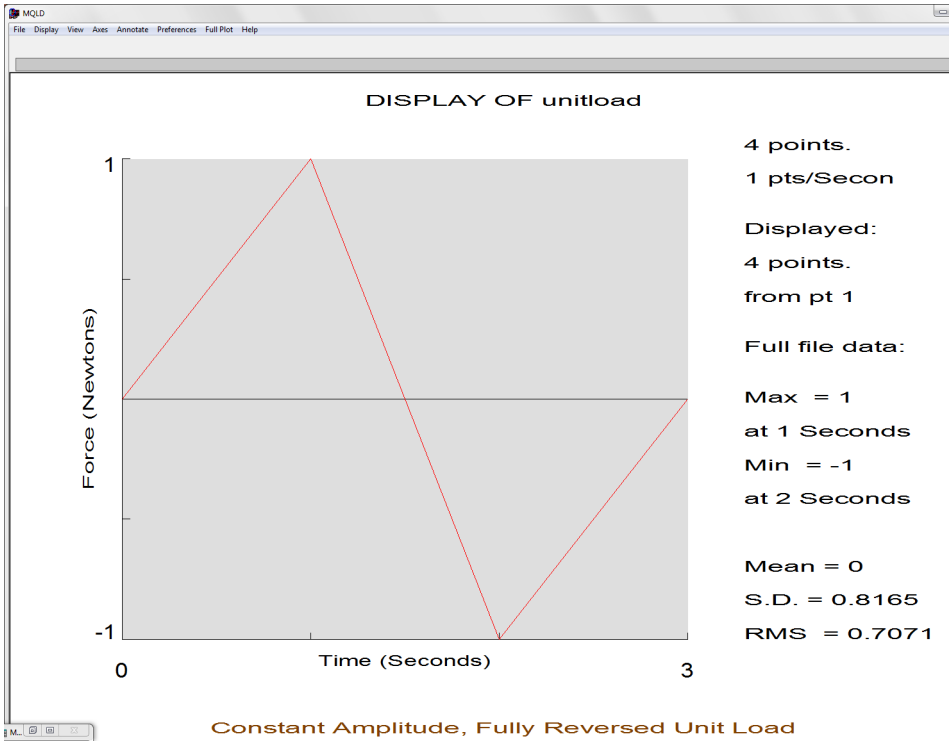


Figure 5-26 Time History Graphical Display Window

Many of the options available from the top pull-down menus are generic to the MSC Fatigue modules and are described fully in Module Operations (Ch. 18) along with the commands that are applicable in the Command databox.

Important:	The difference between the Graphical edit/create tools and this option is that the facilities available in this option only permit data inspection and hardcopy generation.

Those pull-down menu options that are specific to this graphical window are explained below. This list is not exhaustive.

Option	Description
DISPLAY	
Replot	This simply replots the plot. The same thing can be accomplished by pressing the P key.
Towers / Join Points	Presents the display of the plot in towers or as straight lines connecting point to point.
Stats On / Stats Off	Turns the statistical display of the time history on or off. The display appears to the right of the plot.
F Stats / W Stats	Displays either the Full (F) statistical data of the entire time history or just the statistical data for the portion of the time history appearing in the current Window (W).
VIEW	
Full Plot	Displays the entire signal within the visible window.
Full X / Full Y	Displays the entire X-axis or Y-axis of the signal within the current window.
Page Left / Page Right	Pages left or right one window of the signal.
Page Up / Page Down	Pages up or down one window of the signal. The signal must be transposed for this option to be enabled.
Zoom Out	Zooms out (away) from the signal 5 times.
Window X / Window Y	Requests a minimum X or Y-axis value and a maximum X or Y-axis value in the Command databox from which the signal is then brought to fit into the current window.
Transpose	Transposes the X and Y axes of the plot.
AXES	
Lox X / Log Y	Converts the X or Y axes to Log scale.
Linear X / Linear Y	Converts the X or Y axes to Linear scale.
dB Y	Scales the Y-axis to Db.
Grid On / Grid Off	Turns the plot grid on or off.
Box On/ Box Off	Turns the box around the plot on or off.
Zeros On / Zeros Off	Removes or plots the line defining the X and Y zero locations.

Option	Description
ANNOTATE	
Set Title / Del Title	Allows for setting a title or deleting a title from the plot. The title must be input through the Command databox. The default title is the description line 1 from the database.
Add Text / Del Text	Allows for adding or removing additional text on the plot. The text is input through the databox and automatically placed at a predefined location on the plot. To delete the text, click on it with the mouse after selecting the Delete Text option. Confirmation of the text will be requested.
Move Text	To place added text use this option. First select the text with the cursor and then use the cursor to place the text in the new location.
Top Label / Side Label	Moves the label of the Y-axis to the top or side.
Labels On / Labels Off	Turns any labels on or off.
MISCELLANEOUS	
P	If the P key is pressed at any time, the current option is terminated and the whole screen is redrawn.
W	If the W key is pressed at any time with the cursor over a menu option, help is displayed for that option.
V	If the V key is pressed at any time with the cursor over a menu option, that menu option is invoked.

Delete Entries Option

This is a Main menu option that takes the user to the delete screen. From this option, the user can either type the name of the entry he wishes to delete or mark entries in the database for deletion. Once the user has marked all files for deletion (denoted by an asterisk to the left of the file), he may execute the delete option.

The initial delete screen looks like Figure 5-30, which asks only for a single file name.

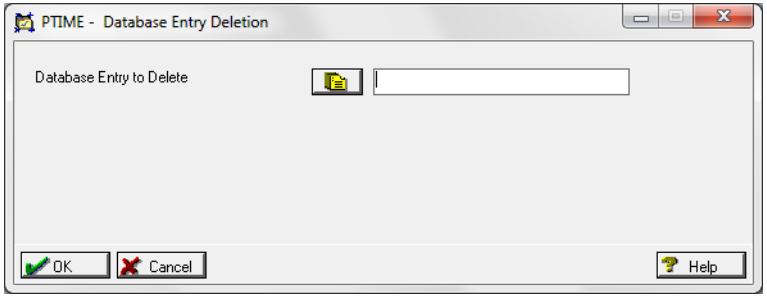


Figure 5-27 The Delete Time History Option

If deletion of more than one file at a time is required or the file name is not known, it is possible to bring up a listbox which is invoked by pressing the List button. This will give a listing of available files to delete along with their descriptions. See Figure 5-28. Once all files have been selected for deletion, click the OK button. A confirmation to delete the files will be requested.

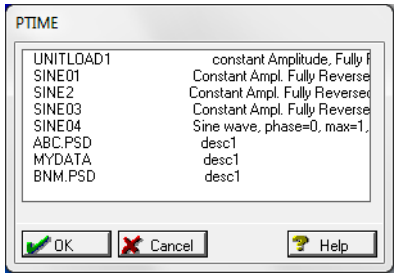


Figure 5-28 The Select List Screen for Deletion

Validate Database Option

The Validate database option takes you to the Validate submenu where two levels of validation are offered. The first is a Quick validation and the second is a Full validation which involves recalculating the statistical information of all the time history data files listed in the database. See Figure 5-29.

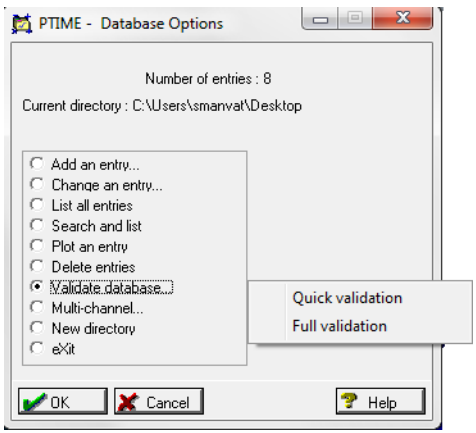


Figure 5-29 The Validate Database Submenu Options

These two options are described below.

Option	Description
Quick Validation	This option checks the local directory to ensure that all the files in the database exist and reloads the database with the statistics of the files from the file header blocks. Statistics include values such as max, min, and mean. In addition the description (line 1) and the equivalent units are stored in the file's extra details area. The description is stored against the keyword GRPTITLE and the equivalent number and units are stored as NUMEQUNI & EQUUNITS. It is envisaged that these files and the specific information may be shared with other users. The items in the extra details area are updated in the file from the PTIME database. The description line 2 is stored only in the PTIME database as this is intended for the analyst to make model specific comments.
Full Validation	The full validation option recalculates the statistics of all the files in the database and reloads the database with the newly calculated values as well as carrying out the quick validation checks. This option may take some time to execute.

Multi-Channel

This option allows for display of multiple time histories simultaneously and also includes a utility for extracting peaks and valleys from existing time history files and reducing them. One of two separate processes are actually spawned when one of these options is specified.

When multiple file display of time histories is selected a MSC Fatigue module called MMFD is run. Operation of this module is fully described in Multi-File Display (MMFD), 251.

When peak valley extraction is selected a MSC Fatigue module called MPVXMUL is run. Operation of this module is fully described in Peak-Valley Extraction (MPVXMUL), 254.

New Directory

This Main menu option allows the user to change the directory in which the loading database will be or is stored. All `.dac, .psd, .chy, .pmx` files and the `ptime.adb` and `ptime.tdb` files must reside in this directory or will be created in the directory indicated in Figure 5-30.

The default is the current directory and is indicated by a dot (.). Enter the full path name of the new directory and end with a slash (/).

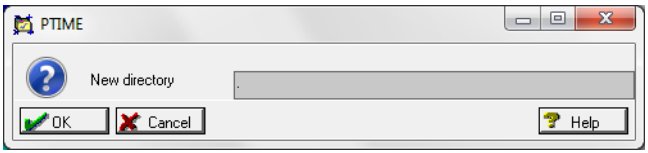


Figure 5-30 The New Directory Option

Exit Option

The Exit option will exit the loading database manager (PTIME) and return control to Patran or return the user to the system prompt. At this stage, a new ASCII database index file (`ptime.adb`) will be produced. This file is necessary for Patran to recognize which loading files are available for job setup.

Multi-File Display (MMFD)

This module is used mainly to compare channels of measurement data in a fast and convenient way, MMFD displays files as graphs. Single files are also easily displayed. Up to 32 files can be plotted, either separately, or at up to 8 files per page. Files can be overlaid (superimposed) on a single set of axes, or even cross plotted against each other to test correlation. Scaling, tracking and scrolling functions are available. To invoke this program from the system prompt type **mmfd**.

The MMFD module can display several data files simultaneously. It is intended primarily to display time series data such as single parameter `.dac` files, however, other data files with a single parameter structure can also be displayed.

As stated earlier, up to 32 data files may be plotted during a single run of the program. The plots may be presented either separately or overlaid. In this way data from a particular measurement campaign can be easily compared and contrasted.

To aid the process of comparison, the Y-axes of the displays can be scaled in any one of three ways:

Field	Description
Local (Self) scaling	Each channel scales to its own maximum and minimum values.
Global (Auto) scaling	All the channels scale to the channel with the largest global maximum or smallest global minimum, auto-scaling. This type of scaling can be achieved by setting the optional scaling toggle on screen 1 of MMFD to Global.
Standard scaling	In this mode the local Y range is used but the x-axis (time) units are set from the lowest minimum to the highest maximum x values extracted from the set of files.

There are three types of plots that can be displayed:

Field	Description
Standard plots	These are normal plots displayed next to each other.
Overlaid plots	These are plots displayed on top of each other
Cross plots	Data files can be cross plotted. This is a process whereby up to 7 files can be plotted against a nominated reference file. The reference file will represent the X-axis and the other files are plotted on the Y-axis. This is an excellent way of assessing the correlation between sets of data. A high degree of similarity between data sets will produce a straight line cross plot. Low correlation will produce a scattered plot.

In cross-plot mode, there are facilities to setup exclusion zones, adjust plotting resolution and slow down plotting by time delay to aid in the viewing (and plotting) of data.

In standard and overlay modes there are facilities for scrolling through the data and tracking individual data sets for coordinate values to aid in the viewing of data.

Input File Names

MMFD can display the contents of a single data file or a group of files. The concept of a generic name for a test represents a quick method of entering channel names for processing. The methodology is based on the fact that data derived from the same test, are usually demultiplexed into individual channel files whose file name structure consists of a generic base (test) name, normally the name of the multiplexed data file itself, with the appropriate channel number appended. For example, if the name of the multiplexed data file happened to be DATA, then the following file names would be generated: `data01.dac, data02.dac,..., datann.dac`.

In the above example, the generic name of the test would be `data`. So entering `data(1-n)` where n is the last number in the series will process all files prefixed by the word `data`. If fewer files exist than are specified then MMFD will display an error message prompt to that effect and proceed to display the data files that do exist.

Since MMFD expects to process standard time series data files (usually a `.dac` file) the above generic test name convention only applies to this data type. Files with the correct internal format but different file extensions, such as spectra with the extension `.psd`, must have their names entered in full, i.e. including the file extension e.g. `data(1-n).psd`.

MMFD expects to find the data files for plotting to be resident in the user's directory, however, other directories can also be accessed if the complete file specification i.e. path name and file name are entered.

If data files unconnected by a generic test name are to be processed then enter their full file names separated by commas.

Entering File Names Using F3/List

A very easy way to select files is by using the List facility. List displays all available files on a scrollable pick list. Files on the list can be selected by pointing and clicking with the mouse pointer. For technical reasons

the files picked from this list are not remembered once you have progressed beyond this screen, although the path is remembered and used in the next run of the program.

Typing File Names

If more than 1 individual file name is typed then the last name in the list is saved to the environment and is used as the default file name the next time the program is run. Also files that are typed are remembered once you have progressed beyond this screen but return to it. Files must be separated by commas.

Up to 32 individual file names may be entered.

When the files have been selected, MMFD will display either the names of the files or the number of files selected.

Alter Setup Yes/No

Generally speaking, as a default, MMFD will plot the entire contents of each data file separately on the screen and scale each data set to its own local maximum and minimum values. If something other than these defaults are required, then answer this question YES. Having requested to alter the default settings up to 8 additional fields will be displayed.

Field	Description
Plots per page	This field allows the default number of plots per screen to be set. The maximum range is 1-8 for separate plots and 2-8 for overlaid and cross plots. The actual range offered depends on the number of files presented. If 12 files are presented the minimum value is 3, i.e. 3 per page for the 4 pages. The selected value is written to the environment and used as the default when the program is next run.
X and Y scaling	X Scaling Linear/Log 10 Y Scaling Linear/Log 10/dB These fields allow the user to choose the X and Y-axis scales.

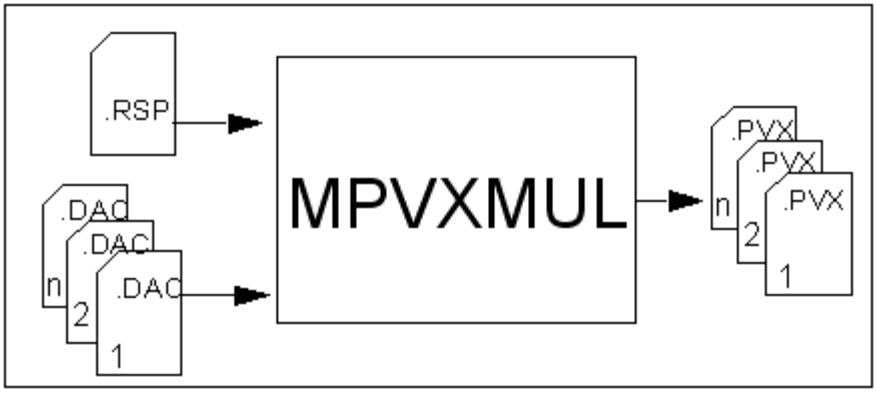
Field	Description
Axes Limits	The axes limits settings can be global, local, or standard.
Maximum/Minimum X/Y	These fields appear if the axes limits are set to global. The default values are the highest maximum and lowest minimum X and Y values across all the files. Note that on color screens or hardcopy devices that support different pen colors, MMFD will plot each data set in a different color. For monochrome devices, each plot will be distinguished by a different line style.
Cross Plot Type	If Cross Plot is selected then X-Scaling and Y-Scaling appear as in Display Type = Separate (above). The new field that only appears for Cross Plot is Plot Type = Scatter - Joined - Both The selection of scattered or joined plots is a matter of preference and clarity, some files look better cross plotted and some are clearer with joined plots. However, please note that scattered plots take 30% longer to plot than joined plots. So if large files are being plotted then selecting joined plots will save time.

Graphical Display

Graphical display operations in MMFD are similar if not identical to that of PTIME. These operations and the description of pull down menu options are described in Plot an Entry Option, 245.

Peak-Valley Extraction (MPVXMUL)

This program performs peak valley extraction i.e. it extracts turning points (maxima and minima or “peaks” and “valleys”) from single parameter files such as `.dac` and on RPC multiple data-channel files. The analysis is performed such that the phase relationship between all the channels is not lost, when peaks and valleys occur on different channels at different times. To invoke this program from the system prompt you type **mpvxmul**.



A hysteresis gate can be set independently for each channel, allowing small oscillations on the data (noise) to be ignored.

Output files are created, one per channel (RPC files), or one per .dac file. All outputs are .dac files. Extra details, such as equivalent units, are automatically transferred to the output file.

Input Data Files, .dac and MTS RPC

Single parameter .dac files can be processed by MPVXMUL. These exist as families of files with a common generic name, but with different numbers appended to the name which denotes the channel number. For example test01.dac for channel 1, test02.dac for channel 2, etc. test is the generic name.

Remote parameter RPC multiplexed files from MTS[tm]can also to loaded into MPVXMUL and each channel converted into .pvx files. Both RPC II and III file types can be processed. See the MTS documentation for full details of their RPC file types.

The Range Method

Whatever the input, an output .pvk (Peak Valley extraction) file will be generated for each channel processed.

Within a time series, a turning point is defined as a local peak or valley, i.e. a value within the time series at which its direction changes. At the start of the peak-valley process, only one turning point can be defined with any certainty and that is the largest absolute value in the signal, be that the signal maximum or minimum. MPVXMUL starts its processing at this point and continues on to the end of the signal, then it loops back to the start of the data until the absolute maximum start point is reached again. The peak-valley values are written out to the output file in the original sequence, i.e. starting with the first. This form of output is exactly what is required by the critical location fatigue analyzer, FEFAT.

During the course of the peak-valley extraction process, the number of turning points detected can be restricted by imposing a hysteresis “gate”. This gate corresponds to the smallest difference between adjacent turning points that can be accepted. For turning points to be counted, they must be separated by a distance greater than the specified gate. By these means, small disturbances or “noise” in the time series may be “gated out” from the set of extracted turning points.

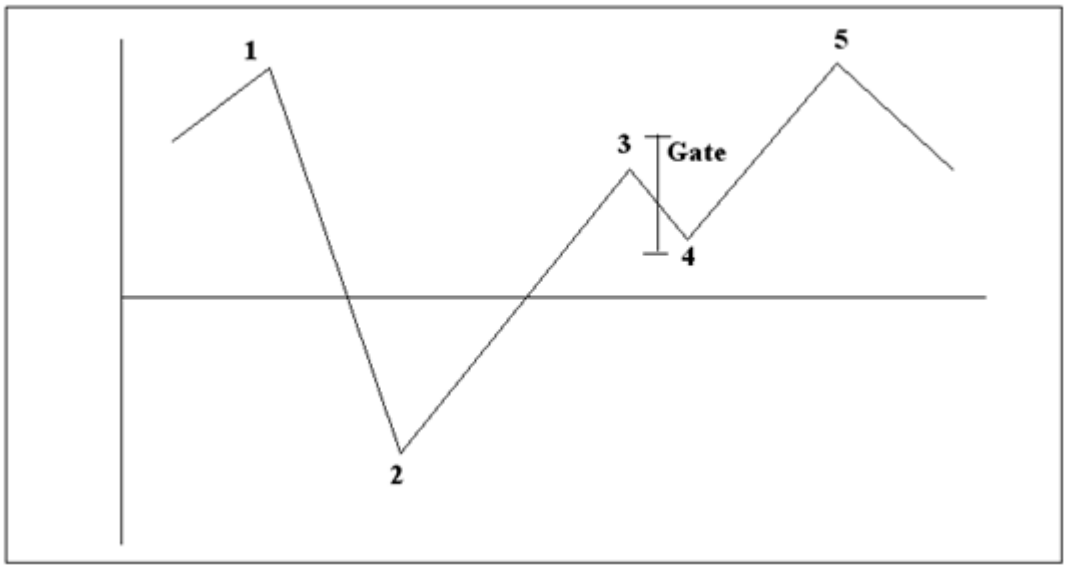


Figure 5-31 Hysteresis Gate

In Figure 5-31 turning points 3 and 4 will not be counted since their separation is less than the specified gate.

MPVXMUL can process the input time series data file in its entirety or just a selected portion. The analysis can begin at a particular time, by default this time is 0.0 i.e. the start of the signal, and end at another time, greater than the start time, usually the end of the signal. By these means it is possible to define a time window for the peak-valley extraction process.

The output data file (.pvx extension) contains turning points in physical units and has the same file format as the input data file (.dac extension). The output file can therefore be manipulated and displayed for example using PTIME or MMFD.

Any information stored in the Extra Details Area of the input file are automatically carried forward to .pvx file. The EQUUNITS and NUMEQUNI values are carried forward so that the fatigue analyzer FEFAT can report predicted limits in equivalent units.

The Rainflow Procedure Method

The rainflow algorithm used by MPVXMUL is based on the standard practice for cycle counting in fatigue analysis as defined by the ASTM designation E 1049-85, (See ASTM standards Vol. 03.01). Under these circumstances, the rainflow cycle count will be identical to a range pair cycle count which itself starts at the largest value. Differences in the results produced by the two procedures only arise if processing starts at some value other than the absolute maximum.

It is possible to illustrate the rainflow cycle counting procedure used by considering the simple strain time series shown in Figure 5-32 below.

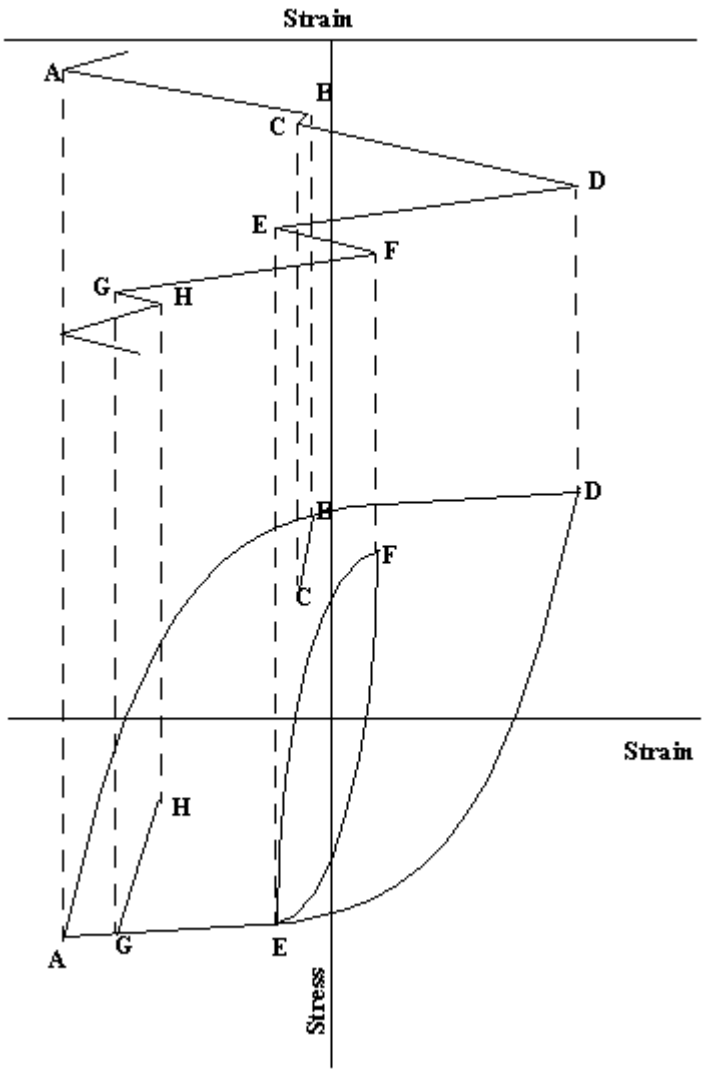


Figure 5-32 Illustration of the Rainflow Procedure

Figure 5-32 shows a strain sequence of four turning points, and the stress-strain response of a material to this sequence. The closed hysteresis loop is a cycle, which may be characterized in terms of its strain range and mean strain.

If the stress axis of this diagram is ignored and only successive strain ranges are considered, then an algorithm can be developed which will extract cycles from a signal whatever its units. The rainflow algorithm is able to extract cycles in the way described above, classify them in terms of their range and mean value, and store them in a range mean matrix.

The term rainflow derives from an algorithm, in which cycles are extracted through a consideration of rain drops flowing down a pagoda roof. Modern algorithms no longer use this concept although the generic name “rainflow” still persists.

Module Operation

The first screen allows the type of input files to be selected. The program will work with multiple .dac files or RPC files.

Normally, file type selection is available, as illustrated above, but the file type may be fixed before running MPVXMUL by setting the environment variable $SIMXTYP equal to DAC or RPC.

There is therefore two distinct paths through MPVXMUL; however, they differ only in detail.

If .dac files are being processed then a generic .dac file name must be supplied.

If RPC files are being processed then only a single file name has to be supplied.

The next few screens presented will request that certain parameters be set. These are explained here.

Field	Description
Generic Input File Name	The family of .dac files which are specified for input must all have names which are constructed as follows: `<generic name><channel number>.dac` where the generic name is the same for all the files. The channel numbers are two or three digits padded with a leading zero where required. The generic name should be entered in this field. A directory path may be included if required.
Channels	Up to 999 channel numbers may be specified by entering them in the form `1,3-15,35-60, etc.` where the hyphen represents an inclusive range of channel numbers. Alternatively, the word 'ALL' may be entered in the answer field to indicate the use of all the channels which match the given generic name.
Output File Name	One output file is created for each input channel processed. The output file names are also constructed according to the generic name plus channel number syntax described above. Enter the generic part of the output file names in this field. This name may include a directory specification if the output files are to be written into a different directory. The extension of all the output files will be .pvx. If one or more of the output files already exist with the specified name, the program will ask whether to overwrite the existing files or not. If the answer is NO, then a different output file name will be requested.
Input data time span	This is purely for display purposes and is a statement of the length of the input data files, and the number of points that each of them contains.

Field	Description
Set Limits As	This field and the following two allow a section of the input file(s) to be processed, rather than processing the complete file(s). This is done by setting a start and end position. This field determines the method by which the start and end position are to be set, the options being Time or Point number. The following two fields, specifying the start and the end position, will be changed to reflect the appropriate units.
Start Position/Time	As described above, this field may be either Start Time or Start Point. Times are measured from the time origin, which is normally, but not necessarily always, the start of the file. Points are numbered from 1 at the beginning of the file. The word 'START' may be entered signifying the start of the data.
End Position/Time	This field sets the end position in the chosen units. The word 'END' may be entered signifying the end of the data.
Gate Method Range\Cycles	Choose the method by which data values will be gated. Range - each consecutive range value is checked against the specified gate. The reference value is the maximum range of the data which MPVXMUL determines before gating out those values that are less than the specified percentage of the maximum range. This is the method used by previous versions of PVX and MPVXMUL and is included for compatibility reasons. Cycle - A rainflow analysis is performed and the size of the cycles are then compared to the specified gate value. This involves an intermediate stage in which a rainflow file is created and means that it is possible for non-consecutive points to be extracted or excluded. This method is smarter and produces superior fatigue analysis results.
Gate Values	When the above fields have been defined, MPVXMUL moves onto the Hysteresis Gate Setting screen. This screen allows the gate value to be set independently for each input channel. The gate value may be set directly in the units of the data on the channel, or may be entered as a percentage. If a percentage, the actual gate value used will be that percentage of the maximum range of the data on the respective channel. This screen may consist of more than one page, since only 16 channels can be displayed at once. To move between the pages, use the Page Up, Page Dn, Home and End keys.

The fields are as follows:

On completion of all input parameters, the analysis will be performed and the results files written to disk.

A processing summary screen is displayed.

Batch Mode (MPVXMUL)

The following keywords are defined for batch mode operation:

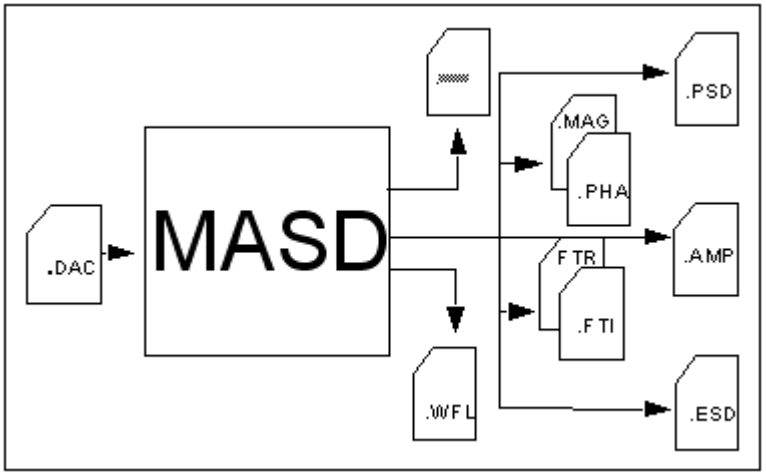
/TYpe	Type of input file either D or R (/TY=D)
/GENeric	Input generic file name (/GEN=test)
/CHAnnels	Channels (/CHA=5,7,9,11)
/OUTput	Output file name (/OUT=fred)
/OVerwrite	Overwrite output files Y, N (/OV=Y)
/METhod	Limit setting method, either T or P (/MET=P)
/STArt	Start position (/STA=10)
/END	End position (/END=20)
/GATMTH	Gate method, either R or C (/GATMTH=R)
/GATe	Gate value (absolute value)
/PGATe	Percentage gate value (% of range)
/CGATxxxx	Gat value for channel xxxx, where xxxx is zero padded, i.e., channel 1 is 0001. This is only available if GAT or PGAT are not used.
/*=TT	If the user wishes to see output sent to the screen, he must include this parameter exactly as shown. Any other value other than TT after the equal sign will send the output to a file by that name.

Example:

```
mpvxmul /ty=d/gen=test/ov=y/cha=all/met=t/sta=start
/end=end/gatmth=r/pgat=50
```

Auto Spectral Density (MASD)

The Auto Spectral Density program, MASD, performs a frequency analysis on a single parameter input file, a `.dac` file for example. MASD produces an output file that indicates the frequency content of the input file.



Most physical events occur at a certain speed, with a certain magnitude, and repeat at a certain rate. Engineers need to analyze such events in terms of all these characteristics. While all events occur in time and space, engineers and mathematicians have devised an alternative way of viewing events. They refer to this as the frequency domain as opposed to the time domain.

The French mathematician J Fourier (1768 - 1830) showed that any periodic motion can be represented by a series of sines and cosines. This can be demonstrated simply by considering a square wave. Such a waveform can be represented by the following Fourier series:

$$Y = a_0 sin(t) + \frac{a_0 sin(3t)}{3} + \frac{a_0 sin(5t)}{5} + \frac{a_0 sin(7t)}{7} + \frac{a_0 sin(9t)}{9} \ldots \quad (5\text{-}2)$$

There are some very good reasons for working in the frequency domain. One of the most important is that some complicated operations in the time domain become simple in the frequency domain, for example convolution in the time domain becomes a simple multiplication in the frequency domain. In addition, the relationship between the excitation and the response of a structure is often more easily understood in the frequency domain.

Real events are analogue (continuous) and are often random and non-periodic. Random processes continue for an infinite length of time, but are usually observed over a finite time. By carrying out a Fourier Transform (FT) of the finite length time domain data segment, the data is transformed into the frequency domain.

If the data is sampled for use in a digital computer, it is not possible to use the continuous form of the Fourier Transform and a discrete equivalent is required (a DFT):

$$x_m = \sum_{k=0}^{N-1} x_k e^{-i2\pi m \frac{k}{N}} \quad \text{m=0,1,...N/2} \quad (5\text{-}3)$$

where N is the number of points in the time history and x_m is the kth Fourier coefficient.

The Inverse Fourier Transform (IFT) also exists which transforms data from the frequency domain into the time domain. This reverse transformation is not discussed here but the reader is referred to standard texts for more information (Refs. 41,42).

One of the problems of the DFT is that calculation times are long when there are a lot of data values, i.e. when N in (5-3) is large. Many optimized methods have been proposed, collectively described as "Fast" Fourier Transforms (FFT). One such method was proposed by Cooley and Tukey (Ref. 43) and is published by the IEEE. One of the compromises made in the FFT is that a particular number of data values must be analyzed at any time. However, the fast Fourier transform and the DFT can be regarded as identical for the purpose of this discussion. (The algorithm used in the frequency analysis tools is based on the IEEE optimized algorithms (Ref. 44)).

The basic FFT process carried out on N data values sampled at F samples per second results in $N/2$ complex numbers representing a one sided spectrum of frequencies with a bandwidth of $F/2$ Hz. These raw complex Fourier coefficients may be postprocessed in a number of ways.

Window Functions

Since it is rarely practical to transform a complete data segment, the analysis is carried out using short buffers of data taken at increasing times through the sample. However, start and end data values in the buffer are rarely identical, resulting in an effective discontinuity in the signal being analyzed. To reduce these effects, the buffer is shaped using a window function.

The default window function if no special window function is applied is a rectangular one, that is to say no data window is applied. Other common types of spectral window functions are shown in Figure 5-33 below. The choice of which window type to use will depend on the type of data to be analyzed. However, it is always a good idea to analyze the data first using the rectangular window function.

If a window function is used, a correction for effective bandwidth is applied to the complex results of the FFT. This correction depends on the type of window and is carried out automatically by the software.

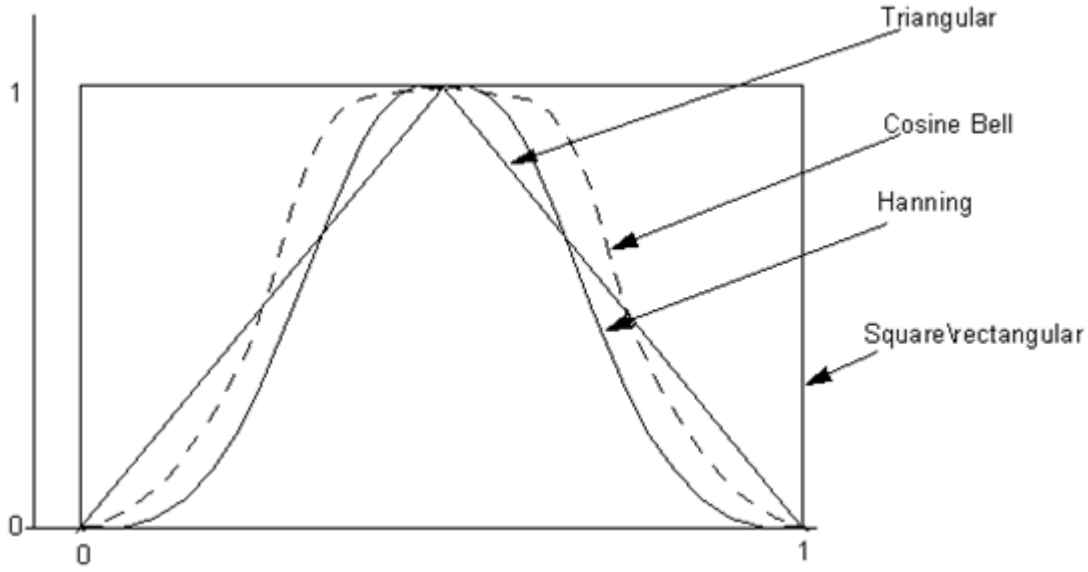


Figure 5-33 Spectral Window Functions (Schematic)

Ensemble Averaging

The FFT coefficients from each buffer are normally summed in some way to produce a result representing the average frequency content over the whole of the sample. The greater the number of buffers contributing to this average, the more confidence may be placed in the result. Since the data windows attenuate data at the start and end of the buffer, some data is not used in computing the average. Consecutive buffers may be overlapped to use all the data in the signal. An overlap of 50% means that data at the end of one buffer is located at the center of the next buffer where the end effects are not present.

If the changes in the frequency components over the length of the whole sample are of interest, the buffers should not be averaged. The results from this type of analysis may be presented using a plot where the results from each buffer form one line in a three dimensional plot of frequency versus time versus spectral estimate. Such a plot is known as a ***waterfall plot*** (see Figure 5-34).

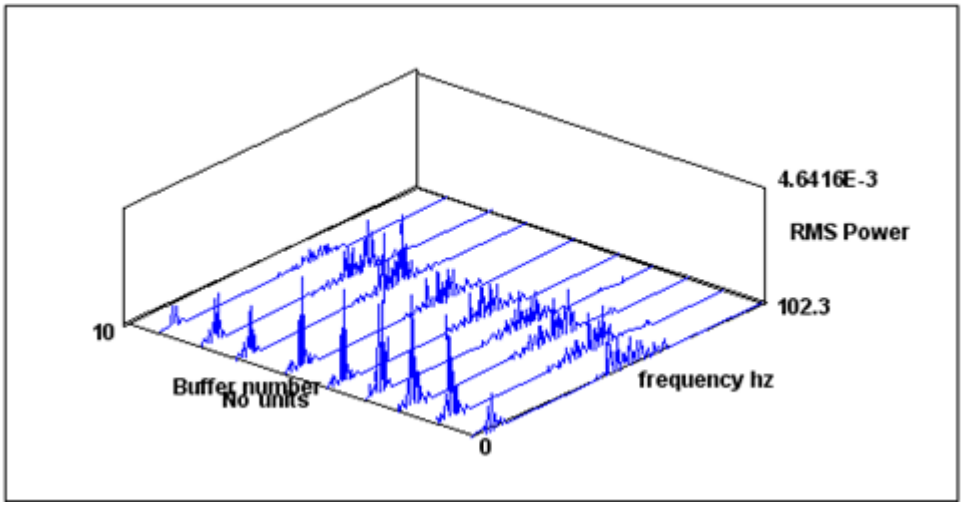


Figure 5-34 A Waterfall Plot.

Module Operation

The MASD module can be run in one of the following three modes:

- From PTIME menu driven system - PSD from time.
- In stand alone mode by typing **masd** at the system prompt
- By incorporating the MASD commands in a batch operation

Once running in interactive mode the MASD module will display the following screen.

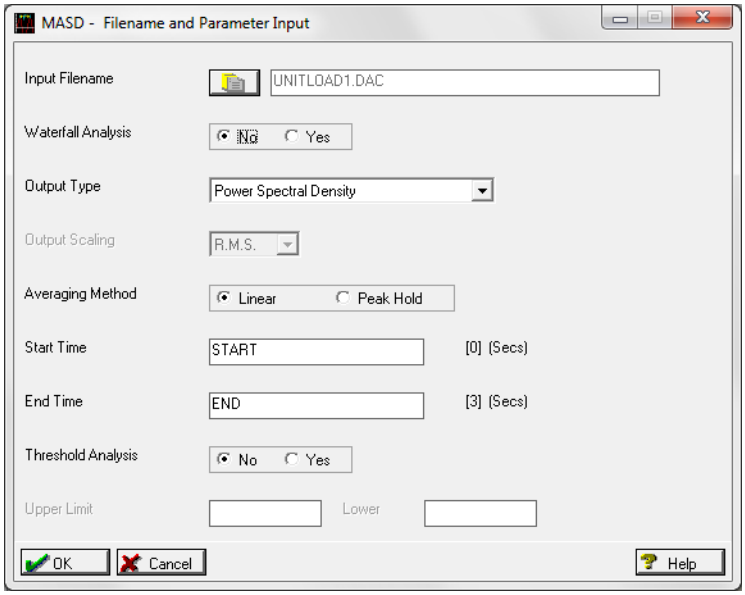


Figure 5-35 The MASD File Name & Parameter Screen

The purpose and usage of each field is explained below. Note that until Input File name is completed and confirmed (by clicking **OK**), the other fields are grayed out.

Field	Description
Input File Name	In this field the user should type the name of an input file (usually a single parameter `.dac` file). By default MASD assumes a `.dac` file extension but if a file with a different file extension is to be processed then enter the file name plus extension in full. MASD carries out a frequency analysis on a time series. MASD will expect to find the input data files resident in the user's directory, however, other directories can also be accessed if the complete file specification i.e. path name and file name are entered, e.g. `/demo/filename`. Probably the easiest way to name an input file is to use the pick list facility.
Waterfall Analysis (Y/N)	Within the section of data being analyzed, a single spectrum can be produced showing the frequency content over the whole section. Alternatively, multiple spectra can be produced showing the change in spectral content as the time series progresses. The multiple spectral display is called a Waterfall Plot. Choose Yes to produce a multiple analysis.
Output Type	The complex spectrum created by this program can be one of three types. Power Spectral Density (PSD) where the magnitude is scaled, squared, and divided by the spectral width. Amplitude Spectrum (MASD) where the magnitude of the FFT is scaled to indicate the amplitude of the original data. Energy Spectral Density (ESD) which is defined as the PSD multiplied by the analysis time. In addition to these options it is possible to output the Real and Imaginary parts of the FFT or the magnitude of FFT. The FFT algorithm returns real and imaginary components for a single sided spectrum. The -ve frequency mirror is ignored although this is compensated for by scaling up the magnitude by a factor of 2. ***These options are only available if the data will fit into a single buffer.***
Output Scaling	MASD allows the user to choose between displaying the amplitude as RMS or as true values. For example a sine wave of amplitude (and true value) 1 has a RMS value of 0.7171. To calculate and display the true amplitude of the frequency components, select Peak, otherwise the default is RMS.

Field	Description
Averaging Method	When many FFT buffers are calculated, the spectral estimates for each buffer can be handled in 2 ways. One simple way is to ***linearly*** average the component values over the number of buffers calculated. This corresponds to the default linear option. The disadvantage of this method is that fast high amplitude events that occur in a single buffer are lost when the average of many buffers is taken. The ***peak hold*** option retains the largest FFT for each component and thus eliminates the disadvantage.
Start Time (secs.)	The frequency analysis may be carried out on a specific portion of the time series. The start and end times for this analysis window are user- selectable and are requested in the units of the time base for the input time series. The default value offered is the base value of the signal. Keywords can be used, for example START+5 will start the analysis 5 seconds after the START of the file.
End Time (secs.)	The end time of the analysis window may be selected by the user or defaulted to the end time that MASD reads in the input time series file. Keywords can be used, for example END-5 will end the analysis 5 seconds before the END of the file. When all the fields shown in Figure 5-35 have been completed and confirmed (by clicking OK) then the screen shown in Figure 5-36 is shown.

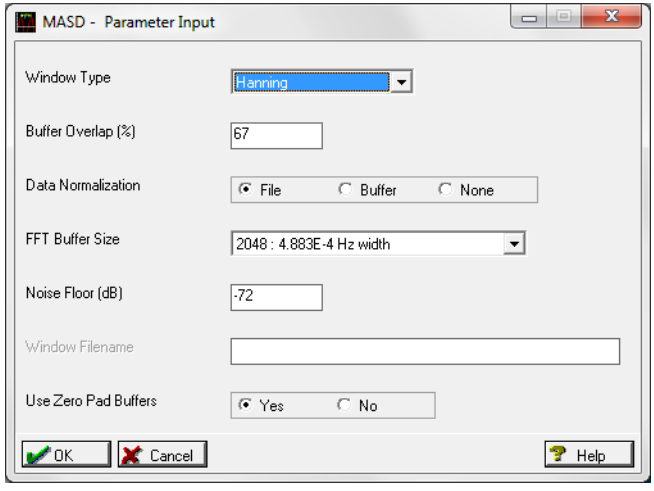


Figure 5-36 MASD's Parameter Input Screen

Fields that are not appropriate to the choices made in Figure 5-35 are not active on Figure 5-36. This means that the Overlap (%), Normalization, and Window File name, fields may be grayed out in certain instances.

Field	Description
Window Type	Each data block is tapered using the window function. A window improves accuracy of the FFT because it reduces the magnitude of the data block in a gradual way and avoids discontinuities at the extremities of data blocks. The window functions are as follows. ■ Rectangular ■ Hanning ■ Kaiser-Bessel ■ Triangular ■ Cosine Bell ■ User Defined If a user-defined window is required then MASD looks for a user definition file with a `.uwf` file extension. A user definition file is created using a `.dac` file creation program such as Graphical Create, 223, or Multi-Channel Editor - (MCOE), 970. Alternatively the user could create the file in ASCII and then use ASCII Convert + Load, 217 to convert it to a `.dac` file. Note that the user-defined file MUST be the same size as the FFT buffer size.
Buffer Overlap (%)	By default spectral windows are overlapped by 67%. The purpose of overlapping spectral windows is to minimize the effects of "leakage" at the edges of each window. Clearly the greater the degree of overlap the longer it will take to process a given signal. ***Negative values of overlap will cause MASD to skip data between spectral windows.***

Field	Description
Data Normalization File/Buffer/None	Normalizing the input time series data prior to analysis reduces the DC contribution (or power at 0 Hz) which can often swamp the rest of the spectrum. Note that selecting normalization prior to analysis will not permanently alter the input signal. **File** means that the mean of the entire file is removed from each buffer, even if the whole file is not selected. **Buffer** means that each individual buffer has its mean calculated and then subtracted from the data in that buffer.
FFT Buffer Size	The minimum buffer size available is 32 and the maximum is 131072. For buffer sizes greater than 8192, a slow FFT is used. The FFT buffer size defines the resolution of the power spectrum. The buffer must be a power of 2 and the longer the buffer, the higher the resolution of the spectral lines. To calculate the resolution divide the Nyquist frequency by half the FFT buffer size. E.g. if Nyquist=178 Hz and the FFT buffer size selected is 1024 the spectral lines are 178 / (1024 / 2) = 0.347 Hz apart. Another use of a smaller buffer size is for short data files as these cannot be adequately analyzed with a big buffer, since there may not be enough data to give a good spectral average. Using a smaller buffer size could give a better spectral average at the expense of spectral line width.
Noise Floor	When the amplitude of a particular frequency component is small, the FFT coefficients become vanishingly small and may cause computational difficulties or distort results; for example, the phase calculation is particularly affected. In order to overcome these difficulties, a value may be defined which represents an effective zero. This cutoff point is normally specified in dB down from the maximum magnitude. The default is -72dB. Note that -20dB is one order of magnitude less than the maximum.
Window File Name	If a user-defined window is required then MASD looks for a user definition file with a `.uwf` file extension. See Window Type above.

When the fields in Figure 5-36 have been completed the screen shown in Figure 5-37 appears.

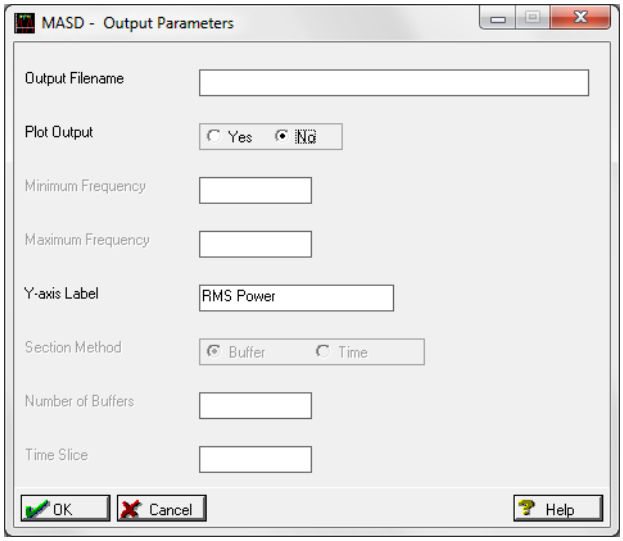


Figure 5-37 The Output Parameters Screen (MASD)

This screen enables the user to specify the output file name, whether to plot the output file, and the minimum/maximum frequency of the plot.

Field	Description
Output File Name	The results of the frequency analysis are written to an output file for later analysis or graphical postprocessing. By default the name of the output file is taken to be the input file name but with a file extension as appropriate to the analysis; for example ■ `.psd` - Power spectral density ■ `.amp` - Amplitude spectrum ■ `.esd` - Energy spectral density ■ `.mag` - Magnitude ■ `.pha` - Phase ■ `.ftr` - Real part of FFT ■ `.fti` - Imaginary part of FFT ■ `.wfl` - Waterfall file If a file with the specified or default name already exists, MASD will prompt for confirmation that the existing file is to be overwritten.
Plot Output Yes/No	Whether or not to plot the postprocessing file using MQLD. If No is selected then the plot parameter fields such as Minimum/Maximum frequency do not appear. The results of processing will NOT be plotted on screen but they will be saved as a disk file. This is also the case when waterfall files are processed.
Maximum / Minimum Frequency	If the output file is to be plotted then the frequency limits for the plot are set here. The Nyquist frequency (half the sampling rate) is offered as the maximum frequency for the spectrum to be displayed initially on the screen. The user may select this limit or specify the required frequency. It is possible to change this later when the power spectrum is displayed graphically.
Waterfall Parameters	
Section Method Buffer/Time	The selection of the file used for each waterfall spectrum can be specified in terms of time or number of buffers.
Number of Buffers	This option enables the user to specify the number of FFT buffers to be used for each waterfall spectrum (maximum buffer size = 8192, minimum = 32). At this stage, MASD will commence to do the analysis according to the user specification. During this process a message indicating the extent of progress through the analysis will be displayed. When the input file has been processed a screen of results is displayed.

A results summary screen will be displayed after all input parameters have been specified and the analysis has been performed.

The calculated power spectrum can be displayed graphically. MASD uses the MSC Fatigue module MQLD to plot non Waterfall (if Display=Y was set) MASD output files (see Graphical Display, 254). To plot Waterfall files, MASD uses module MP3D which is described in Matrix Options, 356.

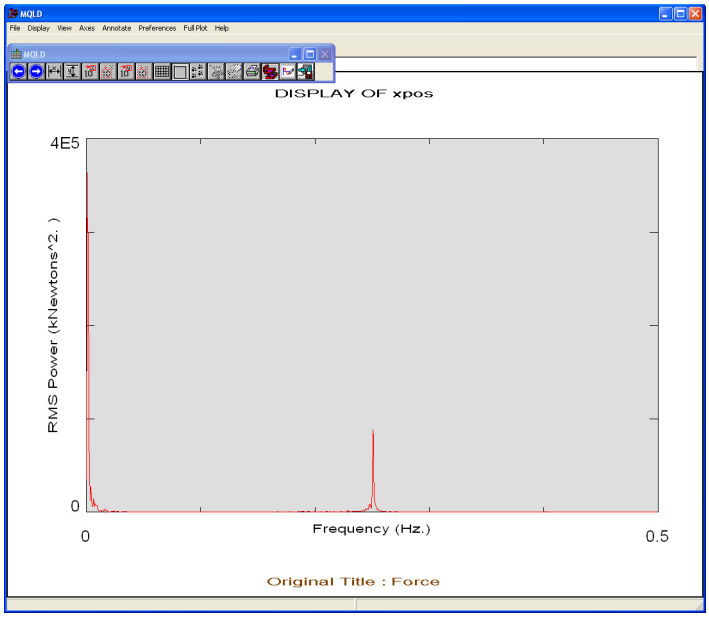


Figure 5-38 Results File Being Displayed by MP3D.

Batch Operation (MASD)

MASD can be run in batch mode as with other MSC Fatigue modules. A list of MASD's batch keywords:

/WFALL	Waterfall yes or no. /WFALL=Y
/INPut	The input file name. /INP=FILE
/OUTput	The output file name required for the results data file. /OUT=RESULT
/OVerwrite	Whether to overwrite an existing results file. /OV=Y
/STArt	The start time for the analysis window. /STA=1
/END	The end time for the analysis window. /END=20
/FMAXimum	The maximum frequency for the power spectrum plot. /FMAX=5.7
/FMINimum	The minimum frequency for the power spectrum plot. /FMIN=2.7
/NORMalize	Option to normalize the input time series. = F, B, or N. /NORM=F
/OVERlap	The amount of overlap of the FFT buffers. /OVER=45

/OTYPE	Output type PSD, ESD, or amplitude (see above). /OTYPE=PSD
/SCALE	Scaling (peak/RMS). /SCALE=PEAK
/AVERaging	The averaging method used i.e. Linear/Peak. /AVER=PEAK
/WINdow	The window type (rectangular, Hanning, user-defined etc.). /WIN=REC
/WFILE	The window file name if user-defined. /WFILE=<FILENAME>.UWF
/FLOor	The noise floor in dB. /FLO=79
/PLOt	To plot or not to plot Yes or No. /PLO=Y
/NBUFfer	Number of buffers. /NBUF=10
/TIME	The waterfall time slice in second. /TIME=0.1
/PLTNAM	Hardcopy file name. /PLTNAM= <FILENAME>
/METHOD	Waterfall sectioning method. /METHOD=B
/FFTsize	The FFT buffer size according to the table below. /FFT=512 FFT Buffer Sizes: 32, 256, 512, 1024, 2 048, 4096, 8192, 16384, 32768, 65636, 131,072

PTIME Central Database Listing

Table 5-3 lists the standard loading time histories that are delivered with the MSC Fatigue system including their descriptions.

Table 5-3 MSC Fatigue Time History Central Database Listing

Name	Description
SAETRN	SAE Standard transmission loading history
SAESUS	SAE Standard suspension loading history
SAEBRAKT	SAE Standard bracket loading history
COLOS	Offshore industry loading history
S1	Offshore industry loading history
S4	Offshore industry loading history
I-N	ASTM Instrumentation & Navigation, typical fighter
A-G	ASTM Air to ground, typical fighter
R-C	ASTM Composite mission, typical fighter
TRANSP	ASTM Composite mission, typical transport
HELIX1	Helicopter standard loading history
FALSTAFF	FALSTAFF standard loading history

Table 5-3 MSC Fatigue Time History Central Database Listing

Name	Description
A-A	ASTM Air to air, typical fighter
SINE01	Sine wave, phase = 0, max = 1, min = –1, time = 1 sec
SINE02	Half sine, phase = 0, min = 0, max = 1, time = 1 sec
SINE03	Half sine, phase = 0, min = –1, max = 0, time = 1 sec

DAC File Format Description

The MSC Fatigue Loading data file has a binary direct access format with a fixed record size of 512 bytes. It is made up of three distinct regions:

- A header region
- A data region
- A footer region

The Format of the Header Region

The file header is contained within the first record, the first 512 bytes, of each data file. Three types of information are stored:

- REAL numbers, floating point values such as 123.16
- INTEGER (16 bit) numbers, 1234
- CHARACTER strings, such as: Seconds
- INTEGER*4 (32 bit) numbers, 1234.

It is convenient to think of the header as being made up of 128, 4-byte elements, making 512 bytes in all. Data can then be stored in each element according to the number of bytes required for each data type; this storage requirement is detailed below:

Data Type	Example	Storage Requirement (Bytes)	Number of Values per Dump Form Element
REAL	150.65	4	1
INTEGER	10	2	2
CHARACTER	A	1	4

In this region, a floating point number will occupy an entire element; an integer will occupy half an element; and a character will occupy a quarter of an element. Broadly speaking, the header is divided into four main areas where data of similar type are grouped. Element numbers 1 to 32 contain exclusively REAL (floating point information), 33 to 65 contain mostly integer information, 66 to 125 textual information and 126 to 128 again contain real numbers. Details of the contents of each of these areas are provided in the following table:

REAL AREA 1:			
Dump Form Element No.		Data Type Stored	Contents of Element
1		REAL	Number of data values in file.
2		REAL	Sample rate for time series data, or for HISTOGRAMS: 1/X-axis bins size.
3		REAL	Base for X-axis data.
4		REAL	Increment between data samples, or for HISTOGRAMS; X-axis bin size.
5		REAL	Data mean, (X-axis for paired).
6		REAL	Data standard deviation, (X-axis for paired).
7		REAL	Mean of Y-axis of paired data files.
8		REAL	Standard deviation of Y-axis for paired data files.
9		REAL	Maximum of data, (X-axis for paired).
10		REAL	Minimum of data, (X-axis for paired).
11		REAL	Maximum of Y-axis data for paired.
12		REAL	Minimum of Y-axis data for paired.
13		REAL	HISTOGRAM: the number of bins along the X-axis, for all other data set to 0.0.
14		REAL	HISTOGRAM: Y-axis base.
15		REAL	HISTOGRAM: Y-axis element size.
16		REAL	Multiple set paired data separator.
17		REAL	Number of footer records.
18		REAL	RMS of data.
19 - 32		RESERVED	
INTEGER AREA: **Notice that some elements of this region contain characters.**			
Dump Form Element No.	**Int. No.**	**Data Type Stored**	**Contents of Element**
33	1	INTEGER	Data type flag: 1 = Real 2 = Histogram 3 = Complex: data is X-Y paired. 30 = Complex: change pen between sets. 31 = Complex: no change pen between sets.
	2	INTEGER	RESERVED
34	3	INTEGER	RESERVED

REAL AREA 1:			
Dump Form Element No.		Data Type Stored	Contents of Element
	4	INTEGER	Y-axis units validity flag: -2 = Y-axis units strings are valid.
	4	INTEGER	Y-axis units validity flag: -2 = Y-axis units strings are valid.
35	5, 6	CHARACTER	Y-axis units chars. 17 - 20
36	7, 8	CHARACTER	Y-axis units chars. 21 - 24
37	9, 10	CHARACTER	Y-axis units chars. 25 - 28
38	11	INTEGER	X-axis units validity flag: -2 = Y-axis units strings are valid.
	12	CHARACTER	X-axis units chars. 17 - 18
39	13, 14	CHARACTER	X-axis units chars. 19 - 22
40	15, 16	CHARACTER	X-axis units chars. 23 - 26
41	17	CHARACTER	X-axis units chars. 27 - 28
	18	INTEGER	-axis units validity flag: -2 = Z-axis units strings are valid.
42	19, 20	CHARACTER	Z-axis units chars. 17 - 20
43	21, 22	CHARACTER	Z-axis units chars. 21 - 24
44	23, 24	CHARACTER	Z-axis units chars. 25 - 28
45	25	INTEGER	File format flag: 0 = Data represent time series structure. 1 = Data represent a histogram structure.
	26	INTEGER	RESERVED
46	27	INTEGER	Statistics validity flag: 1 = Calculated statistics ARE valid.
	28	INTEGER	RESERVED
47 - 56			RESERVED
57	49	INTEGER	RESERVED
	50	INTEGER	File edited flag.
58	51	INTEGER	File smoothed flag.
	52	INTEGER	Number of smoothing operations.
60-65			RESERVED
CHARACTER AREA:			

REAL AREA 1:		
Dump Form Element No.	Data Type Stored	Contents of Element
66	CHARACTER	Original file name chars. 1-4
67	CHARACTER	Original file name chars. 5 - 8
68	CHARACTER	Original file name chars. 9 - 12
69	CHARACTER	Original file name chars. 13 - 16
70	CHARACTER	Original file name chars. 17 - 20
71	CHARACTER	Original file name chars. 21 - 24
72	CHARACTER	Original file name chars. 25 - 28
73	CHARACTER	Original file name chars. 29 - 30
74	CHARACTER	Edited file name chars. 1 - 4
75	CHARACTER	Edited file name chars. 5 - 8
76	CHARACTER	Edited file name chars. 9 - 12
77	CHARACTER	Edited file name chars. 13 - 16
78	CHARACTER	Edited file name chars. 17 - 20
79	CHARACTER	Edited file name chars. 21 - 24
80	CHARACTER	Edited file name chars. 25 - 28
81	CHARACTER	Edited file name chars. 29 - 30
82 - 89	RESERVED	
90	CHARACTER	Y-axis title chars. 1 - 4
91	CHARACTER	Y-axis title chars. 5 - 8
92	CHARACTER	Y-axis title chars. 9 - 12
93	CHARACTER	Y-axis title chars. 13 - 16
94	CHARACTER	Y-axis title chars. 17 - 20
95	CHARACTER	Y-axis title chars. 21 - 24
96	CHARACTER	X-axis title chars. 1 - 4
97	CHARACTER	X-axis title chars. 5 - 8
98	CHARACTER	X-axis title chars. 9 - 12
99	CHARACTER	X-axis title chars. 13 - 16
100	CHARACTER	X-axis title chars. 17 - 20
101	CHARACTER	X-axis title chars. 21 - 24
102	CHARACTER	Z--axis title chars. 1 - 4
103	CHARACTER	Z--axis title chars. 5 - 8

REAL AREA 1:		
Dump Form Element No.	Data Type Stored	Contents of Element
104	CHARACTER	Z--axis title chars. 9 - 12
105	CHARACTER	Z--axis title chars. 13 - 16
106	CHARACTER	Z--axis title chars. 17 - 20
107	CHARACTER	Z--axis title chars. 21 - 24
108	RESERVED	
109	CHARACTER	Y--axis title chars. 1 - 4
110	CHARACTER	Y--axis title chars. 5 - 8
111	CHARACTER	Y--axis title chars. 9 - 12
112	CHARACTER	Y--axis title chars. 13 - 16
113	CHARACTER	X--axis title chars. 1 - 4
114	CHARACTER	X--axis title chars. 5 - 8
115	CHARACTER	X--axis title chars. 9 - 12
116	CHARACTER	X--axis title chars. 13 - 16
117	CHARACTER	Z-axis title chars. 1 - 4
118	CHARACTER	Z-axis title chars. 5 - 8
119	CHARACTER	Z-axis title chars. 9 - 12
120	CHARACTER	Z-axis title chars. 13 - 16
121 - 125	RESERVED	
REAL AREA 2:		
126	REAL	Point no. of maximum.
127	REAL	Point no. of minimum.
128	REAL	Point no. of absolute maximum.

Format of the Data Area

The body of a MSC Fatigue loading file contains the actual number to be used in the analysis. As with the header region, this information is stored in fixed length 512 byte records. The numbers themselves are held in a floating point binary format so that each record contains 128 values.

Three generic data structures are currently supported:

- **Time series structure**
- **Histogram structure**
- **Paired X-Y structure**

Format of the Time History (.dac) Structure

The "time series" structure is used to describe single channel data where the increment between samples is constant.

A true time series, where each value represents a sample scaled in physical units and the "gap" between samples represents reciprocal sample rate, is the most common data type normally found in this category. Files of this sort are easily identified by their file extension of "`.dac`".

As long as the rule of constant increment between samples is obeyed, other types of data can also be represented in time series format; a case in point would be an auto spectrum file with the extension `.asd` or `.psd`. In this case each data value represents a power (or amplitude) and the "gap" between values a constant increment in frequency.

A single dimensional histogram representing a time at level (`.tlv` files) or probability density distribution, (`.pdd` files) generated by the amplitude distribution module, MADA, also fall into this category.

Within this category of files, the default file extension expected by is `.dac`.

Note:	MQLD, the quick look display and MMFD, the multi-file display modules are specifically designed to display data files in time series format.

DAC file revision - Larger file capability

Pre V8 DAC files have a problem storing the number of points accurately when the number of points exceeds **16 mega-points**. To solve this, the following header values for V8 on:

- The current number of points (CB(1)) is retained and used if the new location is not found.
- The integer control block location ICB(32), or the second element of the real location 48, should be set to 2 for a new file.
- The real area now ends at CB(28) instead of CB(32).
- A new INTEGER*4 area has been created between CB(29) and CB(32), called I4CB.
- I4CB(1) or CB(29) is set to the location of the maximum value
- I4CB(2) or CB(30) is set to the location of the minimum value.
- I4CB(3) or CB(31) is set to the location of the absolute maximum value.
- I4CB(4) or CB(32) is set to the number of points (V5 files on).

Program MFILMNP has been modified to auto-correct old files. Old files will work as before (i.e. accurately to 16 mega-points.).

Format of Histogram Structure

Histogram data files are those which contain a two-dimensional matrix such as might result from a rainflow cycle count. The counts within each element of the X-Y matrix are stored in the form of REAL floating point numbers. The sequence in which the matrix elements are stored can be seen by considering the 16 elements of the, 4 x 4, matrix illustrated below:

x-axis y-axis	Min			Max
Min	1	2	3	4
	5	6	7	8
	9	10	11	12
Max	13	14	15	16

Values from this matrix would be stored within a histogram file starting with element number 1 and proceeding sequentially through to element number 16. In the general case, values within each X-axis column, for each Y-axis row, are stored sequentially, starting at the minimum of both X and Y-axes and ending at the maximum of both the axes.

Numbered among files with this generic format are rainflow (`.cyh`), range mean (. `rmn`), Markov (`.mkv`) and range pair (`.rph`) files.

Within this category of files, the default file extensions expected are, `.cyh`, `.rmn`, `.mkv`, and `.rph`.

Note:	MP3D, the two dimensional histogram display module is specifically designed to display these type of data files.

Format of Paired X-Y Structure.

Paired X-Y type data files consist of consecutive pairs of X-Y values stored as REAL floating point numbers. The two parameter creation module, MTPC, can be used to create such files.

Within this category of files, the default file extension expected is `.mdf`.

An additional subclass of this structure, in which several individual X-Y data sets can be stored simultaneously in the same file, is also supported. The structure is very similar to that described above except that a unique delimiter is inserted between each data set. The value of this delimiter is defined in the header region at dump form element number 16. If for some reason this delimiter ceases to be unique as a result of some manipulation by an MSC Fatigue module, then the offending module will replace it by another value which is again unique.

Note:	MTPD, the two parameter display module is specifically designed to display these type of data files.

Minimum Header Requirement for a Loading Type File

All 128 elements of the header area should be set to zero and then, as a minimum requirement the following elements defined in the REAL area:

Dump Form Element No.	Data Type Stored	Contents of Element
REAL AREA 1:		
1	REAL	Set to the number of data values in file.
2	REAL	Set to the sample rate for time series data or reciprocal of the increment between samples.
3	REAL	Set to the X-axis origin.
4	REAL	Set to the increment between data samples.

As a minimum requirement, the following field from the INTEGER area should be set:

Dump Form Element No.	INT. No	Data Type Stored	Contents of Element
33	1	INTEGER	Set to 1.

The X and Y-axis label and units fields can be set, by using the MFILMNP module. As long as the first integer field of dump form element number 46 (INTEGER field number 27) is set to zero, the statistics fields will be automatically calculated by the first MSC Fatigue module to access the file.

Minimum Header Requirement for a Histogram Type File

All 128 elements of the header area should be set to zero and then, as a minimum requirement the following elements defined in the REAL area:

Dump Form Element No.	Data Type Stored	Contents of Element
REAL AREA 1:		
1	REAL	Set to the number of data values in file.
2	REAL	Set to 1 / X-axis bin size.
3	REAL	Set to the X-axis origin.
4	REAL	Set to the size of the X-axis bins.
13	REAL	Set to the number of X-axis bins.
14	REAL	Set to the Y-axis origin.
15	REAL	Set to the size of the Y-axis bins.

Note:	The number of bins along the Y-axis is automatically calculated by dividing dump form element number 1 by element number 13 and so this value need not be stored.

As a minimum requirement, the following field from the INTEGER area should be set:

Dump Form Element No.	INT. No.	Data Type Stored	Contents of Element
33	1	INTEGER	Set to 2.
45	25	INTEGER	Set to 1.

The X and Y-axis label and units fields can be set, by using the MFILMNP module. As long as the first integer field of dump form element number 46 (INTEGER field number 27) is set to zero, the statistics fields will be automatically calculated by the first module to access the file.

Minimum Header Requirement for a Paired Data Type File

All 128 elements of the header area should be set to zero and then, as a minimum requirement the following elements defined in the REAL area:

Dump Form Element No.	Data Type Stored	Contents of Element
	REAL AREA 1:	
1	REAL	Set to the number of data values in file, X + Y.
2	REAL	Set to 1.
4	REAL	Set to 1.

If multiple X-Y data sets are being stored then element 16 should be set:

Dump Form Element No.	Data Type Stored	Contents of Element
16	REAL	Set to a unique value which will be taken to a delimiter between adjacent data sets within a file containing multiple data sets.

As a minimum requirement, the following field from the INTEGER area should be set:

Dump Form Element No.	INT. No.	Data Type Stored	Contents of Element
33	1	INTEGER	Set to 3 for single data set or 30 for pen changes between multiple data sets or 31 for no pen change pen between multiple data sets.

The X and Y-axis label and units fields can be set, by using the MFILMNP module. As long as the first integer field of dump form element number 46 (INTEGER field number 27) is set to zero, the statistics fields will be automatically calculated by the first MSC Fatigue module to access the file.

Loading and Units

MSC Fatigue has been designed to appear to use units in the same way that FE modeling does (i.e., independent of any specific units system). However, the models for fatigue life estimation are not independent of the units system and so MSC Fatigue actually uses the S.I. system internally to carry out its computations. In order to appear independent of units, the relationship between S.I. units and the chosen units system has to be defined for different kinds of loading. This definition is held in two ASCII tables which are located in the MSC Fatigue central system directory in the `ptime` directory.

The tables have the following file names:

```
ltypes.ind loading types definition
utypes.ind units definition
```

Loading Types

The loading types table will have a form similar to Table 5-4.

Table 5-4 Loading Types Definition File

Load Number	Object Load Group	Description Load Name
1	1	Point Load
2	2	Pressure
3	3	Temperature
4	4	Acceleration
6	6	Displacement
7	7	Angular Speed
8	8	Speed
9	9	Uncalibrated
10	10	Scalar
11	11	Moment

The basic form of the loading types table is defined within the file `ltypes.ind`. Each new type of loading is given a unique index number, a group number which is the same as the index number and a title. The title will appear at various places in MSC Fatigue wherever loading types are being manipulated (e.g., when defining a new loading time history in PTIME).

The list of loading types may be extended by adding a new line to the bottom of the `ltypes.ind` file. The user should NOT insert a loading type in the middle of the existing list and renumber the loading types since

this will seriously interfere with any existing MSC Fatigue jobs and make the loading types incompatible with previous jobs. The maximum load number cannot exceed 99.

As well as defining a new loading type in the `ltypes.ind` file, the user must also define the units system for this loading type. This is explained in Units Types, 284.

Units Types

The units types table will have a form similar to Table 5-5. The base units types for any loading type is the S.I. units system as defined below:

Note: Kilograms	for mass
Meters	for length
Seconds	for time
Kelvin	for temperature

All other units may be derived from these (i.e., stress base units are N/m or Pascals). Once a new loading type has been added, the user must also define the relationship between the units for this loading type. Failure to do so will produce an error message in the MSC Fatigue software. The units relationship is held in the `utypes.ind` file. To add new lines to the `utypes.ind` file, the user must first understand the format of the file. There are five columns of information required.

Table 5-5 Units Types Definition File

Units Number	Load Group Number	Conversion Factor	Conversion Offset	Units Name
0	1	1	0	Newtons
1	1	1000	0	kNewtons
2	1	4.448	0	lbs force
3	1	9964	0	Tons force
4	1	9807	0	Tonnes force
10	2	1	0	Pascals
11	2	1E6	0	MPa
12	2	6894.65	0	PSI
13	2	6894650	0	KSI
14	2	1.3788E7	0	TSI
15	2	9806.81	0	kgf/m^2
20	3	1	0	Degrees Kelvin
21	3	1	273	Degrees Celsius
22	3	0.55556	255	Degrees Fahrenheit

Table 5-5 Units Types Definition File

Units Number	Load Group Number	Conversion Factor	Conversion Offset	Units Name
30	4	1	0	m/s^2
31	4	9.80655	0	g
32	4	0.0254	0	$feet/s^2$
40	6	1	0	m
41	6	0.001	0	mm
42	6	0.0254	0	inches
43	6	0.0254E-3	0	milliinches
44	6	0.0254E-3	0	microinches
50	7	1	0	rps
51	7	0.0166667	0	rpm
52	7	2.7778E-4	0	rph
53	7	0.159155	0	rad/s
54	7	2.6526E-3	0	rad/min
60	8	1	0	m/s
61	8	0.001	0	mm/s
62	8	0.27778	0	kph
63	8	0.0254	0	ips
64	8	0.4470	0	mph
70	9	1	0	none
80	10	1	0	none
81	10	1	0	%
82	10	1	0	levels
90	11	1	0	Nm
91	11	0.73759	0	Ft lbs
92	11	0.001	0	Nmm

Units Number: This is a unique number and will usually be the next number which leaves room for later expansion if it becomes necessary. The maximum number allowable is 999.

Load Group Number: This number MUST correspond to the loading group number allocated in the `ltypes.ind` file (see above).

Conversion Factor: This is the multiplier which converts the new units into S.I. units (see note on units).

Conversion Offset: This is the offset number which converts the new units into S.I.units (e.g., 273 to convert temperature from Kelvin to degrees Celsius).

Units Name: This is the text which will appear whenever a units option is presented.

Aerospace Spectrum File Support

This section describes the aerospace spectrum file support in MSC Fatigue. Load time histories are not required as stress spectra are generated entirely from the information in ASCII editable Spectrum (extension .spe) and Load Control files (extension .lcs). In essence, the stress spectrum at each analysis location (nodes or elements) is generated by stepping through a sequence of static load cases that are defined in the Load Control file.

Aerospace spectra are not currently supported for the following analysis types:

- Strain data in FES file
- Seam weld
- Spot weld
- Vibration fatigue
- Multi-analysis (duty cycle analysis)
- Fast Analysis
- Critical Plane & Multiaxial Analysis

File Definitions

Spectrum file

Spectrum files are ASCII and must have a **.spe** extension. Format, explanation of each input, and a typical file are shown below.

```
<Spectrum file V2>
<Description>
<Scatter Factor>
<Number of cycles>
<Number of flights>
<Stress Factor>

Flight, <Flight number>, <Number of sequences>, <Number of
repeats>
<Sequence name>, <Repeats>, <Case 1>,...,<Case J>
<Sequence name>, <Repeats>, <Case 1>,...,<Case K>
.
.
```

The header "Spectrum File V2" must be the first line in the .spe file. Scatter factor accounts for scatter in materials and loads data. The calculated life will be divided by this factor.

The Number of Cycles and Stress Factor parameters have not been implemented. Therefore, these values should be set to 1.

The "#" character is used to denote comments. All text appearing after the "#" are ignored.

Example Spectrum File

```
Spectrum file V2
Example
1
1
2
1
Flight, 1, 3, 15                          # 'Flight', Flight Number,
No. of repeats
Push Back, 7, 1, 2, 0, 3, 1, 2, 1  # Sequence name, repeats,
cases,...
Engine Run-up, 9, 0, 5, 0, 1, 5, 1
Taxi, 20, 0, 2, 0, 1, 4
Flight, 2, 2, 10
Take-off, 1, 10, 11, 0, 10, 50
Gear retract, 5, 0, 25, 0
```

Load Control file

Load Control files are ASCII and must have a **.lcs** extension. Format, explanation of each input, and a typical file are shown below.

```
<Description>
<Units>
<Number of load cases>
<Number of FE cases>
<Divisor 1>,...,<Divisor m>
<ID1>, <Desc1>, < scale 1>,..., <scale m>
...
<IDn>, <Descn>, <scale 1>,..., <scale m>
```

The number of FE cases has to be greater than or equal to 2. Blank lines after the 5 header lines are ignored, as is anything after the # character. The ID must be a positive integer. The divisor values are the scale factors (as divisors) for normalizing the stresses and must not be zero.

Example Load Control File

```
Title
Units
10                                  # Number of load cases
5                                  # Number of FE cases
1.0,1.0,1.0,1.0,1.0    # Divisor Values for normalizing stresses
0,Load 0,0,0,0,0,0
1,Load 1,-1.000,.5125, 0.2,0.4,-1.8
2,Load 2,.55,-.76, 0.3,0,1.1
```

```
3,Load 3,0.5,-0.4,1.8,3.3,1.0
4,Load 4,-2.000,.5, 0.6,0.7,1.9
5,Load 5,.65,-.77,3.4,9.1,2.5
10,Load 10,1.1,2.5,6.8,9.0,-2.0
11,Load 11, 4.1,1.4,-4.3,-2.4,-1.0
25,Load 12, 4.2,2.4,-3.4,-2.4,-1.0
50,Load 50, 4.4,-0.4,-6.5,-2.4,-1.0
```

In the example files above, the spectrum file is made up of 2 flights that references one or more of the 10 load cases. Each load case is comprised of 5 static FE Cases that are factored to compute the stress for the given load case.

6 Total Life and Crack Initiation

- Introduction 290
- Job Setup(dtfat) 297
- Multi-Curve Materials (dtfat) 307
- Temperature Dependent Fatigue (dtfat) 308
- dtfat Batch Operation 316
- FE Fatigue Analysis Options (FEFAT) 317
- Reviewing Results (PFPOST) 363
- Fast Analysis (FASTAN) 368
- Description of Files 369
- FEFAT Batch Operation 417

Introduction

This chapter on total life and crack initiation consists of descriptions of various MSC Fatigue modules. These modules can be accessed from subordinate forms of the MSC Fatigue main form within the MSC Patran program.

There are two basic approaches for performing a fatigue analysis in MSC Fatigue. One approach is to use the new DTLib-based solution, dtfat, which is constantly being updated and extended. This solution integrates the latest solver technology from HBM nCode. The other approach is to use the more special-purpose Classic modules. The Classic modules may be more familiar to long-time MSC Fatigue users, and the modules also have the advantage of having a interactive GUI and an "Optimization" tool.

It is possible to carry out a MSC Fatigue analysis with the Classic modules outside the pre- and post-processing environments such as the Patran environment by executing the analysis programs from the system prompt. The main reason for doing this is to provide an alternative and sometimes, faster route for carrying out multiple computations with only one or two small changes to the analysis parameters. This can be achieved because the MSC Fatigue job parameters and the nodal or elemental fatigue data are stored in ASCII files. For completeness, a general schematic showing the analysis route is shown in Figure 6-2. The aspects of this figure are described throughout this chapter. The crack growth analysis route is also shown in Figure 6-2 but is specifically described in Crack Growth (Ch. 8).

Terminal Definition

MSC Fatigue runs on a wide range of computers and graphics devices. The parameters used by each graphics device must be defined by using the MENM module, prior to the first use of MSC Fatigue. For details, please see Module Operations (Ch. 18). This is automatically accomplished when running MSC Fatigue from a pre- and post-processing environment such as Patran and is transparent to the user and defaults to the Motif driver.

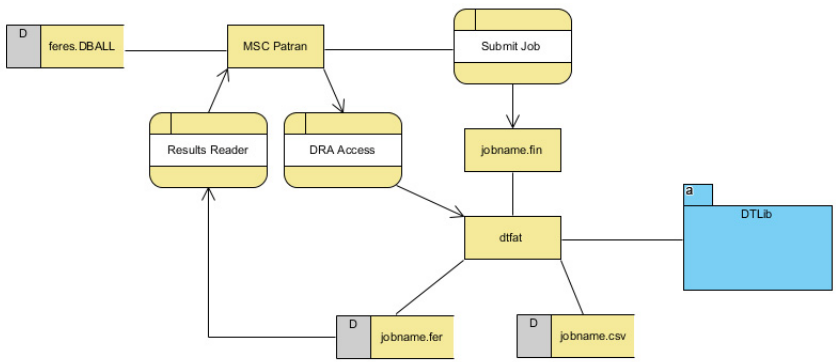


Figure 6-1 Job Submittal Process for DTLib-Based Solver

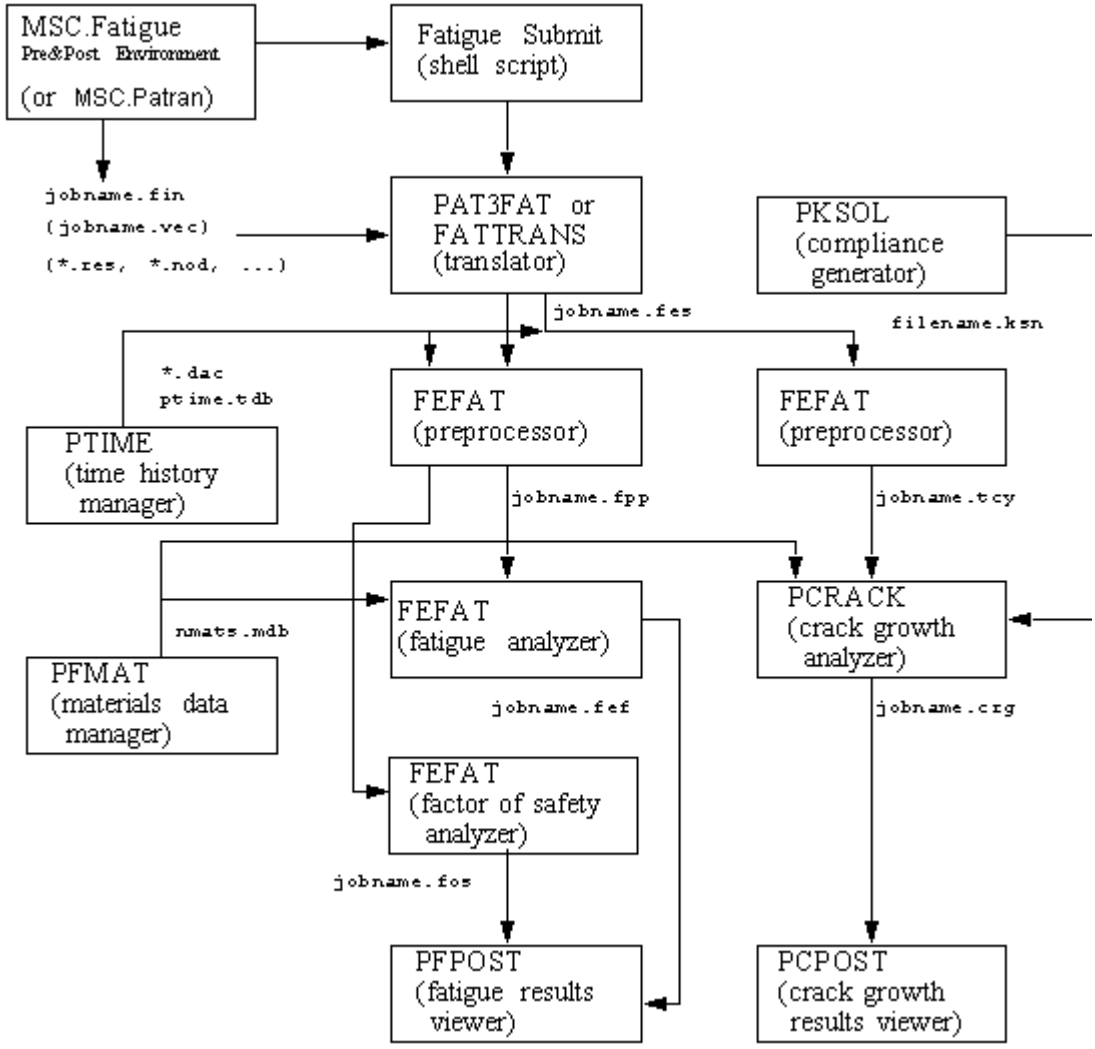


Figure 6-2 A MSC Fatigue Submittal Schematic

Basic Information

All programs in the MSC Fatigue system may be executed by typing the name of the program or its symbol.

Interaction with the dtfat solver is normally handled through the MSC Fatigue GUI, since it doesn't have its own GUI. Since dtfat is a multi-disciplinary solver, a different solver module is not required to perform different types of analysis. More detail is given in the following sectionsJob Setup(dtfat).

The Classic modules, which have their own GUI, may ask questions which are not normally presented to you since they are executed as batch jobs when called from the pre-/postprocessing environment. The modules normally used in a typical or basic fatigue analysis are listed below.

1. Data Preparation

dtmat	Materials Database Manager
PFMAT	Materials Database Manager and BS5400 Weld Classification Advisor
PTIME	Time History Database Manager and ASCII Time History File Convertor
dcbuilder	Duty Cycle tools for the DTLib-based solver
PVXMUL	A Peak-Valley Extraction Program for Reducing Lengthy Time Histories
MMFD	A Multi-file Display Program

2. Global Multi-Node/Element Analysis

PAT3FAT	Model database (Patran) to Fatigue Input Translator
FATTRANS	A new model database (Patran) to Fatigue Input Translator
dtfat	multi-disciplinary fatigue solver
FEFAT	Fatigue Preprocessor (rainflow cycle counting)
MFATFE	second-generation FE-Fatigue Analyzer (damage summation)
FEFAT	FE-Fatigue Analyzer (damage summation)
PFPOST	Global Fatigue Results Postprocessor

3. Factor of Safety Analyzer

MFATFE	second-generation FE-Fatigue Analyzer (damage summation)
FEFAT	Factor of Safety Analyzer

4. Design Optimization Analyzer

FEFAT	Single Node/Element Total Life (S-N) and Crack Initiation (e-N) Analyzers and Cycle/Damage 3-Dimensional Histogram Display

5. General Utilities

FEFAT	FES File ASCII/Binary Convertor
PFTRM	Terminal Driver
MCONFIL	Binary to Binary File Convertor
FASTAN	Utility to run a peak-valley extraction and subsequent analysis to speed up execution time

Analysis Route

The actual programs necessary to complete a global multi-node or element total life or crack initiation analysis aside from the pre/postprocessor are:

FatigueSubmit	Shell script (necessary for submittal from Patran)
PAT3FAT	Translator (creates the fatigue input file `filename.fes`)
FATTRANS	New Translator (creates the fatigue input file `filename.fes`)
FEFAT	Fatigue Preprocessor (superposition of multiple load cases and rainflow cycle counting)
FEFAT	Fatigue Analyzer (total life and crack initiation)
FEFAT	Factor of Safety Analyzer (optional)
MFATFE	second-generation Fatigue Analyzer (total life and crack initiation)
dtfat	Fatigue Analyzer (multi-discipline)

The programs and options must be used in this sequence with exception of the factor of safety option which may run directly after the fatigue pre-processing. The results may be reviewed using PFPOST or by inspecting the ASCII nodal or element results file (`jobname.fef` or `jobname.fos`) using a text editor or by inspecting fringe plots directly within the pre/postprocessor. PFPOST is generally not compatible with output files from MFATFE or dtfat.

Necessary Files

When a global multi-node fatigue analysis is set up using the MSC Fatigue pre-/post-processing menus, these are the files necessary to run the analysis and the files that are created.

jobname.fin	This file contains all the analysis parameters that were defined in the main and subordinate MSC Fatigue forms, (e.g., loading time history data file names). In addition, the analysis type and job titles are also defined in this file. A full description of this file is contained in The Job Information File (jobname.fin), 369.
Database	Other pertinent information such as the nodes or elements for which to calculate fatigue life is contained in the database in the form of a group or groups. The component stresses or strains from these locations will be used, scaled, superimposed and resolved (dependent on various parameter requests) and used in the fatigue calculations.
Results Files	The actual stress or strain results used in the fatigue analysis can be stored in the database or can come from external results files such as Abaqus, Ansys, Marc, Nastran, or other attached results files.
Additional Files	Other files that are necessary to complete a successful fatigue analysis are the time history files (`ptime.adb`, `ptime.tdb` and `*.dac`) and the materials database (`nmats.mbd, iceflow_standard.mxd`) which is generally held in a central location and not necessary to be located in the user's local directory.

The first of these files is ASCII and may be edited using a standard text file editor. Although this method of defining the MSC Fatigue job parameters is not as automated as using the MSC Fatigue menus, it does offer a simple and rapid method of changing a few job parameters without the encumbrance of a menu structure.

For a DTLib-based solver run, the job setup data is contained in the fin file. This file is parsed by the dtfat solver, and if there are no input errors a **jobname.config** file is created which echos the DTLib-based software objects used in the analysis. After the job configuration phase, the fatigue job is handed-off to the DTLib library, which performs a validation step to check whether FE results are available for the requested entities, etc. If the validation completes successfully, the analysis run proceeds and the fatigue results are created. The user may request that a fes file is created, rather than directly accessing the FE results and model data from Patran and any attached FE results files. However, this is not required and is provided for legacy purposes.

By default, the dtfat solver does not output FE results to an intermediate file. The FE results data are accessed directly from the Patran database or from the attached FE results files. The Fatigue Preferences can be modified to force MSC Fatigue to perform a translation step and output FE results to a fes file by setting the **FE Results Access** option = **fes file**. Note, however, this setting will be ignored for analysis types that require usage of the **Direct Access** option. This will be stated where applicable.

For a Classic module job run, after the translator has been run (described in The Translator (PAT3FAT or FATTRANS), 294) and the fatigue input file (`jobname.fes`) has been created, the fatigue preprocessor, FEFAT, is run. A `jobname.fpp` file is created which is a scaled, rainflow cycle count for stresses or strains of the loading time history. The fatigue analysis is performed also by the program FEFAT. When run in interactive mode, this program asks for a number of input parameters which are passed in through the `jobname.fin`, `jobname.fes` and `jobname.fpp` files when run from the MSC Fatigue menus. A full description of file content is provided in Description of Files, 369.

The Translator (PAT3FAT or FATTRANS)

MSC Fatigue uses a translator to combine the information in the database, the possible external results files and the job information file (`jobname.fin`) to create the fatigue input file (`jobname.fes`). For this reason, the translator must be run each time any information is changed in the database and/or the results and information files such as if the FEA analysis has been rerun.

To run the translator, the following command should be used:

```
pat3fat jobname
        or
fattrans jobname
```

PAT3FAT and FATTRANS both produce a `jobname.fes` file which is a binary file. An ASCII to binary and binary to ASCII file convertor is provided for the `jobname.fes` file. The convertor program is part of the FEFAT module and is described in Utilities, 360. A description of the ASCII and binary versions of the fatigue input file (`jobname.asc/fes`) is provided in The Fatigue Input File (jobname.fes), 401. Remember that the fatigue executables can only read the binary version and the ASCII version must always be converted back to binary via FEFAT's utility options.

The FATTRANS Translator

Although FATTRANS, the new (since Release 2005) translator that handles large models more efficiently will ultimately replace the older translator, PAT3FAT is still the default translator for most of the fatigue analysis types. FATTRANS is currently the default for only the following analysis types:

Aero-Spectrum Loading analysis

Seamweld analysis using stress tensor results

Isothermal fatigue analysis

Multi-mean SN analysis

The PAT3FAT translator cannot be used to create a fes file for dtfat, and FATTRANS is used automatically in this event. The fes file created for dtfat is generally not compatible with the Classic modules.

FATTRANS can be made the default translator for all analysis types, by setting the Fatigue Environment Varianble **Default Translator** = ,**fattrans**.

Handling Very Large Models

One goal of the FATTRANS development was to ensure that the application performs well no matter how large the user's model is, and no matter how many time increments in results history data. The translator should never utterly fail for lack of memory.

In order to perform the translation of very large models, the FATTRANS translator computes the memory needs for the model, and automatically creates sub-groupings of entities that fit into user-defined available memory. The translator then performs the results fetch/write operations on a given sub-group at a time. For small or moderate size models, no subgrouping is needed.

To allow for maximum flexibility, the maximum amount of memory to be used by the FATTRANS translator can be modified with a the Fatigue Environment Variable **Max Memory** = **value**.

Where:

value-is the maximum amount of memory (in words) to be used by fattrans

And:

value = 10E7-is the default value set internally at 10E7 words.

Since most machines use some type of virtual memory, there should not be a problem with setting **Max Memory** to a number higher than the physical memory available on a given machine. However, setting the number too high may lead to excessive use of virtual memory for very large models, and thus result in poor performance.

Conversely, setting **Max Memory** to too small a number may result in too many subgroups being created. A large number of sub-groups can lead to degraded translator performance. The best procedure is to only specify **Max Memory** when translating very large models or moderately sized models with a large number of result time steps.

The memory usage estimate for model translation is written from fattrans to the $jobname.msg file. This is illustrated from the following excerpt:

Message: Group 1 contains 733 entities. At 9 words per ent,

Max memory allocation is 6597 words.

In the case where the default **Max Memory** is exceeded, (i.e., in the case of a very large model), the model is automatically divided into smaller subgroups. Each subgroup contains no more memory than that specified by **Max Memory**, which in this case would be the default value of 10 megawords. The following example message is written to the $jobname.msg file:

Message: Group 2 sub-divided into 6 subgroups;

Max memory for each sub-group is 10000000 words.

Large Model Example

During product testing, a crankshaft model was used which contained 32,000 nodes and 37 load increments. For one test, the default memory usage value was used.

By examining the jobname.msg file, the memory required for the translation could be determined as shown below:

(excerpt from crankshaft.msg)

Message: Group 1 contains 32000 entities. At 333 words per ent,

Max memory allocation is 10656000 words.

In another test with the same model, a smaller value for maximum memory was used. In this case, the maximum amount of memory to be used by fattrans was set at 1 megawords. For a 32-bit machine, this is equivalent to 32 megabytes.

Max Memory =1E6

During the run, the following messages were written to the msg file:

(excerpt from crankshaft.msg)

Message: User modifiable variable FAT_MAX_NUM_WORDS has been changed from 10000000 (default) to 1000000.

Message: Group 1 contains 32000 entities. At 333 words per ent,

Max memory allocation is 10656000 words.

Message: Group 1 sub-divided into 73 subgroups;

Max memory for each sub-group is 1000000 words.

Warning: Translator performance may be seriously degraded with exessive number of subgroups (i.e., 10+ subgroups).

Setting the FAT_MAX_NUM_WORDS environment variable to a number too small can cause excessive number of sub-groups.

In this case, the translator took aproximately 5 times longer to perform the translation. This was due to the results fetch routine having to be performed 73 times-once for each subgroup. This example shows that playing with the FAT_MAX_NUM_WORDS variable is not advisable. The default should not be over-ridden arbitrarily, and smaller default values may lead to increased translation times.

Job Setup(dtfat)

The MSC Fatigue analysis module **dtfat** is a multi-discipline fatigue analysis program. The name dtfat refers to fatigue analysis using the solver technology of the HBM nCode DTLib-library

Solution Parameters (S-N)

The following table lists DTLib solution parameters that are associated with S-N analysis as shown on the Solution Parameters form below:

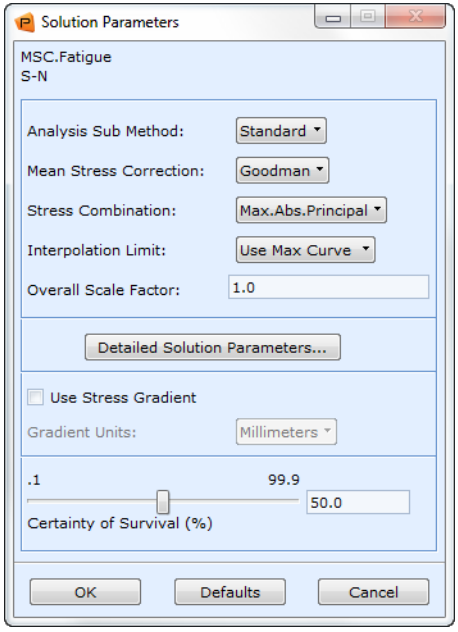


Table 6-1 Solution Parameters

Parameter	Options	Description	Default
Analysis Sub-Method		A number of new material models and S-N methods are available. The interpolation-type methods (or multi-curve methods) allow the user to correct for mean stress effects using the measured material properties rather than rule of thumb approaches, such as Goodman. Please note that a limited number of multi-curve data sets is included with MSC Fatigue. See the section on the **dtmat** material manager for more information on how to add user material data sets. Note that when the Haigh diagram multi-curve method is chosen, the mean stress correction is automatically set to **Interpolate**.	**Standard**
	Standard	Use the standard nCode S-N method and standard S-N material data sets	
	Multi-Mean Curve	Use Multi-Mean curves to determine S-N material properties for a given mean stress. Use Multi-Mean curve material data sets.	
	Multi R-Ratio Curve	Use Multi R-Ratio curves to determine S-N material properties for a given r-ratio of stress. Use Multi R-Ratio curve material data sets.	
	Haigh Diagram	Use Haigh Diagram constant life curves to determine S-N material properties for a given mean stress. Use Haigh Diagram curve material data sets.	
	Bastenaire	Use the Bastenaire probabilistic material data sets for S_N and E-N.	
	Multi-Temp Curve	Use the Temperature Dependent material data sets.	
Mean Stress Correction			**None**
	None	No mean stress correction applied.	
	Goodman	Goodman method of mean stress correction applied.	
	Gerber	Gerber method of mean stress correction applied.	

Table 6-1 Solution Parameters

Parameter	Options	Description	Default
	Interpolate	Use multi-curve method. Note that this only applies to Multi-Mean Curve, Multi R-Ratio Curve, and Haigh Diagram S-N analysis sub-methods. Automatically selected for these Haigh diagram S-N analysis sub-methods.	
	FKM	The FKM method as implemented is based on the method described in the FKM Guideline “Analytical Strength Assessment of Components in Mechanical Engineering”, Tr. E. Haibach. 2003.	
	Goodman Tension-Only	Goodman method of mean stress correction applied in tension only.	
	Gerber Tension-Only	Gerber method of mean stress correction applied in tension only.	
Stress Combination Method			**Max Abs Principal**
	Max Abs Principal	The absolute Maximum Principal stress is defined as the principal stress with the largest magnitude	
	Signed Max Shear	The Signed Max Shear stress is the Maximum Shear stress (Tresca Criterion), forced to take the sign of the Absolute Maximum Principal stress. A factor of 2 is applied to ensure that the parameter has the same value as the Absolute Maximum Principal stress under uniaxial loading conditions.	
	Signed Von Mises	The Signed von Mises stress is the von Mises stress, but forced to take the sign of the Absolute Maximum Principal stress.	
	Critical Plane	The Critical Plane option is available only for stress flagged as 2-D. The normal stress is calculated and rainflow counted on multiple planes.The critical plane is the plane with the most predicted fatigue damage. The planes on which the normal stress is determined have normals that lie in the plane of the physical surface, i.e., in the x-y plane of the 2-D stress results coordinate system. The orientation of each plane is defined by the angle ϕ made with the local x-axis.	
	Max Principal	Max Principal is the maximum principal stress. It may be smaller in magnitude than the minimum principal stress and should be used with caution.	

Table 6-1 Solution Parameters

Parameter	Options	Description	Default
	Von Mises	VonMises is the von Mises stress. It is always positive (not signed by the absolute maximum principal stress) and should be used with caution.	
	Shear	The Shear stress is the Maximum Shear stress (Tresca Criterion). It is not signed and should be used with caution.	
Interpolation Limit		This option is used to specify how the multi-curve method compute values at the extrema of the material data sets.	**Use Max Curve**
	Use Max Curve	Material property values are not extrapolated from the material data set values. Rather, the values nearest the extrema are used.	
	Extrapolate	Material property values are extrapolated from the material data set values.	
Overall Scale Factor	0 < **Value** <= 1e10	The scale factor to apply to the combined data prior to performing damage calculation.	**1.0**
Use Stress Gradient	**Enabled, Disabled**	This option allows to apply a stress correction based on the gradient in stress in the direction normal to the free surface. Only applies to surface resolved stresses on the free surface. The stress gradient correction is based on gradients computed in MSC Fatigue.	**Disabled**
Gradient Units	**Millimeters, Meters, Inches, Feet**	Length units for stress gradients.	**Millimeters**
Certainty of Survival	0 <= **Value** <= 100	Percentage certainty of survival	**50**

Solution Parameters (E-N)

The following table lists DTLib solution parameters that are associated with E-N analysis as shown on Solution Parameters form below:

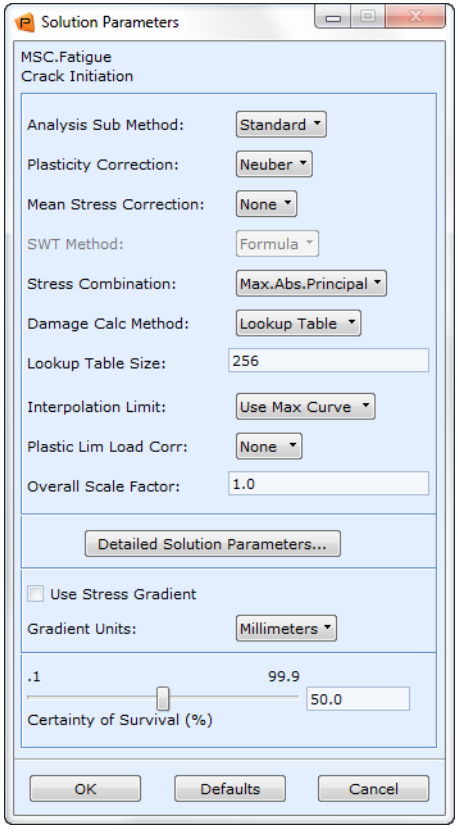

Solution Parameters
MSC.Fatigue
Crack Initiation
Analysis Sub Method:
Standard
Plasticity Correction:
Neuber
Mean Stress Correction:
None
SWT Method:
Formula
Stress Combination:
Max.Abs.Principal
Damage Calc Method:
Lookup Table
Lookup Table Size:
256
Interpolation Limit:
Use Max Curve
Plastic Lim Load Corr:
None
Overall Scale Factor:
1.0
Detailed Solution Parameters...
Use Stress Gradient
Gradient Units:
Millimeters
.1
99.9
50.0
Certainty of Survival (%)
OK
Defaults
Cancel

Table 6-2 Solution Parameters

Parameter	Options	Description	Default
Analysis Sub-Method		A number of new material models and E-N methods are available. The interpolation-type methods (or multi-curve methods) allow the user to correct for mean stress effects using the measured material properties rather than rule of thumb approaches, such as Goodman. Please note that a limited number of multi-curve data sets is included with MSC Fatigue. See the section on the **dtmat** material manager for more information on how to add user material data sets.	**Standard**
	Standard	Use the standard nCode E-N method and standard E-N material data sets	
	Multi-Mean Curve	Use Multi-Mean curves to determine E-N material properties for a given mean stress. Use Multi-Mean curve material data sets.	
	Multi R-Ratio Curve	Use Multi R-Ratio curves to determine E-N material properties for a given r-ratio of stress. Use Multi R-Ratio curve material data sets.	
	Multi-Temp Curve	Use the Temperature Dependent material data sets.	
Plasticity Correction		Elastic-plastic corrections are often required because the local strain approach requires total (elastic-plastic) strain as well as stress, yet for efficiency reasons, calculations often have to be based on linear elastic FE calculations. Notch corrections allow elastic plastic strains and stresses to be estimated based on elastic FE results.	**Neuber**

Table 6-2 Solution Parameters

Parameter	Options	Description	Default
	Neuber	The Neuber method provides a simple way of estimating the total elastic-plastic strain and stress at an "average" stress concentration, based on the local elastic stress/strain. Consider a simple notched specimen subjected to a uniaxial loading. As long as the yield stress is not exceeded, linear FE analysis gives (assuming a good model) a reasonably accurate estimate of the strain and stress at the root of the notch. However, once the yield stress is exceeded, the elastic solution becomes increasingly unrealistic. In practice, as yielding occurs, there will be a redistribution of stress and strain around the notch, so that the real strain will be greater than the elastic value and the real stress less than the value from elastic analysis. The true solution must lie somewhere on the material stress-strain curve.	
	Hoffmann-Seeger	The Hoffmann-Seeger method is a modified version of the Neuber method that takes into account the state of stress, allowing it to be extended to proportional multiaxial loadings. If the loading is very non-proportional, then most of the assumptions implicit in this method are violated; hardening is really kinematic and not isotropic, and the principal axes may not only rotate, but the principal stress and strain axes are no longer necessarily aligned with each other. The Hoffmann-Seeger method requires 2-D stresses or strains (e.g., from thin shells, or surface resolved) and knowledge of the biaxiality ratio.	
	None	No elastic-plastic correction will be carried out. This option is valid only when both elastic-plastic strains and stresses are available from the FE.	
Mean Stress Correction			**None**
	None	No mean stress correction applied.	
	Morrow	Morrow method of mean stress correction applied.	
	Smith-Watson-Topper	SWT method of mean stress correction applied.	
	Interpolate	Use multi-curve method. Note that this only applies to Multi-Mean Curve, Multi R-Ratio Curve, and Haigh Diagram S-N analysis sub-methods. Automatically selected for these Haigh diagram S-N analysis sub-methods.	

Table 6-2 Solution Parameters

Parameter	Options	Description	Default
SWT Method		There are two possible interpretations of the SWT (Smith-Watson-Topper) method – **Iterative** and **Formula**.	**Iterative**
	Formula	In the **Formula** method, the an equation is solved for life/damage.	
	Iterative	In the **Iterative** method, a fully reversed cycle that has the same value of PSWT as the cycle being analyzed is sought	
Stress Combination		CombinationMethod defines how a damage parameter for fatigue calculation will be extracted from the strain tensor history.	**Max Abs Principal**
	Max Abs Principal	The absolute Maximum Principal stress is defined as the principal stress with the largest magnitude	
	Signed Von Mises	The Signed von Mises stress is the von Mises stress, but forced to take the sign of the Absolute Maximum Principal stress.	
	Signed Shear	The Signed Shear stress is the Maximum Shear stress (Tresca Criterion), forced to take the sign of the Absolute Maximum Principal stress. A factor of 2 is applied to ensure that the parameter has the same value as the Absolute Maximum Principal stress under uniaxial loading conditions.	
	Critical Plane	The Critical Plane option is available only for stress flagged as 2-D. The normal stress is calculated and rainflow counted on multiple planes.The critical plane is the plane with the most predicted fatigue damage. The planes on which the normal stress is determined have normals that lie in the plane of the physical surface, i.e., in the x-y plane of the 2-D stress results coordinate system. The orientation of each plane is defined by the angle ϕ made with the local x-axis.	
	Type-B Critical Plane Shear Strain	The Type B Critical Plane Shear Strain is derived from the out-of-plane shear strain on planes inclined at 45 degrees to the free surface. It can be used only with 2-D FE Results (typically from thin shells or resolved to the surface).	

Table 6-2 Solution Parameters

Parameter	Options	Description	Default
Damage Calc Method		The solutions of the equations required to predict life/damage in the local strain approach require numerical methods. There are two options – **Single Shot** and **Lookup Table**.	**Lookup Table**
	Lookup Table	As the name suggests, linearizes the curves into a lookup table. When there are a lot of cycles, the use of the lookup table may speed up the calculation, but at the cost of a slight loss of precision, depending upon the size of the table.	
	Single Shot	uses an iterative (Newton-Raphson) method,	
LookUp Table Size	2 < **Value** <= 1000	Sets the number of values used in the lookup table (see above). Larger values will give increased accuracy at the expense of some speed.	**256**
Interpolation Limit		This option is used to specify how the multi-curve method compute values at the extrema of the material data sets.	**Use Max Curve**
	Use Max Curve	Material property values are not extrapolated from the material data set values. Rather, the values nearest the extrema are used.	
	Extrapolate	Material property values are extrapolated from the material data set values.	
Plastic Limit Load Correction		Simple notch rules such as the Neuber and Hoffmann-Seeger rules estimate the redistribution of stress and strain around a notch with limited yielding, but with no reference to the geometry of the component. In practice, plasticity may become more widespread, and these simple notch rules may then significantly underestimate the plastic strains.	**None**
	None	There is no effect of shape factor considered. The property has no effect if **Elastic Plastic Correction** = **None**	
	Seeger-Heuler	The Seeger-Heuler method provides a variation on the Neuber or Hoffmann-Seeger methods that increases the amount of plasticity estimated when net section yielding is predicted, based on a Shape Factor.	
Overall Scale Factor	0 < **Value** <= 1e10	The scale factor to apply to the combined data prior to performing damage calculation.	**1.0**

Table 6-2 Solution Parameters

Parameter	Options	Description	Default
Use Stress Gradient	**Enabled, Disabled**	This option allows to apply a stress correction based on the gradient in stress in the direction normal to the free surface. Only applies to surface resolved stresses on the free surface. The stress gradient correction is based on gradients computed in MSC Fatigue.	**Disabled**
Gradient Units	**Millimeters, Meters, Inches, Feet**	Length units for stress gradients.	**Millimeters**
Certainty of Survival	0 <= **Value** <= 100	Percentage certainty of survival	**50**

Material Information Updates

A few entries are available in the Materials Information spreadsheet for the DTLib-based solver that may not be available for the Classic Modules.For completeness, all available entries are described in the table below.

Table 6-3 Material Information

Table Entry	Option	Description	
Material		Material Name	**No Default**
Finish		Surface finish and treatment can have a significant effect on fatigue behavior. Rough surface finishes, e.g., due to machining marks, will in general reduce the fatigue strength, whereas surface treatments are often applied to increase the fatigue strength.	**Polished**
	Polished		
	Ground		
	Machined		
	Poor Machined		
	Rolled		
	Cast		
	Enter K-Roughness	The value of the K-Roughness property is used directly	
	Enter Surface Roughness	The value of the Surface Roughness property is used to calculate the surface roughness factor	
Roughness	0 <= **Value**	Value of K-Roughness or Surface Roughness; applicable only for Finish options "Enter K-Roughness" and "Enter Surface Roughness"	**0.**
K-Treatment	0 <= **Value**	User surface treatment factor	**0.**

Table 6-3 Material Information

Table Entry	Option	Description	
Region		Material Group to which material properties apply	**No Default**
Layer		Layer is generally associated with shell element surface (z1, z2). The z1/z2 nomenclature is the Nastran definition for bottom/top shell surface. The layers available are the only ones available in the model. If solids only, appears as un-layered. If **Composite** option is chosen, a model with multiple intermediate layers can have 1 – n layers.	**No Default**
Kf		User surface factor	**1.0**
Shape Factor		This property is used by the EN analysis engine only, and then only when the **Seeger-Heuler** plastic limit load correction is invoked.	**No Default**
Weld			**N/A**
Multiplier		The scale factor to apply to this material	**1.0**
Offset		A stress offset for this material	**0.0**

Multi-Curve Materials (dtfat)

The **Analysis Sub Method** property on the Solution Parameters form defines the format of the S-N or E-N data to be used for the analysis, and the method for calculating damage. Aside from the standard S-N or E-N material data, an number of multi-curve methods are available.

For S-N analysis, the options are:

- Multi Mean Curve
- Multi RRatio Curve
- Haigh
- Bastenaire
- Multi Temperature Curve

For a detailed discussion of the format of the different S-N data types, the meaning of the parameters and their use, please see SNMethod (Ch. 5) in the Theory Guide.

For E-N analysis, the options are:

- Multi Mean Curve
- Multi RRatio Curve
- Multi Temperature Curve

For a detailed discussion of the format of the different S-N data types, the meaning of the parameters and their use, please see ENMethod (Ch. 5) in the Theory Guide.

Notes on Mean Stress Correction and the Multi-Curve Methods

Whether or not a particular mean stress correction method can be applied depends upon the formulation of the S-N data (SNMethod) that is being used. The allowable combinations are detailed in the table below: For a more detailed discussion, please see MeanStressCorrection (Ch. 5) in the Theory Guide.

Table 6-4 Allowable S-NMethod/MeanStressCorrection Combinations

		SNMethod					
		Standard	MultiMean Curve	MultiRRatio Curve	Haigh	Bastenaire	Multi-Temperature Curve
Mean Stress Correction	None	P	✓*	✓*	O	P	P
	Goodman	P	✓*	✓*	O	P	P
	Gerber	P	✓*	✓*	O	P	P
	Interpolate	O	P	P	P	O	O
	FKM	P	✓*	✓*	O	P	P
	GoodmanTensionOnly	P	✓*	✓*	O	P	P
	GerberTensionOnly	P	✓*	✓*	O	P	P

✓= Allowed
×= Not allowed
✓* = Allowed but a curve for R = -1 or zero mean stress must be present

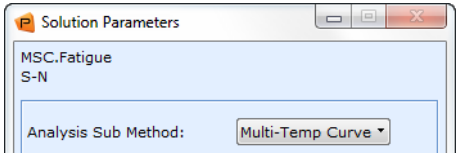
Each of the mean stress correction methods is described below.

Temperature Dependent Fatigue (dtfat)

Temperature corrected fatigue analysis enables the calculation of uniaxial E-N, S-N, or Vibration fatigue through utilization of temperature corrected materials data at non-ambient temperatures. Temperatures may be assigned to the analysis group globally, by group (region) or extracted from a Thermal case in the Patran database.

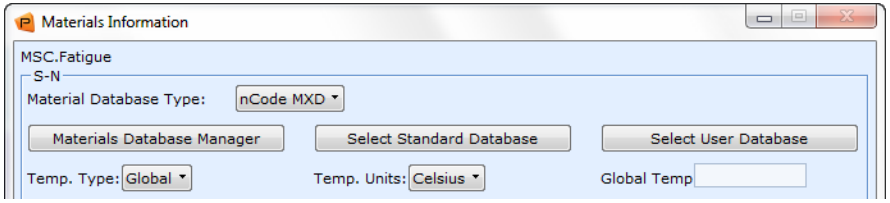
The temperatures are steady state (i.e. constant with time) and do not include time varying or creep effects but may vary across the FE model.

In order to perform a Temperature Dependent fatigue analysis, first the Temperature dependent material type must be selected on the Solution Parameters form. For S-N analysis, the Analysis Sub-Method **Multi-Temp Curve** is selected as shown below.

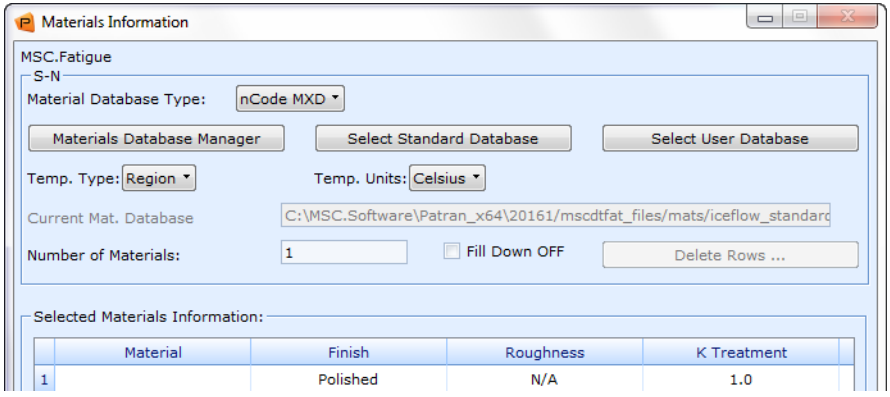


The temperature dependent material selection, temperature data, and other information are specified on the Materials Information form. Several data inputs must be specified, depending on the **Temp. Type** selected on the Materials Information form. The different options for **Temp. Type** are shown below.

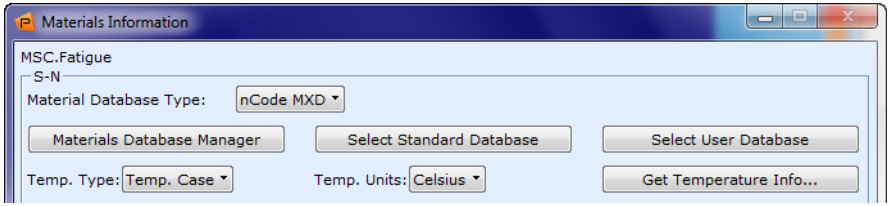
The most basic option is **Temp. Type** = **Global**. For this option, all nodes/elements are assigned the temperature value specified in the **Global Temp** data box. The specified temperature is given in units specified with the **Temp Units** option.



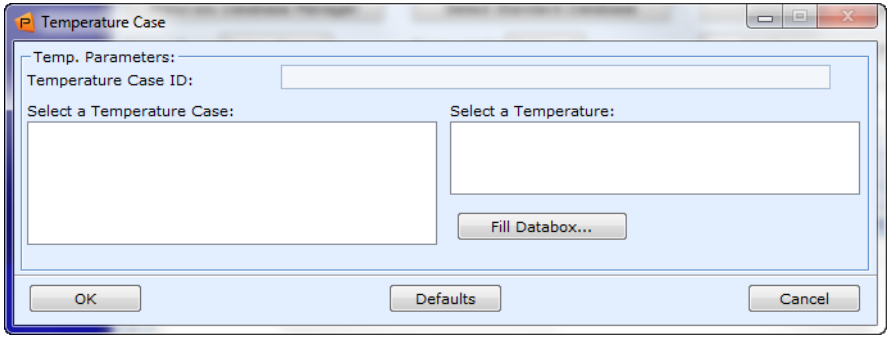
A slightly less granular option is **Temp. Type** = **Region**. For this option, all nodes/elements in a particular material group are assigned the temperature value specified in the **Temperature** cell on the Selected Materials Information spreadsheet. The specified temperature is given in units specified with the **Temp Units** option.



The least granular option is **Temp. Type** = **Temp Case**. For this option, all nodes/elements are assigned the temperature associated with a selected temperature results case. The specified temperature is given in units specified with the **Temp Units** option. The temperature results case can be selected on the Temperature Case form, which is accessed by clicking the **Get Temperature Info** button on the Materials Information form.



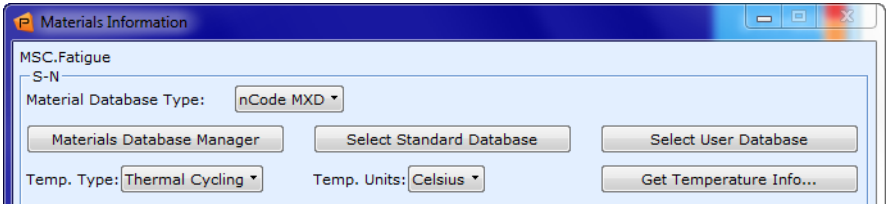
Using the Temperature Case form, the user can highlight a temperature results case in the **Select a Temperature Case** data box, and then make this results case the selection by clicking the **Fill Databox** button. Clicking **OK** finalizes the selection. To overwrite the currently selected results case, select a different results case in the **Select a Temperature Case** data box and then click the **Fill Databox** button.



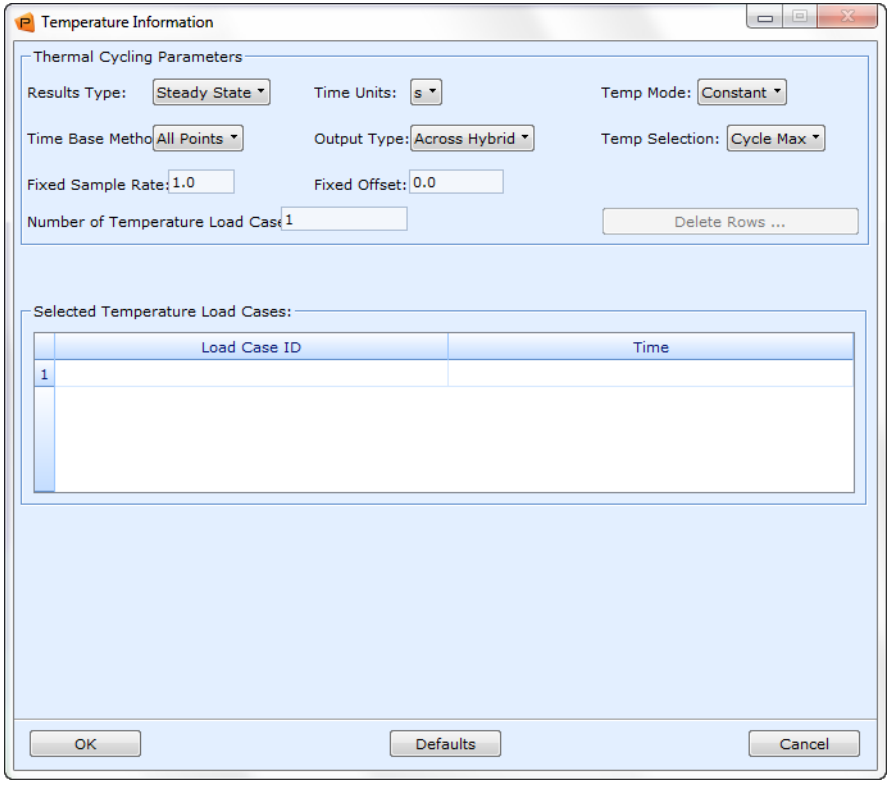
Thermal Cycling (dtfat)

Components under cycling loading can be adversely affected by high temperatures. MSC Fatigue has two methods available for which temperature effects can be included in the fatigue analysis. A thermal cycling approach can be used with S-N analysis. With the thermal cycling approach, temperatures can be defined at different points in time, with a number of options for how the temperature varies between points.

In order to use themal cycling approach, the **Temp. Type** = **Thermal Cycling** should be chosen on the Materials Info form.



The settings for the thermal cycling capability can be set on Temperature Infomation form. This form can be accessed by clicking the **Get Temperature Info** button on the Materials Information form.



The Thermal Cycling fatigue analysis requires the introduction of two new concepts in MSC Fatigue, the Hybrid Load Provider and the Temperature Load Provider. The Hybrid Load Provider allows loads from different sources to be combined to provide loading cycles for fatigue analysis. The Temperature Load Provider allows definition of thermal conditions to be included in the fatigue analysis. Some important method options available to control DTLib for thermal cycling fatigue are given in the below.

In general, the thermal cycling feature has two built-in functionalities for getting temperature data for the fatigue analysis. The first is controlled by the **Temperaure Mode** setting. This setting determines which temperature data is passed on to the analysis engine. The associated options are:

Constant - assumes that the temperature distribution in the model is not time-varying; one temperature case should be selected.

Max, **Min** or **Median**- as the names suggest, pass on the maximum, minimum or median (average of maximum and minimum) temperature for each FE entity to the analysis engine.

All - generates a temperature time history, where the time value of each temperature step is also used. This mode can be used to perform cycleby-cycle temperature-corrected calculations.

The second functionality is provided by the analysis engine itself. This functionality controls the temperature value, or values, that are used in the calculation when a hybrid load provider is configured that provides multiple temperature-time values.

With the **All Available** option, the temperature time history is used point-by-point alongside the stress time history. In the other modes, the maximum, minimum or median value from the temperature data is used for each point in time as a constant temperature. This has the same effect as using the equivalent options in the hybrid load provider.

Note that for performance reasons, the Median is estimated as the average of the maximum and minimum values.

The settings and options available on the Temperature Information form are given in the table below.

Parameter	Description	Options	Default
Results Type	This option allows static thermal results cases to be selected and a time to be associated with each static results case. In this way, a pseudo-time dependent temperature profile can be established.	**Steady State** A single selected temperature case is used to provide a temperature for each entity. Currently, this is the only option	**Steady State**
Time Units	This property specifies the time units of the generated data of the hybrid load provider, and also the time units of all of the subload providers. Care should be taken to ensure that all of the sub-load providers do have the correct matching time units. For example, if one of the sub-load providers is a time series, the time series data files should have X axis units that match the units selected here. No checking of this is done by the hybrid load provider.	**s** Seconds **mins** Minutes **hours** Hours **days** Days **weeks** Weeks	**s**
Temp. Mode	Mode of operation for the temperature load provider. Defines how to select temperatures from the selected temperature steps. Note that this is a LoadProviderTemperature parameter.	**Constant** A single selected temperature case is used to provide a temperature for each entity. **Max** For each entity, the maximum temperature from the selected steps is used for analysis. **Min** For each entity, the minimum temperature from the selected steps is used for analysis. **Median** For each entity, the median temperature from the selected steps is used for analysis. **All** This option generates a temperature time history, where the time value of each temperature step is also used. This mode can be used to perform cycle by cycle temperature corrected calculations.	**Constant**

Parameter	Description	Options	Default
Time Base Method	The method to use for generation of the time base of the load provider. Note that this is a LoadProviderHybrid parameter.	**All Points** An output point will be generated for all time points from all of the history load providers. If two or more load providers define a data point at the same time, that time will only be output once. This method means that the data will not be equispaced in time and therefore time data will not be available to the analysis. This mode should not be used if a time dependent analysis, such as creep, is being performed. **Fixed Sample Rate** In this case, the time values to generate will be defined independently using the **Fixed Sample Rate** and **Fixed Offset** properties. Data will be linearly interpolated from each load provider. This mode will provide time data suitable for analysis such as creep. Care should be taken when setting the offset and sample rate to ensure that peaks in the load providers are not lost. If one of the load providers is a rapidly changing time series, and another is, say, temperature, then setting the sample rate and offset to match the sample rate and base time of the time series is likely to preserve peaks correctly **Auto Sample Rate** This is similar to **Fixed Sample Rate** but the sample rate is determined automatically by looking at all the contributing load providers. If at least one time series or constant amplitude load provider is available then the sample rate is taken from that. If more than one is available then they must all have the same sample rate and base offset. If none are available then an error is generated.	**All Points**

Parameter	Description	Options	Default
Output Type	This property determines the method used to generate the multi-column output from the hybrid load provider (if requested).	**Across Hybrid** **Per Loadprovider** **Per Loadcase**	**Across Hybrid**
Temperature Selection	Determines how to process temperature data. Temperature Mode and Time Base Method contains settings that control how temperature data will be made available to the solver. **Temperature Selection** contains settings that control how the solver will use temperature data made available to it. For this reason, it is advised to set **Temperature Mode** = **All** and use **Temperature Selection** to further control what temperature data is actually used. Note that **Temperature Mode** = **All** and T**emperature Selction** = **All Available/Cycle Max/Cycle Min** are required for **thermal cycling**. Any other selection results in an isothermal case.	**Overall Max** An isothermal calculation is done using the maximum value of the temperature data. **Overal lMin** An isothermal calculation is done using the minimum value of the temperature data. **Overal lMedian** An isothermal calculation is done using the median value of the temperature data. The median is defined as (Max+Min)/2. **Cycle Max** A cycle by cycle temperature correction is made using a time series from the temperature load provider. The maximum temperature for each cycle is used to make the correction. **Cycle Min** A cycle by cycle temperature correction is made using a time series from the temperature load provider. The minimum temperature for each cycle is used to make the correction. **All Available** Used only for ChabocheTransient method. Normalizes the stress time series using the temperature time series before applying the Chaboche fatigue calculation.	**Cycle Max**

Parameter	Description	Options	Default
Fixed Sample Rate	This property is only used if **Time Base Method** is set to **Fixed Sample Rate**. In that case, this property defines the sample rate of the generated data. The time units for the sample rate are based on the **Time Units** property, e.g. if Time Units is set to Seconds, then the sample rate should be specified in points/second.	**Value** > 0.	**1.**
Fixed Offset	This property is only used if T**ime Base Method** is set to **Fixed Sample Rate**. In this case, data points are generated at equally spaced time intervals defined by the **Fixed Sample Rate property**. This property defines a time offset to apply. For example, if the sample rate was set to 1 and the offset to 0.3, then data points would be generated at 0.3, 1.3, 2.3 etc. seconds. Note that the actual time range for which data points are generated is governed by the time range of the underlying loading histories. So it is possible that no value will be generated at the **Fixed Offset** position if this lies outside the time range of the underlying load histories. The units of the offset are specified in the **Time Units** property.	**Value**	**0.**

dtfat Batch Operation

The MSC Fatigue analysis module **dtfat** is a multi-discipline fatigue analysis program. The name dtfat refers to fatigue analysis using the solver technology of the HBM nCode DTLib-library. Recall that dtfat runs in batch-only mode, and has no built-in GUI. Please see Command Line Usage for more information.

The DTLib-based solver has the capability to output stress time history and cycle count information. This can be done by running the solver from the command line, and using the appropriate command line options. Since this can result in a large amount of data, it is recommended that this optional output is only performed for a few entities of interest. The entities can be selected via the command line option /ENTIDS or using hot-spot filtering techniques.

In order to output stress time history, the /TSOUT command line option is required. The command line option /OUTTXT can be used to switch between text-based output (i.e., csv file) and binary output (i.e., dac file). An example command line is as follows:

```
dtfat /FERES=PDA /TSOUT=Y /ENTIDS=1-2 /INP=jobname.fin
```

The following text-based stress time history data files are created for each entity:

`jobname_ts_1_.csv`

`jobname_ts_2_.csv`

Using the command line option /OUTTXT=N, as shown below, results in dac file output:

```
dtfat /FERES=PDA /TSOUT=Y /ENTIDS=1-2 /OUTTXT=N /INP=jobname.fin
```

The following binary stress time history and metadata files are created for each entity:

`jobname_ts_1_01.dac`

`jobname_ts_2_01.dac`

`jobname_ts_1__dac.nmd`

`jobname_ts_2__dac.nmd`

Similarly, cycle count information can be output using the /CYCOUT command line option. The command line option /OUTTXT can be used to switch between text-based output (i.e., csv file) and binary output (i.e., nCode s3m file). Note, however, that MSC Fatigue currently doesn't have a viewer for the s3m file. An example command line is as follows:

```
dtfat /FERES=PDA /CYCOUT=Y /ENTIDS=1-2 /INP=jobname.fin
```

The following text-based cycle list data files are created:

`jobname_cyc_1_01.csv`

`jobname_cyc_2_01.csv`

Using the command line:

```
dtfat /FERES=PDA /CYCOUT=Y /ENTIDS=1-2 /OUTTXT=N /INP=jobname.fin
```

The following binary cycle list data files are created:

`jobname_cyc_1_.s3m`

`jobname_cyc_2_.s3m`

FE Fatigue Analysis Options (FEFAT)

The MSC Fatigue analysis module FEFAT performs many different tasks from simple file conversions to full blown fatigue analysis. Module operation of each of these tasks is described in detail in this section. The name FEFAT refers to FE-Fatigue which implies fatigue analysis from finite element (FE) data.

FEFAT handles all the pre-processing, data file import and export, all S-N and crack initiation analysis including factor of safety analysis. The fatigue design optimization analyzer is also available from the FEFAT menu.

The operation of FEFAT can be in two modes: within the MSC Fatigue and Patran environments or in stand alone mode from the system prompt. The only difference is that in stand alone mode, the user must supply the jobname when asked to preform the analysis. (In direct mode from a preprocessor such as Patran, these

are passed to FEFAT automatically.) FEFAT can be accessed directly from the operating system prompt by typing the symbol **fefat**. Once FEFAT has been initiated in either of these modes, two windows will be presented. The top, small form is a generic form and allows for general program control. This is discussed in detail in Module Operations (Ch. 18) for the Motif driver.

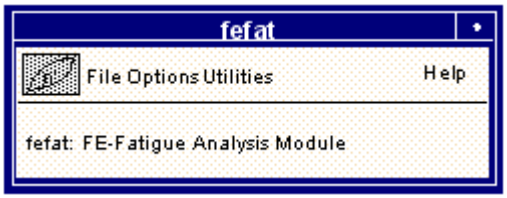


Figure 6-3 FEFAT Utility Form

The main menu appears as follows. Each item is discussed in this section.

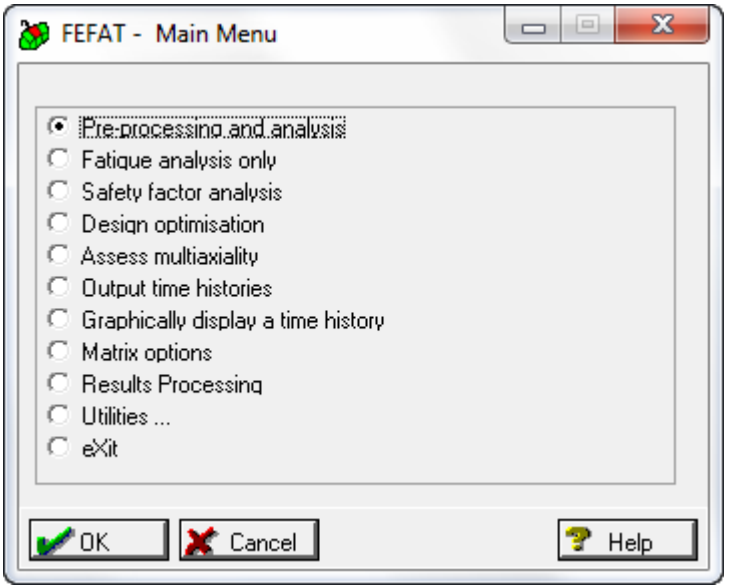


Figure 6-4 .FEFAT Main Menu Form

Fatigue Preprocessing

The complete fatigue analysis is actually split into two parts: the pre-processing (cycle counting) and the analysis. This is mainly because it affords more flexibility in the design optimization analysis. When running the global multi-node/element analysis, however, the FEFAT pre-processing option will run the analysis also if requested.

Important:	Remember that the fatigue analysis may take some time; so it may be desirable to ensure that the terminal is available for a long interactive session. It may be worth considering operating FEFAT in batch; batch operation of these programs is discussed in FEFAT Batch Operation, 417.

When the pre-processing option has been selected, the user will be presented with a number of questions. The first question asks for the input file name. Click the **OK** button once a file name (`jobname.fes`) has been selected. Use the List button to list all available fatigue input files. These files have been created by the PAT3FAT or FATTRANS translator. The default will be the last `jobname.fes` created. Once a valid file name has been entered, the user will be presented with a summary of the `jobname.fes` file that has been opened. Each of these parameters can be changed or edited.

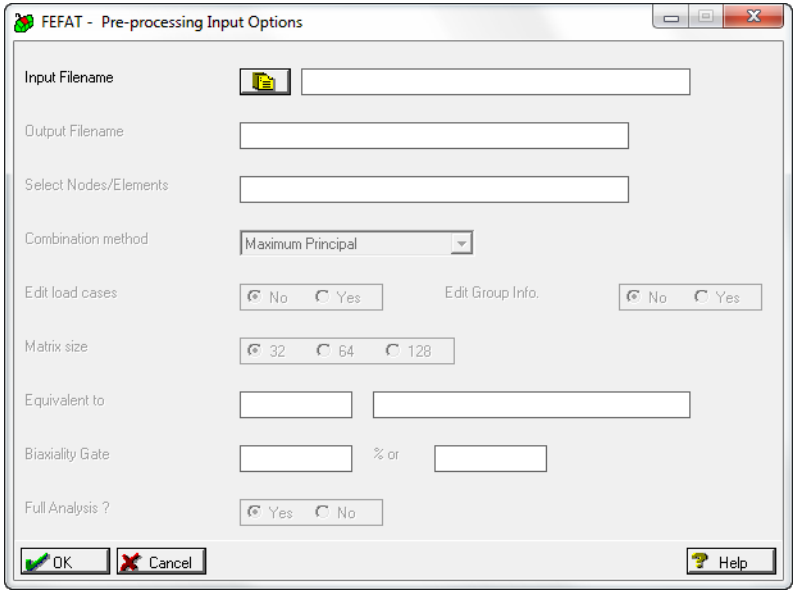


Figure 6-5 FEFAT Preprocessor Form

The following table explains each entry on the previous form.

Field	Description
Input Filename	This is the fatigue input file (`jobname.fes`) to be used in the fatigue pre-processing. The job must have already run at least through the PAT3FAT or FATTRANS translator to produce a `jobname.fes` file. This is achieved by carrying out a full, partial, or translate only submission from the job submit options in the MSC Fatigue menus, or by running PAT3FAT or FATTRANS in stand alone mode (see The Translator (PAT3FAT or FATTRANS) (Ch. 6)). A fatigue input file can also be created using FEFAT's Utilities, 360 or by running the separate module FEFTRN, 109. To select a jobname from a list of available jobs, use the List button. FEFAT accepts all types of jobs. Once the file name has been supplied and the screen inputs accepted, the rest of the initial input options will be displayed. These are described below.
Output Filename	The default is the jobname. After pre-processing, a file called `jobname.fpp` will exist. You will be requested to overwrite any existing output file of the same name if one exists. The main analyzers produce results files that have `.fes` or `.fos` extensions for fatigue results and factor of safety results respectively.
Select Nodes/Elements	This field accepts either a single node or element or the word *ALL*. The job automatically detects whether a nodal or elemental based analysis is to be performed from the `jobname.fes` file. The pre-processing can be time consuming for many nodes or elements, especially with multiple load cases and complicated time histories. Nodes or element numbers may be specified in the following ways: 1,2,3,4 - or the equivalent - 1:4 - or `@pfatigue.ents` - which is an ASCII file containing a list of node or element id numbers. A complete list of all nodes/elements is available by pressing the List button.
Combination method	The stresses and strains at a node/element are stored in component form. For an analysis a time history file needs to be extracted from the stresses and strains. In order to do this the stresses need to be resolved to a specific plane or tensor parameter, which are available on this list. The default is the absolute maximum principal (Abs. Max. Principle).
Edit load cases	If this toggle is on, you will be allowed to modify the load case information. You will be prompted to change the Time History, Applied Load, Scale Factor, and Offset for each load case.
Edit Group Info.	If this toggle is on, you will be allowed to modify the group information.

Field	Description
Matrix size	This sets the number of bins for cycle counting; more bins means a higher matrix resolution which will increase accuracy but also increase processing time and file sizes. The matrix size is remembered and any further jobs that are run in the same directory will use the size set here.
Equivalent to	In this field the number of equivalent units is set. If, for example, the time history is equivalent to 3 laps around a test track, then Laps would be a suitable name and 3 would be a suitable number (although you may enter any name or number).
Biaxiality Gate	A biaxiality analysis can be sped up by excluding small, and therefore less significant stress vectors, from the calculation. The default gate is 20% of the materials ultimate tensile strength (UTS). An absolute value in MPa can also be entered. The larger of the two values will be used in the calculation.
Full Analysis?	If Yes, then the full fatigue analysis will be executed automatically. If No, the job will stop and you will be returned to the Main menu of FEFAT. A fatigue analysis can be carried out independently from the **Fatigue analysis only** option on the Main menu.

If a partial analysis is specified then there is no post-analysis form. In the case of a full analysis then a results form is displayed with the results of the most damaged nodes similar to that presented by the results viewing module PFPOST. See Figure 6-33.

Fatigue Analysis

This option allows a pre-processed job to be analyzed (or reanalyzed). The input file will be an intermediate results file produced within pre-processing and analysis as described in the previous section. The file extension is `.fpp`.

The **Fatigue analysis only** option has two principal forms. The first allows the input and output file names to be specified, and offers you the option of editing the preparedness job. Editable parameters are shown on Figure 6-6. If no parameters are to be edited the second form will not appear.

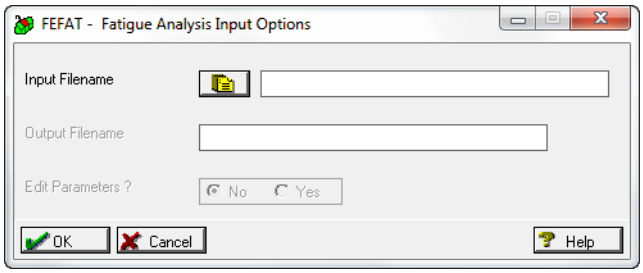


Figure 6-6 FEFAT Fatigue Analysis Input Form

Field	Description
Input File Name	This is the fatigue preprocessor file (`jobname.fpp`) to be used in the fatigue analysis. The job must have already run through the fatigue preprocessor to produce a `jobname.fpp` file. This is achieved by carrying out a full, or partial submission from the job submit options in the MSC Fatigue menus, or by running PAT3FAT or FATTRANS in stand alone mode (see The Translator (PAT3FAT or FATTRANS) (Ch. 6)) and then the FEFAT Preprocessor option. To select a jobname from a list of available jobs, use the List button. FEFAT accepts two types of job: total life and crack initiation. Once the File name has been supplied and the screen inputs accepted, the rest of the initial input options will be displayed. These are described below.
Output File name	The default is the jobname. After the analysis, a file called `jobname.fef` will be created. You will be requested to overwrite any existing output file by the same name if one exists.
Edit Parameters?	If Edit is set to Yes then the edit form shown in Figure 6-7 will be displayed.

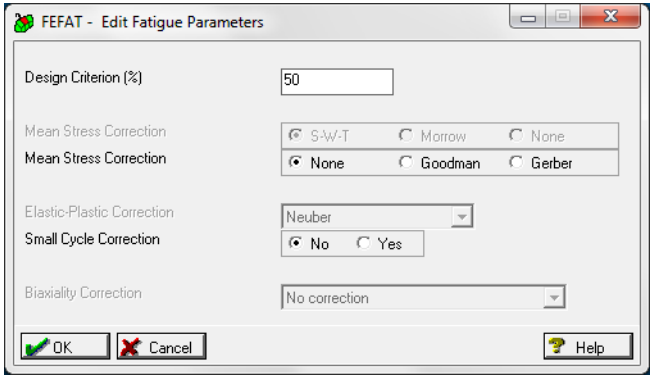


Figure 6-7 FEFAT Edit Fatigue Parameters Form

Field	Description
Design Criterion (%)	This value is the percentage certainty of survival and is a statistical parameter between 0.1 and 99.9%. It is based on the scatter in life curves which is described by the standard error parameter stored in the material data set for the stress or strain life curves.
Mean Stress Correction (**crack initiation**)	The Smith-Watson-Topper and Morrow correction methods are used when the local strain (low cycle fatigue) approach is used to predict lives. They take into account the effect of non-zero mean stresses on the strain life curve. These effects are fully discussed in the technical overview, but in brief it is true to say that in mainly tensile mean stresses the SWT approach is more conservative and in mainly compressive situations the Morrow approach is the more conservative. It is advisable to check all methods before selecting the final basis for a life estimate.
Mean Stress Correction (**S-N**)	Most fatigue cycles do not have a mean value that is zero. This is significant because S-N curves are obtained using tests where the loading has a zero mean. The fatigue cycles therefore have their amplitude adjusted and reset to a zero mean. For S-N curves there are two correction methods available - Goodman or Gerber. The Goodman method gives the most conservative answers. For a full explanation of the Goodman and Gerber methods see the theory section.

Field	Description
Elastic-Plastic Correction	The correction factor selected allows the FEFAT to convert the elastic stresses and strains from their FEA values to true stresses and strains for use in crack initiation modelling. By default the Neuber correction is used but the Mertens-Dittmann and Seeger-Beste modifications to the Neuber method can give better, i.e. more conservative, results for unnotched geometries and those where the plasticity is not highly localized. The Seeger-Beste method gives the most conservative results, but both it and Mertens-Dittmann require a shape factor which is shape and loading dependent.
Small Cycle Correction	If this is set to Yes then the Heibach correction is carried out. The Heibach correction accounts for the fact that small fatigue cycles which lie at a level below the endurance limit do become damaging if they follow a large cycle. If the largest cycle is above the endurance limit of the S-N curve then all cycles will be damaging because the S-N curve is changed such that the slope beyond the endurance limit cut-off is greater than zero.
Biaxiality Correction	This is the correction method for proportional biaxiality and methods available are the Hoffmann-Seeger and Parameter correction. They should only be used where shell element results or surface resolved solid nodal results are available. If No correction is selected then FEFAT will use the Neuber method and the chosen strain combination on the uniaxial cyclic stress-strain curve. The Hoffmann-Seeger method uses the biaxiality ratio to convert the combined strain parameter to an equivalent strain (based on the von Mises strain) before carrying out the Neuber correction and then recalculating the elastic-plastic stresses and strains. It is applicable to the absolute maximum principal and signed Tresca strain conditions. The Parameter Modification method works by calculating on the basis of the mean biaxiality ratio a new cyclic stress-strain curve for each node or element. The new curve relates absolute maximum principal stress and strain amplitudes and should only be used with this strain combination (set during pre-processing).

After an analysis, a results form is displayed with the results of the most damaged nodes similar to that presented by the results viewing module PFPOST. See Figure 6-33.

Factor of Safety Analysis

The MSC Fatigue factor of safety analysis option in FEFAT provides a set of semiautomatic tools to assess stress factors based on S-N or ε-N fatigue curves. It uses the pre-processed fatigue data from the global multi-node/element analysis (`jobname.fpp` file) to carry out its analyses. Both total life (S-N) and crack initiation (ε-N) calculation methods are supported by FEFAT.

This option calculates a factor of safety, i.e. over-design, which is reflected in the excess of the estimated life over the target life. Two analysis methods are provided: stress-based and life-based. Only the life-based method is available for crack initiation jobs.

FEFAT presents its results in the form of analysis summary reports. The factors reported are color coded for easy interpretation in PFPOST.

The stress-based method needs a fatigue endurance limit, and calculates the amount by which the stress must be factored to exceed that limit.

Factors less than 1.0 indicate a nonconservative design. For variable amplitude loading the worst scale factor is reported based on an analysis of the largest load cycle.

The life-based method performs a full fatigue back calculation to determine the stress scaling factor.

Analyses may be performed on a single node/element or for the entire data-set and every loading cycle.

Important:	This analysis is based on a calculation of a stress scaling factor needed to meet a given design life. There are other factors that can affect the safety of structures other than stress such as material properties and manufacturing variations and processes. All such factors should be taken into account in a safety analysis.

The first screen to be presented when the program starts is shown in Figure 6-8.

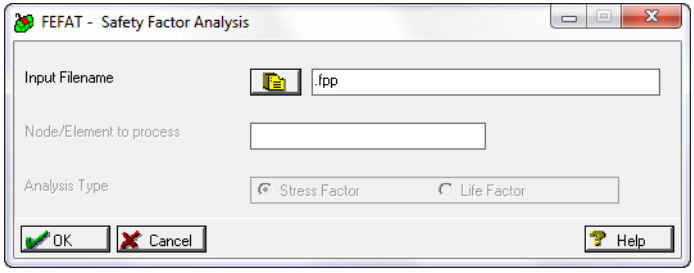


Figure 6-8 Factor of Safety Job Entry Screen

The fields on this screen are described below.

Field	Description
Jobname	This is the name of the job which is to be used in the fatigue Factor of Safety analysis. The job must have already run at least through the fatigue preprocessor, FEFAT, to produce a `jobname.fpp` file. This is achieved by carrying out a full or partial submit from the job submit options in the MSC Fatigue menus, or by running PAT3FAT (or FATTRANS) and FEFAT in stand alone mode. To select a jobname from a list of available jobs, use the List button. FEFAT accepts two types of job: Total Life (Material S-N only) and Crack Initiation. Once the jobname has been supplied and the screen inputs accepted, the rest of the initial input options will be displayed. These are described below.
Node/Element to process	This field accepts either a single node or element or the word *ALL*. The job automatically detects whether a nodal or elemental based analysis is to be performed from the `jobname.fpp` file. If All is selected, then factors for all nodes or elements in the above file will be calculated. This can be time consuming for many nodes or elements.
Analysis Type	This switch indicates which Factor of Safety analysis type is to be performed. This switch is disabled for crack initiation analysis and allows only a *life-based* factor analysis.

The next screen to appear allows for the setup of various parameters dependent on the analysis type selected. For a life-based factor of safety analysis, the form in Figure 6-9 appears.

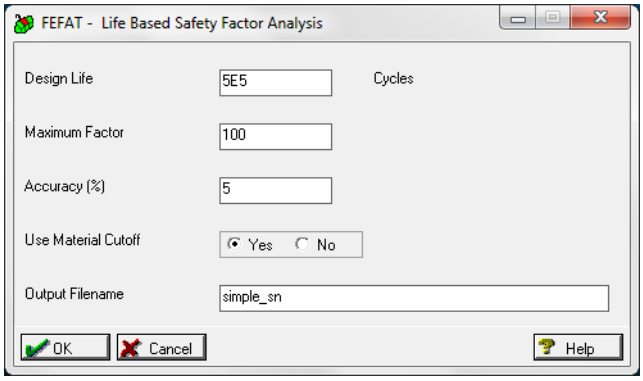


Figure 6-9 Life-Based Factor of Safety Setup Screen

The fields to the life-based form are described here. See Fatigue Theory (Ch. 16) on the formulation of the factor of safety equation.

Field	Description
Design Life	This value is required and specifies the target life expected. A resulting stress factor of unity would indicate that this design life was exactly achieved.
Maximum Factor	The default for this field is 100 meaning that stress factor calculations will be terminated if the factors exceed 100. To speed up an analysis this maximum factor can be lowered.
Accuracy(%)	Specifies the accuracy of the stress factor calculation. The default is set to 5%. Again the analysis can be accelerated by requesting a less accurate result (more than 5%) or the accuracies can be increased (numbers less than 5%), but solution times will increase and the results may not be meaningful in the context of the errors associated with fatigue life estimation. Back calculation accuracy can be set between 0.1 and 20.2%.
Use Material Cutoff	Specifies whether to use the material cutoff parameter from the cyclic material properties. The default is YES. With this parameter turned off, all cycles from the loading time history are considered in the factor of safety analysis. When on, those cycles below the material cutoff value are ignored.
Output File Name	If the user has requested to calculate factors for all nodes or elements then the results are sent to a file as well as the screen. The default name of this file is `jobname.fos` but can be changed in this field.

For a stress-based factor of safety analysis the form appears as in Figure 6-10.

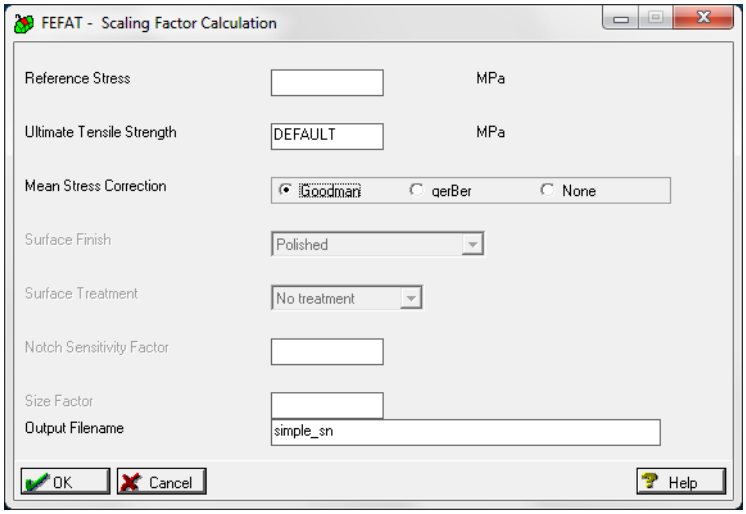


Figure 6-10 Stress-Based Factor of Safety Setup Screen

The stress-based parameters are described below. See Fatigue Theory (Ch. 16) on the formulation of the factor of safety equation

Field	Description
Reference Stress	This value is required and is the endurance or fatigue limit of the material or the stress below which no damage occurs for an S-N analysis. The Haigh diagram uses an endurance stress to define the relationship between the stress range and mean stress. The reference stress is the allowable stress amplitude at the required design life.
Ultimate Tensile Strength	This value is passed through the `jobname.fpp` file from the material information at each node/element but which can be changed here if desired. Note that if a single value is applied and many nodes are being analyzed then the value will apply to all nodes (if DEFAULT were set then each node would have its own UTS value).
Mean Stress Correction	The stress -based factor of safety analysis is based on a ratio of the actual stress amplitude and the allowable stress amplitude at a given mean stress. The allowable amplitude is taken from the Goodman or Gerber equations. When 'None' is selected the calculated factor is independent of the material's UTS.

Field	Description
Surface Finish	Surface finish has an important effect upon fatigue life. To model this effect the slope of elastic strain life or stress life curve is adjusted using a factor calculated from the UTS and a set of correction curves. All surface finishes are available for a single node or element analysis, although only 1 can be applied per analysis. For a global analysis (if ALL was entered) the defaults are used from the `jobname.fpp` file.
Surface Treatment	Surface treatment has an important effect upon fatigue life. To model this effect the slope of elastic strain life or stress life curve is adjusted using a factor calculated from the UTS and a set of correction curves. All surface treatments are available for a single node or element analysis, although only 1 can be applied per analysis. For a global analysis (if ALL was entered) the defaults are used from the `jobname.fpp` file.
Notch Sensitivity Factor	This factor is set to 1 by default for a single node or element. It acts as a scale factor to the factor of safety equation. It is only applicable to single node analyses and must be greater than 1.
Size Factor	This factor is set to 1 by default for a single node or element. Works the same as the Notch Sensitivity Factor.
Output File Name	If the user has requested to calculate factors for all nodes or elements, then the results are sent to a file as well as the screen. The default name of this file is `jobname.fos` but can be changed in this field.

The safety factors are in a simple tabular format as shown below.

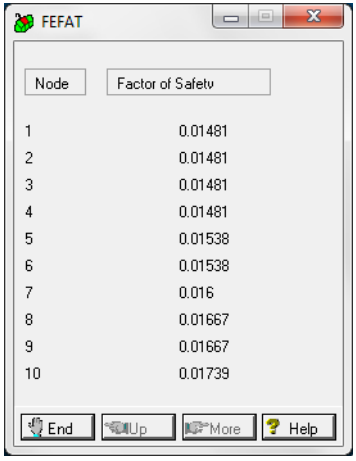


Figure 6-11 Factor of Safety Results Report

Results of unity suggest that the target life is exactly achieved and there is no safety factor margin. Results less than unity should be looked into for possible design changes immediately.

Design Optimization

Having completed a global multi-node or multi-element analysis, the user will have identified an area of the structure that is either liable to fail at a life less than the design life or has such a long life that he wishes to explore manufacturing options using more cost effective methods and materials which will still achieve the target life. Alternatively, the user may already have some options for material selection or geometry detail which he wants to assess in terms of their effect on fatigue life.

The MSC Fatigue design optimization FEFAT provides a set of semi-automatic tools to assess fatigue design options. It uses the pre-processed fatigue data from the global multi-node/element analysis (`jobname.fpp`) file to carry out its analyses quickly. It supports a number of options including back calculation of parameter values which meet a target life, sensitivity studies on critical parameters, and an automatic material selection option based on fatigue criteria. Both total life and crack initiation calculation methods are supported by FEFAT including material and component S-N analyses.

Having selected the node or element of interest, FEFAT will carry out a single fatigue calculation based on the default parameters from the global multi-node/element analysis and present the results in more comprehensive form than that available in the global analysis. The design optimization analysis options are then presented on a main analysis page from which the user can set up the optimization calculations.

FEFAT presents its results in the form of analysis summary reports, 3-dimensional cycle or damage histograms, fatigue life sensitivity tables, and life versus parameter plots.

Stage 1 Module Operation

The operation of FEFAT from within the Patran environment is almost identical to its operation in stand alone mode. The only difference is that in stand alone mode, the user must supply the jobname. (In direct mode from MSC Fatigue or Patran, this is passed to FEFAT automatically.)

The first screen to be presented when the program starts is shown in Figure 6-12. This screen will be skipped if FEFAT is entered from Patran and the screen in will be presented instead. This is because FEFAT already knows the jobname as specified from the MSC Fatigue menus. The node/element number will also be passed to FEFAT (see next page).

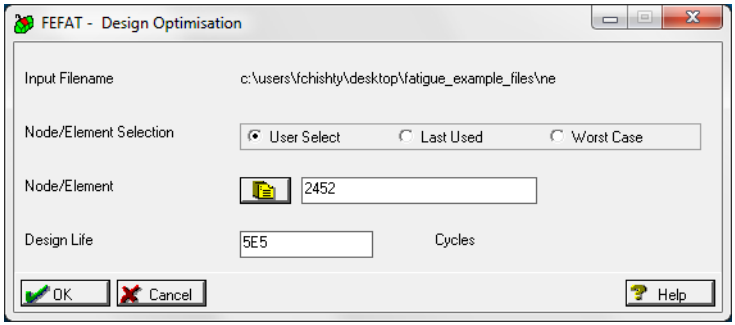


Figure 6-12 Design Optimization Job Entry Screen

The fields on this screen are described below.

Field	Description
Input File Name	This is the name of the job which is to be used in the fatigue design optimization analysis. The job must have already run at least through the fatigue preprocessor of FEFAT, to produce a `jobname.fpp` file. This is achieved by carrying out a full or partial submit from the job submit options in the MSC Fatigue menus, or by running PAT3FAT (or FATTRANS) and FEFAT in stand alone mode. To select a jobname from a list of available jobs, use the List button. FEFAT accepts two types of jobs: Total Life Analysis or S-N, and Crack Initiation Analysis or Strain-Life. Once the jobname has been supplied and the screen inputs accepted, the rest of the initial input options will be displayed. These are described below.
Node/Element Selection	There are three options offered on this field: *Last node/element* used recalls the number of the node or element used in the last job. This number is shown in the Node Number field. If the last job used a different geometry model, this option is unlikely to offer a meaningful node or element number. *User entry* allows for typing in a number in the Node/Element Number field shown below the Node/Element Entry menu. A list of possible node or element numbers is available using the List button. The *Worst case* node or element option is only available if a valid `jobname.fef` file exists. When this option is selected, the `jobname.fef` file is searched to find the node or element with the most damage as calculated by the global fatigue analysis. Once the critical node or element is found, its number is presented in the Node/Element Number field.
Node/Elem No.	The number displayed in this field depends upon the choice made in the Node/Element Entry described above. Use the List button to display a list of valid node or element numbers.
Design Life	The design life is a target life which is associated with the component or structure being analyzed. The life should be specified in the user units. These units and the number of these units equivalent to 1 repeat of the time history is displayed under the Design Life field. **Note:** A design life MUST be entered here.

When all fields are filled in appropriately, click the **OK** button. At this stage, FEFAT carries out an initial analysis using the original fatigue analysis parameters defined when the fatigue job was set up. The life computed from this "stage 1" analysis is used as a benchmark against which all subsequent optimization calculations can be judged.

The results from this analysis are presented in a summary table on the screen and also written, with additional information, to the `pfatigue.prt` file. See Figure 6-13.

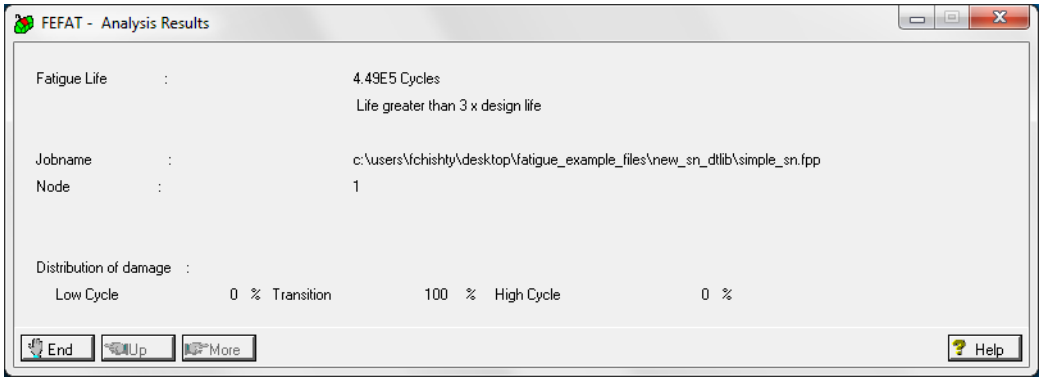


Figure 6-13 Results of the "stage 1" Fatigue Analysis for a Crack Initiation Job

The fatigue life is reported as a mean since the fatigue analysis is based on a rainflow cycle histogram where the damage for each range-mean bin in the histogram can lie between a minimum and maximum value. In practice, for real variable amplitude random loading, the mean damage is almost identical to the absolute damage (i.e., the damage calculated from the exact peak-valley data).

If a design life has been defined, a message will be written under the life result indicating whether the design life has been met or not. The three possible messages are:

- Design life exceeded
- Life within a factor of 3 of the design life
- Life less than the design life

The screen also shows the jobname and the node/element selected. Addition details are presented in the pfatigue.pat file. These details summarize the analysis parameters and will be specific to the type of job being carried out. Not all parameters will always be present.

Some of the parameters are not defined in the global analysis such as the stress concentration factor, residual stress, and the Miner's constant. While these parameters are not available for editing at this stage, they are provided as analysis options in the design optimization input screens described later.

The stress concentration factor allows an additional stress raiser to be defined even though it is not modeled in the FEA and the residual stress allows for the local modeling of manufacturing or assembly stresses. Miners' constant is the value at which failure occurs when the sum of the damage fractions from all the cycles equals this value. Some applications require this value to be different from one (see later). The generic histogram name is made up from the jobname and the node or element number. This name is used for the cycles and damage files which have the following naming conventions:

`jobnamenn.cyh`	Rainflow cycle histogram for node or element nn
`jobnamenn.dhh`	Corresponding damage histogram for node or element nn

On the bottom of the screen is a distribution of damage summary. This feature helps to establish the nature of the fatigue problem and possible solutions. For a discussion of low- and high-cycle fatigue, see High Cycle versus Low Cycle Fatigue, 1342.

The algorithm for calculating the high- and low-cycle fatigue contributions is based on a consideration of the magnitude of each cycle and the part of the life curve used to calculate damage for that cycle. The reasoning behind this logic is that structures experiencing low-cycle fatigue problems require more ductile materials to extend their lives whereas high-cycle fatigue problems are solved by increasing the strength of the material and by carrying out surface treatments. An example of this dependence on the type of fatigue problem is shown in Table 6-5 (Note that the magnitude of the percentage drop in life is dependent on other fatigue analysis parameters).

Table 6-5 An Example of the Effect of Surface Finish on Fatigue Lives for High- and Low-Cycle Fatigue Problems

Percentage of life for polished surface		
Surface Finish	88% High Cycle	76% Low Cycle
Polished	100	100
Ground	71	84
Good Machined	55	74
Poor Machined	34	57
Forged	15	30

Stage 2 Module Operation

After going through the initial reanalysis of a particular node or element, the main analysis screen is shown in Figure 6-14. From this menu, all the analysis options are available. The current jobname and node or element identity is shown at the top of the screen together with the Analysis and design life. The type of analysis that is reported on the screen depends on what action was last performed. Menu options which are followed by three dots indicate the existence of another form to be filled out.

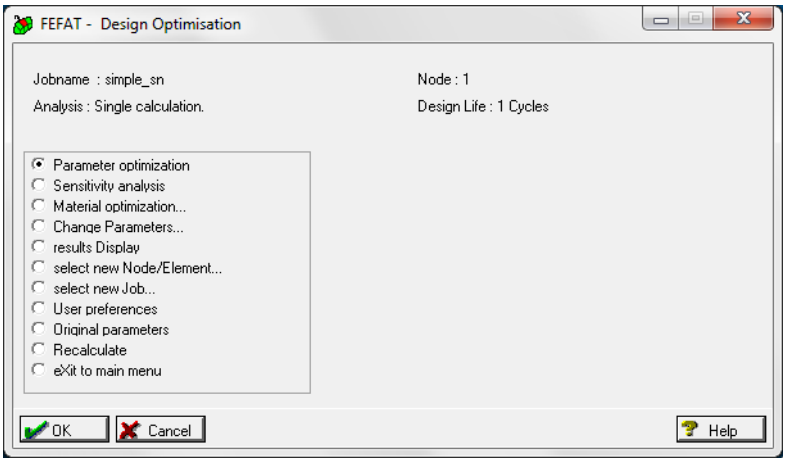


Figure 6-14 Fatigue Design Optimization Main Menu

To use this menu, choose the required option, set up the analysis parameters, and finally, when ready, select the Recalculate option to submit the analysis. A percentage complete message will inform the user of the progress of the calculations. A description of each menu pick follows.

Parameter Optimization

This option is the back calculation facility where a design life is supplied and FEFAT's automatic routines calculate the value of the chosen parameter that will achieve the target life; see Figure 6-15. There are five fatigue analysis parameters which may be used in this type of calculation though not all parameters are available for all analysis types (material S-N, component S-N, and crack initiation). The parameters on which back calculation may be carried out are:

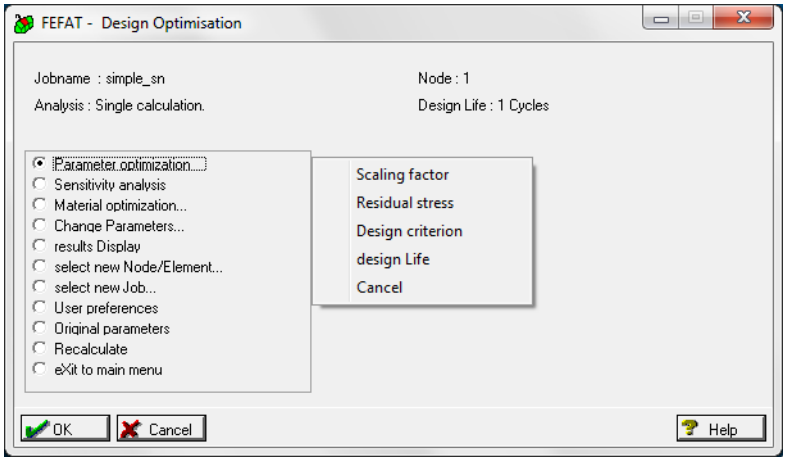


Figure 6-15 Parameter Optimization Submenu

Option	Description
Scaling Factor	This factor can be thought of as a multiplier of the combined superimposed load input or of the local stresses or strains. Errors in the FE mesh could be investigated using this feature. This factor has also been provided to allow for additional stress concentrations not modeled in the finite element analysis. The value should be the elastic stress concentration factor for the geometric detail.
Design Criterion	This is the confidence of survival parameter which is based on the standard error of the S-N or ε-N curves. Using this parameter will tell how much confidence the user can have in the product reaching the target life. However, the user should also consider the error in other parameters such as the stress computed in the FE analysis which may cause the life to be different from the estimate.
Residual Stress	Localized residual stresses may have a significant effect on life either detrimental or advantageous. Use this option to estimate the required residual stress to achieve a given life. Note that compressive stresses are usually beneficial.
Design Life	This is not an optimization parameter but is used as a target for the optimization process. The design life may be changed or defined using this option. If a design life has not been indicated initially, the user will be prompted for one before being able to take advantage of any of the above back calculation options.

Having set up one of the optimization calculations, it is necessary to unset it in order to carry out any other kind of analysis. The easiest way to do this is to select the *Original parameters* option from the main menu.

The other way is to select Change parameters followed by the parameter that was last set to back. The original default value will be offered and if accepted, the back calculation facility will be turned off. See also the section under User Preferences, 349 on Back calculation accuracy.

Sensitivity analysis

A sensitivity analysis allows the effect of variation in any of the input parameters on fatigue life to be explored, see Figure 6-16. For some parameters, all possible values are used in the sensitivity analysis. Parameters which fall into this category are:

- surface finish
- surface treatment
- mean stress correction method

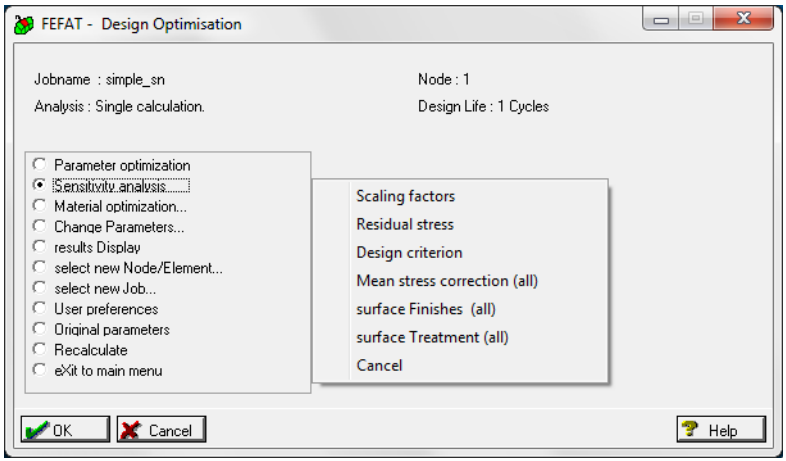


Figure 6-16 Sensitivity Analysis Submenu

To use one of these types of analysis, simply select the option followed by Recalculate on the main menu. For other parameters which are specified in a numerical form, the user is requested to enter a range of values for the chosen parameter. Parameters which fall into this category are:

- scaling factor
- residual stress
- design criterion

To select one of these types of analysis, simply select the option which will then present the user with a data input form at the bottom of the screen. In the box on this form, the user will be asked to provide a range of numbers for the parameter. Having done this, it is necessary to select the Recalculate option on the main menu

Option	Description
Scaling Factors	This factor can be thought of as a multiplier of the combined superimposed load input or of the local stresses or strains. The user may enter a single value in the data input form that appears at the bottom of the screen or a range of value separated by spaces or commas. The user may also specify a range of values by inputting a start value, end value, and increment, i.e., (1,10,2). A sensitivity plot can be created from this calculation. See Results Display, 344 for more detail. This factor has also been provided to allow for additional stress concentrations not modeled in the finite element analysis. The user may enter a single value in the input bar that appears at the bottom of the screen or a range of value separated by spaces or commas. The user may also specify a range of values by inputting a start value, end value, and increment, i.e., (1,10,2). A sensitivity plot can be created from this calculation.
Design Criteria	This is the confidence of survival parameter which is based on the standard error of the S-N or ε-N curve. Using this parameter will tell the user how much confidence he can have in the product reaching the target life. However, the user should also consider the error in other parameters such as scaling factor which may cause the life to be different from your estimate. The user may enter a single value in the input bar that appears at the bottom of the screen or a range of value separated by spaces or commas. The user may also specify a range of values by inputting a start value, end value, and increment, i.e., (1,10,2). A sensitivity plot can be created from this calculation. See Results Display, 344 for more detail.
Residual Stresses	Localized residual stresses may have a significant effect on life either detrimental or advantageous. The user may enter a single value or a range of value separated by spaces or commas. The user may also specify a range of values by inputting a start value, end value, and increment, i.e., (-200,200,50). A sensitivity plot can be created from this calculation. See Results Display, 344 for more detail.
Mean Stress Correction (all)	All mean stress correction methods are automatically calculated and a table of corresponding lives given when the *Recalculate* option is invoked.
Surface Finishes (all)	All surface finishes are automatically calculated and a table of corresponding lives given when the *Recalculate* option is invoked.
Surface Treatment (all)	All surface treatments are automatically calculated and a table of corresponding lives given when the *Recalculate* option is invoked. The changes in material properties are not modeled here but are available from the material optimization form.

Material Optimization

The material optimization allows for changing to a different material, editing the parameters associated with the current material dataset, and searching for a better or worse material. These tools facilitate the optimization of the materials selection in terms of fatigue performance. For example, a new material may be

found that offers the same fatigue performance but has a lower raw material cost and is easier to work with in the manufacturing process. Alternatively, a possible fatigue failure could be designed out of the product by switching material and these tools would give a selection of alternative materials based on a fatigue selection criterion. Figure 6-17 shows the Material optimization form.

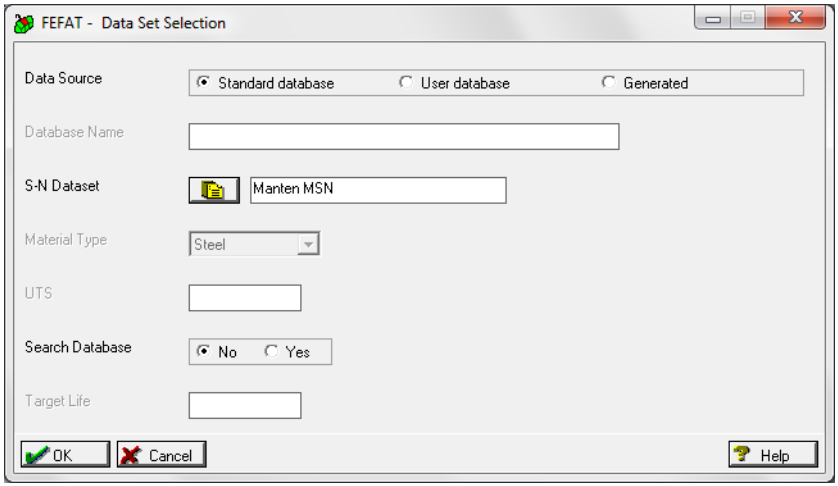


Figure 6-17 Material Optimization Form

The fields on this screen are defined below.

Options	Description
Data Source	There are three sources of materials data in all MSC Fatigue analyzers. They are: The Standard Database, which can be the central database or a user specific local database (which is usually a modified copy of the standard database). A user database which contains data in the format of the standard database but which is specific to the user, i.e. a custom database. Generated - which are generated from the UTS - the results of this type of calculation are an approximation, they should NOT be used in a final sign off.
Database Name	This field becomes live if User Database is selected. The user database is created using the tools in PFMAT documented in Material Management (Ch. 4).
Material Name	This field becomes live if Standard or User Database is selected. All materials currently available can be viewed using the List button.

Options	Description
Material Type	This field becomes live if 'Generated' is selected. Options available are steel, aluminium, titanium, and 'Other'. If Other is selected, then a Young's modulus and Area Reduction must be supplied in addition to the UTS.
UTS	See Material Type above.
Young's Modulus	See Material Type above.
Area Reduction (%)	See Material Type above.
Search Database	If search database is set to Yes then a range of materials will be evaluated and the 10 best will be listed in a pick list. One of the ten should be selected for further consideration.
Target life	A target life is required so that the search database option can use it as a benchmark against which it can compare the relative performances of all the materials in the chosen database.

When the material choice has been optimized you the user will be returned to the main Design Optimization menu.

In the case of an analysis of a B57608 steel or aluminum weld, the material optimization route would follow that described in Material Management (Ch. 4).

Change Parameters

This design optimization option allows for changing individual parameters or to reset individual parameters back to their original values. The Change Parameters form is shown in Figure 6-18.

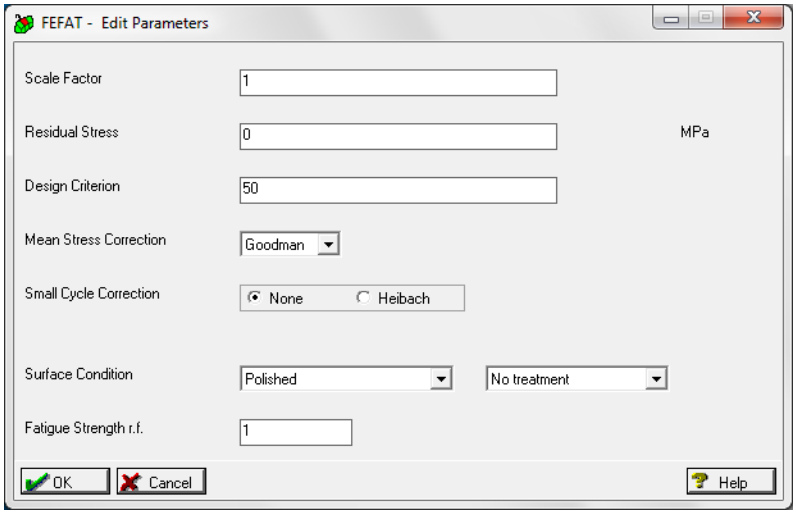


Figure 6-18 Change Parameters Form

The fields on this screen are defined below.

Option	Description
Scale Factor	This factor can be thought of as a multiplier of the combined superimposed load input or of the local stresses or strains. You can accept the default to reset to the original value or you can supply a single scale factor. Up to 30 values can be entered.
Residual Stress	Localized residual stresses may have a significant effect on life either detrimental or advantageous. Use this option to estimate the required residual stress to achieve a given life. Note that compressive stresses are usually beneficial. The user can accept the default to reset to the original value or he can supply a single residual stress value.Up to 30 multiple values can be entered. Negative values shift the mean range down, positive values shift it up.
Design Criterion	The % certainty of survival is a statistical parameter between 0.1 and 99.9% which is based on the standard error of the S-N or ε-N curve. Using this parameter will tell you how much confidence you can have in the product reaching the target life. A low confidence is associated with long lives whereas the probability of reaching a short life is high. However, you should also consider the error in other parameters such as scaling factor which may cause the life to be different from your estimate. You can accept the default to reset to the original value or you can supply a single design criterion.

Option	Description
Mean Stress Correction	This field will have different values depending on what the analysis type of the job is. For a crack initiation analysis, the Smith-Watson-Topper and Morrow methods are used when the local strain (low cycle fatigue) approach is used to predict lives. They take into account the effect of non-zero mean stresses on the strain life curve. These effects are fully discussed in the technical overview, but in brief it is true to say that in mainly tensile mean stresses the SWT approach is more conservative and in mainly compressive situations the Morrow approach is the more conservative. If an S-N job is being processed then the user must decide between the Goodman and the Gerber method for calculating the mean stress correction. It is advisable to check all methods before selecting the final basis for a life estimate.
Small Cycle Correction	This field only appears when a S-N job is being processed. The Heibach correction accounts for the fact that small fatigue cycles which lie at a level below the endurance limit do become damaging if they follow a large cycle. If the largest cycle is above the endurance limit of the S-N curve then all cycles will be damaging because the S-N curve is changed such that the slope beyond the endurance limit cut-off is greater than zero.
Elastic-Plastic Corr.	This field only appears when a crack imitation job is being processed. The correction factor selected allows FEFAT to convert the elastic stresses and strains from their FEA values to true stresses and strains for use in crack initiation modelling. By default, the Neuber correction is used but the Mertens-Dittmann and Seeger-Beste modifications to the Neuber method give better, i.e. more conservative, results for unnotched geometries, and those where the plasticity is not highly localized. The Seeger-Beste method gives the most conservative results, but both it and Mertens-Dittmann require a shape factor which is shape and loading dependant.

Option	Description
Shape Factor	This field only appears when a crack imitation job is being processed. This factor is required by the Mertens-Dittmann and Seeger-Beste plasticity correction methods. It is an elastic strain concentration or shape factor which is a function of the shape of the cross section of the component and the type of loading.
Biaxiality Correction	This is the correction method for proportional biaxiality and is only available when processing a crack imitation job. The methods available are the Hoffmann-Seeger and Parameter correction. They should only be used where shell element results or surface resolved solid nodal results are available (see Technical Overview). If No correction is selected then FEFAT will use the Neuber method and the chosen strain combination on the uniaxial cyclic stress-strain curve. The Hoffmann-Seeger method uses the biaxiality ratio to convert the combined strain parameter to an equivalent strain (based on the von Mises strain) before carrying out the Neuber correction and then recalculating the elastic-plastic stresses and strains. It is applicable to the absolute maximum principal and signed Tresca strain conditions. The Parameter Modification method works by calculating on the basis of the mean biaxiality ratio a new cyclic stress-strain curve for each node or element. The new curve relates absolute maximum principal stress and strain amplitudes and should only be used with this strain combination (set during preprocessing).
Surface Condition (finish and treatment)	You may choose to change this surface finish or reset it back to its original value by accepting the default. The user will be presented with a submenu with the list of choices. These are: *Polished*, *Ground*, *Good Machined*, *Average Machined*, *Poor Machined*, *Hot Rolled*, *Forged*, *Cast*, *Water corroded*, *Seawater corroded, User-defined.* You may choose to change the surface treatment or reset it back to its original value by accepting the default. The user will be presented with a submenu with the list of choices. These are: *No treatment*, *Nitrided*, *Cold rolled*, *Shot peened.*

Option	Description
Fatigue Strength r.f.	Since not all stress raisers may be modeled correctly in the FE analysis the effect of additional stress concentration factors on fatigue life may be modeled. Normally this is modified to a fatigue strength reduction factor Kf. Alternatively, an elastic stress concentration factor Kt can be entered and are available from standard reference texts such as Peterson's book. Kf factors must be in the range 1 to 100 inclusive.

Results Display

The presentation of the results in both tabular and graphical form is handled from this menu. The options available are shown in Figure 6-19 and discussed below:

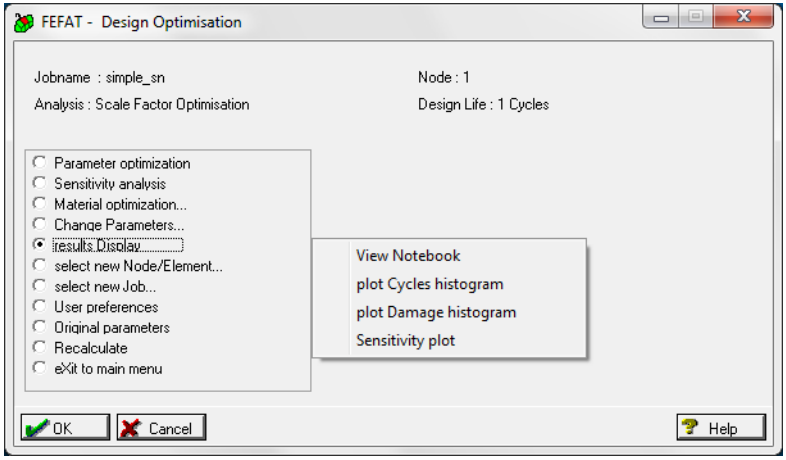


Figure 6-19 Results Display Submenu

Option	Description
View Notebook	Allows the review of the results of all analyses written to the Notebook (including the latest analysis if Notebook is set to On). To view the Notebook FEFAT uses whichever text processor has been nominated, e.g. vi on a Linux platform.
Plot Cycles Histogram	Plots the 3-dimensional rainflow cycle counted histogram after scaling to the local stresses at the node or element being analyzed. A description of the graphical histogram display is given in Matrix Options, 356.
Plot Damage Histogram	Plots the 3-dimensional damage histogram which is related to the rainflow cycle counted histogram at the node or element being analyzed. The units of damage for the histogram is user-configured and is set using the damage histogram units option in the User Interface submenu. A description of the graphical histogram display is given in Matrix Options, 356.
Sensitivity Plot	Displays an x-y sensitivity plot using the MTPD module when one of the options under *Sensitivity analysis* from the main selection screen is chosen. This plotting option is only accessible immediately after running a sensitivity analysis. Various files are created which allow this plot to also be created under the Results button of the main MSC Fatigue form. An example of this type of display is shown in Figure 6-20. Also see Sensitivity Plots, 107.

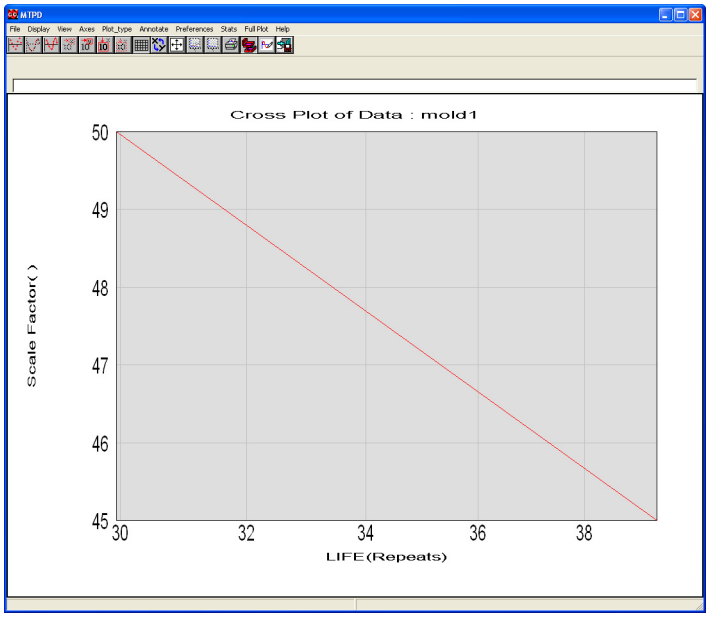


Figure 6-20 Sensitivity Plot and Menu Option Bar

Many of the options available from the top pull-down menus are generic to the MSC Fatigue modules and are described fully in Module Operations (Ch. 18) along with the commands that are applicable in the Command databox. Those specific to this display in the MTPD graphical module are described here.

Field	Description
DISPLAY	
Join / Points	Displays the plot as either a continuous line by joining the points together or displays only the data points.
Join Points	Displays both the lines joining the data points and the data points themselves.
VIEW	
Full Plot	Displays the entire signal within the visible window.
Full X / Full Y	Displays the entire X-axis or Y-axis of the signal within the current window.
Page Left / Page Right	Pages left or right one window of the signal.
Page Up / Page Down	Pages up or down one window of the signal. The signal must be transposed for this option to be enabled.
Zoom In / Out	Zooms in or out (away) from the plot 5 times. Once the plot is fully displayed it will not zoom out any farther.

Field	Description
X Window / Y Window	Requests a minimum X- or Y-axis value and a maximum X- or Y-axis value in the Command databox from which the signal is then brought to fit into the current window.
Transpose	Transposes the X- and Y-axes of the plot.
AXES	
Lox X / Log Y	Converts the X- or Y-axes to Log scale.
Linear X / Linear Y	Converts the X- or Y-axes to Linear scale.
dB Y	Scales the Y-axis to dB (decibels).
Grid / No Grid	Turns the plot grid on or off.
Dash Grid	Turns the grid on and dashes the grid lines. To undash the grid lines, set the grids off and back on again.
Box / No Box	Turns the box around the plot on or off.
Zeros On / Zeros Off	Removes or plots the line defining the X and Y zero locations.

Field	Description
PLOT TYPE	
Show Set	Displays data for a set which is not already displayed.
Hide Set	Hides a currently displayed data set.
Shapes On / Shapes Off	Shows the data points as crosses rather than shapes.
Point Skip	Allows the display to be plotted for every nth data point. For example if every other data point is to be plotted use 2.
ANNOTATE	
Set Title / Delete Title	Allows for setting a title or deleting a title from the plot. The title must be input through the Command databox.
Add Text / Delete Text	Allows for adding or removing additional text or titles on the plot. The text is input through the databox and automatically placed at a predefined location on the plot. To delete the text, click on it with the mouse after selecting the Delete Text option. Confirmation of the text will be requested.
Move Text	To place added text or titles use this option. First select the text with the cursor and then use the cursor to place the text in the new location.
Top Label / Side Label	Moves the label of the Y-axis to the top or side.
MISCELLANEOUS	
P	If the P key is pressed at any time, the current option is terminated and the whole screen is redrawn.
W	If the W key is pressed at any time with the cursor over a menu option, help is displayed for that option.
V	If the V key is pressed at any time with the cursor over a menu option, that menu option is invoked.

Select New Node/Element

Normally, design optimization will be carried out on the node or element which has the shortest life based on the assumption that the lives at all other nodes and elements will show at least the same change in life as the critical node/element. However, the lives at other nodes or elements will need checking especially where the surface parameters or additional local effects such as mean stress or stress concentration are different from those at the critical node or element. When this option is selected, a new node or element entry screen is presented with the same select options used on the main input screen such as already shown in . Having selected a new node or element, the user will be returned to the Design Optimization Analysis menu.

Select New Job

This option returns the user to the first input screen where the jobname is requested (see Figure 6-12). The current jobname is presented as a default.

User Preferences

The preferences that may be set here are generally items which are not normally changed for every analysis (i.e., they are not job specific). Each parameter is described in Figure 6-21.

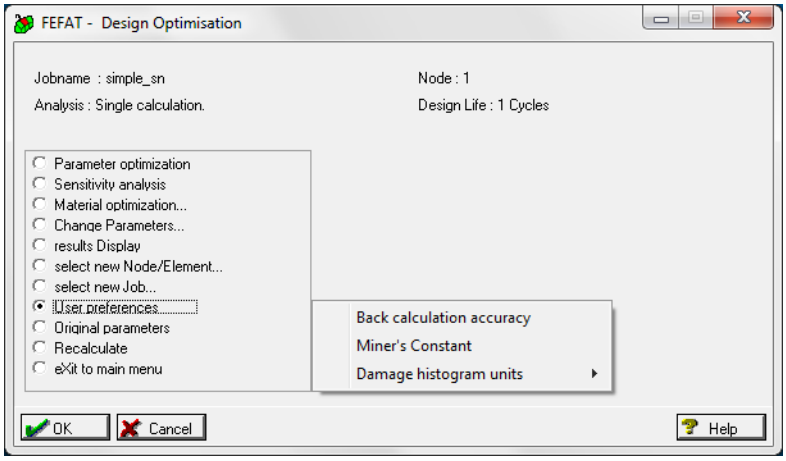


Figure 6-21 User Preferences Submenu

Option	Description
Back Calculation Accuracy	The normal convergence accuracy for the back calculation is 5% (i.e., the iteration will stop once the life is within 5% of the target or design life). Note: Higher accuracy will take longer for the calculation to converge.
Miner's Constant	This constant is normally set to a value of 1.0. Some situations may call for it to be set to a different value, usually less than 1.0 for more conservative life predictions. Caution: Once this value has been set, it will be used in all fatigue calculations carried out in the current directory including global multilocation jobs.
Damage Histogram Units	There are three types of scaling available for the z-axis of the damage histogram. The Normalized scaling is where the sum of all the damage from all the bins is equal to 1.0. Percentage damage is where the normalized damage is multiplied in each bin by 100. Uncalibrated is the basic Miner damage fraction representation for each bin.

Original Parameters

If at any stage in the design optimization, the user wants to recall the original analysis parameters as defined in the global analysis, then this option will do this. This facility is particularly useful for turning off a previously defined back or sensitivity analysis setup. If the user only wants to reset certain parameters, then he should use the Change Parameters main menu pick.

Recalculate

Once the new analysis parameters have been defined, it is necessary to pick this option to start the re-analysis. Once this option has been selected, a message will appear to tell the user that the analysis parameters are being written to the `pfatigue.prt` file. A "fatigue analysis xx complete" message is used to report the stage of the analysis, where "xx" is a number between 0 and 100.

Exit to Main Menu

Picking this option causes FEFAT to return to the main menu, saving the analysis results summaries in the `pfatigue.prt` file.

Assess Multiaxiality

This option allows for a multiaxial assessment of the stress state in your FE model subject to the service loadings. Various parameters can be investigated to give a good understanding of the stress state, whether it be in a uniaxial, proportional or non-proportional loading situation. Much discussion is given in Crack Initiation Solution Parameters, 54 and Multiaxial Fatigue (Ch. 7), particularly Multiaxial Fatigue Theory, 481 as to the usefulness of these parameters.

Briefly these parameters are the biaxiality ratio, a_e which is defined as σ_2/σ_1 where σ_1 is the largest in-plane absolute magnitude principal stress and σ_2 is the other in-plane principal of a state of plane stress on the surface of the model, the out-of-plane principal being zero, and ϕ, being the angle that σ_1 makes with the local x-axis. These parameters tell us the following:

1. If a_e is zero and ϕ is constant a state of uniaxial loading exists. Normal fatigue theories apply to this situation.
2. If a_e is non-zero but constant and ϕ is constant a state of proportional loading exists. Corrections to the uniaxial case need to be made based on a_e.
3. If a_e is variable and ϕ is variable a state of non-proportional loading exists. The techniques in Multiaxial Fatigue (Ch. 7) need to be resorted to in order to perform the fatigue analysis.

Before resorting to a multiaxial fatigue analysis, an assessment should be made first. By selecting this option you are presented with this form.

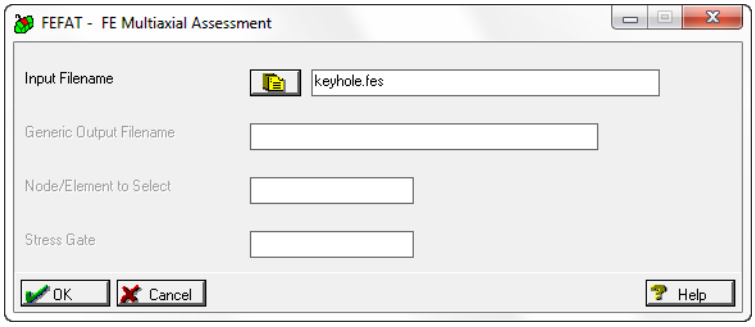


Figure 6-22 Multiaxial Input Form

The fields are:

Option	Description
Input File Name	This is a jobname.fes file created with the PAT3FAT or FATTRANS translator.
Generic Output File Name	Any files created will have this name in front of any extension.
Node/ Element to Select	This is a node or element number on which to do the assessment. Only one may be entered. You may list all available entities by pressing the List button next to this field.
Stress Gate	Entering a stress gate will ignore any cycles with stress ranges below this gate. This speeds up the analysis and removes spurious results due to low stress levels.

A summary screen will be presented once the assessment is complete which lists the following information:

- Input/Output File name(s)
- Stress Gate Used
- Biaxiality Average (over the entire loading history)
- Most Popular Angle (in degrees)
- Angle Spread (in degrees)
- Maximum Stress Range

Once you have accepted the summary page you are presented with a menu for various plotting options.

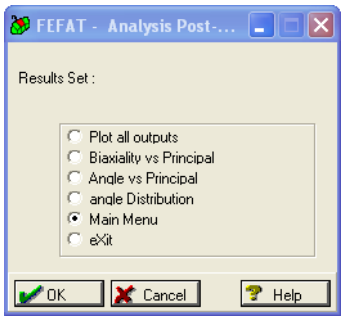


Figure 6-23 Multiaxial Assessment Plotting Options

The relevance of these plots is:

Option	Description
Plot all Outputs	This is a multi-display using MMFD showing the time variation of all the multiaxial assessment parameters: Maximum Principal Stress, Minimum Principal Stress, Absolute Maximum Principal Stress, Signed von Mises Stress, Signed Maximum Shear Stress, Angle to X-axis, Biaxiality Ratio.
Biaxiality vs. Principal	This is a cross plot using MMFD of the biaxiality ratio, a_e, against the Maximum Absolute Principal Stress. This will either show a scatter (indication of non-proportional loading) or for higher stress values a lining up around zero (uniaxial) or some non-zero biaxiality ratio (proportional loading). See Figure 7-50 through Figure 7-53.
Angle vs. Principal	This is also a cross plot using MMFD but of the angle, ϕ, against the Maximum Absolute Principal Stress. Lining up at a particular angle indicates uniaxial loading if a_e is zero or proportional loading if a_e in not zero. Again see Figure 7-50 through Figure 7-53.
Angle Distribution	This plots up a distribution of the angle vs. the number of times encountered throughout the time series. A spike around a certain angle (with all else approaching zero) will indicate stationarity (proportional and possibly uniaxial loading).

Output Time Histories

Output Time history provides an important link between test and analysis in the integrated durability management of which MSC Fatigue is a core tool.

It allows the fully pre-processed time history at a node or element to be exported and written to a standard time history `.dac` format. One or more nodes can be processed in a single run of this option. The local stress or strain histogram is produced by cycle counting the time history `.dac` file.

The time history `.dac` file is extremely useful to the understanding of local stress or strain response when working with strain measurements taken from a fatigue test component. These combined time history responses can be used in subsequent, more detailed, single location analyses such as those provide by Advanced Fatigue Utilities, 1083.

The option automatically plots *single* time histories graphically. For multiple time history creations use the 'Graphically display a time history' option on the main menu.

Time History Creation consists of a form on which you specify the input `jobname.fes` file and output `.dac` file names, and optionally edit the load cases that the new `.dac` file creation is based upon. See Figure 6-24.

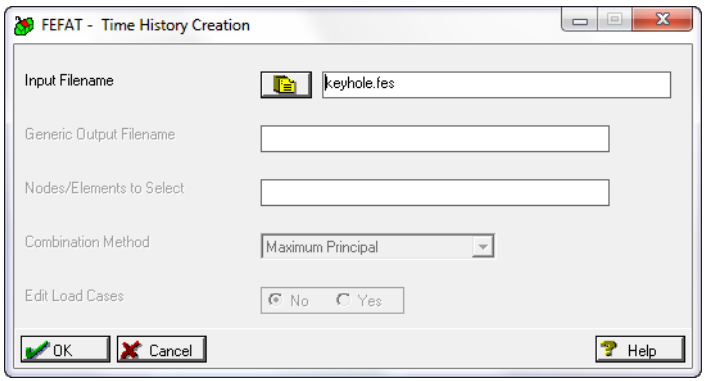


Figure 6-24 Time History Creation Form

The fields are as follows:

Field	Description
Input File Name	The name of a binary MSC Fatigue input file (`.fes`) is needed here. Such a file is normally produced from PAT3FAT (or FATTRANS) or by converting an ASCII file to `jobname.fes` format using Utilities/Binary create from the FEFAT main menu.
Generic Output File Name	The time histories that will be created for each node or element will have a file name which is comprised of a generic root, taken from the input file name, and a node or element id number. A `.dac` extension will be automatically appended to each output.
Nodes/Elements to select	Time histories will be created at a user-defined number and designation of nodes or elements. Up to 100 nodes can be processed. Ranges can be entered in the normal way but the word ALL will not be accepted because ALL could exceed the limit of 100 nodes/elements.
Combination Method	Since the stresses and strains at a node or element are stored in component form, in order to extract a time history the stresses need to be resolved to a particular plane or tensor parameter. Select the stress or strain parameter most appropriate to the job. The default is absolute maximum principal since this is the most commonly used in fatigue. See the Technical Overview for more information about the various combination methods.
Edit Load Cases	The load case information consists of the: load case number time history name applied load scale factor offset All can be edited except for the load case number. The edit facility allows sensitivity analyses to be carried out.

If Edit Load Cases is set to Yes the edit form will appear. See Figure 6-25.

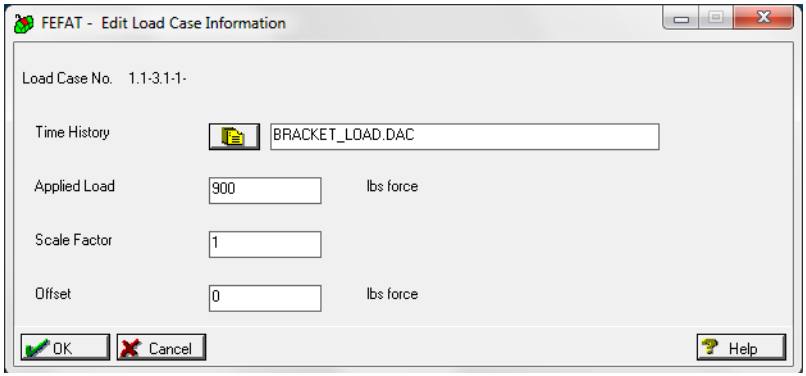


Figure 6-25 Edit Load Case Information Form

Various operations can be carried out on the time history. They are applied according to the following formula:

[(Time history*scale factor) +offset]/Applied load

Its fields are as follows:

Field	Description
Time History	The name of the time history associated with the load case is entered here, but you may enter the name of a different time history: However, the new file *must* have the same sample rate and number of points.
Applied Load	The load history applied to the FEA will be used to scale the time history according to the above equation. To ensure engineering integrity the applied load must have the same units as the time history.
Scale Factor	The scale factor is a multiplier of all the values in the time history. Scale factors can be less than 1 or negative.
Offset	The offset is applied to the time history according to the above equation. To ensure engineering integrity the applied load must have the same units as the time history.

When the time histories have been created a page of results will be displayed. The results will be maximum and minimum stresses per node/element.

For details about time history plotting and plot manipulation see the next section.

Graphical Display of Time Histories

Once a time history has been created from the preprocessed data or via the Output Time Histories option or created by any other means within PTIME, the time history can be graphically displayed. The MQLD module is used to display these plots. The explanation of time history graphical displays is identical to that already discussed in Plot an Entry Option, 245.

Matrix Options

This option enables the user to extract and view a rainflow matrix for a particular node or element using both graphical and text based tools. The display option also allows cycle (`jobnamenn.cyh`) or damage (`jobname.dhh`) matrix to be plotted.

See Figure 6-26 for the matrix options form.

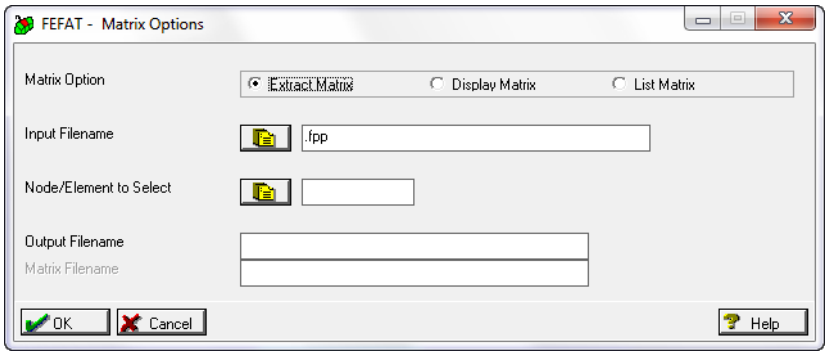


Figure 6-26 Matric Options Form

The fields are as follows:

Field	Description
Matrix Option	Extract Matrix will write a file containing a single rainflow matrix at a single node or element. The extracted file will be automatically plotted. Display matrix will plot a rainflow matrix in 3-dimensional form. List matrix will list the numbers in a tabular (text based) format.
Input File Name	Matrices are stored in a compact format (with a `.fpp` extension) and must be extracted and converted to displayable form before the Display Matrix option can be used.

Field	Description
Node/Element to select	Any individual node or elements number can be specified. **Note:** ALL is not valid and if a range is entered only the first member of the range is displayed.
Output File Name	The matrix extracted from the input file named above will be stored in binary format with a `.cyh` extension. The default name of the output file will comprise the same root name as the input file plus the node/element number,(e.g., `jobname134.cyh` is job1 node 34).
Matrix File Name	This field is enabled if an existing matrix is to be displayed, i.e. if Matrix Option= Display Matrix. Both cycles and damage matrices can be displayed; they have `.cyh` and `.dhh` extensions respectively.

Plotting, or listing a matrix are further described below (extraction is dealt with on the form above).

Cycle/Damage Histogram Display (MP3D)

When the user picks the **plot Cycles histogram** or the **plot Damage histogram** options from the **results Display** main menu pick, he is presented with a 2 and 3D graphical histogram presentation tool - the MP3D module. Both cycle and damage histograms produced by the design optimization fatigue analyzers may be viewed. The histograms are presented graphically and may be manipulated in a number of ways.

The functions of the plot pull-down menus are described in the table following Figure 6-27. Many of the options available from the top pull-down menus are generic to the MSC Fatigue modules and are described fully in Module Operations (Ch. 18) along with the commands that are applicable in the Command databox. Those specific to the form displayed in Figure 6-27 are described below.

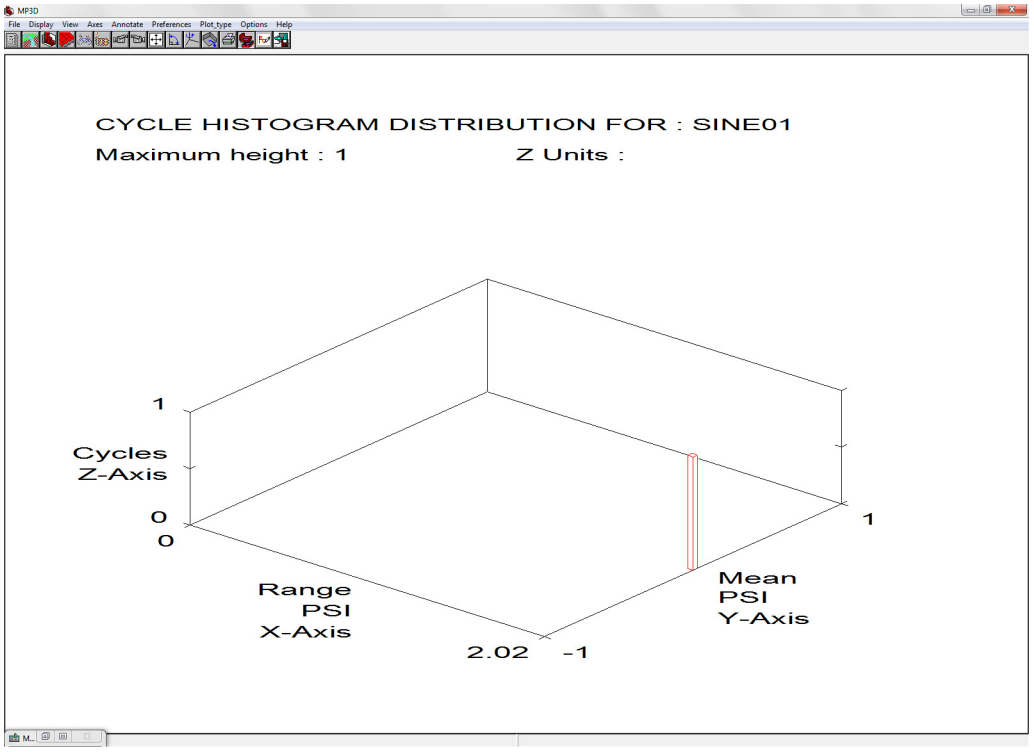


Figure 6-27 Histogram Display and Menu Option Bar

Field	Description
DISPLAY	
Hide On / Hide Off	This option turns the hidden line feature on and off. If on, histogram lines that are hidden from view will not be drawn, If off, the histogram will appear "see-thru" and all lines will be drawn.
Replot	This replots the histogram.
VIEW	
X/Y/Z window	This option allows for specification of a minimum and maximum value for the x, y, or z-axis thus providing a x, y, or z-axis zoom. When chosen, the user will be prompted in the box at the top of the screen to enter the new maximum and minimum values in the units which are relevant to the current histogram display.
Quadrant	Determines the quadrant (1-4) of the plot in which to view the histogram.
Rotate	The histogram may be rotated using this option through 20 to 50 degrees. When chosen, the user will be prompted in the box at the top of the screen to define the number of quadrants through which to turn the histogram.

Field	Description
Tilt	The angle of rotation of the histogram may be changed using this option. When chosen, the user will be prompted in the box at the top of the screen to enter a new angle tilt. Angles should be specified in degrees and are normally less than 90 degrees.
Full Plot	Attempts to display the entire plot, cancelling any windowing or scaling.
AXES	
BPlane On / BPlane Off	This option allows the user to turn on and off the back plane (z border). When turned off, the histogram will appear only on an X-Y surface.
Log Z / Linear Z	The Z scale may be toggled between a normal and logarithmic (base 10) scale. The logarithmic scale is helpful in determining the magnitude of the short towers.
ANNOTATE	
Add Text / Delete Text	Allows for adding or removing additional text or titles on the plot. The text is input through the databox and automatically placed at a predefined location on the plot. To delete the text, click on it with the mouse after selecting the Delete Text option. Confirmation of the text will be requested.
Move Text	To place added text or titles use this option. First select the text with the cursor and then use the cursor to place the text in the new location.
PLOT TYPE	
Surface	The surface option causes the histogram to be presented as a continuous surface using a grid to show the contours. This kind of display is also known as a carpet plot. The converse to surface is histogram (see below).
Histogram	The histogram option causes the histogram to be presented as a set of towers, where each tower corresponds to one location on the x-y grid and the tower height to the number of cycles or amount of damage. (A cycle is one fully reversed event as identified by the cycle counting algorithm. The latter is used to reduce the local random stress or strain time history to a set of discrete cycles).
View left	This option reduces the 3D histogram to a two dimensional view by summing the histogram as viewed from the left, looking right. When the histogram is set to the default histogram orientation, this option will show strain RANGE along the x-axis. The y-axis in this 2D view will correspond to the z-axis for the 3D view.
View right	This option reduces the 3D histogram to a two dimensional view by summing the histogram as viewed from the right, looking left. When the histogram is set to the default histogram orientation, this option will show MEAN strain along the x-axis. The y-axis in this 2D view will correspond to the z-axis for the 3D view.
Cycles / Damage	This option toggles the display between a cycle histogram and its corresponding damage histogram. The histograms show cycles or damage on the z-axis and applied strain or stress range and mean on the x and y axes respectively.

Field	Description
OPTIONS	This options brings up a form where all items explained in this table may be set simultaneously.
MISCELLANEOUS	
P	If the P key is pressed at any time, the current option is terminated and the whole screen is redrawn.

List Matrix

As can be seen from, this option lists the values of all the range and mean bins, and the number of cycles in each bin Figure 6-28.

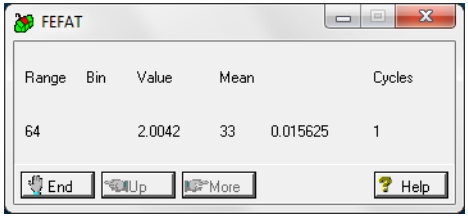


Figure 6-28 Histogram Matrix Listing

Results Processing

Once a fatigue analysis has been completed, the results may be viewed tabularly using a module called PFPOST.

Detailed descriptions of the operation of PFPOST are given in Reviewing Results (PFPOST), 363.

Utilities

This option consists of 5 utilities for converting `.fes` fatigue input files between various formats. Once the program has started, the user will be presented with a menu of options as described in Figure 6-29.

The options menu looks like this:

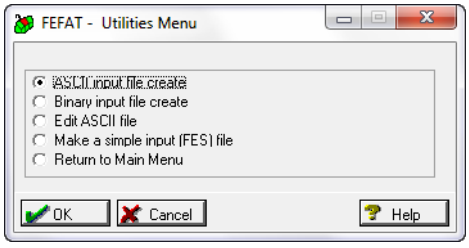


Figure 6-29 Utilities Menu Options

An explanation of each follows:

Field	Description
ASCII input file create	This is a `jobname.fes` file to ASCII file conversion utility. MSC Fatigue uses a compact binary input data file which has a file type of `.fes`. An ASCII version of a `jobname.fes` file must be created in order to view or modify the input data. Conversion back to `.fes` format is via the Binary input file create utility. The form of ASCII input file create is show below.
Binary input file create	This is a `jobname.asc` file to `fes` file conversion utility. MSC Fatigue uses a compact binary input data file which has a file type of `.fes`. A `jobname.fes` version of an ASCII file must be created in order to process the ASCII file within MSC Fatigue. Conversion back to ASCII format is via the ASCII input file create utility. The form of ASCII input file create is shown below.
Edit ASCII file	This option allows the use of an ASCII editor to make changes to the current FES file. The program converts the FES file to a temporary ASCII equivalent, which is then edited by the user selected editor. After editing, the FES file is recreated.
Make a simple input (FES) file	This utility simply leads you through a few set up pages asking for parameters to a fatigue analysis. Stresses are input manually and not taken from any finite element generated results file(s). There is a limit of one load case and one material at one stress location. It is possible to edit this file after converting it to ASCII form if more load cases, materials and stress locations are desired.

In all cases above the user will be presented with a form asking for the appropriate input file and an output file name. An example of this file is shown below.

```
MSC Fatigue Input file - ASCII format
Job name : mask_ci
File revision=3
#
# ----------------- Job Descriptors -----------------------
#
Crack Initiation of Catcher's Mask
#
# ----------------- Job Information -----------------------
#
Fatigue code=1# Local Strain
FE results type=0# Linear Static
Stress/strain=0# Stresses
Results location=0# @Node
Results co-ordinate system=0 # Full 3D
Stress/strain combination=3 # Abs Max principal
Number of load cases=1
```

```
Number of materials=2
Number of nodes=5601
#
Stress units=0 # MPa
Certainty of survival=50%
Damage model=0 # Smith-Watson-Topper
Elastic-plastic correction=0 # Neuber
Biaxiality correction=0 # None
#
# ---- Load case information ( 8 lines per load case ) -----
#
#
#Number of load cases=1
#
#
Load case number=1
Description=1.2-Default, Static Subcase
Load type=1
Load units=2
Applied FE load=0.16
Scale factor=1
Offset=0
Time history name=BALL_HITS
#
# --------------- Material/Group Information ---------------
#
#
#Number of materials=1
#
#
Group ID number=1
Material source=0 # Local/central databank
Group description=hex_mesh.25
Database name=
Material name=MANTEN
Sub-name=
Surface finish=12
Surface treatment=1
Stress/strain raiser (multiplier)=1
Stress/strain offset=0
Fatigue limit reduction (Kf) =1
Plastic stress concentration factor (Formzahl)=0 # 0 means Neuber
#
# ------------------- Stress Information -------------------
#
Node number=1 Group ID=1 Multiplier=1 Offset=0 Temp.=0
0.300007E0 -1.57632E1 1.825162E0 -5.864792E0 1.835311E0 0.587022E0
Node number=2 Group ID=1 Multiplier=1 Offset=0 Temp.=0
0.300619E0 -1.128119E1 5.086284E0 0.821963E0 3.335781E0 1.55936E0
Node number=3 Group ID=1 Multiplier=1 Offset=0 Temp.=0
```

```
1.967203E0 -3.0951E1 3.67726E0 -7.270879E0 1.536527E0 1.258015E0
Node number=-1
```

Reviewing Results (PFPOST)

Results may be reviewed in two different ways aside from using the MSC Fatigue results menus. The results are contained in the file called `jobname.fef` (and `jobname.fos` for factor of safety results) which is written as a 6-column or 12-column Patran nodal or elemental results text file. This file can be inspected with an editor, or it can be postprocessedprocessed using the standard Patran postprocessing options once it has been read into the database. See Using MSC Fatigue - The Classic Modules (Ch. 3).

Results files can also be reviewed using the results postprocessor program, PFPOST. To execute this program, type the symbol **pfpost** at the operating system prompt. This program is described in detail in List Results, 106.

Introduction to PFPOST

When performing a crack initiation or total life analysis this option executes the external MSC Fatigue module PFPOST (a Tabular Results Postprocessor for the global multi-node/element analysis), which allows various listing options for the specified results file.

Search facilities assist in locating the region with the maximum damage and to find nodes or elements with lives less than a user-specified life.

PFPOST supports results files for both total life and crack initiation fatigue analysis. Two types of fatigue results files are supported from these analyses: those with biaxiality results and those without biaxiality results. It also supports results files from Factor of Safety analyses performed by the MSC Fatigue module FEFAT.

In addition to being selected through this List Results option, it may also be run from the operating system prompt by typing the symbol **pfpost**.

Once initiated, PFPOST will present a set of forms which may be manipulated using the keyboard and mouse. The program operates interactively in Motif or mask mode and menu selections are made according to the rules defined in Module Operations (Ch. 18).

PFPOST Module Operation

When first invoked PFPOST appears displaying two forms.



The top, small form is a generic form and allows for general program control. This is discussed in detail in Module Operations (Ch. 18) for the Motif driver. The first screen to appear along with the main pull-down menu form is a jobname selection form. Once the jobname has been chosen, another form displaying the preferences is displayed. This form is shown in Figure 6-30. The fields to this form are explained in the following table.

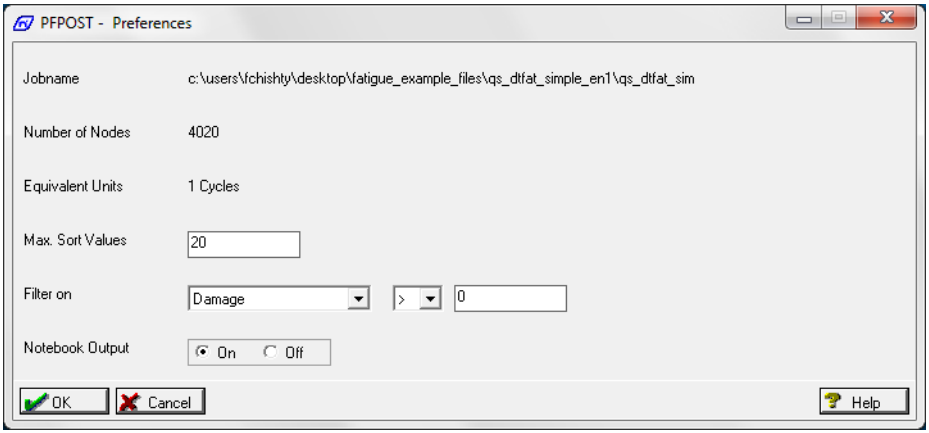


Figure 6-30 PFPOST Work File Input Screen

When the fields are filled out properly click on the **OK** button. Clicking on the *Cancel* button will exit the program at this point.

Parameter	Description
Jobname Equivalent Units Number of Nodes/Elements	These fields are displayed for information only. The equivalent units and the number of nodes/elements fields are taken from the input file and cannot be changed.
Max. Sort Values	Set this to the number of nodes or elements that you would like to be sorted based on the Filter selection. For example, if 20 is supplied, and damage is the filter, the 20 nodes with the most damage will be displayed.
Filter on	You may filter the output based on Damage, Life, the node or element number or any other result value that may have been calculated during the analysis. The filter can be an equality check or a greater than or less than criteria.
Notebook Output	If this is ON, all output will be placed in the `pfatigue.prt` file.

The next window to appear is the main PFPOST form which allows for the various tabular fatigue results displays. Normal fatigue results listing options are displayed in Figure 6-31, where as Factor of Safety results listing options are shown in Figure 6-32. Each of these options is discussed below.

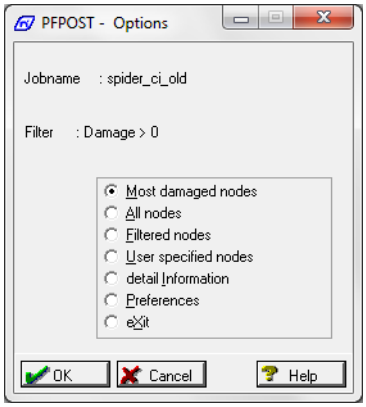


Figure 6-31 The PFPOST Results Listing Menu

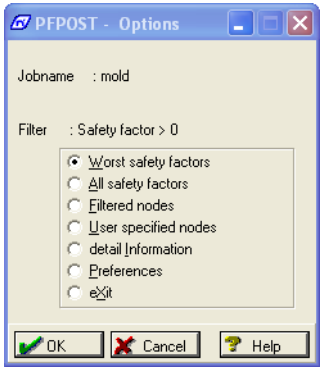


Figure 6-32 PFPOST Factor of Safety Listing Options

Most Damaged Nodes/Elements - Worst Safety Factors

This option searches the results file for the most damaged nodes or elements and presents them in ordered tabular form, similar to what is shown in Figure 6-33, where the design life was 1000 hours, and in this case, for nodes. If results are from a Factor of Safety analysis, then the worst safety factors are reported.

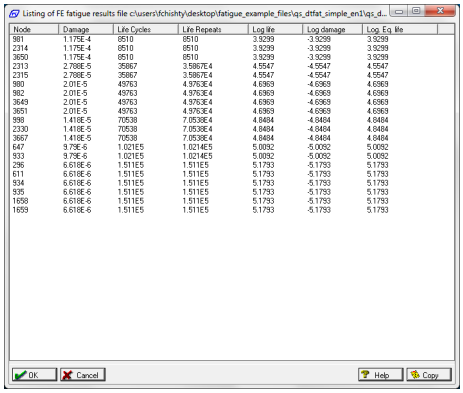


Figure 6-33 Display of Most Damaged Nodes for a Design Life of 1000 Hours

All Nodes/Elements or All Safety Factors

This option will produce a listing of the results for all nodes or elements in the order that they appear in the results file. Nodes or elements which show the words 'Beyond Cutoff' have lives greater than the fatigue limit or user-defined cutoff (i.e., a crack would not initiate or grow in this location within the bounds of the materials data used in the modelling). Stress safety factors are also reported in a like manner.

Filtered Nodes/Elements

This option lists the result values of all nodes or elements passing the filter.

User Specified Nodes/Elements

When this option is chosen, the user will be asked to supply a set of node or element numbers, separated by commas, spaces, slashes, or other separators (i.e., 1,5/8 (12]). The lives for the nodes or elements will be tabulated in the usual manner.

List Detail Information for the Current Job

This option displays the overall statistics for the data file as shown in Figure 6-34. Stress safety factor information is also reported in a form similar to this.

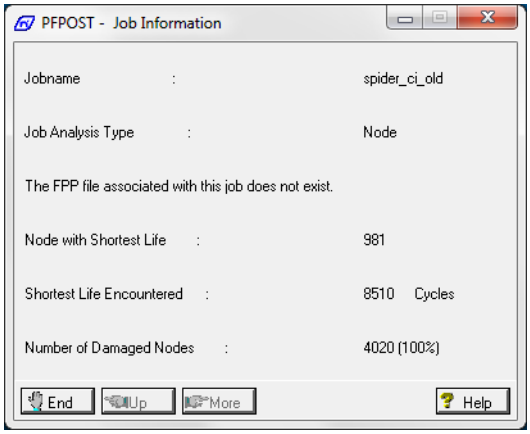


Figure 6-34 Fatigue Life Results Statistical Summary

Biaxiality Parameters

If a biaxiality analysis has been requested, then biaxiality parameters will be present in the results file. If they are not present, this option will not be presented. A listing such as that shown in Figure 6-35 appears. See Multiaxial Fatigue (Ch. 7) or the explanation on Read Results, 101 for a discussion on the meaning of these parameters.

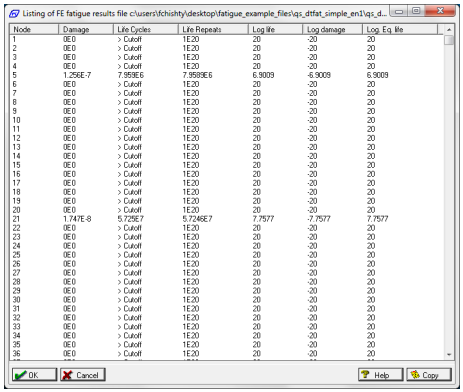


Figure 6-35 Display of Biaxiality Parameters

eXit

This option will terminate the PFPOST program.

Fast Analysis (FASTAN)

A separate utility function is delivered with MSC Fatigue called FASTAN which is a program which can dramatically speed up a full analysis. This program is generally called from the MSC Fatigue forms when performing a full analysis. This is explained in Fast Analysis, 87.

The operation of this module is as follows. The program first determines whether a single or multiple load case job is being submitted. If a single load case is being submitted, the time history file is rainflow counted and then the ensuing fatigue analysis at each location proceeds using the rainflow matrix. For a multiple load case job, the following steps are performed:

1. The original time history files are preserved for later use.
2. The original time history files are run through the peak-valley extraction program (Peak-Valley Extraction (MPVXMUL), 254) and reduced as specified by the user using a specific gate value, number of points to retain, or a specified reduction percentage. These files are retained as `opt_xx.pvx`, where xx is a 2 digit number like 01, 02, ..., 99.
3. The fatigue pre-processing (FEFAT) is performed using the reduced time histories.
4. The most damaged locations are calculated and retained in a file called `jobname.ents`. The number of damaged locations is specified by the user.
5. The pre-processing and fatigue analysis is performed again on only the damaged locations as specified by the user using the original time history signals.

When run in batch. Three parameters are required, and three methods are available.

Method 1 sets a specific gate value and is called as follows:

```
fastan /job=<jobnam> /method=1 /gate=<gate_val> /ents=<maxent>
```

Method 2 iterates until the modified file lengths after peak-valley slicing are reduced to a given value.

```
fastan /job=<jobnam> /method=2 /len=<length> /ents=<maxent>
```

Method 3 iterates until the peak sliced files have been reduced from the original file length by a given percentage.

```
fastan /job=<jobnam> /method=3 /reduct=<% reduct> /ents=<maxent>
```

<maxent> is optional, and specifies the maximum number of entities to be used in the abbreviated analysis.

Description of Files

Various files are created during a fatigue analysis. Some of these files are editable and require detailed descriptions of their formats and/or parameters. These files include:

Parameter	Description
jobname.fin	This is referred to as the job information file. It contains all of the job information necessary to set up a fatigue analysis. All information filled out in a form in MSC Fatigue or Patran for a MSC Fatigue analysis is contained in this file. This file is read by the PAT3FAT or FATTRANS translator to create the fatigue input file, `jobname.fes`.
FatigueSubmit	This is not a file that most users will need to worry too much about. It is a Linux shell script that controls the analysis when submitted directly from MSC Fatigue or Patran. It can be edited and changed to customize the submittal process and is included here for completeness. A similar file is `FatigueExecute` which is also a Linux shell script and is used when the user is submitting jobs via the Patran Analysis Manager module.
jobname.fes	This is referred to as the fatigue input file and is generally created by the PAT3FAT or FATTRANS translator. It is binary and is not an editable file, however, the format of this file is included in case someone desperately wishes to write their own.
jobname.asc	This is the ASCII version of the fatigue input file and is created using FEFAT's utility programs. It is fully editable and fully described in this section.
jobname.fpp	This is the pre-processed fatigue input file. It is binary and a description is included here in case a user desires to read this file for his own purposes.
jobname.fef/fos	This is the fatigue results file and is in a typical Patran results file format which is also described in this section. It can also be output as a Universal file format by setting the keyword **FEFTYPE** to the value **UNIVERSAL** using the MSC Fatigue environment module **MENM**. See Modifying the MSC Fatigue Environment (MENM), 1463.

The Job Information File (jobname.fin)

This file has a form as shown in Figure 6-36 where the data is defined in a parameter-driven format (i.e., PARAMETER=VALUE). The first line always must be #3.x (where x=1, 2, or 4), to distinguish the input file as a MSC Fatigue `jobname.fin` file. (Upward compatibility does not exist for versions less than 2.5.) Parameters in the `jobname.fin` file may be in any order with the exception of the ANALYSIS TYPE line,

the MATERIAL and the LOAD CASE data. All load case data must be sequential beginning with NUMBER OF LOAD CASES = as well as material information after MATERIAL # =. Comments may be inserted after the parameter and value by using the ! character.

```
#3.4
ANALYSIS TYPE = CRACK INITIATION
FEA RESULTS LOCATION = NODE
AVERAGING = GLOBAL
TITLE = MYMODEL
P3DATABASE = myjob.db
MATERIAL 1 = MANTEN
        M_DIRECTORY = CENTRAL
        FINISH = No Finish
        TREATMENT = No Treatment
        KF = 1.0
        ALPHAP = 0.0
        REGION = default_group
        MULTIPLIER = 1.0
        OFFSET = 0.0
TENSOR TYPE = STRESS
STRESS UNITS = MPA
STRESS COMBINATION = MAX ABS PRINCIPAL
ANALYSIS METHOD = SMITH-WATSON-TOPPER
DESIGN CRITERION = 50.
PLASTICITY = NEUBER
FEA ANALYSIS TYPE = STATIC
FEA RESULTS TYPE = DATABASE
DATABASE = myjob.db
TRANSFORMATION = NONE
EQUIVALENT UNITS = 0.
SHELL SURFACE = TOP
STRAIN TYPE = TENSOR
!DCMETHOD = MULTI
!DCCFGFILE = jobname.cfg
!FOSTOGGLE = FALSE
!END OF FILE
```

Figure 6-36 A Typical jobname.fin File

The valid parameters, corresponding values, and their meanings in the `jobname.fin` file are described in the next few pages.

General Setup Parameters

ANALYSIS TYPE =	This must be the first line of the file after the #3.4 line. It indicates the type of fatigue analysis to be performed. NOMINAL STRESS CRACK INITIATION CRACK GROWTH VIBRATION SPOT WELD MULTIAXIAL FOS MULTIAXIAL CI MAG WELD SEAM WELD
FEA RESULTS LOCATION =	Identifies whether stress/strain results are to be found at nodes or element centroids. Also tells MSC Fatigue to expect nodal or elemental results files if external FEA results are under consideration. Calculated fatigue results will also be either nodal or elemental depending on the value of this parameter. Valid entries are: NODE ELEMENT CENTROID NODE AND ELEMENT (used for SPOT WELD analysis only)
AVERAGING	Can be set to GLOBAL or GROUP. If set to GLOBAL all nodal stress or strain results will be averaged at the nodes for each contributing elemental node. GROUP refers to the current group With GROUP set, averaging will not take into account contributions of nodal results from elements that are not in the current group. The current group is always referred to on the title bar of the Patran graphics window.
TITLE =	This is a descriptive text string which will be passed on through the analysis and presented together with the results. It may be used to assist in the identification of the job. The maximum length is 80 characters.
P3DATABASE	Refers to the Patran database from which group information will be read and also stress or strain data.
TENSOR TYPE = RESULTS TYPE =	This identifies the type of data retrieved from the FEA analysis. Valid values are STRESS, STRAIN or EPINPUT (for CRACK INITIATION or MULTIAXIAL CI). For SPOT WELD analysis, this parameter is RESULTS TYPE = FORCE.

STRESS UNITS = FORCE UNITS =	If STRESS is chosen in the tensor type parameter, as seen above, the units of stress must be defined here. Acceptable values are: MPA(N/mm2 or MN/m2) PASCALS(N/m2) PSI KSI KG/M**2 For SPOT WELD analysis the FORCE UNITS can be: Nm Nmm lbfin kipin
STRAIN TYPE =	TENSOR or ENGINEERING. The difference is whether the shear strains have been multiplied by two (2). If TENSOR is specified then the shear strains will be scaled. If ENGINEERING is specified it is assumed that the scaling has take place already.
STRESS COMBINATION =	When strains are involved the parameter is STRAIN COMBINATION=. Selections marked with an asterisk (*) are not recommended and disabled from the MSC Fatigue setup forms. However, the `jobname.fin` file can be edited to use these combinations if necessary. MAX PRINCIPAL* MIN PRINCIPAL* MAX ABS PRINCIPAL VON MISES* TRESCA* SIGNED VON MISES SIGNED TRESCA SIGNED MAX SHEAR X NORMAL Y NORMAL Z NORMAL X-Y SHEAR Y-Z SHEAR Z-X SHEAR SURFACE
!FOSTOGGLE =	This tells you whether or not a Factor of Safety run is going to be made.

Nominal Stress-Life Specific Parameters

DESIGN CRITERION =	This is the probability of survival expressed as a percentage (valid numbers are in the range 0.1 to 99.9).
MEAN STRESS CORRECTION =	NONE GOODMAN GERBER
BIAXIALITY=	TRUE FALSE
VIBRATION METHOD =	DIRLIK NARROW BAND ALL

Crack Initiation Specific Parameters

ANALYSIS METHOD =	NONE SMITH-WATSON-TOPPER MORROW
DESIGN CRITERION =	This is the probability of survival expressed as a percentage (valid numbers are in the range 0.1 to 99.9).
PLASTICITY=	NEUBER MERTENS-DITTMAN SEEGER-BESTE
BIAXIALITY=	TRUE FALSE
METHOD BIAXIAL=	This parameter is only written to the .fin file when the BIAXIALITY PARAMETER is set to TRUE. The possible values for this parameter are: NONE MATERIAL PARAMETER HOFFMAN-SEEGER

Crack Growth Specific Parameters

CRACK LENGTH UNIT =	INCHES METERS MILLIMETERS MILLIINCHES
CRACK LENGTH INITIAL =	Initial crack length in user's units. If zero the Kitagawa minimum crack theory will then be used to determine the minimum valid initial crack size for linear elastic fracture mechanics.
CRACK LENGTH FINAL=	Final crack length in user's units. Must be larger than the initial crack length.

COMPLIANCE FUNCTION =	The name of the file containing the data defining the K-solution or compliance function. This file must have a KSN extension. It is not necessary to input the extension.
NOTCH DEPTH =	The depth of the notch in user's units. This parameter may be set to zero in which case the next two parameters are ignored in the analysis. If the initial crack size is set to zero then a non-zero value for the notch depth must be entered.
NOTCH RADIUS =	The radius of the notch in user's units.
SHARP CRACK RADIUS =	The sharp crack tip radius in user's units.

Material Parameters

S-N DATA SET n= MATERIAL n =	The names of the materials datasets are held here where n is the material number being defined. Up to 100 materials may be defined, with each material name not exceeding 32 characters in length. Only one material (MATERIAL 1=) can be specified for crack growth. Also for crack growth, the environment must follow the material name as in the following example: MATERIAL 1 = BS4360-50D:seawater CP-850mV
M_DIRECTORY=	CENTRAL LOCAL USERS
FINISH=	Polished Ground Good Machined Ave. Machined Poor Machined Hot Rolled Forged Cast Water Coreoded Seawater Corr. User defined No Finished
TREATMENT=	No Treatment Nitrided Shot Peened Cold Rolled
KF=	A concentration factor. Can be any number greater than zero. Default is one.
ALPHAP=	Shape factor for Mertens-Dittman and Seeger-Beste plasticity corrections. Any number greater than 1.0 is valid. Default is infinity which forces Neuber correction.

REGION=	The Patran group of nodes and/or elements to be used for this combination of material/surface finish and treatment.
MULTIPLIER=	A multiplier value. Any real number is valid. Default is one.
OFFSET=	An applied offset. Any real number is valid. Default is zero.
WELD=	YES NO NA Valid for component S-N analyses only.
ENVIRONMENT=	The name of the environment for crack growth jobs.
S-N TYPE =	S-N curves can be MATERIAL or COMPONENT. The difference is explained in Component vs. Material S-N Curves, 169.
SPOT WELD Analysis	The following two parameters are necessary for SPOT WELD analysis: DIAMETER and THICKNESS. Each group of spotwelds that share the same material parameters are grouped into 3's. you may have up to 100 groups of 3's. For example the first grouping will appear as: MATERIAL 1 = spot_nugget_generic M_DIRECTORY = CENTRAL DIAMETER = 1.0 THICKNESS = 1.0 MULTIPLIER = 1.0 REGION = spotwelds MATERIAL 2 = spot_sheet_generic . . . MATERIAL 3 = spot_sheet_generic . . . where the first material describes the nugget and the second two describe the top and bottom sheets. The next group will be MATERIAL 4, 5, and 6.
DIAMETER =	Diameter of the spot weld nugget for a particular material for SPOT WELD analysis. This is only important for the first material in a group.
THICKNESS =	Thickness of the sheet for a particular material for SPOT WELD analysis. This is only important for the second two materials in a group.

FEA Results Parameters

FEA ANALYSIS TYPE =	STATIC TRANSIENT TRANSFER FUNCTION PSD
FEA RESULTS TYPE =	The source of the FEA stress analysis results: DATABASE P3/FEA EXTERNAL(e.g., Patran results files) XDB SDRC UNIVERSAL XDB and SDRC files can be read when the `jobname.fin` file is created using FEFTRN only. See FEFTRN, 109.
TRANSFORMATION =	BASIC NONE
EQUIVALENT UNITS =	This is used for calculating and reporting life in the user's units.
DATABASE =	The name of the Patran database from which results will be extracted.
RESULTS JOB NAME =	This is the results jobname of a Patran FEA analysis. The name must correspond to the name of a `jobname.res` file created from a Patran FEA analysis. It is not necessary to specify the `.res` suffix. It is also the name of the XDB or SDRC Universal file.
RESULTS FILE NAME =	If the FEA results are from an external code other than the Patran database or from Patran FEA, then this parameter is used to specify the Patran nodal or elemental results file names. These files are specified by placing a # character in the place of the subcase number (or time step number for a transient analysis). For example if the results files are `sub1.nod`, `sub2.nod`, `sub3.nod`, then the parameter should be specified as: RESULTS FILE NAME= `sub#.dis`

RESULTS COLUMNS =	Used to define in which columns the various stress or strain tensors are located in the results files produced by the translator when using an external FEA code. All six component stresses or strains must be specified, separated by commas, even if one or more of these columns are zero due to the nature of the FE analysis. The columns must be specified in order (i.e., X,Y,Z,XY,YZ,ZX directional stresses/strains, respectively). The following example shows how they must be specified: RESULTS COLUMNS=1,2,3,18,19,20 where Column 1 points to X directional stress Column 2 points to Y directional stress Column 3 points to Z directional stress Column 18 points to XY directional stress Column 19 points to YZ directional stress Column 20 points to ZX directional stress
SHELL SURFACE =	If the analysis is from Patran FEA and there are stresses/strains from shell elements to be considered in the fatigue analysis, then top or bottom results must be specified. This is also true of results from the Patran database regardless of element type. Top results will always be used if this parameter is not specified. TOP BOTTOM
NUMBER OF TIME STEPS = NUMBER OF FREQUNCY STEPS =	An integer number specifying the number of time steps if the analysis is transient or the number of frequencies in a frequency response or random vibration analysis. For an external FE code, there must be the same number of results files as number of time steps specified.
SCALE FACTOR =	A scale factor that may be applied to the results of a transient analysis.
NUMBER OF LOAD CASES = NUMBER OF INPUT =	An integer number specifying the number of load cases. For an external FE code, there must be the same number of results files as number of load cases specified. For frequency response analysis this parameter is NUMBER OF INPUT = referring to the number of input PSD loads.

Load Case Parameters

The next data group is repeated for each load case. All parameters are order independent with the exception of the load case information. All load case information must follow the NUMBER OF LOAD CASES= parameter and all information for each load case must follow the LOAD CASE= parameter. Load case information is not necessary for transient analyses and will be ignored if present.

LOAD CASE = INPUT ID =	An integer between 1 and 40. Must be sequential beginning with 1 and must be repeated for every load case. For frequency response analysis this parameter is INPUT ID = and can only range from 1 to 20.
FEA LOAD CASE ID = FREQUENCY ID # = TIME STEP ID # =	An integer (or group of integers) specifying the load case ID as specified in the FE analysis or the internal IDs of the result in the database. When results are specified from the database, they are X.Y-A.B-C- where: X = Result Case ID Y = Subcase ID A = Primary Result B = Secondary Result C = Layer ID
TIME HISTORY = LOAD FILE =	This is the name of the loading time history which describes the time variation of the loading for the load case under consideration. The loading time history must be defined in the time history database. For frequency response analysis this parameter is LOAD FILE = and is the input load psd name as also defined using the time history database which is really a loading database.
TH_DIRECTORY=	The directory name where the time history files are located.
SHIFT=	This is an offset value to be applied to the load case results. Can be any valid number. Default is zero.
SCALE FACTOR=	This is a scale factor to be applied to the load case results. Can be any valid number. Default is one.
LOAD MAGNITUDE =	This is the scalar magnitude of the loading applied in the FEA analysis for this load case (or the magnitude of a related quantity). The value must be in the units defined under LOAD UNIT= parameter below.
LOAD TYPE =	The type of load. This number will correspond to the number allocated in the `ltypes.ind` file which is contained in the `ptime` central directory. When shipped, the index numbers are as follows: 0! Static 1! Force 2! Pressure 3! Temperature 4! Acceleration 6! Displacement 7! Rotational Velocity 8! Velocity 9! Uncalibrated 10! Scalar 11! Moment

LOAD UNIT =	The load unit type as defined by the loading time history database. This number will correspond to the number allocated in the `utypes.ind` file which is contained in the `PTIME` central directory. When shipped, the index numbers are as follows: 0! Newtons 1! kNewtons 2! lbs force 3! Tons force 4! Tonnes force 10! Pascals 11! MPa 12! PSI 13! KSI 14! TSI 15! kgf/m^2 20! Degrees Kelvin 21! Degrees Celsius 22! Degrees Fahrenheit 30! m/s^2 31! g 32! feet/s^2 40! m 41! mm 42! inches 43! milliinches 44! microinches 50! rps 51! rpm 52! rph 53! rad/s 54! rad/min 60! m/s 61! mm/s 62! kph 63! ips 64! mph 70! undefined 80! none 81! % 82! levels 90! Nm 91! Ft lbs 92! Nmm

When a Duty Cycle analysis is run a master FIN file shown in **Figure 5-37** is written along with a .cfg file, see **Figure 5-38**. These two files provide all the information that is needed to create the multiple .fin files that are needed for a Duty Cycle run. Note that the lines listed in the master FIN file are the same as those listed for a single analysis run. Only the Load Case information has been replaced with two lines indicating that this is a Duty Cycle run and the location of the CFG file.

```
#3.4
ANALYSIS TYPE = CRACK INITIATION
FEA RESULTS LOCATION = NODE
AVERAGING = GLOBAL
TITLE = MYMODEL
P3DATABASE = myjob.db
MATERIAL 1 = MANTEN
        M_DIRECTORY = CENTRAL
        FINISH = No Finish
        TREATMENT = No Treatment
        KF = 1.0
        ALPHAP = 0.0
        REGION = default_group
        MULTIPLIER = 1.0
        OFFSET = 0.0
TENSOR TYPE = STRESS
STRESS UNITS = MPA
STRESS COMBINATION = MAX ABS PRINCIPAL
ANALYSIS METHOD = SMITH-WATSON-TOPPER
DESIGN CRITERION = 50.
PLASTICITY = NEUBER
FEA ANALYSIS TYPE = STATIC
FEA RESULTS TYPE = DATABASE
DATABASE = myjob.db
TRANSFORMATION = NONE
EQUIVALENT UNITS = 0.
SHELL SURFACE = TOP
STRAIN TYPE = TENSOR
!DCMETHOD = MULTI
!DCCFGFILE = jobname.cfg
!FOSTOGGLE = FALSE
!END OF FILE
```

Figure 6-37 A Typical jobname.fin file for a Duty Cycle run

Duty Cycle Parameters

!DCMETHOD = MULTI	Indicator that a Duty Cycle analysis is going to be run.
!DCCFGFILE =	Specifies the directory and name of the .cfg file that will be read during the Duty Cycle run.

```
##
## MSC.Fatigue - Multiple Analysis/Load Setup file
##             for STATIC Case
##
## Version 2.0
##
##  File created on 11-Apr-05 at 12:05:16
##
TEMPLATE JOBNAME = jobname
NUMBER OF RESULTS ENTITIES PER JOB = 733
#
NUMBER_OF_SEQUENCES = 1
#
SEQUENCE_NAME_1 = seq1
NUMBER OF REPEATS OF SEQUENCE_NAME_1 = 1
NUMBER OF EVENTS = 2
LAYER SELECTION = BOTH
NUMBER OF JOBS REQUESTED = 4
#
#FIN FILES CREATED
#
jobname_seq1_evt1_z1
jobname_seq1_evt1_z2
jobname_seq1_evt2_z1
jobname_seq1_evt2_z2
#
#EVENT DETAILS
#
EVENT_NAME_1 = evt1
NUMBER OF REPEATS OF EVENT_NAME_1 = 1
NUMBER OF TIME HISTORIES = 1
##
## TIME HISTORY/RESULTS EXTRACT INFORMATION
##
## LAYER 1
##
LOAD CASE = 1
        FEA LOAD CASE ID = 1.1-3.1-2-
        LOAD TYPE = 10
        LOAD UNIT = 80
        TIME HISTORY = sine01.dac
        TH_DIRECTORY = D:\fat_test
        SHIFT = 0.0
        SCALE FACTOR = 1.0
        LOAD MAGNITUDE = 1.0
##
## LAYER 2
##
LOAD CASE = 1
        FEA LOAD CASE ID = 1.1-3.1-3-
        LOAD TYPE = 10
        LOAD UNIT = 80
        TIME HISTORY = sine01.dac
        TH_DIRECTORY = D:\fat_test
        SHIFT = 0.0
        SCALE FACTOR = 1.0
        LOAD MAGNITUDE = 1.0
```

```
END EVENT_NAME_1
#
EVENT_NAME_2 = evt2
NUMBER OF REPEATS OF EVENT_NAME_2 = 1
NUMBER OF TIME HISTORIES = 2
##
## TIME HISTORY/RESULTS EXTRACT INFORMATION
##
## LAYER 1
##
LOAD CASE = 1
        FEA LOAD CASE ID = 1.1-3.1-2-
        LOAD TYPE = 10
        LOAD UNIT = 80
        TIME HISTORY = sine01.dac
        TH_DIRECTORY = D:\fat_test
        SHIFT = 0.0
        SCALE FACTOR = 1.0
        LOAD MAGNITUDE = 1.0
LOAD CASE = 2
        FEA LOAD CASE ID = 1.1-3.1-2-
        LOAD TYPE = 10
        LOAD UNIT = 80
        TIME HISTORY = sine02.dac
        TH_DIRECTORY = D:\fat_test
        SHIFT = 0.0
        SCALE FACTOR = 1.0
        LOAD MAGNITUDE = 1.0
##
## LAYER 2
##
LOAD CASE = 1
        FEA LOAD CASE ID = 1.1-3.1-3-
        LOAD TYPE = 10
        LOAD UNIT = 80
        TIME HISTORY = sine01.dac
        TH_DIRECTORY = D:\fat_test
        SHIFT = 0.0
        SCALE FACTOR = 1.0
        LOAD MAGNITUDE = 1.0
LOAD CASE = 2
        FEA LOAD CASE ID = 1.1-3.1-3-
        LOAD TYPE = 10
        LOAD UNIT = 80
        TIME HISTORY = sine02.dac
        TH_DIRECTORY = D:\fat_test
        SHIFT = 0.0
        SCALE FACTOR = 1.0
        LOAD MAGNITUDE = 1.0
END EVENT_NAME_2
#
END SEQUENCE_NAME_1
#
!END OF FILE
```

Figure 6-38 Typical jobname.cfg file for a Duty Cycle run

CFG File Specific Parameters	
TEMPLATE JOBNAME =	The jobname.
NUMBER OF RESULTS ENTITIES PER JOB =	An integer number specifying the number of result entities in the analysis group.
NUMBER_OF_SEQUENCES =	An integer number specifying the number of sequences that have been defined in the CFG file.
SEQUENCE_NAME_# =	This lists the sequence number and the name of that Sequence.
NUMBER OF REPEATS OF SEQUENCE_NAME_# =	An integer number specifying the number of repeats that are going to be applied to the current sequence.
NUMBER OF EVENTS =	An integer number specifying the number of events in the current sequence.
LAYER SELECTION =	Currently this card has two values BOTH and SINGLE. Note: This will change in future versions where the information will be extracted from the analysis group(s) and recorded here.)
NUMBER OF JOBS REQUESTED =	An integer number specifying the number of jobs or .fin files that will be created.
EVENT_NAME_# =	This lists the event number and the name of that event.
NUMBER OF REPEATS OF EVENT_NAME_# =	An integer number specifying the number of repeats that are going to be applied to the current event.
NUMBER OF TIME HISTORIES =	An integer number specifying the number of time histories that will be used for the current event.
END EVENT_NAME_#	Signifies the end of the current event.
END SEQUENCE_NAME_#	Signifies the end of the current sequence.

The Submit Script (FatigueSubmit)

MSC Fatigue uses a submit script to manage the execution of MSC Fatigue modules during an analysis. For completion and reference purposes, this script is shown here for the Linux bourne shell. This script may be edited and modified by the user if necessary. This a only an example of the submit script and it may vary from computer system to computer system and from release to release.

```
#! /bin/sh
#set -x
```

```
#~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~
~~~~~~~~~
#
#  FatigueSubmit
#
#
#
# Abstract:
#    This procedure executes a MSC/FATIGUE analysis by running
the the
#    MSC/FATIGUE translator and solver in the proper order.  This
#    procedure is called from within MSC/PATRAN when the user
submits a job
#    for analysis.
#
#
#  Arguments:
#
#    -j jobname         Analysis job name
#
#   -a iat            The analysis type:     1 = crack initiation
#                                              2 = spot weld
#                                              3 = crack growth
#                                             4 = nominal stress
#                                              44 = rotational
analysis
#                                           5 = vibration fatigue
#                                              6 = Mag Weld
#                                              7 = Multiaxial CI
#                                            8 = Multiaxial FOS
#
#    -p partial         Partial Analysis Flag: 0 = full analysis
#                                          1 = pre-processor only
#
#    -y plane           Plane_stress:          0 = none
#                                              1 = plane stress
correction
#
#    -f fos_opt         Safety factor option:  0 = none
#                                              1 = stress based
#                                              2 = life based
#
#    -s str_life        Stress reference or design life (actual
value)
#
#   -m max_fac        Maximum factor to calculate (actual value)
#
#    -c cut_off         Use material cutoff:   0 = use cuttoff
#                                           1 = don't use cutoff
#
#    -z fast            Fast analysis option:  0 = none
#                                              1 = gate
#                                              2 = points
#                                              3 = reduction
```

```
#
#    -g pvx_gate        Percentage gate to use in PV eXtraction
for quick analysis
#                     or the number of points or reduction factor
#
#    -n num_ents        Number of entities to keep for fast
analysis
#
#    -t temp_dpnd       Use temp dependent mats:    0 = no
(default)
#                                                   1 = yes
#
#    -b mmean_mats      Use multi-mean correction:  0 = no
(default)
#                                                   1 = yes
#
#    -d spect_lod       Use spectrum loading input: 0 = no
(default)
#                                                   1 = yes
#
#    -e seamw_2004      Use seamw2004 solver:       0 = no
(default)
#                                                   1 = yes
#
#
#  Modification History:
#     11/24/93 Initial Release
#     11/26/94 Update for 1.4
#     09/12/95 Update for 6.0
#     10/22/01 Update Multiaxial (ljd)
#     12/13/01 Fix femlf submit error (dt1)
#
#~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~
~~~~~~~~~

# DEFINE A LOCAL SHELL FUNCTION FOR DISPLAYING VALID PARAMS FOR
THIS SCRIPT
abort_usage( ) {
   cat 2>&1 <</
Usage:  $Cmd -j jobname -a iat -p partial [optional flags]
    flags:  -v             for verbose output
    -j jobname     to specify a jobname (required)
    -a iat         to specify analysis type:
                              1 = crack initiation (default)
                              2 = spot weld
                              3 = crack growth
                              4 = total life
                              44 = rotation
                              5 = vibration fatigue
                              6 = Mag Weld
                              7 = MultiCI
                              8 = MultiFOS
    -p partial     to specify full or partial analysis:
                              0 = full analysis (default)
```

```
                                1 = partial analysis (stops after
rainflow)
    -y plane       to specify plane_stress correction:
                                0 = no (default  (for crack
                                1 = yes           growth only)
-f fos_opt     to specify factor of safety option:
                                0 = none (default)
                                1 = stress based
                                2 = life based
           -s str_life    to specify reference stress or design
life
                                (this is a factor of safety
parameter)
            -m max_fac     to specify maximum stress factor
            -c cut_off     to specify material cutoff usage:
                                0 = use cutoff (default)(used for
factor
                               1 = don't use cutoff     of safety
only)
            -z fast        to specify fast analysis using PV
eXtraction
                                0 = none (default      (Not valid for
                                1 = percentage gate     crack growth
or
                                2 = number of points    rainflow
matrices )
                                3 = reduction factor
           -g pvx_gate    to specify actual percentage gate for
PV eXtraction
                              or number of points or reduction factor
            -n num_ents    to specify number of FE entities for
keep for
                               fast analysis
            -t temp_dpnd   Use temp dependent mats:
                                0 = no (default)
                                1 = yes
            -b mmean_mats  Use multi-mean correction:
                                0 = no (default)
                                1 = yes
            -d spect_lod   Use spectrum loading input:
                                0 = no (default)
                                1 = yes
            -e seamw_2004  Use seamw2004 solver:
                                0 = no (default)
                                1 = yes
/
   exit 1
}

# DEFINE A LOCAL SHELL FUNCTION FOR CHECKING STATUS
check_status( ) {
   if [ -f $jobname.abo ] ; then
      rm -f $Semaphore
      rm -f $jobname.abo
```

```
      echo "Abort detected ... aborting $Cmd"
      exit 2
   fi
   if [ "$1" -ne 0 ] ; then
#     trap "No damaging nodes" as a non-error
      code=0
      code=`tail -5 $jobname.msg|grep -c "No damaging nodes"`
      if [code -ne 1] ; then
         rm -f $Semaphore
         echo "Error detected ... code is <$1> ... aborting $Cmd"
         exit 2
      fi
   fi
}

# DEFINE A LOCAL SHELL FUNCTION FOR DOING A GENERAL-PURPOSE EXIT
exit_normal() {
   rm -f $Semaphore
   rm -f $jobname.rmn
   echo "$Cmd complete"
   exit 0
}

# DETERMINE THE COMMAND NAME OF THIS SCRIPT
Cmd=`echo $0 | sed 's;^.*/;;'`

# SET UP PROPER PATHS TO THE EXECUTABLES
if [ -z "$p3fpath" ] ; then
   if [ -z "$P3_HOME" ] ; then
      ExecDir=/msc/patran75/mscfatigue_files/bin
      FatDir=/msc/patran75/mscfatigue_files
   else
      ExecDir=$P3_HOME/mscfatigue_files/bin
      FatDir=$P3_HOME/mscfatigue_files
   fi
else
   ExecDir=$p3fpath/bin
   FatDir=$p3fpath
fi
if [ -z "$npath" ] ; then
   NpathDir=$ExecDir
else
   NpathDir=$npath/nsoft
fi

# set nCode environment variables
if [ -z "$SHLIB_PATH" ] ; then
   SHLIB_PATH=$P3_HOME/mscfatigue_files/bin ; export SHLIB_PATH
else
   SHLIB_PATH=$P3_HOME/mscfatigue_files/bin:$SHLIB_PATH ; export
SHLIB_PATH
fi

# ASSIGN THE DEFAULT ARGUMENT PARAMETER VALUES
```

```
jobname=''
Verbose=''
iat=1
partial=0
plane=0
fos_opt=0
str_life=0
max_fac=100
cut_off=0
fast=0
pvx_gate=0
num_ents=0
temp_dpnd=0
mmean_mats=0
spect_lod=0
seamw_2004=0
Status=0

# PARSE THROUGH THE INPUT ARGUMENTS
while [ $# -ne 0 ] ; do
   case "$1" in
      -v) Verbose=yes    ; shift   ;;
      -j) jobname=$2     ; shift 2 ;;
      -a) iat=$2         ; shift 2 ;;
      -p) partial=$2     ; shift 2 ;;
      -y) plane=$2       ; shift 2 ;;
      -f) fos_opt=$2     ; shift 2 ;;
      -s) str_life=$2    ; shift 2 ;;
      -m) max_fac=$2     ; shift 2 ;;
      -c) cut_off=$2     ; shift 2 ;;
      -z) fast=$2        ; shift 2 ;;
      -g) pvx_gate=$2    ; shift 2 ;;
      -n) num_ents=$2    ; shift 2 ;;
      -t) temp_dpnd=$2   ; shift 2 ;;
      -b) mmean_mats=$2  ; shift 2 ;;
      -d) spect_lod=$2   ; shift 2 ;;
      -e) seamw_2004=$2  ; shift 2 ;;
      -*) abort_usage           ;;
   esac
done

# MAKE CERTAIN THAT A JOBNAME PARAMETER WAS SPECIFIED
if [ -z "$jobname" ] ; then
   abort_usage
fi

# MAKE CERTAIN THAT AN ANALYSIS TYPE WAS SPECIFIED
if [ -z "$iat" ] ; then
   abort_usage
fi

# DISPLAY VERBOSE INFORMATION DESCRIBING PARAMETERS IF REQUESTED
TO DO SO
if [ ! -z "$Verbose" ] ; then
```

```
   cat <</

$Cmd parameters ...
   Jobname               = $jobname
   Analysis Type         = $iat
   Full(0)/Partial(1)    = $partial
   Plane Stress          = $plane
   Factor of Safety      = $fos_opt
   Ref. Stress/Life      = $str_life
   Use Cutoff            = $cut_off
   Fast Analysis         = $fast
   PVX % Gate            = $pvx_gate
   Num. FE Entities      = $num_ents
   Temperature dependent= $temp_dpnd
   Multi-mean correction= $mmean_mats
   Spectrum Load Input   = $spect_lod
   Use Seamw2004         = $seamw_2004
/
fi

# DELETE ANY OLD FILES OF THE SAME JOBNAME
rm -f $jobname.fpp
rm -f $jobname.fos
rm -f $jobname.fef
rm -f $jobname.fdf
rm -f $jobname.tcy
rm -f $jobname.crg
rm -f $jobname.rmn
rm -f $jobname.fos_tmpl
rm -f $jobname.fef_tmpl
rm -f $jobname.fdf_tmpl

# ASSERT THE SEMAPHORE FOR THIS JOB TO LET THE WORLD KNOW THAT
WE'RE BUSY
Semaphore=$jobname.sta
StatusFile=$jobname.sta
echo "Job has begun execution" >$Semaphore

# ALSO SEND ALL LICENSE MESSAGES TO A LOG FILE
LogFile=$jobname.log
date >$LogFile

# DEFINE SOME USEFUL GLOBAL VARIABLES
PCRACK=$ExecDir/pcrack
PFPOST=$ExecDir/pfpost
FEFAT=$ExecDir/fefat
FATFE=$ExecDir/mfatfe
FEROT=$ExecDir/ferot
FEVIB=$ExecDir/fevib
FASTAN=$ExecDir/fastan
SPOTW=$ExecDir/spotw
SEAMW=$ExecDir/seamw
MULTI=$ExecDir/femlf
EntFile='pfatigue.ents'
```

```
rm -f $EntFile
rm -f fastan.msg

# FUNCTION TO PERFORM THE ANALYSIS
SubmitAnalysis()
{
   Semaphore=$jobname.fpr
   echo "Executing MSC/FATIGUE job \"$jobname\"" >$Semaphore

 # DO THE CRACK GROWTH ANALYSIS:
   if [ $iat = 3 ] ; then
       # PRE-PROCESS CRACK GROWTH:
       $PCRACK /mscopt=p/INP=$jobname/OV=Y/\*=$jobname.msg
>>$LogFile
       check_status $?
       if [ $partial = 0 ] ; then
          # DO THE FULL ANALYSIS:
          if [ -f $jobname.tcy ] ; then
             if [ $plane = 1 ] ; then
                # PLANE STRESS CORRECTION:
 $PCRACK /mscopt=a/INP=$jobname/PLANE=Y \
/OV=Y/\*=$jobname.msg >>$LogFile
                check_status $?
             else
                # NO PLANE STRESS CORRECTION:
                $PCRACK
/mscopt=a/INP=$jobname/OV=Y/\*=$jobname.msg >>$LogFile
                check_status $?
             fi
          else
             check_status 1
          fi
       fi
   fi

 # DO THE SPOT WELD ANALYSIS:
   if [ $iat = 2 ] ; then
      if [ -f $jobname.fes ] ; then
         $SPOTW  /OPT=e/INP=$jobname/OV=Y/\*=$jobname.msg
>>$LogFile
         check_status $?
      else
         check_status 1
      fi
   fi

# DO THE MAG WELD ANALYSIS:
   if [ $iat = 6 ] ; then
      if [ $seamw_2004 = 1 ] ; then
          if [ -f $jobname.fes ] ; then
             $SEAMW  /OPT=e/INP=$jobname/OV=Y/\*=$jobname.msg
>>$LogFile
             check_status $?
          else
```

```
            check_status 1
         fi
      else
         $FATFE  /job=$jobname/ana=y/OV=Y/\*=$jobname.msg
>>$LogFile
# Until fatfe solver writes status, use manual job-end message
         echo "Job execution has ended" >$StatusFile
         check_status $?
      fi
   fi

 # CHECK FOR A pfatigue.ent FILE
   if [ -f $EntFile ] ; then
      location=@$EntFile
   else
      location=all
   fi

 # DO THE ROTATION ANALYSIS:
   if [ $iat = 44 ] ; then
         if [ $partial = 0 ] ; then
            # DO FULL ANALYSIS
            if [ -f $jobname.fes ] ; then
               $FEROT
/OPT=a/INP=$jobname/OV=Y/OPT=X/\*=$jobname.msg >>$LogFile
               check_status $?
            else
               check_status 1
            fi
         fi
   fi

 # DO THE CRACK INITATION OR TOTAL LIFE ANALYSIS:
   if [ $iat = 1 -o $iat = 4 ] ; then
      # DO THE FAST ANALYSIS IF REQUESTED
      if [ $fast != 0 ] ; then
         if [ $fast = 1 ] ; then
              type=GATE
         fi
         if [ $fast = 2 ] ; then
              type=LEN
         fi
         if [ $fast = 3 ] ; then
              type=REDUCT
         fi
         $FASTAN /JOB=$jobname/METHOD=$fast/$type=$pvx_gate \
/ENTS=$num_ents/\*=$jobname.msg >>$LogFile
         cat fastan.msg >>$jobname.msg
         rm -f fastan.msg
      else
        if [ $temp_dpnd = 1 -o $mmean_mats = 1 -o $spect_lod = 1
] ; then
            # DO FULL ANALYSIS
            $FATFE  /job=$jobname/opt=p/popt=a/dobax=n \
```

```
/ov=y/opt=a/ana=y/\*=$jobname.msg >>$LogFile
             echo "Job execution has ended" >$StatusFile
             check_status $?
         else
             # PRE-PROCESS FIRST:
             $FEFAT /OPT=p/INP=$jobname/LOC=$location \
/OV=Y/ANAL=NO/\*=$jobname.msg >>$LogFile
             check_status $?

             if [ $partial = 0 ] ; then
                # DO FULL ANALYSIS
                if [ -f $jobname.fpp ] ; then
                   $FEFAT
/OPT=f/INP=$jobname/OV=Y/\*=$jobname.msg >>$LogFile
                   check_status $?
                else
                   check_status 1
                fi
             fi
         fi
      fi
   fi

 # DO THE FACTOR OF SAFETY ANALYSIS:
   if [ $fos_opt != 0 ] ; then
      if [ $temp_dpnd = 1 -o $mmean_mats = 1 -o $spect_lod = 1 ]
; then
         if [ $partial = 0 ] ; then
            # DO ONLY IF FULL ANALYSIS REQUESTED:
            if [ -f $jobname.fpp ] ; then
               echo "FOS calculation has started" >$StatusFile
               if [ $fos_opt = 1 ] ; then
                  # REFERENCE STRESS BASED FACTOR OF SAFETY:
                  $FATFE  /job=$jobname/opt=p/popt=a/dobax=n \
/ov=y/opt=a/saf=S/dlif=$str_life/\*=$jobname.msg >>$LogFile
                  echo "FOS calculation has ended" >$StatusFile
                  check_status $?
               elif [ $fos_opt = 2 ] ; then
                  if [ $cut_off = 0 ] ; then
                     # DESIGN LIFE BASED FACTOR OF SAFETY W/
CUTOFF:
                     $FATFE/job=$jobname/opt=p/popt=a/dobax=n \
/MAXFAC=$max_fac/USECUT=Y/ov=y \
/opt=a/saf=L/dlif=$str_life/\*=$jobname.msg >>$LogFile
                    echo "FOS calculation has ended" >$StatusFile
                     check_status $?
                  else
                     # DESIGN LIFE BASED FACTOR OF SAFETY W/O
CUTOFF:
                     $FATFE /job=$jobname/opt=p/popt=a/dobax=n \
/MAXFAC=$max_fac/USECUT=N/ov=y/opt=a/saf=L \
/dlif=$str_life/\*=$jobname.msg >>$LogFile
                    echo "FOS calculation has ended" >$StatusFile
                     check_status $?
```

```
                  fi
               else
                  check_status 1
               fi
            fi
         fi
      else
         if [ $partial = 0 ] ; then
            # DO ONLY IF FULL ANALYSIS REQUESTED:
            if [ -f $jobname.fpp ] ; then
               if [ $fos_opt = 1 ] ; then
                  # REFERENCE STRESS BASED FACTOR OF SAFETY:
                  $FEFAT
/OPT=s/INP=$jobname/ANAL=S/STRESS=$str_life \
/OV=Y \/*=$jobname.msg >>$LogFile
                  check_status $?
               elif [ $fos_opt = 2 ] ; then
                  if [ $cut_off = 0 ] ; then
                     # DESIGN LIFE BASED FACTOR OF SAFETY W/
CUTOFF:
                     $FEFAT
/OPT=s/INP=$jobname/ANAL=L/LIFE=$str_life/OV=Y \
/MAXFAC=$max_fac/USECUT=Y/\*=$jobname.msg >>$LogFile
                     check_status $?
                  else
                     # DESIGN LIFE BASED FACTOR OF SAFETY W/O
CUTOFF:
                     $FEFAT
/OPT=s/INP=$jobname/ANAL=L/LIFE=$str_life \
/OV=Y/MAXFAC=$max_fac/USECUT=N/\*=$jobname.msg >>$LogFile
                     check_status $?
                  fi
               else
                  check_status 1
               fi
            fi
         fi
      fi
   fi


 # DO THE VIBRATION FATIGUE ANALYSIS:
   if [ $iat = 5 ] ; then
            # DO FULL ANALYSIS
            if [ -f $jobname.fes ] ; then
               $FEVIB  /OPT=G/INP=$jobname/OV=Y/\*=$jobname.msg
>>$LogFile
               check_status $?
            else
               check_status 1
            fi
   fi

# DO THE MULTIAXIAL CI ANALYSIS:
```

```
   if [ $iat = 7 ] ; then
      if [ -f $jobname.fes ] ; then
#        $MULTI  /OPT=c/INP=$jobname/OV=Y/\*=$jobname.msg
>>$LogFile       # dt1
         $MULTI
/OPT=c/INP=$jobname/JOB=$jobname/OV=Y/\*=$jobname.msg >>$LogFile
         check_status $?
      else
         check_status 1
      fi
   fi

# DO THE MULTIAXIAL FOS ANALYSIS:
   if [ $iat = 8 ] ; then
      if [ -f $jobname.fes ] ; then
#        $MULTI  /OPT=S/INP=$jobname/OV=Y/\*=$jobname.msg
>>$LogFile       # dt1
         $MULTI
/OPT=S/INP=$jobname/JOB=$jobname/OV=Y/\*=$jobname.msg >>$LogFile
         check_status $?
      else
         check_status 1
      fi
   fi

}

eval SubmitAnalysis

# SUCCESSFUL EXIT OF SCRIPT
exit_normal
```

The Analysis Manager Execution Script (FatigueExecute)

MSC Fatigue uses an execution script to manage the execution of MSC Fatigue modules during an analysis submitted by the Analysis Manager. For completion and reference purposes, this script is shown here for the Linux bourne shell. This script may be edited and modified by the user if necessary. This a only an example of the submit script and it may vary from computer system to computer system and from release to release.

```
#! /bin/sh
#set -x

#~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~
~~~~~~~~~
#
#  FatigueExecute
#
#
# Abstract:
#    Script to submit MSC/FATIGUE to a local or remote host via
the
```

```
#    ANALYSIS MANAGER.  This script will call FatigueSubmit on
the
#   appropriate host.  All FatigueSubmit parameters are obtained
from
#    the jobname.job file.
#
#    The current version does not move files from submit node to
analysis node.
#    This version handles file conversion & launching of
FatigueSubmit.
#
#  Limitations:
#
#  Arguments:
#
#    -j jobname         Analysis job name
#
#    -h remhost         The of the submitting host (where the job
originated).
#
#    -d remdir         The directory on the remote host where the
job originated.
#
#  Modification History:
#     08/30/04 Initial Release
#
#~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~
~~~~~~~~~

# DEFINE A FUNCTION FOR DISPLAYING VALID PARAMS FOR THIS SCRIPT
#~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~
abort_usage( ) {
   cat 2>&1 <</
Usage:  $Cmd -j jobname -h remhost -d remdir
    flags:   -v             for verbose output
             -j jobname     to specify a jobname (required)
             -h remhost     to specify the remote hostname
(required)
             -d remdir      to specify the remote directory
(required)
           -e exename     to specify a solution type (optional)
/
   exit 1
}
#~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

# DEFINE A FUNCTION FOR CHECKING STATUS
#~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~
check_status( ) {
   Status=$1
   if [ "$1" -ne 0 ] ; then
      echo "Error detected ... aborting $Cmd"
      exit 2
   fi
```

```
}
#~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

# DEFINE A FUNCTION FOR DOING A GENERAL PURPOSE EXIT
#~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~
exit_normal( ) {
   rm -f $jobname.fpr
   echo "$Cmd complete"
   exit 0
}
#~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

# DETERMINE THE COMMAND NAME OF THIS SCRIPT
Cmd=`echo $0 | sed 's;^.*/;;'`
# SET UP PROPER PATHS TO THE EXECUTABLES
if [ -z "$P3_HOME" ] ; then
    echo "No path to fatigue executables"
    check_status -1
else
    ExecDir=$P3_HOME/bin
fi

# ASSIGN THE DEFAULT ARGUMENT PARAMETER VALUES
jobname=''
exename=''
remhost=${MSCP_FATIGUE_HOST:-LOCAL}
remdir=${MSCP_FATIGUE_SCRATCHDIR:-.}
Status=0


# PARSE THROUGH THE INPUT ARGUMENTS
if [ $# -lt 2 ] ; then
  abort_usage
fi

while [ $# -ne 0 ] ; do
   case "$1" in
      -v) Verbose=yes  ; shift   ;;
      -j) jobname=$2   ; shift 2 ;;
      -h) remhost=$2   ; shift 2 ;;
      -d) remdir=$2    ; shift 2 ;;
      -e) exename=$2   ; shift 2 ;;
       *) abort_usage ;;
   esac
done

# MAKE CERTAIN THAT THERE IS A JOBNAME
if [ -z "$jobname" ] ; then
   abort_usage
fi
if [ -z "$FAT_COMMAND_STR" ] ; then
   echo "Env variables not being set from rcf file"
   check_status -1
fi
```

```
if [ -z "$FAT_SUBHOST_TYPE" ] ; then
   echo "Env variables not being set from rcf file"
   check_status -1
fi

exename=$FAT_EXE_NAME
if [ -z "$exename" ] ; then
   exename=FEFAT
fi


# DISPLAY VERBOSE INFORMATION DESCRIBING PARAMETERS IF REQUESTED
TO DO SO
if [ ! -z "$Verbose" ] ; then
   cat <</

$Cmd parameters ...
   Exename               = $exename
   Jobname               = $jobname
   Remote Host           = $remhost
   Remote Directory      = $remdir
/
fi


# Determine type of analysis machine;
# only two choices since this script only on these machines.
OsName=`uname`
case "$OsName" in
   Linux)
      AnalysisHostType="LINUX"
      DestType=3
      ;;
   *)
      AnalysisHostType="Linux"
      DestType=12
      ;;
esac

# Determine type of submit machine;
# env FAT_SUBHOST_TYPE is defined in rcf file & exported to this
machine
case "$FAT_SUBHOST_TYPE" in
   WINNT)
      SubmitHostType="WINNT"
      SourceType=3
      ;;
   LINUX)
      SubmitHostType="LINUX"
      SourceType=3
      ;;
   *)
      SubmitHostType="Linux"
      SourceType=12
```

```
      ;;
esac


LogFile=$jobname.log
date >$LogFile
CONFIL=$ExecDir/mconfil
FASTAN=$ExecDir/fastan
FATFE=$ExecDir/mfatfe
FEFAT=$ExecDir/fefat
FEROT=$ExecDir/ferot
FEVIB=$ExecDir/fevib
MULTI=$ExecDir/femlf
PCRACK=$ExecDir/pcrack
SEAMW=$ExecDir/seamw
SPOTW=$ExecDir/spotw


# convert ascii fes if needed
if [ "$AnalysisHostType" != "$SubmitHostType" ] ; then
    case "$exename" in
FASTAN)
if [ -f $jobname.asc ] ; then
   $FASTAN /OPT=u/UOPT=b/INP=$jobname \
/OUT=$jobname/OV=Y/\*=$jobname.msg  >>$jobname.log
   check_status $?
fi
;;
MFATFE)
if [ -f $jobname.asc ] ; then
   $FATFE /OPT=u/UOPT=f/JOB=$jobname \
/OV=Y/\*=$jobname.msg  >>$jobname.log
   check_status $?
fi
;;
FEFAT)
if [ -f $jobname.asc ] ; then
   $FEFAT /OPT=u/UOPT=b/INP=$jobname \
/OUT=$jobname/OV=Y/\*=$jobname.msg  >>$jobname.log
   check_status $?
fi
;;
FEROT)
if [ -f $jobname.asc ] ; then
   $FEROT /OPT=u/UOPT=b/INP=$jobname/OUT=$jobname \
/OV=Y/\*=$jobname.msg  >>$jobname.log
   check_status $?
fi
;;
FEVIB)
if [ -f $jobname.asc ] ; then
   $FEVIB /OPT=u/UOPT=b/INP=$jobname/OV=Y/\*=$jobname.msg
>>$jobname.log
   check_status $?
```

```
fi
;;
FEMLF)
if [ -f $jobname.asc ] ; then
   $MULTI /OPT=u/UOPT=b/INP=$jobname/OV=Y/\*=$jobname.msg
>>$jobname.log
   check_status $?
fi
;;
PCRACK)
if [ -f $jobname.asc ] ; then
   $PCRACK /MSCOPT=u/UOPT=b/INP=$jobname/OUT=$jobname \
/OV=Y/\*=$jobname.msg  >>$jobname.log
   check_status $?
fi
;;
SEAMW)
if [ -f $jobname.asc ] ; then
   $SEAMW /OPT=u/UOPT=b/INP=$jobname/OV=Y/\*=$jobname.msg
>>$jobname.log
   check_status $?
fi
;;
SPOTW)
if [ -f $jobname.asc ] ; then
   $SPOTW /OPT=u/UOPT=b/INP=$jobname/OV=Y/\*=$jobname.msg
>>$jobname.log
   check_status $?
fi
;;
*)
abort_usage
;;
    esac
fi


# DETERMINE NUMBER OF SUPPORTING FILES USED
numdac=0
numpsd=0
numcyh=0
numksn=0
numspe=0
numlcs=0
nummdb=0
numhtd=0
nummnd=0
numdac=`ls *.dac | wc -l` > /dev/null 2>&1
numpsd=`ls *.psd | wc -l` > /dev/null 2>&1
numcyh=`ls *.cyh | wc -l` > /dev/null 2>&1
numksn=`ls *.ksn | wc -l` > /dev/null 2>&1
nummdb=`ls *.mdb | wc -l` > /dev/null 2>&1

# DO CONVERSIONS IF THEY CAME FROM BINARY INCOMPATIBLE SYSTEMS
```

```
if [ "$AnalysisHostType" != "$SubmitHostType" ] ; then
   if [ "$numdac" -ne 0 ] ; then
      find . -name \*.dac -print | awk -F/ '{print $2}' > names
      for i in `cat names`
      do
         $CONFIL /INP=$i/TYP=11 - DAC/SOU=$SourceType \
/DES=$DestType/OV=Y/\*=$jobname.msg  >>$jobname.log
      done
      rm -f names
   fi
   if [ "$numpsd" -ne 0 ] ; then
      find . -name \*.psd -print | awk -F/ '{print $2}' > names
      for i in `cat names`
      do
         $CONFIL /INP=$i/TYP=11 - DAC/SOU=$SourceType \
/DES=$DestType/OV=Y/\*=$jobname.msg  >>$jobname.log
      done
      rm -f names
   fi
   if [ "$numcyh" -ne 0 ] ; then
      find . -name \*.cyh -print | awk -F/ '{print $2}' > names
      for i in `cat names`
      do
         $CONFIL /INP=$i/TYP=11 - DAC/SOU=$SourceType \
/DES=$DestType/OV=Y/\*=$jobname.msg  >>$jobname.log
      done
      rm -f names
   fi
   if [ "$numksn" -ne 0 ] ; then
      find . -name \*.ksn -print | awk -F/ '{print $2}' > names
      for i in `cat names`
      do
         $CONFIL /INP=$i/TYP=11 - DAC/SOU=$SourceType \
/DES=$DestType/OV=Y/\*=$jobname.msg  >>$jobname.log
      done
      rm -f names
   fi
   if [ "$nummdb" -ne 0 ] ; then
      find . -name \*.mdb -print | awk -F/ '{print $2}' > names
      for i in `cat names`
      do
         $CONFIL /INP=$i/TYP=74 - MDB/SOU=$SourceType \
/DES=$DestType/OV=Y/\*=$jobname.msg  >>$jobname.log
      done
      rm -f names
   fi
fi

# Perform the analysis;
# env FAT_COMMAND_STR is defined in rcf file & exported to this
machine
$ExecDir/FatigueSubmit $FAT_COMMAND_STR ; check_status $?
```

```
# SUCCESSFUL EXIT OF SCRIPT
exit_normal
```

The Fatigue Input File (jobname.fes)

The file is stored in 512 byte blocks, the first of which is a standard nCode header. The file is a mixture of 16-bit integers, 32 -bit integers (INTEGER*4), 32 bit real numbers and character information stored as 16-bit integers (two characters per integer). The date format is 7 INTEGER values, in order Year, Month, Day, Hour, Minute, Second, Hundredths.

Record 1 (Header record)

WORD	CONTENTS
1-7	Creation date (7xINTEGER)
8-14	Update date (7xINTEGER)
15	Operating system which created the file. (INTEGER), 2=VMS,10=DEC Linux,12=Other Linux, 101=Windows
16	File ID number (89=FES File) (INTEGER)
17	Major Revision level (3=Current level) (INTEGER)
18	Minor Revision level (5=Current level) (INTEGER)
19-34	Program name which last wrote the file (CHARACTER*32)
35	FES Type (Integer): 0=Full FES file, 1=Partial FES file
36	Data Source (Integer) 0 = MSC/PATRAN or MSC/FATIGUE 1 = Hypermesh 2 = SDRC 3 = Mechanica 4 = ANSYS 5 = NASTRAN OP2 6 = ABAQUS 7 = MEDINA 8 = FEDEM 9 = ADAMS
37-50	Unused
51	Record number containing load data LODREC (usually record 3) (INTEGER*4)
53	Record number containing group/mat. data (INTEGER*4)
55	Record number containing stress data (INTEGER*4)
57	Number of calculation points (nodes or elements) (INTEGER*4)
59-256	Unused

Record 2 (Parameter record)

WORD	CONTENTS
1	Fatigue Calculation Type (INTEGER): 1=Local Strain, 2=S-N, 3=Crack Growth, 4=Spot Weld, 5=Vibration analysis, 6=Seam Weld S-N, 7=Multiaxial E-N, 8=Multiaxial safety factor, 9=Multiple mean stress S-N
2	FE results type (INTEGER): 0=Linear Static, 1=Linear Time Step (Transient dynamic, etc.), 2=Transfer function, 3=PSD, 4=Elastic-plastic Time Step, 5=Rotating analysis (wheels)
3	Results Type (INTEGER): 0=Stress, 1=Strain (Local strain method only), 2=Forces (Spot weld only), 3=Stress and Strain (from Elastic-Plastic)
4	Results at (INTEGER): 0=Node, 1=Element centroid, 2=Node and element, 3=Element centroid and corners
5	Results co-ordinate system (INTEGER): 0=Full 3D, no defined co-ordinate system, 1=2D resolved to surface.
6	Results combination method (INTEGER): 1=Max. Principal, 2=Min. Principal, 3=Abs. Max. Principal (*), 4=Von Mises, 5=Max. Shear (stress) or Tresca (strain), 6=Signed Von Mises (*), 7= X component (*), 8=Y component (*), 9=Z component (*), 10=XY component (*), 11=YZ component (*), 12=ZX component (*), 19=Signed Max. Shear Stress (*), 20=Signed Tresca Strain(*), 21=Mid Principal, 22=Min. Abs. Principal, 23=Critical plane analysis (set for multiaxial, non-changeable). The (*) indicates approved options.
7	Number of load inputs (static) or time steps (time-step analyses) or load inputs (vibration) or load conditions (rotating analysis) (INTEGER). If number of time steps=-1, see offset in word 17.
8	Number of groups/materials (INTEGER)
9	Design Criterion, in % (REAL)
11	Stress units (INTEGER): 0=MPa, 1=Pascals, 2=PSI, 3=KSI, 4=kg/m^2 Force/length units (Spot Weld only): 0=Newtons/mm, 1=Newtons/m, 2=lbs/ins, 3=kips/in
12	Damage Model (INTEGER): Local Strain: 0=SWT, 1=Morrow, 2=Strain-Life; S-N: 0=No correction, 1=Goodman, 2=Gerber, 3=Interpolate (multiple mean stress only); Spot Weld: 0=only valid number; Seam Weld: 0=No mean stress correction, 1=Mean stress correction; Multiaxial E-N: 0=Normal strain, 1=SWT-Bannantine, 2=Shear stress, 3=Fatemi-Socie, 4=Wang-Brown, 5=Wang-Brown with Mean; Multiaxial safety factor: 0=Dang Van, 1=McDiaemid
13	Elastic-plastic correction (INTEGER): 0=Neuber, 1=Mertens/Dittmann, 2=Seeger/Beste, 3=None (for elastic-plastic results), 4=Glinka (not supported yet)
14	Biaxial correction method (INTEGER): 0=None, 1=Ratio, 2=Hoffmann-Seeger
15	S-N small cycle correction (INTEGER): 0=None, 1=Heibach relative Miner.
16	Vibration Analysis (INTEGER): 0=Dirlik, 1=Narrow Band, 2=All

17	Number of time steps (v3.2 or above) (INTEGER*4)
19	Temperature dependent (INTEGER): 0=No, 1=YES
20	Temperature units (INTEGER): 0=Celsius, 1=Fahrenheit, 2=Kelvin
21	Shell element data type (INTEGER): For FEMAP: 1=Element centroid Top+Bottom, 2=Element centroid top, 3=Element centroid bottom, 4=Element centroid & corners (top+bottom), 5=Element centroid & corners (top), 6=Element centroid & corners (bottom), 7=Nodial average top, 8=Nodal average bottom
22	Solid element data type (INTEGER): For FEMAP: 1=Nodal average (default), 2=Element corners
23	Critical plane angle increment (REAL)
25	Seam weld analyses - Miners sum (REAL)
27	Unused
29	Hardening Parameter for Dang Van (REAL)
31	Crack Growth Initial crack length (REAL)
33	Crack Growth Final crack length (REAL)
35	Crack Growth Sharp crack radius (REAL)
37	Crack Growth Notch length (REAL)
39	Crack Growth Notch root radius (REAL)
41-56	KSN filename (CHARACTER*32)
57-68	Unused
69	Vibration fatigue PSD data (INTEGER): 0=1 combined or directional PSD per node, 1=6 component PSD per node
70	Number of vibration frequency steps (INTEGER), maximum value 8192
71-86	Vibration analysis - Load matrix filename (CHARACTER*32)
87-100	Unused
101-140	Job ID string 1 (CHARACTER*80)
141-180	Job ID string 2 (CHARACTER*80)
181	Number of equivalent units (REAL): Default 0.0; if 0.0, look in time histories extra details, or ignore if no time histories
183	Equivalent units (CHARACTER*80)
223-256	Unused

Frequency step records (start at record 3)

Only applicable for vibration analysis. Contains N real *4 values, one per frequency step. N is defined in record 2, element 70. The number of records occupied is defined by NREC=(N-1)/128 + 1.

The start of the load records is defined by LODREC=3+NREC.

Load description records (start at LODREC - usually record 3)

For load cases:

WORD	CONTENTS
1	Load case index number (INTEGER)
2-41	Load case descriptor (CHARACTER*80)
42	Load case type (INTEGER): 1=constant amplitude, 0=static, >0=see value in LTYPES.IND
43	Load case units (INTEGER)
44-103	Load case time history name (static), Load case PSD input (vibration) (CHARACTER*120)
104	Channel number (for RPCIII input) (INTEGER)
105	Load case applied value (REAL)
107	Load case scaling factor (REAL)
109	Load case offset (REAL)
111	Constant amplitude type (INTEGER): 0=Peak-peak, 1=Zero-peak, 2=User Defined
112	Constant amplitude maximum value (REAL)
114	Constant amplitude minimum value (REAL)
116	Time history file type (INTEGER): 0=DAC, 1=RPC 111 (see word 104)
117-128	Unused
129-256	Load case #2, structure as in 1-128.

For loading conditions

WORD	CONTENTS
1	Load conditions index number (INTEGER)
2-41	Load condition descriptor (CHARACTER*80)
42	Number of wedges (INTEGER)
43-104	Unused
105	Design life (REAL)
107	Load condition scaling factor (REAL)
109	Load condition offset (REAL)
111-128	Unused
129-256	Load condition #2, structure as in 1-128

Record LODREC+1 (if NLOADS > 2)

1-128	**Load case #3, etc. up to maximum of 200 load cases**

If the results are from a time step analysis or PSD type vibration analysis, there is only one load case record. Only the description, scale factor and offset are used.

First Material record (=LODREC + TRUNC(NLOADS/2 + 0.5)

WORD	CONTENTS
1	Material source (INTEGER): 0=Database local/central, 1=Database external, 2=Generated, 3=In FES file (not supported), 4=External ASCII file, 6=No material (e.g. BS5400)
2	S-N analysis type (INTEGER): 0=Material based, 1=Component based, 2=BS5400 welded, 3=BS5400 non-welded, 4=Spot Weld, 5=Seam Weld
3-42	Database file name (CHARACTER*80)
43-58	Material name for Spot Weld, sheet material name (CHARACTER*32)
59-74	Sub name (e.g., for Spot Weld: nugget material name; for Crack growth: environment name) (CHARACTER*32)
75	Analysis code parameter (INTEGER). If analysis code=2 or 3, then the following codes are used to define the weld classes : 1 = B, 2 = C, 3 = D, 4 = E, 5 = F, 6 = F2, 7 = G, 8 = S, 9 = W
76	Type of material as per PFMAT codes (INTEGER)
77	UTS (REAL)
79	Young's modulus (REAL)
81	Elastic Poisson's Ratio (REAL)
83	Cutoff (REAL)
85	Standard error for generated data (REAL)
87-128	Space for 21 material params if option 3 (REAL x 21)
129	Surface finish ID (INTEGER): 1=No finish/Polished, 2=Ground, 3=Good Machined, 4=Average Machined, 5=Poor Machined, 6=Hot rolled, 7=Forged, 8=Cast, 9=Water corroded, 10=Seawater corroded
130	Surface treatment ID (INTEGER): 1=No treatment, 2=Nitrided, 3=Cold rolled, 4=Shot peened
131	Multiplier on stress (REAL)
133	Stress offset (REAL)
135	Fatigue limit reduction factor, kf (REAL)
137	Plastic stress conc. factor (REAL)
139	M1/M2 Ratio (Seam weld, v3.4 onwards) (REAL), must be >0.0

141	Nugget diameter (Spot weld) (REAL), if 0.0, use the value in the data area. If -1.0, use the spotweld.sys file
143	Sheet thickness (Spot weld) (REAL), if 0.0, use the value in the data area
145	R-ratio transition ratio (Seam weld) (REAL), 0.0<=R-ratio <=1.0
147	Materials data type (INTEGER): 0=standard, 1=isothermal
148	Unused
149	Temperature for this group(REAL)
151-172	Unused
173-212	Group name for reference (CHARACTER*80)
213-256	Unused

This repeats, one group per block up to 25 groups.

Elements 76-128 are not used and can be set to 0, unless material source option=2, in which case type, E and UTS are required.

Data Records

These records start at the first record after the last material block.

WORD	CONTENTS
1	Node (INTEEGER*4)
3	Element (INTEGER*4). For "Node at element" either both may be set, or only one. If Node is set to 0, then the result is element only.
5	Pointer to group number (material etc.) (INTEGER)
7	Multiplier for this node (REAL)
9	Offset for this node (REAL)
11	Temperature @node (REAL)
13	For Spot Weld analyses - nugget diameter, for Seam weld analysis - element width (REAL)
15	For Spot Weld analyses - sheet thickness, for Seam weld analyses - element thickness (REAL)
17	Element type (INTEGER): 0=solid, 1=shell, 2=bar element (Spot Weld)
18	Element location if type=1 (INTEGER): 1=bottom, 2=mid, 3=top

19-32	Unused
33-44	X,Y,Z,XY,YZ,XZ stresses/strains for LC1 (6*REAL) and so on for other LCs. This block repeats for nodes/elements. For time step, LC=time step. For spot welds, these 6 values are used for the forces and moments: Fx, Fy, Fz, Mx, My, Mz. For a Spot Weld analysis, the element number must be the same for 3 consecutive entries (nugget, sheet1 and sheet2). The node number for the nugget must be zero. The node numbers for sheet1 and sheet2 correspond to the node numbers of the shell to which the spot weld beam is connected. For Seam welds analysis, the first 2 values are stress on top and bottom surface of shell respectively (the rest are assumed to be 0): S@ top, S@ bottom For vibration analysis, the six components of the transfer function (as complex numbers) must be stored for each frequency step, for each input. The order is as follows: Frequency step 1: Input 1: Sx1,Sx2,Sy1,Sy2,Sz1,Sz2,Sxy1,Sxy2,Syz1,Syz2,Sxz1, Sxz2 Input 2: …… ……. Frequency step 2 Input 1 : and so on where the frequency steps are in the order defined in the header. If the FE results type is PSD, there is only one load input. Since the value is load, not a transfer function there is only one value per frequency step, not twelve. For elastic-plastic results, each time step would also contain 12 results, the first six for strain and the next six for stress. For rotating analysis results, the six stresses for each load case for each wedge are stored as follows: Load case 1: Wedge 1: Sxx, Syy, Szz, Sxy, Syz, Sxz Wedge 2: ... Load case 2: Wedge 1: Sxx, Syy, Szz, Sxy, Syz, Sxz Wedge 2: .. and so on. Node/Element = -1 indicates end of file. It must be present even if the number of valid nodes is stored in the header.

ASCII Version (jobname.asc)

The format of the file is reasonably straightforward due to the fact that keywords have been used together with the data. The information in this file is a combination of information from the Patran database, the `jobname.fin` and FEA results files as combined and translated by the PAT3FAT translator. FEFAT's file conversion utility is used to create the jobname.asc file from a jobname.fes and convert back. The following is a brief explanation of each of the parameters:

File revision=	Should be set to 3 (REAL)
Fatigue code=	(INTEGER) Valid settings are 1 (Crack initiation), 2 (Component S-N), 3 (Crack growth), 4 (Spot weld S-N), 5 (Vibration), 6 (Seam weld S-N), 7 (Multiaxial E-N), 8 (Multiaxial safety factor), and 9 (Multiple menu stress S-N)
FE results type=	FE results type (INTEGER): 0=Linear Static, 1=Linear Time Step (Transient dynamic, etc.) 2=Transfer function, 3=PSD, 4=Elastic-plastic time step, 5=Rotating analysis (wheels)
Stress/strain=	Valid settings are 0 and 1 for stress or strain, respectively.
Results location=	Results at (INTEGER): 0=Node, 1=Element Centroid, 2=Node and Element, 3=Element centroid and corners
Results co-ordinate system=	Results co-ordinate system (INTEGER): 0=Full 3D, no defined co-ordinate system, 1=2D resolved to surface.
Stress/strain combination=	Results combination method (INTEGER): 1=Max. Principal, 2=Min. Principal, 3=Abs. Max. Principal (*), 4=Von Mises, 5=Max. Shear (stress) or Tresca (strain), 6=Signed Von Mises (*), 7= X component (*), 8=Y component (*), 9=Z component (*), 10=XY component (*), 11=YZ component (*), 12=ZX component (*), 19=Signed Max. Shear Stress, 20=Signed Tresca Strain(*), 21=Mid Principal, 22=Min. Abs. Principal, 23=Critical plane analysis. The (*) indicates approved options.
Number of load cases= **Number of time steps=**	Number of load cases or time steps (INTEGER)
Number of materials=	Number of materials (INTEGER)
Number of nodes=	Number of nodes with stress or strain results. Can refer to number of elements also if element based analysis (INTEGER)
Stress units=	Stress units (INTEGER): 0=Mpa, 1=Pascals, 2=PSI, 3=KSI, 4=kg/m^2
Force units	Force units (INTEGER): 0=Nmm, 1=Nm, 2=lbs ins, 3=kip in
Certainty of survival=	Design criterion in percentage (REAL). This must be 50 for crack initiation.
Damage model=	Damage Model (INTEGER): Local Strain: 0=SWT, 1=Morrow, 2=Strain-Life; S-N: 0=No correction, 1=Goodman, 2=Gerber, 3=Interpolate (multiple mean stress only); Spot weld: 0=only valid number; Seam weld: 0=No menu stress correction, 1=Menu stress correction on; Multiaxial E-N: 0=Normal strain, 1=SWT-Bannantine, 2=Shear stress, 3=Fatemi-Socie, 4=Wang-Brown, 5=Wange-Brown with menu.
Elastic-plastic correction=	Elastic-plastic correction (INTEGER): 0=Neuber, 1=Mertens/Dittmann, 2=Seeger/Beste, 3=None (not supported), 4=Glinka; for Multiaxial E-N: 0=Glinka, notch correction
Biaxiality correction=	Biaxial correction method (INTEGER): 0=None, 1=Ratio, 2=Hoffmann-Seeger
Initial crack length=	Crack Growth Initial crack length (REAL)

Final crack length=	Crack Growth Final crack length (REAL)
Sharp crack radius=	Crack Growth Sharp crack radius (REAL)
Notch length=	Crack Growth Notch length (REAL)
Notch root radius=	Crack Growth Notch root radius (REAL)
K Solution filename=	KSN file name (CHARACTER*32)
Load case number=	Load case index number (INTEGER)
Description=	Load case descriptor (CHARACTER*80)
Load type=	Load case type (INTEGER) See the description of the `jobname.fin` file for LOAD TYPE=
Load units=	Load case units (INTEGER) See the description of the `jobname.fin` file for!! UNIT TYPE=
Time history name=	Load case time history name (CHARACTER*32)
Applied FE load=	Load case applied value (REAL)
Scale factor=	Load case scaling factor (REAL)
Offset=	Load case offset (REAL)
Group ID number=	The group ID number which is the same as the material number (INTEGER)
Material source=	Material source (INTEGER): 0=Database local/central, 1=Database external, 2=Generated, 3=In FES file (not supported), 4=External ASCII file, 6=No material (e.g. BS5400)
Group description=	Group name for reference (CHARACTER*80)
S-N Analysis type=	S-N analysis type (INTEGER): 0=Material based, 1=Component based, 2=BS5400 welded, 3=BS5400 non-welded, 4=Spot weld, 5=Seam weld
Database name=	Database file name (CHARACTER*80)
Material name=	Material name (CHARACTER*32)
Sub-name=	Sub name (e.g. environment) (CHARACTER*32)
Weld class=	Analysis code parameter (INTEGER). If analysis code=2 or 3, then the following codes are used to define the weld classes: 1 = B, 2 = C, 3 = D, 4 = E, 5 = F, 6 = F2, 7 = G, 8 = S, 9 = W
Surface finish=	Surface finish ID (INTEGER)
Surface treatment=	Surface treatment ID (INTEGER)
Stress/strain raiser=	Multiplier on stress (REAL)
Stress/strain offset=	Stress offset (REAL)
Fatigue limit reduction (Kf)=	Fatigue limit reduction factor (REAL)
Plastic stress concentration factor (Formzahl)=	Plastic stress conc. factor (REAL)

Node number= **Element number=**	The node or element number. A node or element number of -1 indicates the end of the file (INTEGER)
Group ID=	The group ID applied to the above node or element (INTEGER)
Multiplier=	The scale factor to be applied to results values for the above node or element (REAL)
Offset=	An offset to be applied to the results values for the above node or element (REAL)
Temp.=	A temperature reference for the above (REAL). Not used.

Six component stresses or strains for each load case or time step appear between each **Node number=** or **Element number=** line.

The Preprocessed File (jobname.fpp)

The file is stored in 256 byte blocks, the first of which is a standard header. The file is a mixture of 16-bit integers, 32 -bit integers (INTEGER*4), 32 bit real numbers and character information stored as 16-bit integers (two characters per integer). The date format is 7 INTEGER values, in order Year, Month, Day, Hour, Minute, Second, Hundredths.

Record 1 (Header record)

WORD	CONTENTS
1-7	Creation date (7xINTEGER)
8-14	Update date (7xINTEGER)
15	Operating system which created the file. (INTEGER): 7=SUN, 8=SGI, 9=IBM, 10=DECS, 11=HP700
16	File ID number (88) (INTEGER)
17	Major Revision level (INTEGER)
18	Minor Revision level (INTEGER)
19-34	Program name which last wrote file (CHARACTER*32)
35-50	**Unused**
51	Pointer to record containing load data (INTEGER*4)
53	Pointer to record containing group/mat. data (INTEGER*4)
55	Pointer to first record containing stress data (INTEGER*4)
57	Number of nodes (INTEGER*4)
59	Bin size used in this file (INTEGER)
60	FPP file has been used (1) or not (0) (INTEGER)
61-128	**Unused**

Record 2 (Parameter record)

WORD	CONTENTS
1	**Fatigue Calculation Type (INTEGER): 1=Local Strain, =Component S-N, 4=Material S-N**
2	FE results type (INTEGER): 0=Linear Static, 1=Linear Time Step (Transient dynamic, etc.)
3	Results Type (INTEGER): 0=Stress, 1=Strain (Local strain method only)
4	Results at (INTEGER): 0=Node, 1=Element Centroid
5	Results co-ordinate system (INTEGER): 0=Full 3D no defined co-ordinate system, 1=2D resolved to surface.
6	Results combination method (INTEGER): 1=Max. Principal, 2=Min. Principal, 3=Abs. Max. Principal (*), 4=Von Mises, 5=Max. Shear (stress) or Tresca (strain), 6=Signed Von Mises (*), 7= X component (*), 8=Y component (*), =Z component (*), 10=XY component (*), 11=YZ component (*), 12=ZX component (*), 19=Signed Max. Shear Stress (*), 20=Signed Tresca Strain (*), 21=Mid Principal, 22=Min. Abs. Principal The (*) indicates approved options.
7	Number of load cases or time steps (INTEGER)
8	Number of groups/materials (INTEGER)
9	Design Criterion, in % (REAL)
11	Stress units (INTEGER), 0=MPa, 1=Pascals, 2=KSI, 3=PSI, 4=kg/m^2
12	Damage Model (INTEGER): Local Strain: 0=SWT, 1=Morrow, 2=Strain-Life; S-N: 0=No correction, 1=Goodman, 2=Gerber
13	Elastic-plastic correction (INTEGER): 0=Neuber, 1=Mertens-Dittmann, 2=Seeger-Beste, 3=Glinka, 4=None
14	Biaxiality correction method (INTEGER): 0=None, 1=Ratio, 2=Hoffmann-Seeger
15	S-N small cycle correction (INTEGER): 0=None, 1=Heibach relative Miner
16-29	Unused
30-69	Job ID string 1 (CHARACTER*80)
70-109	Job ID string 2 (CHARCATER*80)
110-128	Unused

Record 3 (Equivalent unit record)

1	Number of Equivalent units (REAL)
2-41	Equivalent units string (CHARACTER*80)
42-128	**Unused**

Record 4(Load description records)

1	Load case index number (INTEGER)
2-41	Load case descriptor (CHARACTER*80)
42	Load case type (INTEGER)
43	Load case units (INTEGER)
44-103	Load case time history name (CHARACTER*32)
104	**Unused**
105	Load case applied value (REAL)
106	Load case scaling factor (REAL)
107	Load case offset (REAL)
108-128	**Unused**

Record 5 (if NLOADS > 1)

1-128	Load case #3, etc. up to maximum of 200 load cases

Time step analyses have only one load record.

First Material record (=2 + NLOADS + 1)

WORD	CONTENTS
1	Material source (INTEGER): 0=Database local/central, 1=Database external, 2=Generated, 3=In FES file, 4=External ASCII file, 5=M/Vision, 6=No material (e.g. BS5400)
2	S-N analysis type(INTEGER): 0=Material based, 1=Component based, 2=BS5400 welded, 3=BS5400 non-welded
3-42	Database file name (CHARACTER*80)
43-58	Material name (CHARACTER*32)
59-74	Sub name (e.g. environment) (CHARACTER*32)
75	Analysis code parameter (INTEGER)
76	Type of material as per PFMAT codes (INTEGER)
77	UTS (REAL)
79	Young's modulus (REAL)
81	Elastic Poisson's Ratio (REAL)
83	Cutoff (REAL)
85	Standard error for generated data (REAL)
87-128	Space for 21 material params if option 3 (REALS)

Second block of info. for material 1

1	Surface finish ID (INTEGER)
2	Surface treatment ID (INTEGER)
3	Multiplier on stress (REAL)
5	Stress offset (REAL)
7	Fatigue limit reduction factor (REAL)
9	Plastic stress conc. factor (REAL)
11-44	**Unused**
45-84	Group name for ref. (CHARACTER*80)
85-128	**Unused**

This repeats, one group per 2 blocks up to 25 groups.

Data Records

These records start at the first record after the last material block.

WORD	CONTENTS
1	Node (INTEGER*4)
3	Element (INTEGER*4) Both may be present for node at element method.
5	Group number (material etc.) (INTEGER)
6-10	**Unused**
11	Temperature @node (REAL)
13-16	**Unused**
17	Minimum on mean axis (REAL)
19	Mean bin size (REAL)
21	Minimum on range axis (REAL)
23	Range bin size (REAL)
25	Stats are valid (1) or not present (0) (INTEGER)
27	Biaxiality ratio average (REAL)
29	Biaxiality standard deviation (REAL)
31	Most popular angle (REAL)
32	Angle spread (REAL)
33	Max. Stress Range (REAL)
34	Gate used in biaxiality analysis. (REAL)
35-48	**Unused**

49	Total number of cycles counted (INTEGER*4)
51	Pointer to next node/element block (INTEGER*4)

From 53 onwards are cycle bin descriptors. These consist of a X/Y bin location and a cycle count for the bin. Only non-zero bins are stored. If range bin=-1 then the records terminate. These records can spill over to a new data block or blocks.

53	Range bin (INTEGER)
54	Mean bin (INTEGER)
55	Cycle count for bin (INTEGER*4)
56-59	Next cycle, etc.

The Results Files (jobname.fef/fos)

The results from a MSC Fatigue analysis are written to a nodal or elemental results file(s) which is a standard Patran results text file. An example of a MSC Fatigue nodal output file (`jobname.fef/fos`) is shown in Figure 6-39. If elemental fatigue results are under consideration, then this file will be a Patran elemental output file with the same name. MSC Fatigue automatically determines which file format the `jobname.fef/fos` file is and compensates accordingly.

```
EVENTS UNITS ARE SET TO
    86     73  0.100000E+01     1     6
12     CYCLES

   730.7271189E-02-.2138395E+010.1375291E+030.2138395E+010.1375291E+03
0.2138395E+01
    10.1014736E-01-.1993647E+010.9854778E+020.1993647E+010.9854778E+02
0.1993647E+01
    20.1145218E-01-.1941112E+010.8731961E+020.1941112E+010.8731961E+02
0.1941112E+01
    30.1274424E-01-.1894686E+010.7846683E+020.1894686E+010.7846683E+02
0.1894686E+01
    40.1399659E-01-.1853978E+010.7144600E+020.1853978E+010.7144600E+02
0.1853978E+01
    50.1519045E-01-.1818429E+010.6583084E+020.1818429E+010.6583084E+02
0.1818429E+01
    60.1625638E-01-.1788976E+010.6151429E+020.1788976E+010.6151429E+02
0.1788976E+01
    70.1729698E-01-.1762030E+010.5781355E+020.1762030E+010.5781355E+02
0.1762030E+01
    80.1813516E-01-.1741479E+010.5514151E+020.1741479E+010.5514151E+02
0.1741479E+01
    90.1895612E-01-.1722251E+010.5275342E+020.1722251E+010.5275342E+02
0.1722251E+01
   100.1949572E-01-.1710061E+010.5129331E+020.1710061E+010.5129331E+02
0.1710061E+01
   110.2002132E-01-.1698507E+010.4994674E+020.1698507E+010.4994674E+02
0.1698507E+01
   120.2019943E-01-.1694661E+010.4950634E+020.1694661E+010.4950634E+02
0.1694661E+01
   130.2055908E-01-.1686996E+010.4864031E+020.1686996E+010.4864031E+02
```

Figure 6-39 A jobname.fef Fatigue Results File

In this example, there are six columns of data. A `jobname.fef` file can have either 6 or 12 columns of data depending on whether biaxiality parameters have been calculated. A jobname.fos file, from a factor of safety analysis, has only one column of data associated with it. The description of these columns is:

`jobname.fes` file:

Column 1	corresponds to *Damage*
Column 2	to *Log of Damage*
Column 3	to *Life in Repeats*
Column 4	to *Log of Life in Repeats*
Column 5	to *Life in User-Defined Units (cycles in this case)*
Column 6	to *Log of Life in User-Defined Units (cycles in this case)*
Column 7	to *Mean of the Biaxiality Ratio*
Column 8	to *Standard Deviation of the Biaxiality Ratio*
Column 9	to *Angle Spread*

Column10	to *Most Popular Angle*
Column 11	to *Maximum Principal Stress Range*
Column 12	to *Gate in Stress Units*

`jobname.fos` file:

Column 1	to *Factor of Safety*

For an explanation of the meaning of these results, see Postprocessing Results, 100.

This file can also be written out as a SDRC Universal file. To create fatigue results in Universal file format as opposed to the standard Patran results file format, set the keyword FEFTYPE to the value UNIVERSAL using the MSC Fatigue environment module MENM. See Modifying the MSC Fatigue Environment (MENM), 1463. Patran cannot read a Universal file.

The format of these Patran results file types is:

A Patran nodal results file is formatted as follows:

```
Record 1:TITLE(80A1)
Record 2:NNODES,MAXNOD,DEFMAX,NDMAX,NWIDTH(2I9 E15.6, 2I9)
Record 3:SUBTITLE1(80A1)
Record 4:SUBTITLE2(80A1)
Record 5 to n+4:NODID, (DATA(J), J=1, NWIDTH)(I8, (5E13.7))
```

Parameter	*Description*
TITLE	80A1 title stored in an 80 word real or integer array
SUBTITLE1	Same format as TITLE
SUBTITLE2	Same format as TITLE
NNODES	Number of nodes (integer)
MAXNOD	Highest node ID number (integer)
DEFMAX	Maximum absolute displacement (real)
NDMAX	ID of node where maximum displacement occurs (integer)
NWIDTH	Number of columns after NODID for nodal information (integer)
NODID	Node ID number (integer)
DATA	Result quantities organized by column index (real)

A Patran element results file is formatted as follows:

```
Record 1:TITLE(80A1)
Record 2:NWIDTH(I5)
Record 3:SUBTITLE1(80A1)
Record 4:  SUBTITLE2(80A1)
Record 5 To N+4:ID, NSHAPE, (DATA(J), J=1,NWIDTH)(2I8, /,
(6E13.7))
```

Parameter	*Description*
TITLE	80A1 Title Stored In An 80 Word Real Or Integer Array
SUBTITLE1	Same format as TITLE
SUBTITLE2	Same format as TITLE
NWIDTH	Number Of Columns Of Data Stored In The File (Integer)
ID	Element Identification Number (Integer)
NSHAPE	Essential Shape Code (Bar = 2, Tri = 3, Quad = 4, Tet = 5, Pyr = 6, Wedg = 7, Hex = 8; Integer)
DATA	Result Quantities Organized By Column Index (Real)

FEFAT Batch Operation

FEFAT Batch Operation for Preprocessor Option

FEFAT can be operated in batch mode also with the following keywords:

/OPT=P	To specify preprocessing option.
/INPut	The name of the input file, usually the jobname for the global multi-node/element analysis run.
/OUTput	The name of the file where the results will be stored. A default answer of the input file name will be offered (Note: the output file type is `jobname.fpp`).
/OVerwrite	Forces an overwrite of any output file with the same name as that specified by the user.
/TIMe history	The name of the loading time history for load case n. This is ignored if a transient analysis is used. If more than one time history is involved, use this keyword multiple times for each time history. They must be entered in the proper ascending order corresponding to the load case numbers.
/UNIts	The equivalent units for the analysis.
/NUNits	The number of equivalent units.
/SIZE	The size of the rainflow matrix created by FEFAT may be specified. The size may be 32, 64 or 128. The high the number, the more accurate the results, however computational speed is the price paid. The default size is 32.
/STIM	If a single node or element has been indicated for processing, the ability to output a stress time history exists. Specify YES or NO with this parameter. If YES, a `jobnamenn.dac` file will be created which can be viewed in PTIME, where nn is the node/element number.
/ANAlysis	Answering YES here will cause the fatigue analyzer of FEFAT, to start immediately after the preprocessing has completed.
/*=TT	If the user wishes to see output sent to the screen, he must include this parameter exactly as shown. Any value other than TT after the equal sign will send the output to a file by that name.

Example:

```
fefat /opt=p/inp=mymodel/ov=yes/tim=load1/anal=yes
```

This batch line would open `mymodel.fes` and use a time history called `load1.dac`. Any files existing with the same name as the output files created would be overwritten. The FE-Fatigue Analyzer option of FEFAT, is automatically started after termination of the preprocessor.

FEFAT Batch Operation for Analysis

If the fatigue analysis is initiated from the preprocessor option, these keywords will not be required.

/OPT=F	To specify fatigue analysis option.
/INPut	The name of the input preprocessed data file, usually the jobname for the global multi-node/element analysis run.
/OUTput	The name of the file where the results will be stored. A default answer of the input file name will be offered. (Note that the output file type is `.fef`.)
/OVerwrite	Forces an overwrite of any output file with the same name as that specified by the user.
/*=TT	If the user wishes to see output sent to the screen, he must include this parameter exactly as shown. Any value other than TT after the equal sign will send the output to a file by that name.

Example:

```
fefat /opt=f/inp=mymodel/ov=yes
```

FEFAT Batch Operation for Factor of Safety

Most of the batch keywords are presented here; however, not all of them are described. To find the appropriate batch keyword for the operation(s) enter FEFAT and use the Help of Question available under Help pull-down menu with the field of interest highlighted or active to find out what the batch keyword is. At the end of each help page that is displayed, is the keyword for that option.

/OPT=S	To specify factor of safety analysis option.
/JOBname=	This is the name of the job to run (/JOB=myjob)
/LOCation=	This is the node or element number. Can be "ALL". (/LOC=13)
/ANALysis=	This is the analysis type: Stress or Life-based. (/ANAL=S)
/LIFE=	The design life. (/LIFE=1000)
/MAXFAC=	The maximum factor. (/MAXFAC=100)
/USECUT=	Use material cutoff (YES/NO). (/USECUT=Y)
/OUTput=	The output file name (`jobname.fos`). (/OUT=myjob)
/STRESS=	The reference stress. (/STRESS=50)
/OV=	Overwrite flag if an existing file is to be overwritten. (/OV=Y)

Example:

```
fefat /opt=s/job=myjob/anal=l/loc=all/out=myjob/ov=y
/life=1000/maxfac=8/usecut=y
```

This example will run FEFAT in batch and will use the file `myjob.fpp` in a life-based factor of safety analysis with the design life set to 1000, the maximum factor allowed set at 100 and will use the material cutoff in the calculation. The results will be written to the file `myjob.fos` and, if it exists, will be overwritten.

FEFAT Batch Operation for Utilities

/OPT=U	To specify utility option.
/UOPTion=	The options are A, B, L, C, V, and R corresponding to the submenu picks under the utilities menu.
/OUTput=	Output file name. Only the jobname need be input. A file suffix of ASC or FES is assumed depending on the main menu option. The default is the input jobname.
/INPut=	The input file name. This is usually the jobname. A file suffix of ASC or FES is assumed depending on the main menu option.
/OVerwrite=	This parameter sets automatic overwrite of existing files on or off. /OV=Y sets overwrite on. /OV=N sets overwrite off. Off is the default.
/*=TT	If it is desirable to see output sent to the screen this parameter can be included exactly as shown. Any value other than TT after the equal sign will send output to a file by that name.

Example:

```
fefat /opt=u/uopt=a/inp=mymodel/out=mymodel/ov=y/*=tt
```

This example selects the utility option in FEFAT and executes the submenu option of creating an ASCII file from a binary file (A), with a jobname of mymodel. It will search for a file called `mymodel.fes` and create an output file called `mymodel.asc`.

7 Multiaxial Fatigue

- Introduction 422
- Multiaxial Factor of Safety (dtfat) 424
- Local Multiaxial Stress/Strain Fatigue Analyzer (MMLF) 438
- Multiaxial Fatigue Theory 481

Introduction

This chapter is dedicated to multiaxial concerns. The stress state in a component and how that stress state changes over time tends to have a great influence on how a crack grows or initiates. The basic fatigue techniques in MSC Fatigue are based on uniaxial stress state assumptions. It is recommended that the user be intimately familiar with these techniques before attempting to solve fatigue problems of structures with multiaxial stress states. The basic uniaxial techniques and usage of the program with these techniques are described in Using MSC Fatigue - The Classic Modules (Ch. 3), Total Life and Crack Initiation (Ch. 6), and Fatigue Theory (Ch. 16).

Two modules are provided in MSC Fatigue for multiaxial fatigue calculations.

1. The first is based on global, multi-location analysis from finite element results described in Multiaxial Factor of Safety (dtfat), 424.
2. The second is a single location analyzer and calculates fatigue life from strain gauge rosette measurements described in Local Multiaxial Stress/Strain Fatigue Analyzer (MMLF), 438.

The real difference between the two is that the first uses a multiaxial notch correction procedure to convert purely elastic FE results into elastic-plastic stresses and strains. This is not necessary when dealing directly with measured responses.

For the FE method, it is not recommended to simply use multiaxial techniques immediately without first running a basic analysis based on uniaxial assumptions along with a multiaxial assessment by running a biaxiality analysis. A multiaxial analysis of an entire model can be a computationally intensive operation and thus should only be resorted to if necessary and then again, only for the critical locations of a model.

MSC Fatigue allows the user, during a basic fatigue analysis, to do a biaxiality analysis. This is described in Using MSC Fatigue - The Classic Modules (Ch. 3) in detail. To summarize here, a biaxiality analysis will give you a good understanding of the stress state in your model and how that stress state varies with time. It uses surface resolved stresses (the state of stress on the surface of the model must be plane stress meaning the out-of-plane stress, σ_z, must be zero). It then uses the two in-plane principal stresses to determine the biaxiality ratio,

$a_e = \sigma_2/\sigma_1$, where σ_1 is the absolute maximum in-plane principal stress and σ_2 is the other in-plane principal. This parameter can take on values between minus one (pure shear) and plus one (fully biaxial).

A biaxiality analysis also will determine the standard deviation of the biaxiality ratio and the angle spread, that is, how the biaxiality ratio is changing over time and whether or not the angle, ϕ, that σ_1 makes with the local x-axis is changing. With these three parameters the stress state can be determined using the following guidelines:

Value of a_e	Value of f	Stress State
0	Constant	Uniaxial
$-1<=a_e<=+1$ Constant but not zero	Constant	Proportional Loading
Varies	Varies	Non-proportional Loading

Uniaxial stress states can be fully handled by the basic techniques of MSC Fatigue as previously stated.

Proportional loading means that you no longer have a uniaxial stress state but that the relative magnitude of σ_2 to σ_1 is not changing with time, i.e. they remain proportional to each other. This case can also be fully handled with the basic techniques of MSC Fatigue by setting the biaxiality correction on and possibly using a signed tresca shear stress parameter as opposed to using the default maximum absolute principal.

Only when the third state of the above table actually exists (non-proportional loading) do the techniques described in this chapter need to be resorted to. To summarize the approach to assess for multiaxiality:

1. Calculate surface resolved stresses. For shell elements this is normally not necessary in that the stresses are already surface resolved. For solid elements this can be done using the Calculate Normals, 94 option. Also see the discussion below in
2. Run a global crack initiation or S-N fatigue analysis using the basic methods with the biaxiality analysis turned on to determine the fatigue critical locations and to calculate the multiaxial assessment parameters such as a_e.
3. Use the post-processing techniques in Patran or MSC Fatigue to evaluate the mean biaxiality ratio, standard deviation of the mean biaxiality ratio, and the angle spread at the critical locations of the model as previously described. Cross plots of the stress parameter versus these multiaxial assessment parameters can also be created at critical locations over the entire time series to help with the assessment.
4. If uniaxial conditions exist, no further steps are necessary.
5. If a proportional loading state exists in the critical locations, correct by turning on the biaxiality correction using the basic analysis.
6. If a non-proportional loading exists in the critical locations, you must resort to the techniques described in this chapter.

Surface Resolved Stresses from FE Models

Many of the multiaxial models used in the module FEMLF are applicable on free surfaces only; in fact all do except the Dang Van model, which does not care. These models require surface resolved stresses and 2-D stress states. When working with shell models, this is not a problem because the stresses are normally already plane stress in a surface resolved coordinate system, with zz, xz and yz stress components being zero. The problem arises when dealing with solid models, where the stresses are surface resolved, but in general, due to the errors in the finite element model, especially in extrapolating stresses to the free surface, the zz, xz and yz stress components will not be zero. There are various possible strategies for dealing with this issue.

1. The first option, and this is the one currently being used, is to ignore the zz, yz and zx stresses. This method has the advantage of simplicity, and is of course correct if the stresses really are 2D. However, in a FE solid model, you would generally expect the displacements of the surface nodes and hence the surface strains to be calculated relatively accurately, and the surface stresses to be less accurate, due to the inaccuracy in extrapolation to the surface. If you calculate the surface strains from the stresses, the accuracy of these will also be compromised.

2. The second option is, when calculating stresses and strains on any plane, always to consider the whole stress tensor. Because the stresses are in any case calculated by the FE from the strains, this approach should give accurate values of the in-plane strains. However, the stresses, and also the out-of-plane strains will have errors due to the non-zero surface normal components, which cannot be present in practice.
3. The third option is to use the whole stress tensor to calculate the in-plane strains, and this will give the same strain values as predicted by the original FE calculation.

$$\varepsilon_x = \frac{\sigma_x}{E} + \frac{\upsilon}{E}(\sigma_y + \sigma_z) \tag{7-1}$$

$$\varepsilon_y = \frac{\sigma_y}{E} + \frac{\upsilon}{E}(\sigma_z + \sigma_x) \tag{7-2}$$

$$\gamma_{xy} = \frac{\tau_{xy}}{G}, \text{ where } G = \frac{E}{2(1+\upsilon)} \tag{7-3}$$

The surface normal strain and direct in-plane stresses could then be recalculated from Hooke's law assuming all the components containing z are zero, thus:

$$\varepsilon_{zz} = -\upsilon(\varepsilon_{xx} + \varepsilon_{yy}) \tag{7-4}$$

$$\sigma_{xx} = \frac{\upsilon E}{(1+\upsilon)(1-2\upsilon)}\{\varepsilon_{xx} + \varepsilon_{yy} + \varepsilon_{zz}\} + \frac{E}{1+\upsilon}\varepsilon_{xx} \tag{7-5}$$

$$\sigma_{yy} = \frac{\upsilon E}{(1+\upsilon)(1-2\upsilon)}\{\varepsilon_{xx} + \varepsilon_{yy} + \varepsilon_{zz}\} + \frac{E}{1+\upsilon}\varepsilon_{yy} \tag{7-6}$$

$$\sigma_{zz} = 0 \tag{7-7}$$

This is generally the best method, because:

1. The in-plane strains are always correct.
2. The surface normal stress is zero.
3. The stresses and strains are consistent with one another.

See Multiaxial Low Cycle Fatigue Analysis, 503 in the Theory section for more details.

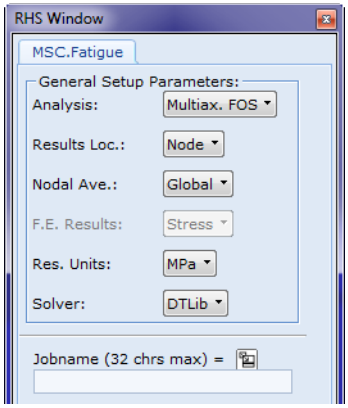
Multiaxial Factor of Safety (dtfat)

In real service conditions, many structures experience multiaxial loadings in critical locations. That is, the stress state in the critical location is characterized as having more than one significant principal stress, and/or the principal stress directions change with time. Methods are available in MSC Fatigue for investigating stress states and making finite life fatigue calculations in multiaxial conditions. However, some components, e.g., engine parts such as connecting rods and crankshafts, are expected to survive a high number of critical loading cycles during their lifetime. It is not realistic to design such components for a finite fatigue life. It is more usual to use a safety factor approach in which the critical loading cycles are compared to a fatigue or endurance limit criterion. The simple uniaxial safety factor approach is fine for many cases, but where the loading is multiaxial, and especially non-proportional, a more sophisticated method, such as the Dang Van

model, is required. The classic solvers support both the Dang Van and McDiarmid methods and this section briefly describes the Dang Van Criterion and changes to the UI for the DTLib based solution.

The Dang Van criterion aims to deal with high-cycle fatigue conditions where the damage occurs at a microscopic level. The formulation uses the current local values of shear stress (t) and hydrostatic pressure p(t), associated with a shakedown state, to calculate an equivalent stress and compares it to a threshold: no damage will occur if $\tau(t) + a \cdot p(t) \leq b$ at any time t in the stabilized cycle where a and b are material parameters calculated from tests on smooth and notched specimen subjected to proportional and non-proportional stress states that characterize the Torsional Fatigue endurance limit and the hydrostatic stress sensitivity factor. For further details on the Dang Van method, please refer to the *MSC Fatigue Theory Guide*.

On the main MSC Fatigue form, set **Solver** to **DTLib** and **Analysis** to **Multiax FOS**. Results Location, Units, etc., can be set as needed.



The Solution Parameters form allows the Dang Van FOS analysis settings to be adjusted for speed and accuracy. The form is shown below, along with a table describing each of the settings.

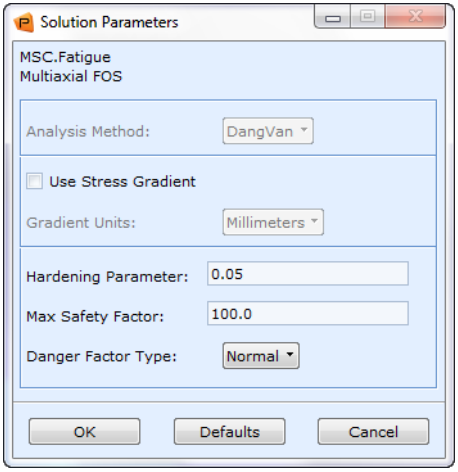


Table 7-1 Solution Parameters

Parameter	Options	Description	Default
Analysis Method		This is currently the only choice.	**Dang Van**
Use Stress Gradient	**Enabled, Disabled**	This option allows to apply a stress correction based on the gradient in stress in the direction normal to the free surface. Only applies to surface resolved stresses on the free surface. The stress gradient correction is based on gradients computed in MSC Fatigue.	**Disabled**
Gradient Units	**Millimeters, Meters, Inches, Feet**	Length units for stress gradients.	**Millimeters**
Hardening Parameter	1E-5 < **Value**	This parameter controls the rate of isotropic hardening, as described above. Decreasing the value of the hardening parameter will reduce the rate of growth of the yield surface used to determine shakedown. This will lead to slower calculation, but increased accuracy.	**0.05**
Max Safety Factor	**Value**	For convenience when viewing and plotting results, this property allows an upper limit to be set on the Safety Factor.	**100**

Table 7-1 Solution Parameters

Parameter	Options	Description	Default
Danger Factor Type		Determines whether the “Normal” or “Oblique” formulation of the Danger Factor is to be calculated.	**Normal**
	Normal		
	Oblique		

Global Multiaxial Fatigue Life Analyzer

This chapter on multiaxial life prediction consists mainly of a description of the multiaxial fatigue analysis module FEMLF. This module can be accessed from Analysis on the Main Fatigue form (Analysis > Multiaxial FOS or Multiaxial CI).

This analysis module allows fatigue life predictions and safety factor calculations to be made, even when the loadings are complex and multiaxial, including both proportional and non-proportional loadings. It should be kept in mind that if you are dealing with proportional or near proportional loadings, you may find the standard MSC Fatigue options give sufficiently good results.

The programs normally used during the execution of a multiaxial fatigue analysis are as follows:

1. Data Preparation

PFMAT	Materials Database Manager and BS5400 Weld Classification Advisor
PTIME	Time History Database Manager and ASCII Time History File Convertor
PVXMUL	A Peak-Valley Extraction Program for Reducing Lengthy Time Histories
MMFD	A Multi-file Display Program

2. Fatigue and Factor of Safety Analysis

PAT3FAT	Model database (Patran) to Fatigue Input Translator
FATTRANS	The new model database (Patran) to Fatigue Input Translator
FEMLF	Multiaxial Fatigue and Safety Factor Analyzer

3. General Utilities

FEFAT	FES File ASCII/Binary Convertor
PFTRM	Terminal Driver
MCONFIL	Binary to Binary File Convertor
FASTAN	Utility to run a peak-valley extraction and subsequent analysis to speed up execution time

4. Results Postprocessing

MPOD	Polar damage display module
MP3D	3-D Histogram display module
MTPD	X-Y plotting
PFPOST	Global Results Listing
MMFD	Multi-File Display

Analysis Route

The actual programs necessary to complete a global multi-node or element crack initiation analysis are:

PAT3FAT	Translator (creates the fatigue input file filename.fes)
FATTRANS	New Translator (creates the fatigue input file filename.fes)
FEMLF	Multiaxial fatigue and factor of safety analyzer

Most of the options in this module require surface resolved stresses, i.e. results either from thin shells or using the surface normal extract option for solid models. The exception is the Dang Van method which accepts 2-D or 3-D stresses.

The results may be reviewed using PFPOST, by reading them back into the pre-/postprocessor for postprocessing, or by using the graphical options in FEMLF.

In summary, to run a multiaxial fatigue or safety factor analysis:

1. Select the analysis option (Multiaxial FOS or Multiaxial CI). Make sure that, if you are using a solid model, you first run the surface normal extract option from the job control menu (this step is not necessary if either you already have 2-D stresses or you are running a Dang Van analysis).
2. Select Solution parameters and fill in the required fields for the analysis Method selected. The input forms are different depending on the analysis option selected above.
3. Select Materials Info and fill in the required fields for the material form. Limited true multiaxial (FOS and CI) materials data is available for Dang Van, Fatemie-Socie and Wang Brown respectively. Users may add multiaxial data with reference to the rules in Problem 4: Ten Simple Notched Geometries (Ch. 15).
4. Select Load information and fill out the loading form.
5. Select Job control and submit the analysis.
6. Post-process the results using FEMLF, PFPOST and by reading the results back into the pre-/postprocessor (MSC Fatigue or Patran).

The analysis should produce a `jobname.fef` or `jobname.fos` result files.

Multiaxial Fatigue Analysis Options

The Multiaxial Analysis options are selected from the main MSC Fatigue form as shown in Figure 7-1.

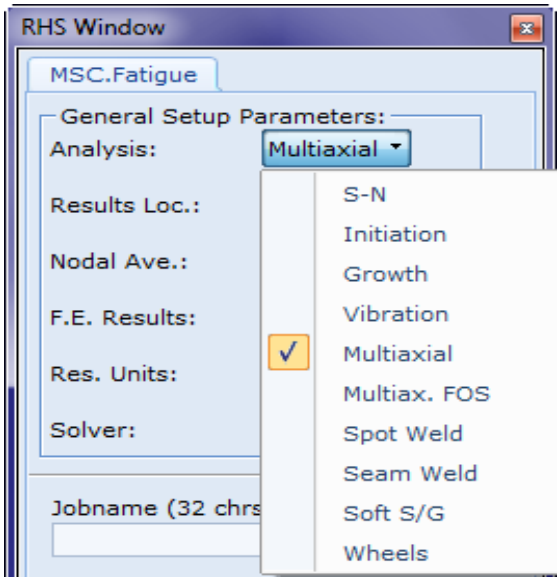


Figure 7-1 MSC Fatigue Main form

Safety Factor Analysis

On the Solution Parameters form select the Analysis method. If Dang Van is selected the Users is required to input a hardening parameter.

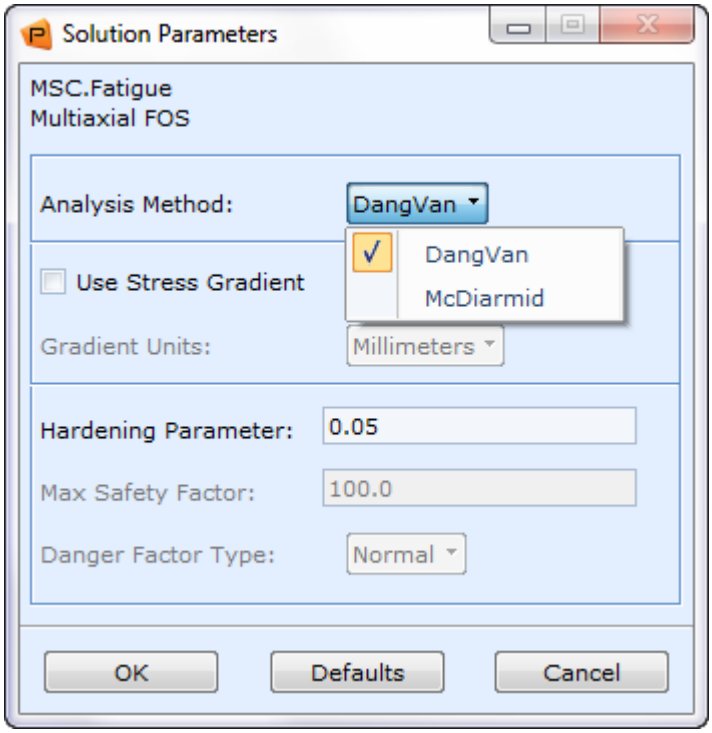


Figure 7-2 Safety Factor Analysis Solution Parameters Form

Dang Van Method

The Dang Van method is a complex method which defines the endurance limit in terms of the microscopic shear stress amplitude and the hydrostatic stress. For details of the method, see the theory on Dang Van Method, 430 in this chapter. The required parameters are the Torsional Fatigue Endurance (or fatigue limit) and the hydrostatic stress sensitivity factor. The Dang Van fatigue limit criterion says that:

$$\tau + a*ph <= t \quad (7\text{-}8)$$

where τ is the microscopic shear stress, a is the hydrostatic stress sensitivity, ph is the hydrostatic stress and t is the torsional fatigue limit (fatigue or endurance limit in a pure torsion fatigue test). You may not have access to parameters a and t. However, these may reasonably be estimated if you have some uniaxial data available. For instance, if you know the bending fatigue limit f, t can be estimated from t = 0.577 * f. (For most materials the factor is somewhere between 0.5 and 0.6). The bending fatigue limit may be around 1.25 times the uniaxial tension-compression fatigue limit.

Now, consider the conditions at the bending fatigue limit, where the shear stress amplitude τ will be 0.5 * f and the hydrostatic stress amplitude f/3. We can therefore rewrite equation (7-8).

$$0.5 * f + a * f/3 = 0.577 * f \quad (7\text{-}9)$$

which gives a = 0.23.

The hardening parameter has a default value of 0.05. The Dang Van method is an iterative method which converges to a solution. The hardening parameter determines the rate at which convergence occurs. A value of 0.05 is a reasonable compromise between accuracy and speed.

When this form is complete, click **OK**, complete the Materials and Loading form, and submit the job from the job control menu. Processing will begin and progress can be monitored using the monitor job function under job control.

When the job is complete a listing of the ten worst safety factors is presented.

If you have only selected one node or element for analysis, the program will show you the following graphical output:

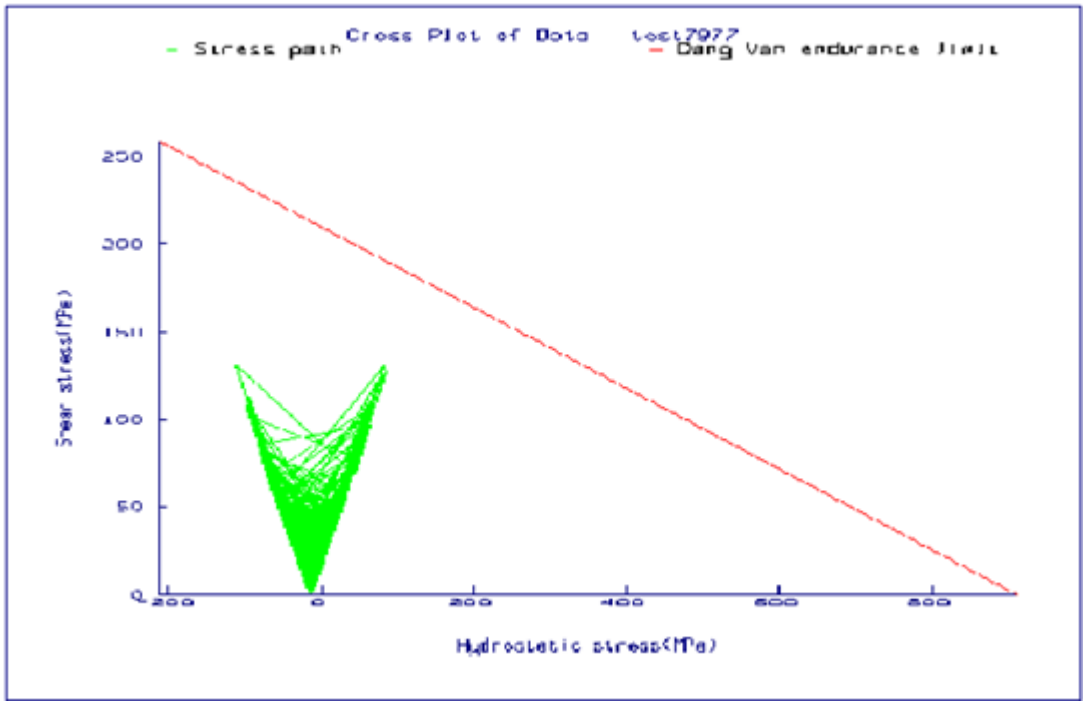


Figure 7-3 Dang Van Plot

It is converted when using the Dang Van Criterion to define a safety factor which may be used to compare the worst loading condition to the fatigue limit criterion.

The definition of safety factor used in FE-Fatigue is:

$$\text{Safety factor} = b/(\text{Max}(\tau_{max}) + a.ph)$$

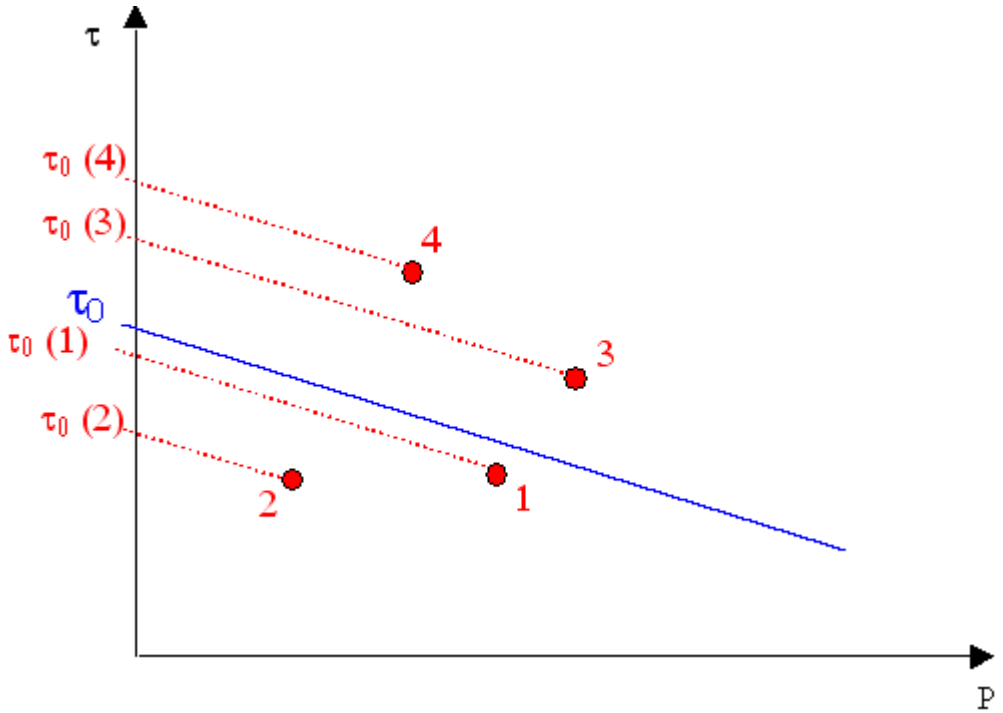
except when the denominator is less than or equal to zero, when the life is predicted to be infinite, and the safety factor is effectively infinite. For the purposes of contour plotting, an upper limit for safety factors is used in results files, with the default being 100, but which can be set as a preference.

In addition to the safety factor, the following parameters are calculated and written to the results file:

τ_{crit} - the value of the microscopic shear stress corresponding to the critical time-history point at which the safety factor is determined

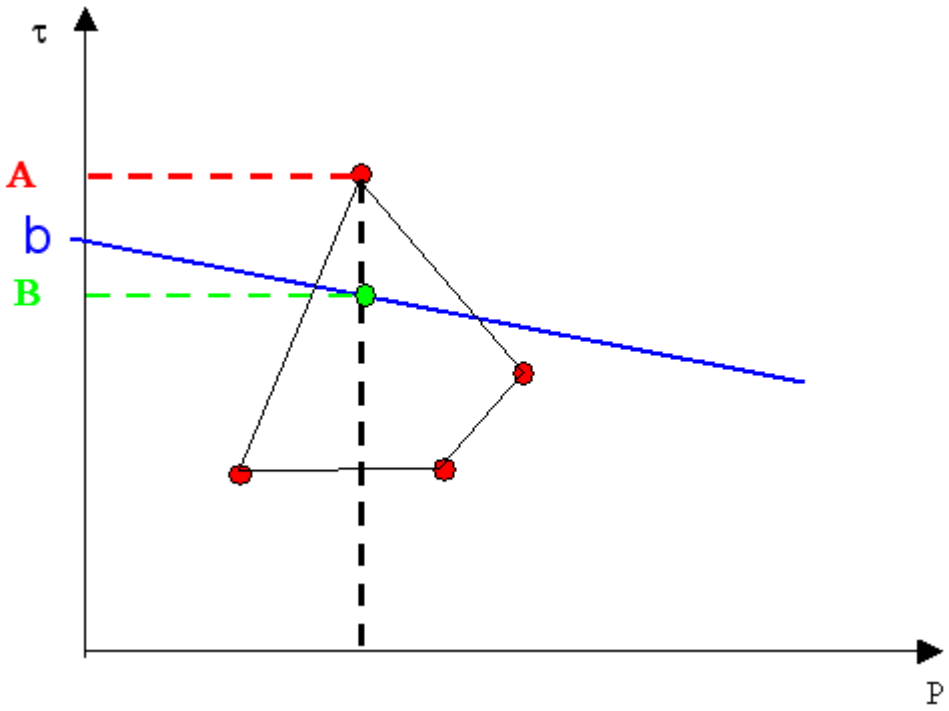
P_{crit} - the value of the hydrostatic stress corresponding to the critical time-history point at which the safety factor is determined

τ_0 - the projection of the most critical point on the Dang Van diagram onto the t axis, parallel to the Dang Van criterion line. $\tau_0 = \tau_{crit} + a*P_{crit}$



DF - the "Danger Factor", defined as (A-B)/B in the following figure, i.e.:

$$DF = \frac{(\tau_{crit} - b + a*P_{crit})}{(b - a*P_{crit})}$$



McDiarmid Method

If the McDiarmid Method is selected no further input is required on the Solution control form. The only requirement is that stresses be surface resolved.

The McDiarmid criterion can be expressed.

$$r_a/t_{A,B}+\sigma_{n,max}/2\sigma_T = 1 \quad (7\text{-}10)$$

Under uniaxial conditions at the fatigue limit, $\sigma_{n,max} = r_a = 0.5*f$ so if the method is to give the correct answer for uniaxial conditions.

$$t_{A,B} = f/(2\text{-}f/2*UTS) \quad (7\text{-}11)$$

By default, the UTS of the material is taken to be that of the material mentioned in the `.fes` file, which can be different for different groups in the job. When the analysis is complete, a form lists the 10 worst safety factors.

Crack Initiation Life Analysis

When Multiaxial Crack Initiation Life Analysis option is selected from the main form Figure 7-1, the Solution Control lists the available analysis methods. Select the desired analysis method, keeping in mind that the first 4 methods (Normal Strain, SWT-Bannatine, Shear Stress, Fatamie-Socie) are critical plane methods while the

Wang Brown method employs a multiaxial rainflow counting algorithm. An explanation of the methods is given below. Mutiaxial properties that have the correct Fatemie-Socie and Wang Brown parameters are available in the materials database parameter for a few materials. Users may add multiaxial data with reference to the rules in Problem 4: Ten Simple Notched Geometries (Ch. 15).

The cyclic plasticity model, necessary for the notch correction procedure, is the Glinka model.

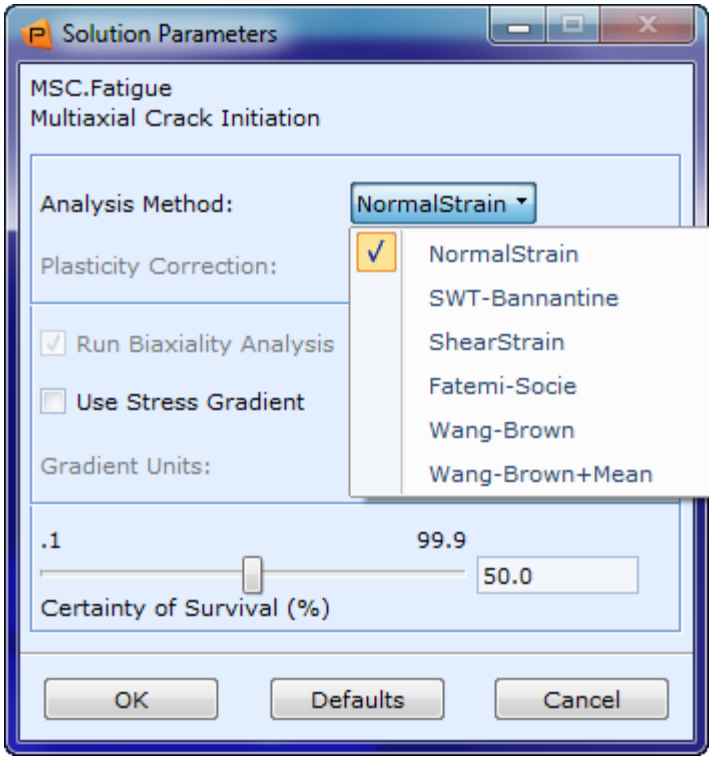


Figure 7-4 Solution Control Form - Multiaxial CI

Complete the Materials and Loading forms as per the other solution sequences and submit the Job.

Multiaxial Methods

Options 1 to 4 are conventional critical plane models which resolve stress and strain components onto particular planes. Options 1 and 2 make multiple fatigue calculations on planes inclined at 90 degrees to the free surface (X-Y plane) at 10 degree intervals, making 18 planes in all. Options 3 and 4, which are based on shear strain, also consider planes inclined at 45 degrees to the surface, making 36 planes in all. The critical plane is considered to be the one with the largest value of predicted damage. The equations of these methods are as follows:

1. Normal Strain. Uses the normal strain on each plane.

$$\frac{\Delta\varepsilon_n}{2} = \frac{\sigma_f'}{E}(2N_f)^b + \varepsilon_f'(2N_f)^c \tag{7-12}$$

2. SWT-Bannantine. This is the Socie-Bannantine "Tensile" model

$$\frac{\Delta\varepsilon_n}{2}\sigma_{n,\,max} = \frac{\sigma_f'^2}{E}(2N_f)^{2b} + \sigma_f'\varepsilon_f'(2N_f)^{b+c} \tag{7-13}$$

3. Shear Stress only

$$\frac{\Delta\gamma}{2} = \frac{(1+\nu_e)\sigma_f'}{E}(2N_f)^b + (1+\nu_p)\varepsilon_f'(2N_f)^c \tag{7-14}$$

4. Fatemi-Socie. This is the Socie-Bannantine "Shear" model, formulated to use uniaxial parameters. The parameter "n" is a material property, usually between 0.4 and 0.6 which dictates the relative importance of the normal stress and the shear strain. The default value of n is 0.6. Note that the Fatemi-Socie method also requires that a yield stress be defined in the material data entry. If this is not defined, an error will occur.

$$\frac{\Delta\gamma}{2}\left(1+n\frac{\sigma_{n,\,max}}{\sigma_y}\right) = \frac{(1+\nu_e)}{E}\sigma_f'(2N_f)^b\left(1+\frac{n\sigma_f'(2N_f)^b}{2\sigma_y}\right) + (1+\nu_p)\varepsilon_f'(2N_f)^c\left(1+\frac{n\sigma_f'(2N_f)^b}{2\sigma_y}\right) \tag{7-15}$$

The last two methods are the Wang-Brown method, with and without mean stress correction. These are probably the best and most sophisticated of the methods, but this is to some extent a matter of opinion. The Wang-Brown method consists of a multiaxial reversal counting method, that uniquely defines rainflow reversals for non-proportional loadings, and a damage model that uniquely identifies a critical plane for each reversal. The total damage is accumulated from many different planes.

5. Wang Brown method. This requires an additional material parameter S, the Brown-Miller parameter which describes the relative importance of the normal and shear strain on the critical plane for Case A loadings. The default is 1 which will generally give a somewhat conservative prediction. S can be determined from the results of uniaxial and torsional tests, if available. If the loading is uniaxial, this method should give the same result as a normal uniaxial calculation using the Coffin-Manson-Basquin equation.

$$\hat{\varepsilon} \equiv \frac{\gamma_{max} + S\delta\varepsilon_n}{1+\nu' + S(1-\nu')} = \frac{\sigma_f'}{E}(2N_f)^b + \varepsilon_f'(2N_f)^c \tag{7-16}$$

6. Wang-Brown method including mean stress term. The mean stress term is the mean normal stress on the maximum shear plane during each reversal. If this method is used for a uniaxial loading it reduces to the Morrow method.

$$\hat{\varepsilon} \equiv \frac{\gamma_{max} + S\delta\varepsilon_n}{1+\nu' + S(1-\nu')} = \frac{\sigma_f' - 2\sigma_{n,\,mean}}{E}(2N_f)^b + \varepsilon_f'(2N_f)^c \tag{7-17}$$

Multiaxial fatigue is a current research area, so there is still a lot of discussion about what the best methods are, and when they are applicable, and new models are still emerging. We have attempted to provide a sample

of what we consider to be the better models. We will also attempt to provide some guidelines as to which models should be used when.

- Methods 1 and 3 are included for completeness only and are not really recommended.
- The SWT-Bannantine method is recommended for cases which are dominated by tensile cracking. Tensile cracking is promoted by tensile loadings, high mean stresses, positive biaxiality ratios, long lives and brittle materials. If the method is applied to cases where the cracking mode is predominantly shear, non-conservative results may be obtained.
- The Fatemi-Socie model is recommended for cases which are dominated by shear cracking. Shear cracking is promoted by shear loadings (or negative biaxialities), compressive mean stresses, short lives and ductile materials.
- The Wang-Brown methods, work for negative and positive biaxiality ratios, and also tend to give better, less non-conservative predictions, for non-proportional loadings. They may be slower than the other methods if long and complex loading histories are calculated, but faster for short ones. The method may be rather conservative for longer life problems, though there is not much evidence to support this.

The next question on the form is the equivalent units. By default, the software will give a fatigue life prediction in Repeats of the loading history. If some other units are defined or desired, enter the number of units in one repeat of the history and the name of the units. FEMLF will then give an answer in the equivalent units.

The next two fields are only used for the Fatemi-Socie method and the Wang-Brown method. Sensible defaults are provided if this information is not available. Apart from these parameters, the only other material information required for these methods is the normal uniaxial cyclic parameters.

When this form is complete, click **OK** to initiate the job. When the job is complete, a form will appear, summarizing the result for the (up to) ten most damaged nodes.

Reviewing the Results

When initial calculations are complete, the results may be reviewed in the following ways:

1. Using PFPOST. The operation of PFPOST is exactly the same as for S-N, Factor of Safety and Crack Initiation Calculations using the standard MSC Fatigue modules.
2. Using Patran (or other postprocessors). The results of calculations may be read back into Patran and Fringe plots of Fatigue Life, Damage and Safety factors created, in exactly the same way as for S-N, Factor of Safety and Crack Initiation Calculations using the standard MSC Fatigue modules.
3. For Dang Van Safety Factor calculations and single node or element calculations only, the results may be viewed as a Dang Van plot. This is described in Dang Van Method, 430 above.

4. When a crack initiation calculation is carried out, the Crack Initiation Summary Results Form appears. If you press End, the following results Display menu appears:

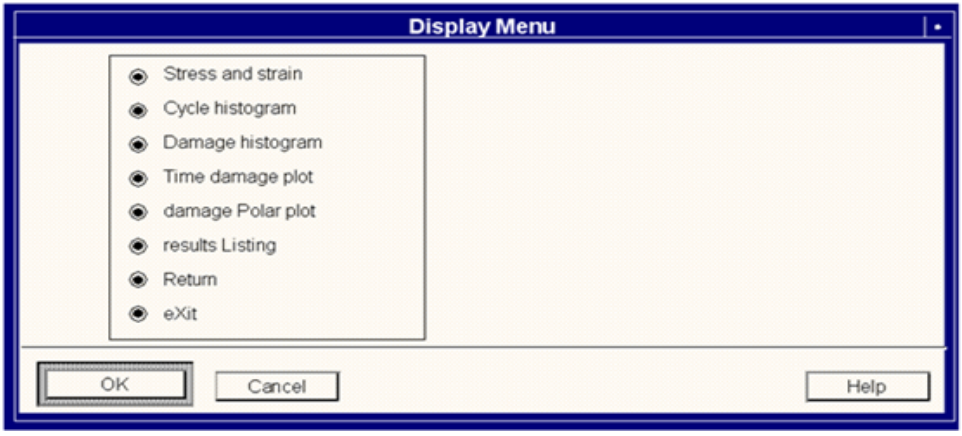


Figure 7-5 Crack Initiation Results Display Form

The options on this form have the following function:

Option	Description
Stress and Strain	This option displays a multi-file display of the 3 surface stress components, σ_{xx}, σ_{yy} and σ_{xy} and the 4 strain components, ε_{xx}, ε_{yy}, γ_{xy} and ε_{zz}. The files are called: `<jobname><node or elem id>.sgx`, `.sgy`, `.txy`, `.epx`, `.epy`, `.gxy` and `.epz`.
Cycle histogram	This option displays a cycle or reversal histogram. The nature of the histogram depends on the damage model used. Histograms are as follows: Normal Strain Range - Mean - Cycles SWT-BannantineRange - Max. Stress - Cycles Shear StrainRange - Mean - Cycles Fatemi-SocieRange - Max. Normal Stress - Cycles Wang-BrownShear Range - Normal Range - Reversals
Damage Histogram	This option displays a damage histogram.
Time Damage Plot	This displays a time correlated damage plot - a plot of the damage plotted against time. It indicates the parts of the loading that are causing the most damage, and is a useful tool in support of fatigue editing. By seeing which parts of the loading cause most damage, the load histories may be edited for instance to generate more efficient rig drive signals which cause the same fatigue damage but in less time. Fatigue editing tools are available from nCode International, the product name being FATIMAS-EDIT.

Option	Description
Polar Damage Plot	The Polar Damage Plot is a polar display of calculated fatigue damage vs. orientation (relative to the local x-axis). For definition of the orientation, refer to the theory guide. For calculation methods 1-4, the damage plotted is the actual damage at each 10 degree angle increment. For methods 5 and 6 the damage values are the sum of damage in 10 degree angle bins. In addition, for methods 2, 4, 5 and 6, there may be two curves on the polar plot relating to Case A and Case B planes. See the documentation on the Polar Display Module MPOD for further details in Polar Display (MPOD), 820.
Results Listing	Repeats the summary results listing.
Return	Returns to the Main menu
Exit	Exits the program

FEMLF Batch Operation

The program FEMLF may be operated in Batch mode. The batch keywords and their meanings are as follows:

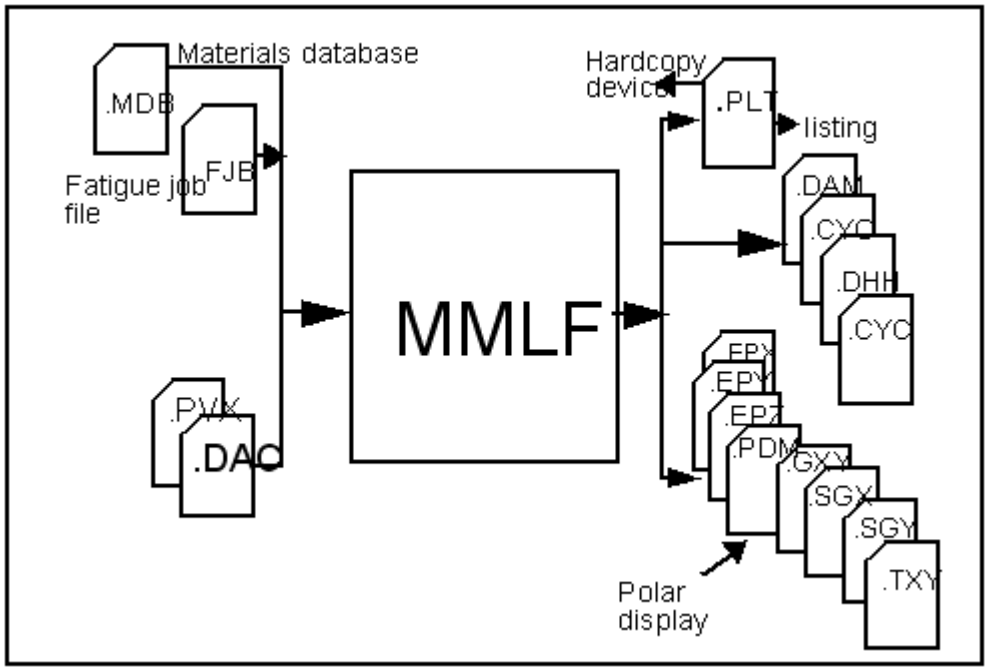
Batch Keyword Summary:	
/OPTion	Main menu option selection: S,C,U,X
/JOB	Job name (FES File)
/OUTput	Output file name
/DAMMOD	The damage model (N,B,S,F,W,M)
/ANAREG	Selected node/element to process nod/el no. or “all”
/LOCations	List of nodes or elements to analyze
/LOPT	Single node/element options

Example:

```
femlf /opt=s/inp=jobname/out=y/dammod=N/anareg=all
```

Local Multiaxial Stress/Strain Fatigue Analyzer (MMLF)

MMLF calculates crack initiation life from data files collected by rosette strain gauges.



MMLF incorporates a Mròz-Garud cyclic plasticity model, together with a generalized form of the Wang-Brown multiaxial rainflow and damage accumulation procedures. The program also incorporates a variety of other critical plane damage models. The program requires measured strains from strain gauge rosettes as input, together with uniaxial cyclic material properties.

The software can handle general non-proportional biaxial loadings.

Summary of Method

Uniaxial methods for life prediction using the local strain approach have been in use for some time, having their roots in the work of Basquin (Ref. 52), Manson (Ref. 53), and Coffin (Ref. 54)(Ref. 55), incorporating rainflow cycle counting and material memory and Miners rule (Ref. 57). Such methods are available in MSC Fatigue. Within the well known limitations of these methods they work quite well for a variety of components where the local loading in the critical area is uniaxial or near-uniaxial. This class of components includes many that are subject to complex multiaxial loading environments (Ref. 49)(Ref. 50). However, there are many other components where a combination of loads and geometric effects generates local loadings which are proportional or non-proportional multiaxial.

The life prediction process from measured strains can be divided into two steps. The first step is to determine the relationship between the measured strains and all the stress and strain components required for the damage calculation, through application of a cyclic plasticity model. The second step is to carry out cycle and damage accumulation. MMLF addresses problems where the strains can be measured with a rosette, i.e. biaxial loading on a free surface. For these problems the process can be summarized by the flow chart in Figure 7-6:

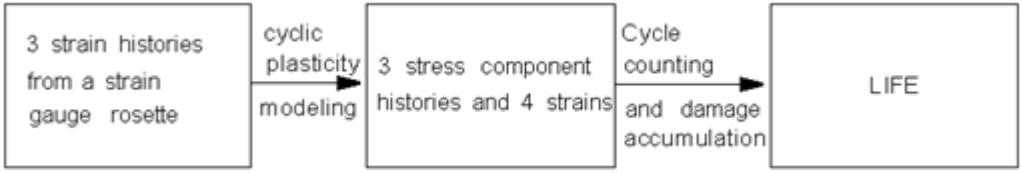


Figure 7-6 Outline of Strain-based Life Prediction Process

The approach for calculations on the basis of elastic finite element analyses will be similar, except that the input strains will be elastic strain components from the surface of the structure, and a notch correction procedure will be required in addition to the cyclic plasticity model to estimate the elastic-plastic stresses and strains (Ref. 59).

The essential calculations made by MMLF are as follows:

1. It takes three strain channels from a strain gauge rosette and convert them to the components of strain ε_x, ε_y and ε_{xy} taking into account the transverse sensitivities of the gauge legs. The gauge coordinate system is defined by x parallel to gauge 1 and z as the outward surface normal.
2. It feeds these 3 strain channels into the Mròz-Garud cyclic plasticity model, the outputs of which are the remaining non-zero strain component ε_z and the in-plane stresses σ_x, σ_y and σ_{xy}.
3. It processes the resulting seven components of stress and strain either by the conventional critical plane methods, or by multiaxial rainflow counting and then accumulating damage using the Wang-Brown methods.

These calculations are described in more detail in the following sections.

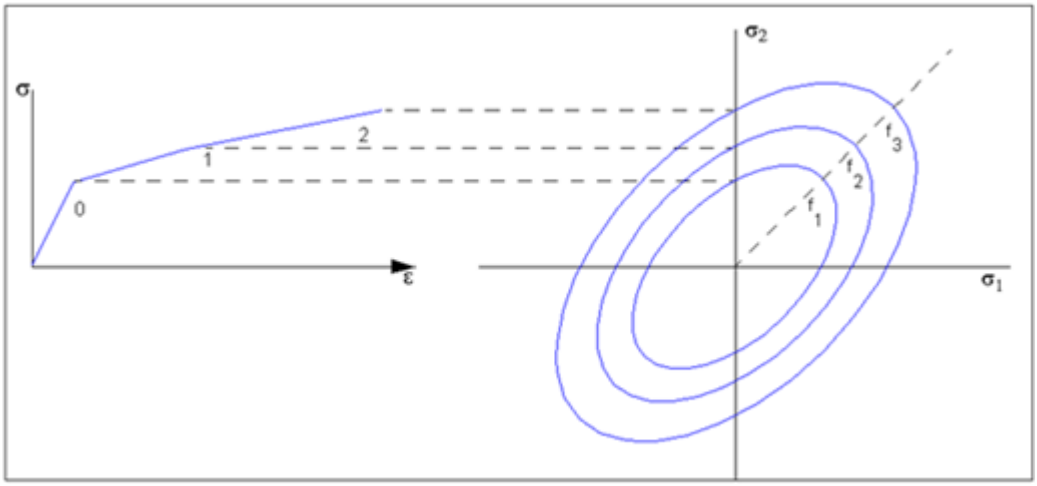
Cyclic Plasticity Modeling

The stresses and strains required by the damage models can be calculated if the relation between the equivalent plastic strain increment $\Delta\varepsilon_{eq}$ and the equivalent stress increment $\Delta\sigma_{eq}$ is known during the application of a given load increment.

However, it is known that the current relation depends on the previous load path and therefore, the plasticity model must deal with loading path dependent material constitutive behavior.

Several models are available in the literature (Ref. 45), (Ref. 46), (Ref. 60), (Ref. 61) of which the model proposed by Mròz and recently modified by Garud are the most popular. Mròz has proposed that the uniaxial stress-strain material curve be represented by a set of plasticity surfaces in three dimensional stress space. In the case of a two dimensional stress state, the plasticity surfaces reduce to ellipses on the plane of principal stresses described by:

$$\sigma_{eq} = \sqrt{\sigma_1^2 - \sigma_1 \cdot \sigma_2 + \sigma_2^2} \qquad (7\text{-}18)$$



and illustrated in Figure 7-7.

Figure 7-7 Linearization of the Material σ–ε Curve and Corresponding Plasticity Surfaces

The load path dependent memory effects are modeled by prescribing a translation rule for the ellipses moving with respect to each other over distances given by the stress increments. It is also assumed that the ellipses move inside each other and they do not intersect. If the ellipses come in contact with one another they move together as a rigid body.

The translation rule proposed by Garud avoids the intersection of the ellipses that could occur in some cases in the original Mròz model. The Garud translation rule is illustrated in Figure 7-8 and can be described by a model consisting, for simplicity, of only two plastic surfaces (ellipses).

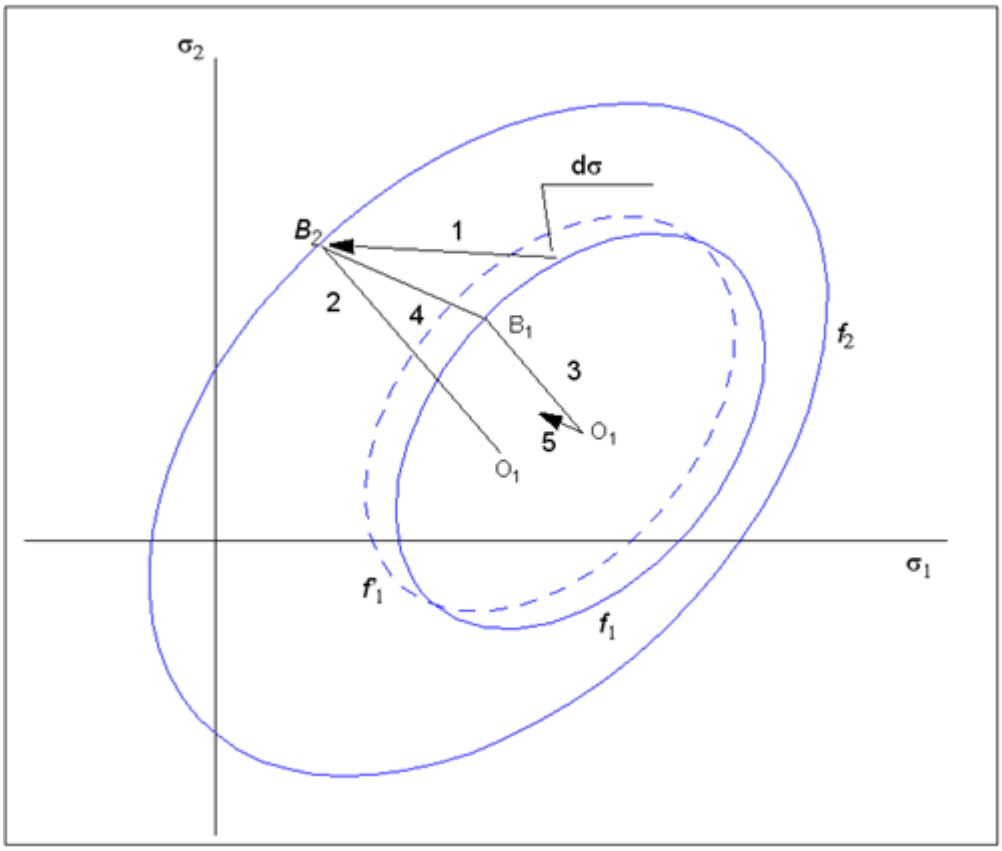


Figure 7-8 Geometrical interpretation of the Mròz-Garud incremental Plasticity Model

In order to predict material response due to the stress increment dσ, the following steps are made:

1. Extend the stress increment dσ to intersect the first external non-active plastic surface f_2 at point B_2.
2. Connect point B_2 and the center of the intersected plastic surface f_2.
3. Find point B_1 on the active plastic surface f_1 by drawing a line parallel to the line O_2B_2 through the centre O_1 of the surface f_1.
4. Connect the conjugate points B_1 and B_2 by the line B_1B_2.
5. Translate the ellipse f_1 in the direction of B_1B_2 from point O_1 until the end of the vector dσ lands on the moving ellipse f_1.

The translation rule assures that the two ellipses are tangential with the common point B_1B_2 without intersecting each other. Two or more tangential ellipses translate as a rigid body and the largest moving ellipse (Figure 7-8) indicates the proper constitutive relation (linear segment) to be used for a given stress increment. The principal of the method is described here for two surfaces. MMLF uses 40, with the corresponding linear

sequences being fitted to the Ramberg-Osgood equation to give a more or less smooth curve. More details of this method can be found in (Ref. 47).

Multiaxial Rainflow Counting

Wang and Brown (Ref. 62) proposed a multiaxial cycle counting method on the basis of strain hardening behavior under non-proportional variable amplitude loading. Relative stresses and strains were introduced so that a pair of turning points define the start and end points of a reversal, where the equivalent relative strain rises monotonically to a peak value. Since plastic deformation generates the driving force for small fatigue cracks, hysteresis hardening provides a physical parameter for cycle counting, analogous to rainflow counting in the uniaxial case.

Each reversal commences with elastic unloading, which is followed by reloading and plastic strain hardening up to the next turning point. The most significant turning point occurs at the highest value of equivalent strain. This is illustrated at time 0 in Figure 7-9, which shows a repeating block of a combined tension/torsion non-proportional load history. The equivalent strain is defined as the von Mises strain.

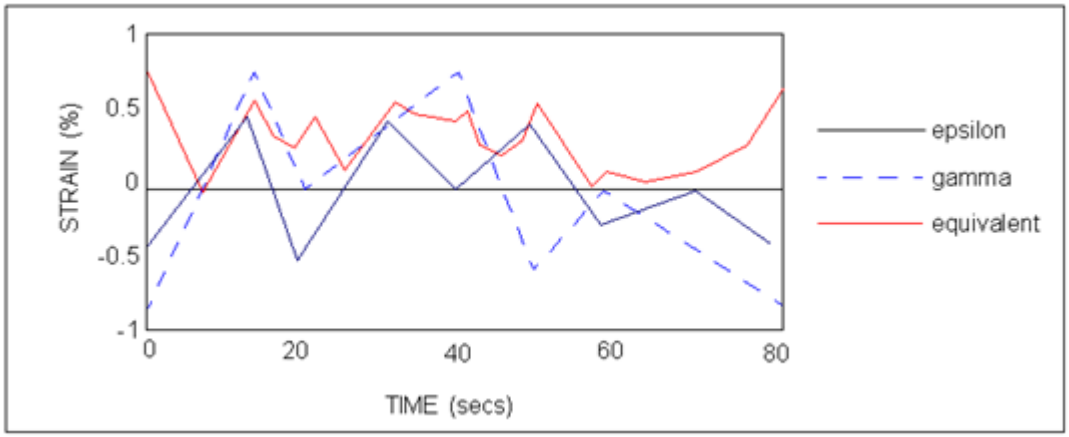


Figure 7-9 A Variable Amplitude Non-proportional Strain History, Tensile (epsilon), Torsional (gamma) and Absolute Equivalent Strain

The cycle counting method is illustrated by the following example. Starting from the most significant turning point, a graph is drawn for the loading block of relative equivalent strain, where relative strain $\varepsilon_{ij}^{*} = \varepsilon_{ij} - \varepsilon_{ij}^{A}$ represents the change of strain since time A. Figure 7-10 shows the relative equivalent strain, with respect to times 0, 10 and 20 seconds. Using the relative strain, a reversal can be defined starting from 0, up to the maximum value 10 seconds. To obtain the second reversal the relative strain is re-plotted starting from the next turning point where unloading commences (at 10 seconds), and the portions of the strain hardening curve for the reversal are selected by a traditional rainflow procedure (Ref. 47). The region of unloading and reloading within that reversal is counted in the next step.

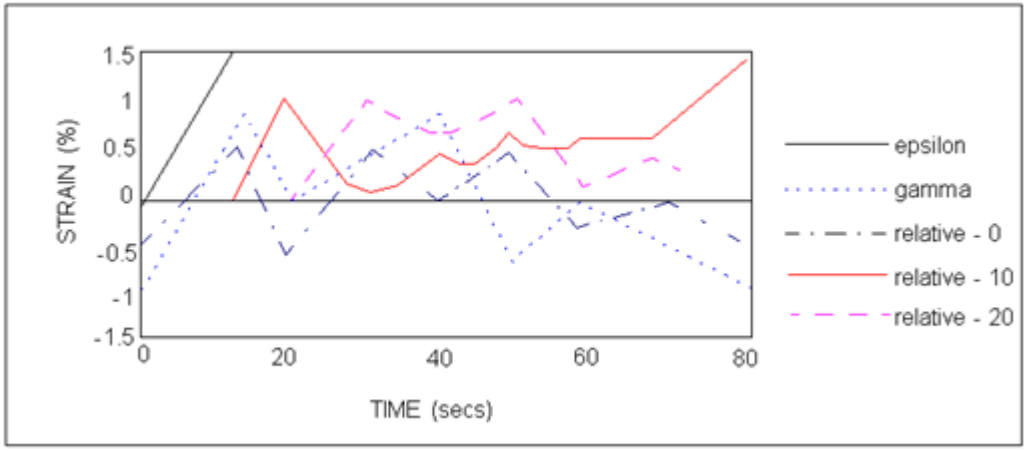


Figure 7-10 The Variable Amplitude History: Equivalent Strains with Respect to Times 0, 10 and 20

Using the next turning point, relative strain is replotted with respect to 20 seconds for the subsequent continuous fragment of strain history, yielding the third reversal. This procedure is repeated for each turning point in chronological order, until every fragment of strain history has been counted. The recursive nature of this process makes this module significantly slower than other analysis modules such as MCLF.

Fatigue Damage Calculation

The counting method described above is independent of fatigue damage parameters, being based on hysteresis deformation behavior. Being unrelated to material properties, it can be integrated with any multiaxial fatigue damage model. If the counted reversals are non-proportional, a fatigue damage parameter that accounts for non-proportional straining effects is required. The path-independent damage parameter proposed by Wang and Brown (Ref. 63) has been shown to provide good correlation for several materials under proportional and non-proportional loading,

$$\hat{\varepsilon} \equiv \frac{\gamma_{max} + S \cdot \delta\varepsilon_n}{1 + \nu' + S(1 - \nu')} = \frac{\sigma_f' - 2\sigma_{n,mean}}{E}(2N_f)^b + \varepsilon_f'(2N_f)^c \qquad (7\text{-}19)$$

where γ_{max} is the maximum shear strain amplitude on a critical plane (proportional or non-proportional), $\delta\varepsilon_n$ is the normal strain excursion between the two turning points of the maximum shear strain (that is the range of normal strain experienced on the maximum shear plane over the interval from start to end of the reversal), and $\sigma_{n,mean}$ is the mean stress normal to the maximum shear plane. This can be omitted if no mean stress correlation is required. If mean stress is included, this is equivalent to the Morrow method. The term S is a material constant determined from a multiaxial test (typically between 1 and 2 for Case A and around 0 for Case B) and v' is the effective Poisson's ratio. The right hand side of the equation is the same as the uniaxial strain life equation, with a Morrow mean stress correction (Ref. 64). Mean stress is measured as the average of the maximum and minimum stress values over the reversal. The total damage induced by a loading history is calculated using Miner's rule.

Figure 7-11 is a plot of predicted life against experimental life for a variety of proportional and non-proportional tests on laboratory specimens, from (Ref. 48).

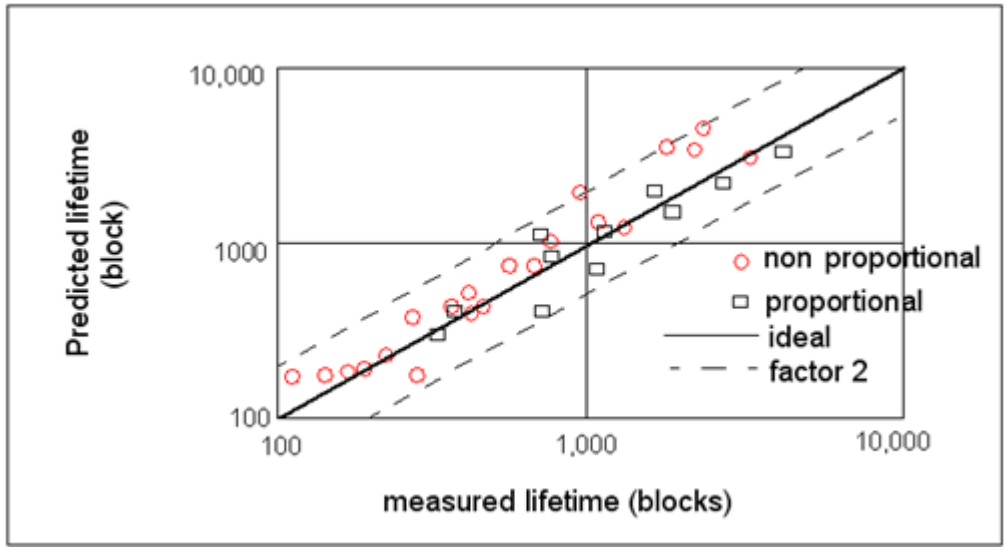


Figure 7-11 Comparison of Experimental and Predicted Results for the Wang-Brown Method

The other multiaxial damage parameters considered are the more conventional critical plane parameters. In these methods, stresses and strains are resolved onto a particular plane, inclined at an angle $\theta = 90$ degrees (Case A) and/or $\theta = 45$ degrees (Case B) to the free surface. Cycle counting (uniaxial) and damage parameter calculation is carried out on the critical plane and the damage accumulated. The orientation ϕ of the projection of the normal to the damage plane is increased by 10 degree increments from 0 to 170 degrees. The plane with the largest accumulated damage is said to be the critical plane. The models are:

1. Normal strain ($\theta = 90$ degrees only):

$$\frac{\Delta\varepsilon_n}{2} = \frac{\sigma'_f}{E}(2N_f)^b + \varepsilon'_f(2N_f)^c \qquad (7\text{-}20)$$

is the strain amplitude normal to the critical plane. Otherwise this is the usual Coffin-Manson-Basquin equation, where $\frac{\Delta\varepsilon_n}{2}$.

2. Shear strain:

$$\frac{\Delta\gamma}{2} = \frac{(1+V_e)\sigma'_f}{E}(2N_f)^b + (1+V_p)\varepsilon'_f(2N_f)^c \tag{7-21}$$

3. Smith-Watson-Topper/Bannantine (θ = 90 degrees only) (Ref. 65):

$$\frac{\Delta\varepsilon_n}{2}\cdot\sigma_{n,max} = \frac{\sigma'^2_f}{E}(2N_f)^{2b} + \sigma'_f\cdot\varepsilon'_f(2N_f)^{b+c} \tag{7-22}$$

where σ_n, max is the maximum normal strain on the critical plane which occurs during each rainflow cycle. Otherwise this is the Smith-Topper-Watson method (Ref. 66).

4. Fatemi-Socie (Ref. 67):

$$\frac{\Delta\gamma}{2}\left(1+n\frac{\sigma_{n,max}}{\sigma_y}\right) = \frac{(1+V_e)\sigma'_f}{E}(2N_f)^b + \frac{n(1+V_e)\sigma'^2_f}{2E\sigma_y}(2N_f)^{2b} + (1+V_p)\varepsilon'_f(2N_f)^c + \frac{n(1+v_p)\varepsilon'_f\sigma'_f}{2\sigma_y}(2N_f)^{b+c} \tag{7-23}$$

Applications

MMLF may be used to predict the life of components and structures, to provide some help in design optimization, and to assist in the design of efficient tests through fatigue editing. The main drawback of the program is that it is necessary to place a strain gauge rosette on the critical location or crack initiation site in order to get a reliable prediction. This is due to the fact that there is no simple transfer function between the measured strains at a non-critical location and the strains at a nearby critical location when the loading is non-proportional.

Three applications of the multiaxial fatigue program will be described to illustrate the method.

Example 1: Fatigue calculations on an automotive wheel

This example is based on strain gauge measurements from an automotive wheel. The wheel was tested on a rig simulating the rotation of the wheel under load, and the strain gauge measurements were taken using a 45 degree rosette placed in the most critical location. This location was identified by previous stress analysis. A small section of the three strain gauge channels is illustrated in Figure 7-12.

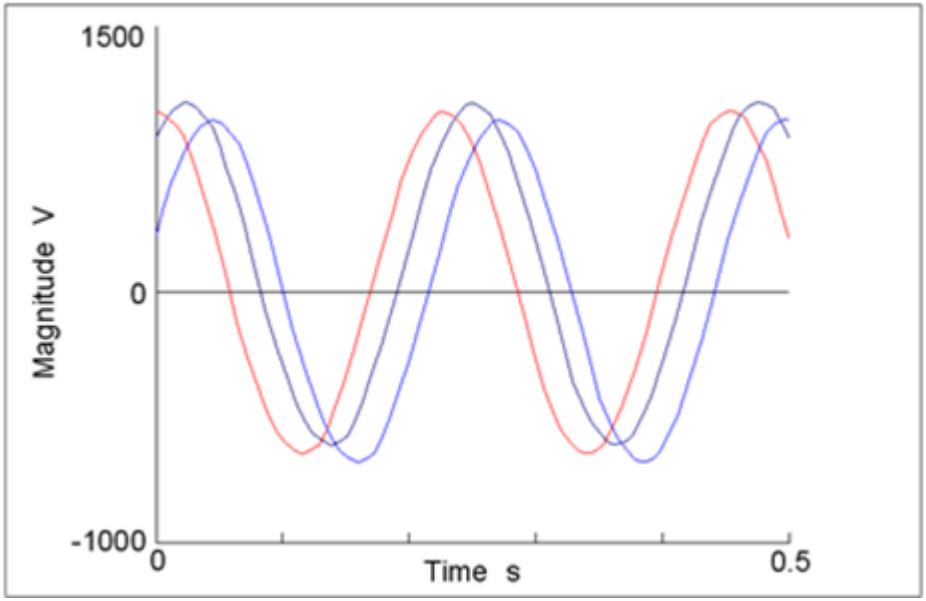


Figure 7-12 Section of Strain Measurements from Wheel Critical Location

The three channels of strain are more or less sinusoidal with a phase difference between them, indicating that the stress state is multiaxial and non-proportional. The two cycles shown above are taken from a longer history. The stress-strain state variations can be better understood qualitatively by looking at the behavior of the principal stresses as illustrated in Figure 7-13, where it can be seen that the two principal stresses are almost in phase, but with slightly different amplitudes and means. Notice that the absolute maximum principal stress (the principal stress with the largest magnitude) "flips" between the maximum and the minimum principals. Whenever it does this of course, the angle also flips through 90 degrees indicating that the state of stress has reached a condition where E.MAX and E.MIN are equal and opposite, i.e. pure shear. In fact the stress state behavior can be summed up by saying that when the stress is at a maximum or minimum, the stress state is almost equibiaxial, and in between times it passes through a condition of pure shear where the stresses are lower. Between the two limits the principal stress axis rotates through 90 degrees, so that there is a complete rotation (180 degrees) per cycle. These files were generated using SSA.

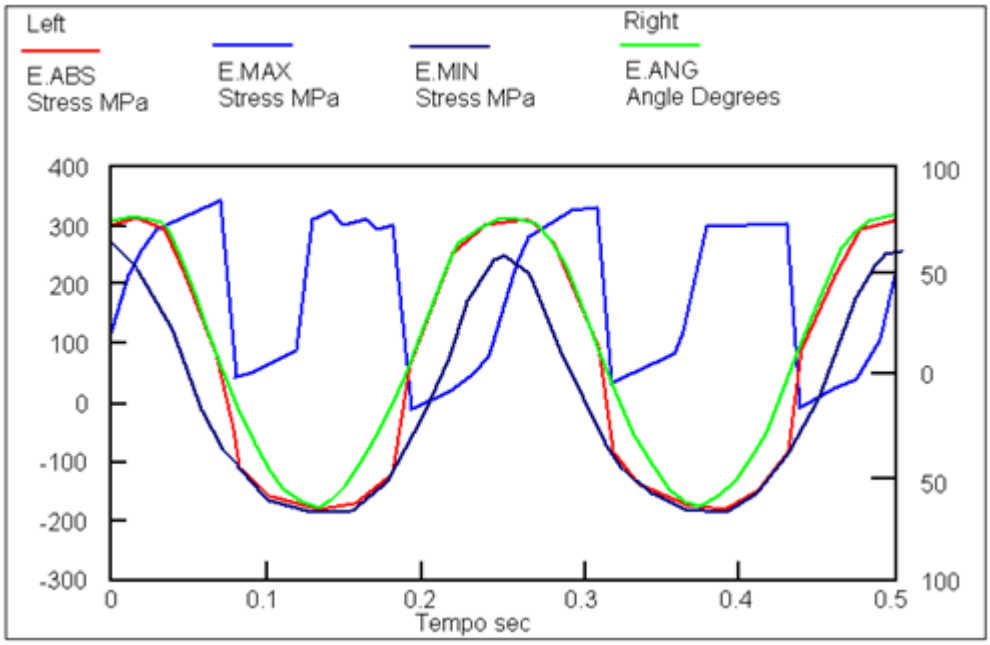


Figure 7-13 Absolute Maximum (E.ABS), Maximum Principal (E.MAX), Minimum Principal (E.MIN) and Angle of the Absolute Maximum Principal Stress to Gauge 1 Against Time

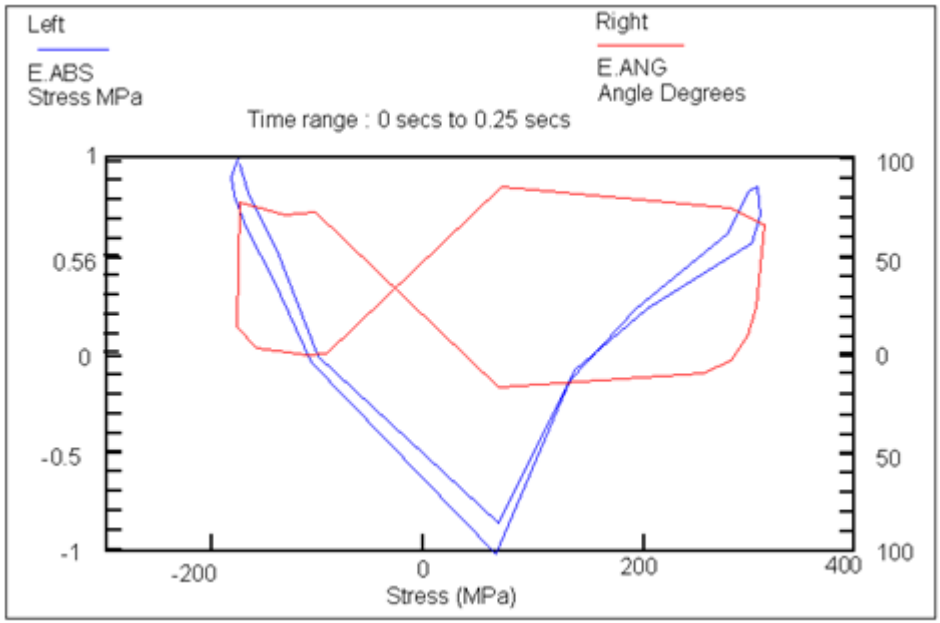


Figure 7-14 Plot of Biaxiality Ratio and Angle Against Absolute Maximum Principal Stress

Another way of looking at this data is to plot the biaxiality ratio (the ratio of the smaller in magnitude to the larger principal stress) and the orientation against the largest principal stress. This is shown in Figure 7-15 for 1 cycle.

The results of analysis of the full loading history are given below with the results of some other methods for comparison.

Method	Life (Rotations)
Wang-Brown (S = 1 for Case A, 0 for Case B)	1.03×10^6
Wang-Brown + Mean (S = 1,0)	7.07×10^5
Normal Strain	5.54×10^6
Smith-Topper-Watson (Bannantine)	9.43×10^5
Shear Strain	5.95×10^5
Fatemi-Socie (n = 0.6)	5.17×10^5
Experiment	> 200000
	(Test stopped with no failure)

These results are all very similar, with the notable exception of the normal strain method which is relatively non-conservative.

It is normal to view the accumulated rainflow cycles in an analysis in the form of a 3D histogram plot. Normally the axes are range-mean or max-min or from-to. With the Wang-Brown method, each cycle is characterized by shear strain range, normal strain range, mean normal stress, and two angle ϕ and θ. The distribution of numbers of reversals and damage may reasonably be visualized in relation to any two of these parameters. Figure 7-15 shows the distribution of reversals in relation to shear strain range and normal strain range.

Another potentially useful way of viewing the results is in the form of a polar plot of damage, as shown in Figure 7-16. In this case there are only Case B damaging cycles, so there is only one curve on the plot. This plot was produced with the polar display program MPOD.

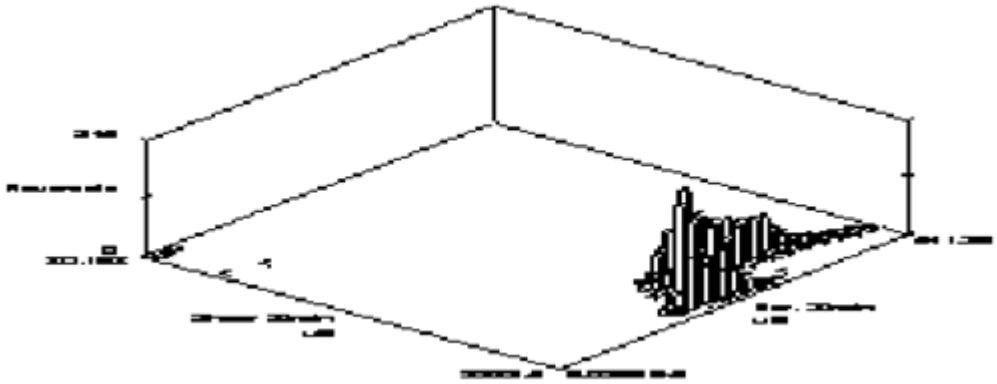


Figure 7-15 One Representation of Rainflow Count.

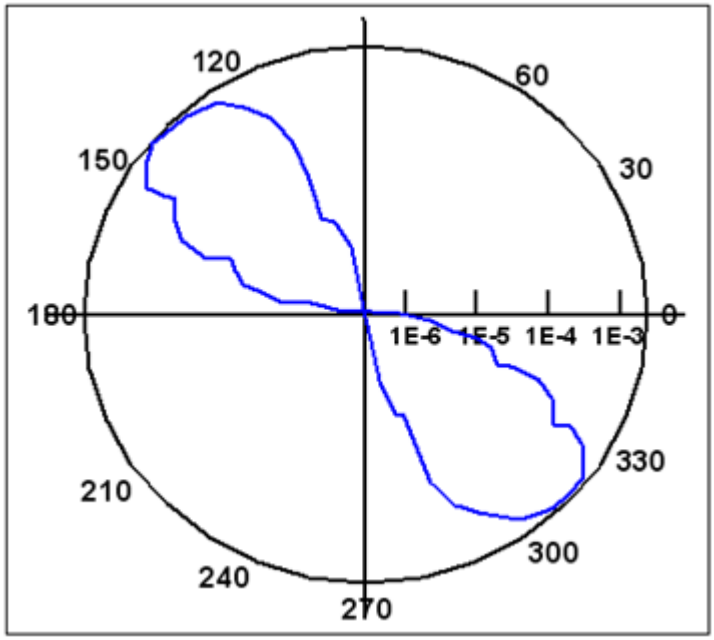


Figure 7-16 Polar Plot of Log (damage) Against Orientation ϕ

Example 2: Fatigue life prediction of a steering rack mounting bracket

This example concerns a sign-off test which is carried out on a front suspension cross-member and steering rack assembly. The steering rack is mounted to the cross-member by means of two small brackets, which are fatigue-sensitive areas. The assembly is tested by fixing the cross-member and applying a constant amplitude, unidirectional load to the steering rack. The resulting stress state variations are illustrated in Figure 7-17.

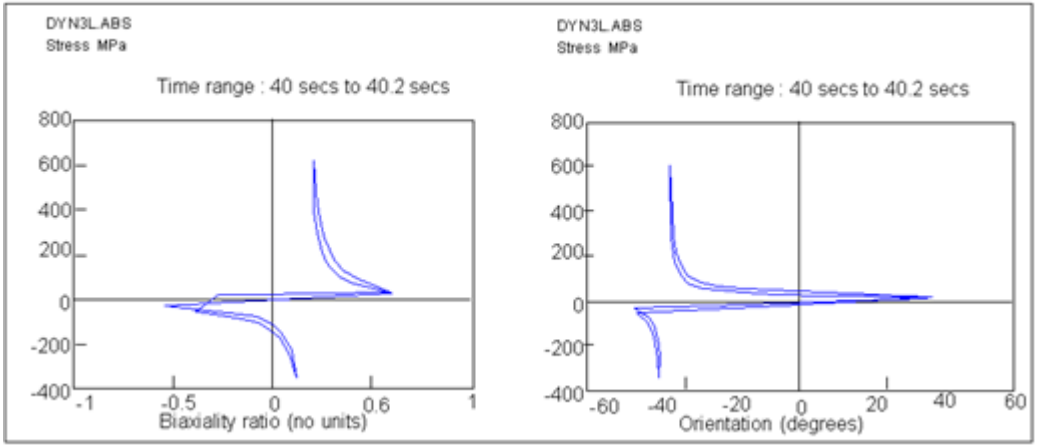


Figure 7-17 Plots of Principal Stress Ratio (left) and Principal Axis Orientation (right) Plotted Against Absolute Maximum Principal Stress

These plots show that the single-axis loads applied generate locally a near-uniaxial, near proportional stress state. The results of analysis with the new program, assuming 99.9% certainty of survival (based on the calculated standard errors on the material parameters) are as given in below.

Method	Life (Repeats)
Wang-Brown (S = 1,0	24
Wang-Brown + Mean (S = 1,0)	23
Normal Strain	33
Smith-Topper-Watson (Bannantine)	22
Shear Strain	23
Fatemi-Socie (n = 0.6)	23

Example 3: Fatigue editing on an off-road vehicle rear axle

This example illustrates how the program can be used to reduce testing times through multiaxial, multichannel fatigue editing of simulation test rig drive files. The component in question is a rear axle from an off-road vehicle. The axle is instrumented with three strain gauge rosettes close to critical locations, and analysis of the measurements from all three strain gauge rosettes show that the loading is non-proportional. It is desired to reduce the testing time by editing the rig drive signals while retaining all the sections of the original sections that cause significant damage. This was carried out using a simple editing procedure.

One of the outputs of the life prediction program is a time-correlated damage file, essentially a time history of damage. This file is a record of the damage accumulated, with damage for each reversal being distributed between the opening and closing point of each reversal. The signal is divided up into a number of equally spaced windows, and the least damaging are discarded until a certain percentage of damage is retained. In this example, 95% was retained. The time slices to be retained are compared across the three channels with a

logical OR operation and all the resulting windows are retained. The resulting edit vector can be used to edit the rig drive signals, inserting suitable joining functions for continuity. It is also important to edit the strain signals and recalculate the life in order to check the likely effect of editing on the fatigue damage. This is important because of the possibility of reversals starting in one window and ending in another, and also because in multiaxial loading the loading path is important as well as the peaks. In some cases, editing the signals may actually cause a slight increase in fatigue damage. This is why methods such as described in (Ref. 68) are not really suitable for non-proportional loadings because they do not retain the important sections of the loading path. The results of the analysis described are summarized below and the original and edited strain signals for Gauge 1 are illustrated in Figure 7-18.

Gauge Number	Original Signal Predicted Life (repeats)	Edited Signal Predicted Life (repeats)	Actual Damage Retained	Data Reduction Factor
1	13866	14079	98.5	8.25
2	29770	30188	98.6	8.25
3	53200	53800	98.8	8.25

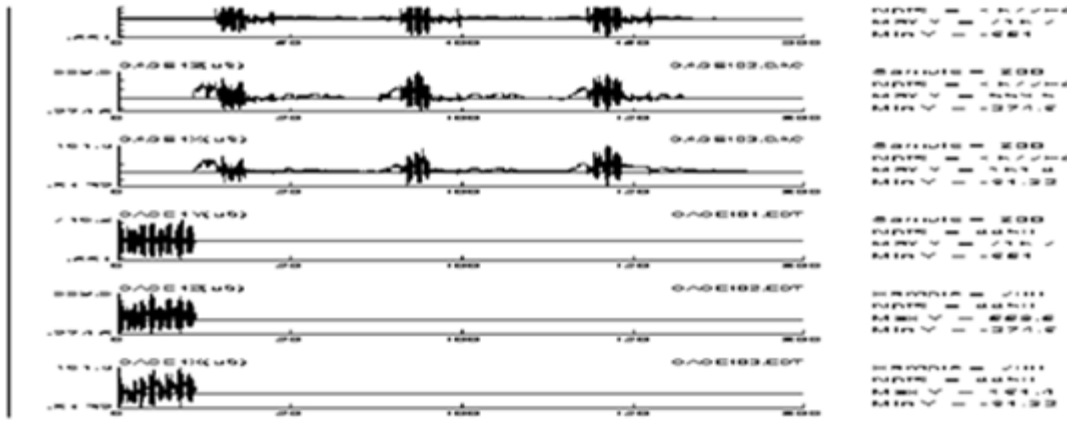


Figure 7-18 Strain Signals Before and After Editing

The advantages of this method are that it retains both the most important sections of loading path, and the essential frequency content of the loading, essential for components with dynamic behavior.

MMLF Module Operation

The MMLF module can be run in one of the following 3 modes:

- From the MSC Fatigue Tools menu pulldown
- In stand alone mode by typing **mmlf** at the system prompt
- By incorporating the **MMLF** commands in a batch operation

Once running in interactive mode the MMLF module will display the following screen.

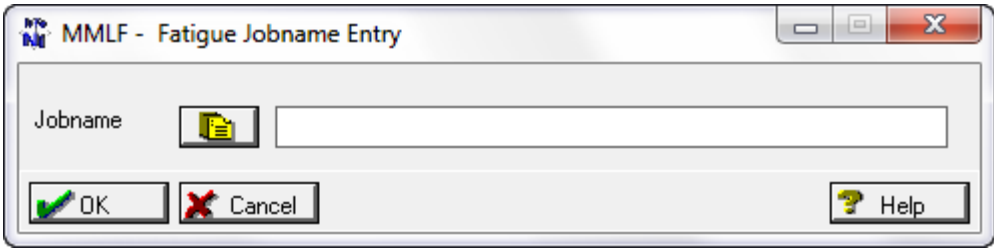


Figure 7-19 The First MMLF Screen

The user must enter the name of an Input Job File, the extension `.fjb` will be appended automatically. If the name of an existing Job File is entered then MMLF will go to the postprocessing menu screen. If a new job is specified then its parameters have to be entered in a series of windows. If no name is entered then the results of processing may NOT be saved.

If a new job is specified, its name is specified in this window and its parameters in subsequent windows.

The flow chart below illustrates the major routes through MMLF.

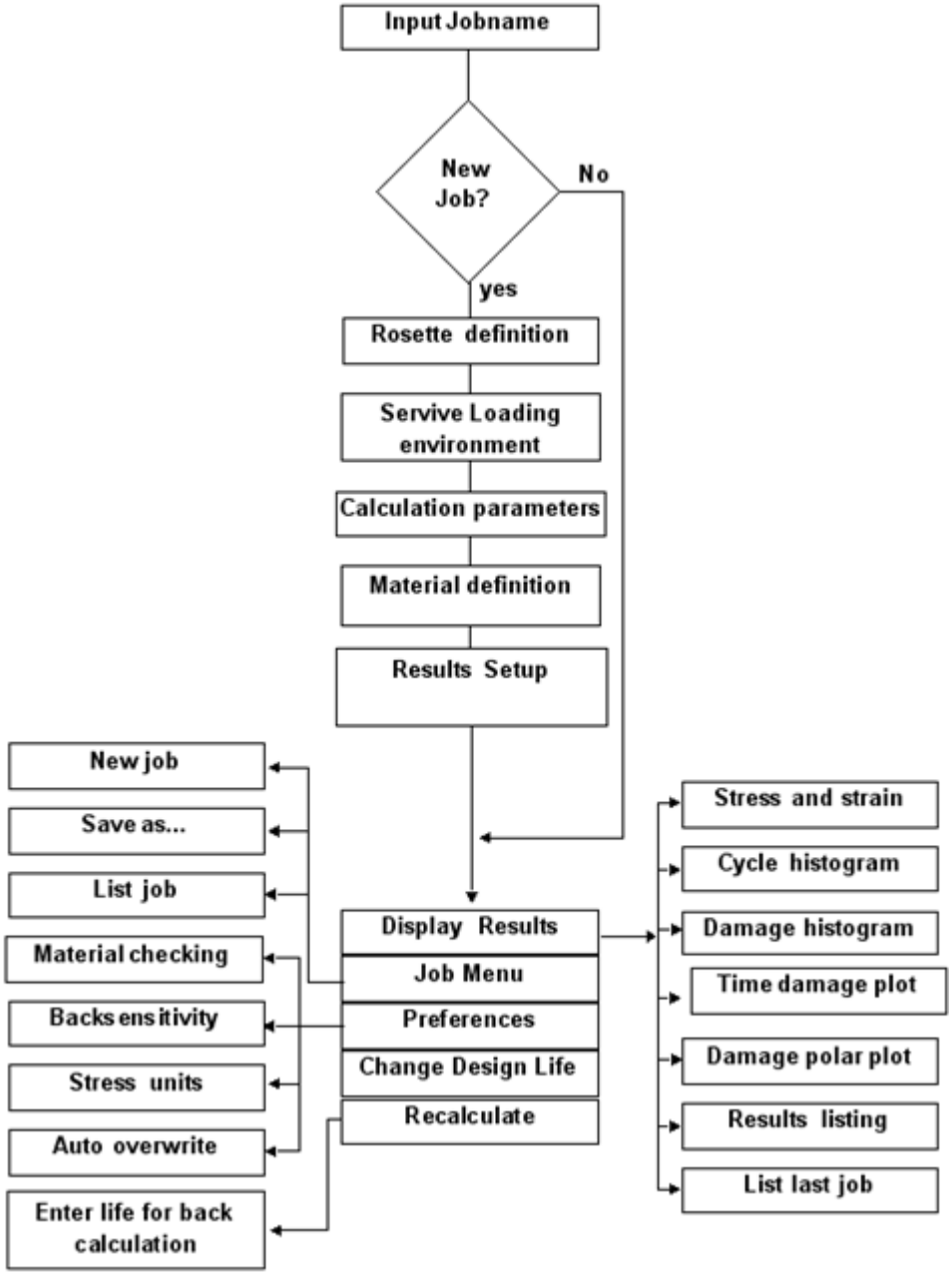


Figure 7-20 The General MMLF Module Structure

The above flowchart is mirrored by the options that are on the Post-processing menu illustrated in Figure 7-31. If a new job is being defined, then the following screen will appear.

Rosette Definition

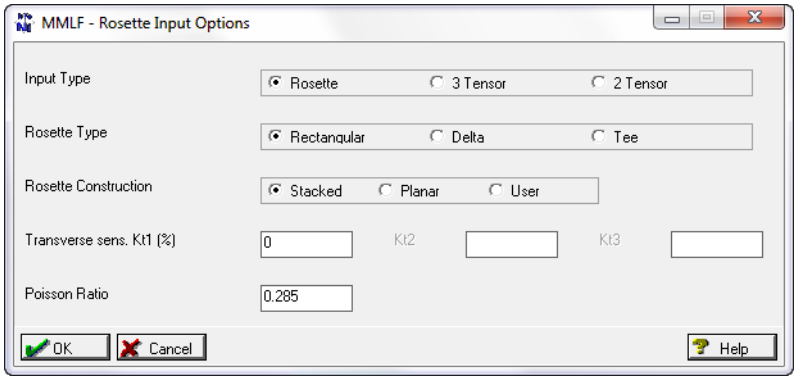


Figure 7-21 The MMLF Geometry Screen

Field	Description
Input Type	Input can be either from a Rosetta or from a set of tensor strains (e.g, from FE).
Rosette Type	MMLF supports three types of rosette configuration: **rectangular**, **delta**, and **tee**. Tee rosettes consist of two mutually perpendicular grids, which should be aligned with the principal stresses. Rectangular rosettes have three grids, with the second and third grids angularly displaced from the first grid by 45 and 90 degrees, respectively. Delta rosettes have three grids, with the second and third grids 60 and 120 degrees away, respectively, from the first grid. See the technical overview of SSA for details.

Field	Description
Rosette Construction	Strain gauge rosettes are manufactured in two forms, **planar**, and **stacked**. Stacked rosettes have the three grids layered on top of each other. Planar rosettes have all three grids laying in the same plane. All the gauges in a stacked rosette have the same gauge factor and transverse sensitivity whilst the grids of a planar rosette will have slightly differing values of these properties. In the latter case, the transverse sensitivities of grids 1 and 3 are generally the same and grid 2 is different. The 'User' field allows for the case when all three transverse sensitivities are different. The rosette manufacturer's data sheet should indicate the type of rosette that the analysis is being set up for. The number of transverse sensitivities (see below) depends upon the choice made in this field.
Transverse sens. Kt1 (%)	The transverse sensitivity coefficient, Kt, is defined as the ratio of the transverse and axial gauge factors. It is used to correct measured strains for the effects of transverse Poisson's strains. It should not to be confused with the elastic stress concentration factor of the same name. The number of values to be entered is... ■ One - Stacked Rosette Configuration ■ Two - Planar Rosette Configuration ■ Three - User Defined Rosette Configuration Enter the value of transverse sensitivity coefficient as a percentage. Please refer to the rosette manufacturer's data sheet for the value(s) of this factor. In practice this may make little difference to the results.
Poisson's Ratio	The **gauge's** (not the test material's) Poisson's ratio (ratio of the fractional contraction in breadth to the fractional increase in length) is entered here. This is the Poisson's ratio of the material on which the gauge factor was measured. For steel the value is usually 0.285. Please refer to the rosette manufacturer's data sheet for the required value if different to the default.

Service Loading Environment

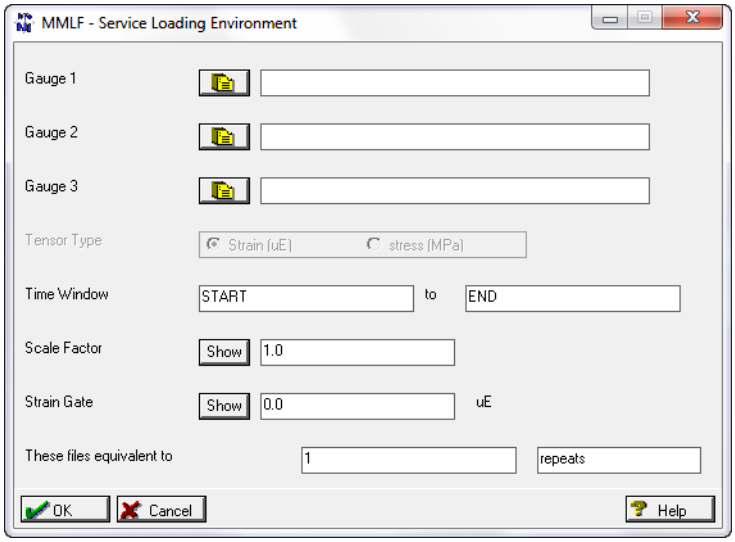


Figure 7-22 The MMLF Loading Environment Screen

The Service Loading fields are as follows:

Field	Description
Gauge1 / Gauge 2 / Gauge 3	The three files that contain the rosette strain gauge data can be typed individually (or picked using the pick list button) into fields Gauge 1, 2 and 3.
Time Window	Not all of the Loading Time History file has to be used in the calculation. If only a part of the file needs to be used then the start point and end point can be specified in the Time Window. The default is to use all of the time history file, which is why the defaults are START and END. If a part only is going to be used, then clear the fields and enter the values, i.e. 10 in the Start field and 1000 in the End. Alternatively, the syntax START+n and END-n' where n is a number of units (e.g. seconds) from the start of the file and n' is a number of units before the end of the file. START and END are usually recorded in the header area of the input file. Please note that START is not necessarily zero.

Field	Description
Scale Factor	The scale factor is an amount by which the history will be multiplied. Multiple scale factors can be entered. For example entering (-500,1000,50) in the field will scale from -500 to +1000 inclusive in steps of 50, i.e. *from to in steps of* -500,1000,50 BACK may also be entered in which case MMLF will calculate the scale factor needed to achieve the required design life. See the online help for further examples of multiple entry and arithmetic operations with keywords.
Strain Gate	The gate is, in effect, a filter that may speed processing by filtering any small cycles that are unwanted (for example signal noise). The gate value should not be set too high or it will filter out relevant parts of the signal. Any signal smaller than the gate value specified in this field will be ignored in the life calculation. Multiple gate values can be entered using the same syntax as the example given for scale factors (above). No results file will be saved if a multiple analysis is carried out. A multiple gate analysis can be used to assess the gate value so that the user can choose the gate threshold that just fails to remove genuine damaging events. The gate threshold value is sensitive to changes in material. Also if a signal is scaled up by 2 the gate is NOT automatically scaled up by 2, the user must manually scale up gate.
These files equivalent to.....repeats	The fatigue analysis to be carried out is through the use of a either a time series, range-mean matrix or a sequence of peak valleys. The results will normally be presented in the form of the number of repeats of these series to failure. An equivalent unit, such as miles, laps, hours, etc. can be attached to the input loading file so that results can be presented in the form of miles, laps or hours to failure. In these fields, enter the units laps, and the number of them that the loading series is equivalent to. Note that the units and their number will be stored within the extra details area of the input data file using EQUUNITS and NUMEQUNI as keywords respectively.

Note:	Only 1 field from Scale Factor, or Strain Gate (see below) can have multiple values entered. Attempts to enter multiple values in more than 1 of these fields will result in an error message. This limitation also affects the % Certainty of Survival field on the Calculation Parameters screen or any field with a SHOW button.

Model Parameters

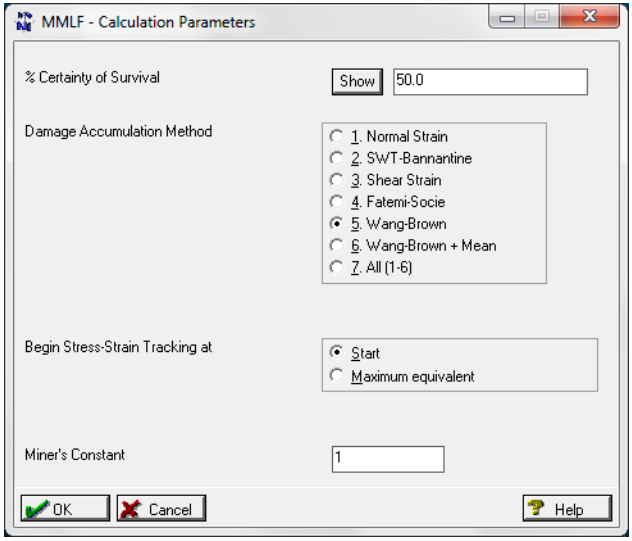


Figure 7-23 MMLF Model Parameters

The fields are as follows:

Field	Description
% Certainty of Survival	Any fatigue life prediction has associated with it a degree of uncertainty. To incorporate the effects of scatter in the strain life data, a % certainty of survival value can be entered here. This option is similar to that in any other MSC Fatigue module.
Damage Accumulation Method	MMLF supports 6 multiaxial damage accumulation methods. Options 1 - 4 are conventional critical plane methods, whereas options 5 and 6 are more sophisticated and are described in more detail in the technical background. The conventional critical plane methods work by calculating the normal or shear strains and normal stresses on a particular plane, before carrying out cycle counting and damage accumulation more or less in the normal way. The orientation of the plane is defined by 2 angles θ and ϕ. The angle θ is the angle made by the free surface normal and the normal to the current plane. The angle ϕ is the angle made by the projection onto the free surface of the normal to the plane and the local x-axis (gauge 1). A variety of angles are calculated and the one with the worst damage is taken to be the critical plane. Because the damage parameters are essentially either normal strain or shear strain based, q takes values of 90 (Case A) or 45 (Case B) degrees. The angle θ can take any value in theory. In the program θ is increased in increments of 10 degrees. If only case A is calculated, as in options 1 and 2, 18 planes are evaluated. Otherwise, 36 planes are considered. The Wang-Brown and Wang-Brown + Mean are considered the best of these methods. If you have a uniaxial loading with no mean stress, all these methods will give almost the same result. The methods are described below this table.
Begin Stress-Strain Tracking at	The Wang Brown method of cycle counting always starts from the maximum equivalent strain. If you wish to start the stress strain tracking from this point also, then select Maximum equivalent. This is better as it minimizes the effect of transient behavior at the beginning of the strain history. Otherwise select Start to begin stress-strain tracking from the start of the strain history file.
Miners Constant	You can adjust Miner's constant as with other MSC Fatigue modules. See the comment below regarding Miner's rule.

The damage accumulation methods are:

1. Normal Strain

This is a simple "critical plane method". The damage parameter is the normal strain ε_n used in conjunction with the Coffin-Manson-Basquin equation.

$$\frac{\Delta\varepsilon_n}{2} = \frac{\sigma'_f}{E}(2N_f)^b + \varepsilon'_f(2N_f)^c$$

2. SWT/Bannantine

   This method is very similar to the normal Smith-Watson-Topper method. In additional to the normal strain, the damage parameter includes the maximum normal stress $\sigma_{n,\,max}$. This is the maximum normal stress occurring in each rainflow cycle. The difference with the normal SWT method is that the maximum normal stress does not necessarily occur at the same time as the maximum normal strain.

$$\frac{\Delta\varepsilon_n}{2}\cdot\sigma_{n,\,max} = \frac{\sigma'^2_f}{E}(2N_f)^{2b} + \sigma'_f\varepsilon'_f(2N_f)^{b+c}$$

3. Shear Strain

   The shear strain method is similar to the first option, except that it uses shear strain only, and the parameters σ'_f and ε'_f are modified accordingly.

$$\frac{\Delta\gamma}{2} = \frac{(1+\nu_e)\sigma'_f}{E}(2N_f)^b + (1+\nu_p)\varepsilon'_f(2N_f)^c$$

4. Fatemi-Socie

   The Fatemi-Socie method uses shear strain and maximum normal stress. Note that it requires two extra parameters: the yield stress of the material and a dimensionless parameter n. Make sure you have defined a yield stress for the material if you want to use this method. These parameters effectively control the relative importance of the shear strain and the normal stress. Typical values of n are from 0.4 to 0.6.

$$\frac{\Delta\gamma}{2}\left(1+n\frac{\sigma_{n,\,max}}{\sigma_y}\right) = \frac{(1+\nu_e)\sigma'_f}{E}(2N_f)^b + \frac{n(1+\nu_e)\sigma'^2_f}{2E\sigma_y}(2N_f)^{2b} + (1+\nu_p)\varepsilon'_f(2N_f)^c + \frac{n(1+\nu_p)\varepsilon'_f\sigma'_f}{2\sigma_y}(2N_f)^{b+c}$$

5. Wang-Brown, and
6. Wang Brown + Mean

   These methods are described in more detail in the Summary of Method, 439.

Regarding Miner's constant, operation at a strain amplitude ε_1 will result in failure in, say, N_1 cycles. Operation at the same strain amplitude for a number of cycles less than N_1, N_j say, will result in a smaller fraction of damage, D_j, which is often referred to as a partial damage. Operation over a spectrum of different strain levels, results in a partial damage contribution D_i from each strain cycle,ε_i. Failure is then predicted when the sum of these partial damage fractions reaches unity so that:

$D_1 + D_2 + \ldots\ldots + D_{i-1} + D_i \geq 1$

The Palmgren-Miner rule asserts that the partial damage at any strain amplitude ε_i , or STW level, is linearly proportional to the ratio of number of cycles of operation, n_i, to the total number of cycles that would produce failure at that strain level, N_i i.e.:

$D_i = n_i / N_i$

Failure is predicted then if:

$$n_1/N_1 + n_2/N_2 + ... + n_{i-1}/N_{i-1} + n_i/N_i \geq 1$$

or

$$\sum_{1}^{i} (ni/Ni) \geq 1$$

The above equations represent a statement of the linear damage rules used by all the MSC Fatigue analyzers. Experience shows that linear damage summation is somewhat of an over-simplification of reality. The most important shortcoming is that no account is made of the sequence in which strain levels are experienced, and damage is assumed to accumulate at the same rate for a given strain level regardless of prehistory. In particular, it appears that large strain amplitudes which precede smaller ones, cause the smaller cycles to become more damaging than expected and visa versa. The net result is that in the first instance, the Miners' sum (total of (n/N)) is measured to be less than 1 and in the latter case, it becomes greater than 1. Since most service environments involve quasi-random loading sequences, the use of the Palmgren-Miner linear damage rule summing to a constant of 1 is mostly satisfactory. However, provisions have been made within MMLF to set the Miners' sum to any desired value.

Material Data Input

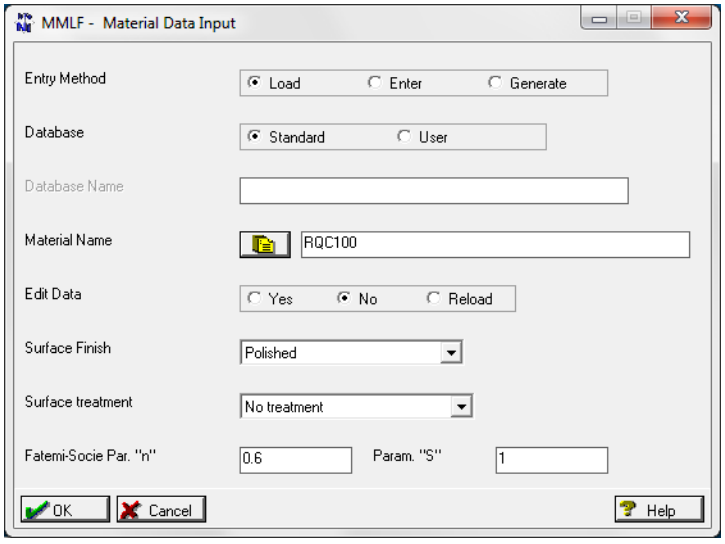


Figure 7-24 The material Data Input screen

The multiaxial local stress-strain fatigue analyzer requires a characterization of the material from which the component or part is manufactured. This characterization is accomplished through a set of parameters generated by strain-controlled testing smooth specimens of the material in question.

This screen allows selection of material data-sets to be made. It is possible to either load the desired parameters from the PFMAT data base, to load them from a user custom database (any name allowed) or to enter them directly from the keyboard, or to generate them from empirical rules based on tensile properties.

Field	Description
Entry Method = Load	Selecting this option allows data-sets to be loaded directly from the material database.
Database	Allows you to specify the database from which materials parameters will be extracted. The default is to look for a local database called NMATS.MDB. If the User option is selected, then a database name must be specified.
Database Name	Specify the name of the database to use.
Material Name	Material data-sets are loaded directly from the material database. Each data-set has a unique name through which it can be referenced. The PICK LIST button, can be used to list all the available data-sets and the cursor used to pick the required set. The data base manager, PFMAT, can also be used to search for materials with specific characteristics. The search operation needs to be carried out outside this program, perhaps in multiaxial mode.
Edit Data Yes/No/Reload	Having selected a material data-set from the database, it is possible to edit the parameters prior to a life estimation. Select yes if you require to edit the material parameters prior to (re)calculation. The edit function itself will be present on exit from this screen. Select reload to undo any previous edits by reloading the named material from the database.
Entry Method = Enter	Material data parameters may be entered directly from the keyboard. The parameters entered will be loaded into the job file and remain active until overwritten or another job selected. They will NOT be loaded into the PFMAT database. If a particular set of parameters is to be used often, then it is recommended that they be loaded within the database by running PFMAT. To enter parameters, move the highlight cursor to the required position and enter the value in the appropriate units. Stress units can be customized in the preferences menu. All of these parameters are described in Material Management (Ch. 4).
Entry Method = Generate (Material Parameters)	Selecting this option will send the user to a separate Material Data Generation screen that prompts the user to enter the data needed to generate a data set. The form in Figure 7-25 is displayed and the various field are explained in the table that follows the figure.
Surface Finish	This field is provided because surface finish can have a significant effect upon fatigue life, especially with higher strength materials. The default entry is 'Polished' as with a laboratory specimen, but a whole range of predefined finishes is provided. The surface finish correction factors that will be applied to the calculation are held in the `/mats` sub-directory. Selecting *User Defined* enables the user to define a surface correction factor. Selecting all will cause MMLF to calculate the fatigue lives for the whole range of Surface Finishes.

Field	Description
Surface Treatment	Treatments that introduce stresses, usually compressive, into the surface of a component have a significant effect upon fatigue life. Correction factors for a range of surface treatments are therefore available within MMLF. Selecting ALL will cause MMLF to calculate the fatigue life for the entire range of surface treatments. 'All' can NOT be entered for both Surface Finish and Surface Treatment at the same time.
Fatemi-Socie Parameter "n"	The Fatemi-Socie method requires two parameters to be defined which are not normally required for a fatigue analysis, namely the yield stress of the material and a dimensionless parameter n. The value of n/σ_y controls the relative importance of the maximum normal stress and the shear strain range for the damage parameter.
Material parameter "s"	The Wang-Brown method uses a damage parameter $\hat{\varepsilon} \equiv \frac{\gamma_{max} + S \cdot \delta\varepsilon_n}{1 + \nu' + S(1 - \nu')}$. The parameter S controls the relative importance of the shear and normal strain range on the critical plane, for Case A only. For Case B, S is set to 0. In general values of S are in the range 1 to 2. A value of 1 will tend to be a little conservative, so is a reasonable default.

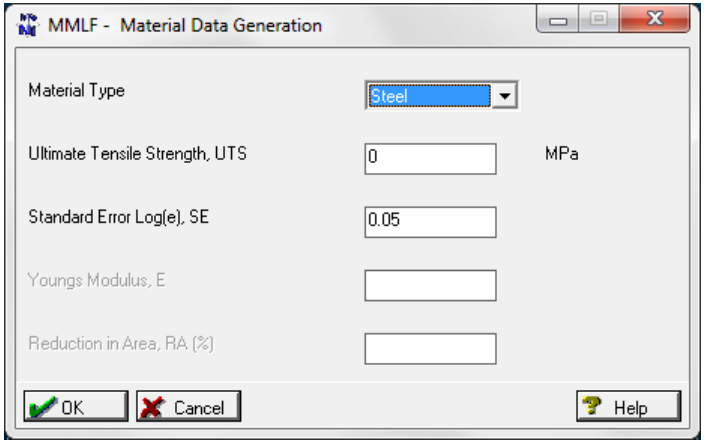


Figure 7-25 Generating a Materials Data Set

Field	Description
Material Type	Steel, Titanium, Aluminum, Other
Ultimate Tensile Strength (UTS)	Enter a value for the UTS in the units specified. The parameters required to characterize cyclic material properties can be generated purely on the basis of this field.
Standard Error of Log(ε) - SE	Enter a value for the standard error of log(ε). The value will apply equally to the elastic life line, the plastic life line, and the cyclic stress strain curve.
Youngs Modulus - YM	Enter a value for the modulus of elasticity (Young's modulus) in the units specified. This field only appears for Material Type = Other.
Reduction in Area - RA %	Enter a value for the percentage reduction in area. This field only appears for Material Type = Other.

Regarding surface finish and treatment: the effect of surface treatment depends on the surface finish. The increase in endurance limit with stress due to the surface treatment is given below.

FINISH	SHOT PEENED	COLD ROLLED	NITRIDED
Polished	+15%	+50%	+100%
Ground	+20%	+0%	+100%
Machined	+30%	+70%	+100%
Hot Rolled	+40%	+0%	+100%
Cast	+40%	+0%	+100%
Forged	+100%	+0%	+100%

Whatever correction was made by the surface finish, then applying a surface treatment will have a subsequent effect taken from the table above. For example if machining reduces the endurance limit by 50%, and it is wished to recover the loss, then from the table it can be seen that cold rolling will increase the limit by +70%.

Results Setup Screen

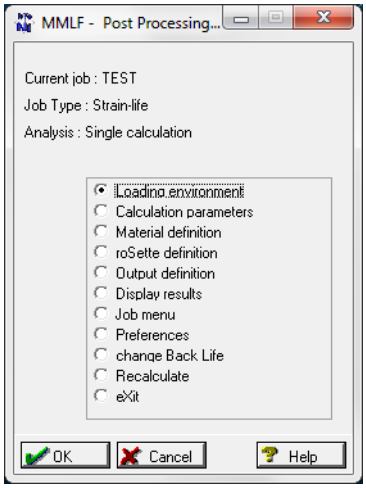


Figure 7-26 Results Setup Screen

The output files generated by any of the combination of settings to which MMLF can be set are shown below. Note that the files generated in any given run of MMLF depend upon the parameters specified in the job file.

Extension	File definition
.cyo	Three parameter cycles file
.cyc	ASCII reversals file (Wang-Brown only). It has 11 columns of data (shear strain, angle ϕ etc.) which can be viewed in a text editor or a spreadsheet.
.dam	Single parameter time correlated damage file
.dhh	Three parameter damage histogram
.epy	Y-component direct strain
.epx	X-component direct strain (parallel to gauge 1)
.epz	Z-strain (surface normal)
.gxy	XY-shear strain (engineering)
.pdm	Polar plot of damage at different critical planes (angle ϕ)
.sgx	X-component stress
.sgy	Y-component stress
.txy	XY shear stress

MMLF can generate a number of different output files depending on the analysis path chosen. If a multiple analysis has been specified in any of the input screens, then a single file, with the results name provided with any one of the following file extensions, depending on the multiple parameter, will be created.

.fal	Scale factor vs. Life
.dcl	Survival probability vs. Life
.gal	Gate vs. life

All these files have the standard X-Y paired data format and can be displayed graphically within MMLF itself or by the X-Y parameter display module, MTPD. No other output files will be created.

If, on the other hand, a single shot analysis has been carried out then MMLF can optionally create a group of files which contain different information, have the same results name and the following file extensions.

.cyo	rainflow matrix suitable for input to MCDA or MP3D
.cyc	ASCII cycles file
.dhh	damage matrix suitable for input to MP3D
.pdm	data file of damage/angle suitable for plotting with MPOD or MTPD
.ep	strain component
.epy	strain y-component
.epz	strain z-component

.gxy	strain xy-component
.sgx	stress x-component
.sgy	stress y-component
.gxy	stress xy-component

In addition, numerical results are written to the extra details area of the damage matrix file, `.dhh` extension. The following table details the keywords and values stored.

Keyword	Value	Description
ANALYSIS	MLF	Program responsible for analysis
METHOD	Wang-Brown+Mean	Mean stress correction method
DCRIT	50.0	Probability of survival used
FACTOR	1	Scale factor
GATE	0.0	Hysteresis gate used
MINER	1	Miner's constant used
MATERIAL	MANTEN	Material used
EQUUNITS	Repeats	Equivalent units
NUMEQUNI	1	Number of equivalent units
UTS	552	Ultimate tensile strength of material use
$CYCHEXT	5161.CYO	Rainflow matrix id, for system use only
MEANLF	333	Mean life calculated

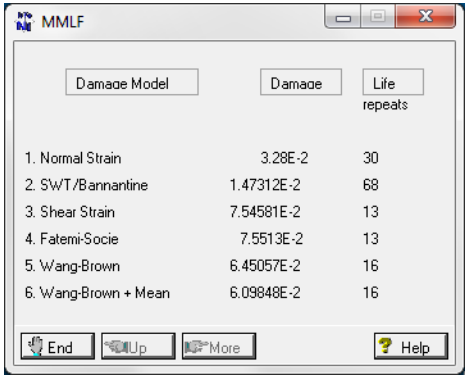


Figure 7-27 Results of a Single Shot Analysis (All Methods)

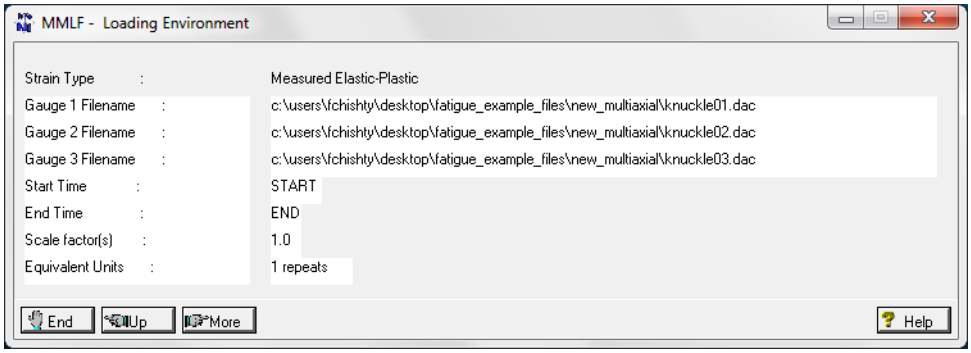


Figure 7-28 Single Shot Analysis of Just One Method

The fatigue life represents the number of repeats of the history to failure. In this context “failure” is taken to be that defined to be the initiation of an 'engineering crack', typically 2mm. The number of cycles or reversals used corresponds to the number of cycles or reversals in the strain record.

The Results From a Multiple Analysis

If a multiple analysis procedure has been specified then, on completion, MMLF will display a table of results on the screen and make a similar entry to the notebook and generate an X-Y file of results. In the case of a certainty of survival analysis, entering **ALL** in response to the question, and introducing scatter (e.g. SD=0.2 in the materials data edit screen), will cause the following table to be displayed:

Design criterion vs. Life analysis		
Number of SD's from mean	% Certainty of survival	Design life (repeats of history)
-3	99.9	1164
-2.5	99.4	1486
-2	97.7	1902
-1.5	93	2437
-1	84	3129
-0.5	69	4024
0	50	5185
0.5	31	6698
1	16	8674

An example of an X-Y plot is shown below.

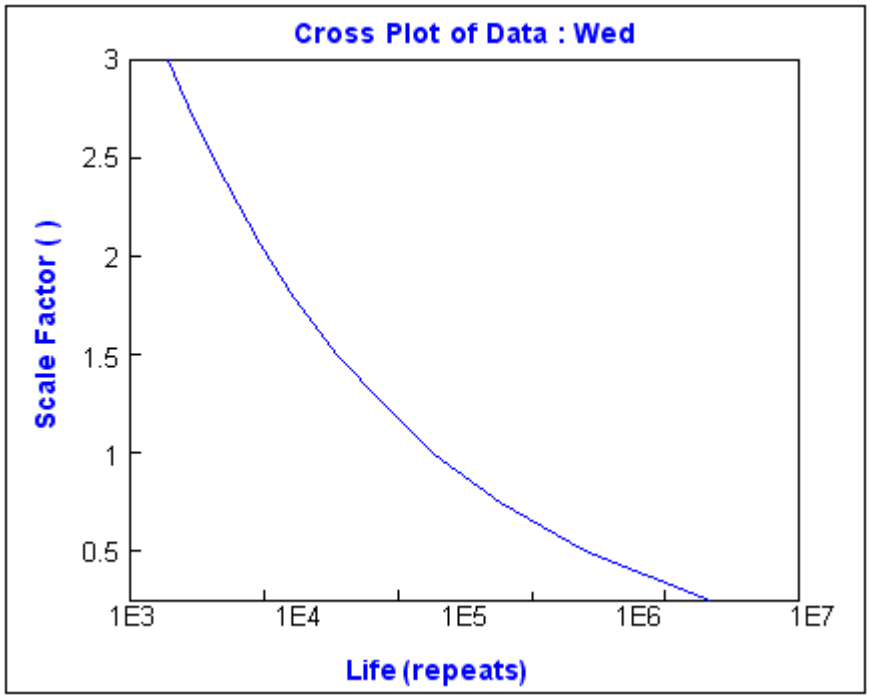


Figure 7-29 Plot of Design Criterion vs. Life

MMLF - Life

Scale Factor	Damage	Life repeats
0.5	5.32672E-	1.88E6
1	1.71035E-5	5.85E4
1.5	7.52006E-5	13298
2	1.90732E-4	5243
2.5	3.75288E-4	2665
3	6.37798E-4	1568

End Up More Help

Figure 7-30 A Multiple Analysis Results Listing

The Postprocessing Menu

The general module structure flowchart in Figure 7-20 at the start of the Module Operation section illustrates how MMLF will always lead the user to the Postprocessing Menu.

The Postprocessing menu looks like this:

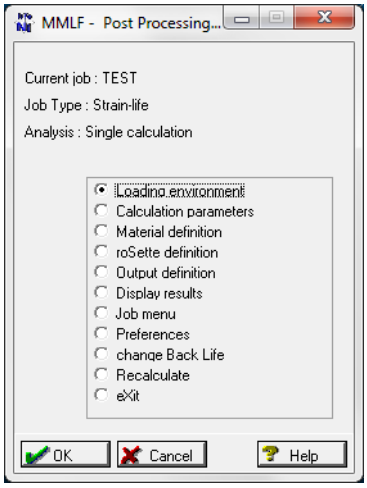


Figure 7-31 The Postprocessing Menu

The menu shown above can be mapped directly to the flow chart in Figure 7-20.

The Postprocessing options screen will be reached when:

- **all of the parameters of a new job have been entered and the calculation has been carried out.**
- **a predefined job was entered as the Jobname (see Figure 7-19). In which case the job definition screens will be skipped.**

The Postprocessing menu allows a job to have one or more of its parameters changed, and to have existing results and parameters changed and/or viewed.

Note: If any parameters are changed, the Recalculate option must be run for the changes to take effect.

The options on the menu that allow current inputs to be altered are:

Field	Description
Loading Environment	Returns control to the loading environment screen (see Figure 7-22).
Calculation Parameters	Returns control to the Model parameters screen (see Figure 7-23).
Material Definition	Returns control to the Material Definition screen (see Figure 7-24).
Rosette Definition	Returns control to the Rosette Definition screen (see Figure 7-21).
Output Definition	Returns control to the Results Setup screen (see Figure 7-26).

The other options may or may not affect the results of the calculation, but are generally of the nature of utilities rather than calculation parameters. They are as follows:

Display Results

This displays the results of the most recently calculated life estimation. Exactly what is available for display depends upon the calculation; if an X-Y file was not created then it cannot be displayed, a multiple calculation will not produce any histogram files, but will produce an X-Y plot, etc.

Essentially this option plots the 7 non-zero stress and strain components. These are:

`.sgx - `$\sigma_x(t)$`, .sgy - `$\sigma_y(t)$`, .txy - `$\sigma_{xy}(t)$`, .epx - `$\varepsilon_x(t)$`, .epy - `$\varepsilon_x(t)$`, .epz - `$\varepsilon_z(t)$`, .gxy - `$\gamma_{xy}(t)$

The full range of possibilities is explained below:

Option	Description
Stress and Strain (X-Y Plot)	Display a two parameter plot from a multiple analysis. MMLF actually loads the two parameter display module MTPD for this type of display.
Cycle Histogram	Display a rainflow cycles or reversals matrix file as a 3D plot. Such a file may have been calculated or entered into the analyzer. MMLF actually loads the plot 3D module MP3D for this type of display. If a damage matrix file (.dhh) was also created during the current analysis it can also be loaded from within MP3D. See Matrix Options, 356. For option 1 and 3 the histogram is stress range, mean stress, cycles. For option 2 and 4 the histogram is strain range-mean, max stress, cycles. For option 5 and 6 the histogram is strain range, normal stress range, reversals.
Damage Histogram	Display a damage matrix file as a 3D plot. MMLF actually loads the Plot 3D module MP3D for this type of display. If a cycles matrix file (.cyh) was also created during the current analysis it can also be loaded from within MP3D. See Matrix Options, 356.
Time-damage Plot	Displays the original input strains, together with a time-correlated damage plot. See , 1143.
Damage Polar Plot	This option displays a polar plot that shows log (damage) against orientation ϕ. The plot is created for the .pdm file, which is a two parameter file. Case A and B damage are plotted separately. See Polar Display (MPOD), 820.
Results Listing	This option will list the full results of the current life estimation in tabular form, i.e. not as a graphic. Such listings are carried out within MMLF. If parameters have been changed but *not* Recalculated, then the unrecalculated data will NOT be listed.
List Last Job	This option will list the full set of parameters used for the current life estimation, in tabular form, i.e. not as a graphic. Such listings are carried out within MMLF. If parameters have been changed but not Recalculated, then the unrecalculated data will NOT be listed.

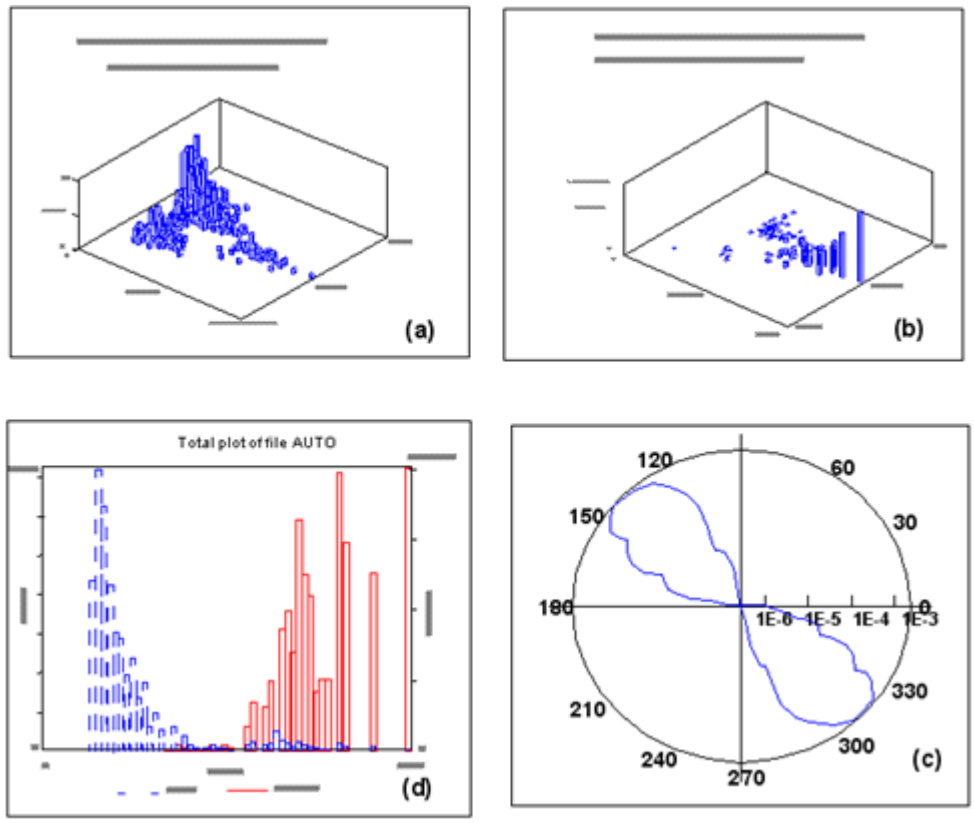


Figure 7-32 Sample MMLF plots. Cycles (a), Damage (b), Polar Plot (c), and Damage Analysis (d)

Job Menu

The Job utilities menu has three options:

Option	Description
New job	Will send the user to screen 1 of MMLF where a new job can be defined.
Save job as	Will save the current job under a new name. The current job remains active. This option is useful for preserving the current job before changing it.
List job	Will list all of the details of the current job (but not the life estimation results).

Preferences

The personal preferences menu allows some aspects of MMLF's operation to be customized. These are:

Option	Description
Material checking	MMLF normally checks material parameters to ensure that they are within certain limits. Such limits ensure that the numbers are within expected ranges, and also trap numbers that may cause arithmetic errors, e.g., the log of a negative number. Such self-checking can be turned off if required.
Back sensitivity	BACK calculations are iterative life estimations which determine the value of certain parameters necessary to achieve a specific life. Normally MMLF considers the calculated life to be estimable to within 5%. This assumption can be altered with this option.
Stress Units	The units in which material parameters are displayed can be selected from a submenu from this option.
Auto-overwrite	This option can be used to specify if existing output files are to be automatically overwritten by new files with the same name. Auto-overwrite=Yes should be used with caution because it suppresses the Overwrite Existing file Yes/No warning, and may cause a loss of data. On the other hand, it can save a lot of disk space.
Curve method	MMLF creates a stepwise cycle stress strain increment table for the plastic section of the curve based on Ramberg-Osgood. The method of curve creation can be selected with this option.

Change Back Life

Allows a value to be entered for the life of the back calculation.

> **Note:** A back calculation may fail if the target life is greater than the cutoff point of the S-N curve or strain life curve currently being used. So exercise caution when specifying large numbers as the target for back calculation e.g. >1E5.

Recalculate

In order to see the results of any change to the parameters in a job file, this option must be selected. Until this option is selected any changes will be ignored, and the job and results files will not be updated.

eXit

Exits from MMLF.

MMLF Environment Keywords

MMLF writes or reads the following entries to or from the local user environment.

Writes		Reads	
.MLFFJB	last job file	PFSTUNI	match units
$PAIRED	last X-Y file	BACKTOLR	back life calculation tolerance
$LASTPRG	last program	MATCHECK	default status of material checking
$HISTOG	last histogram file	FATAUTOV	default status of auto-overwrite
.CYH	last cycles histogram file		

MMLF Batch Operation

It is recommended that, by default, the /OV=Y keyword be included in every batch command line, since if it is omitted and an output file with the specified name already exists, batch operation will cease. To finish an MMLF batch line use /OPT=X to exit from the postprocessing screen.

When batch processing with a series of different inputs, it is necessary to use a new batch line and option definition for each new input. The new line must specify the option from the postprocessing menu into which the new input will go.

A list of MMLF's batch keywords:

/JOB	Job File name
/CREate	Confirm Creation of New Job. Y,N
/STYPE	Strain Type. M,E
/FTYPE	File input method. I,G
/IINP1,/INP2,/INP3	Input file names
/GEN	Generic file name
/CHA	Channel numbers
/EQUNITs	Equivalent Units
/NUMEQUnits	Number of Equivalent Units
/STArt	Start Time for Analysis
/END	End Time for Analysis
/GATE	Strain cycle gate
/FACTor	Scale Factor for input Loading
/MATENTry	Mode of Material Parameter Entry. L,E,G
/MATSRC	Source of materials data. S,U

/DBNAME	Material Database Name
/MATname	Material Parameter Dataset Name
/EDIT	Edit Material Dataset. Y,N
/FSP	Fatemi-Socie Parameter
/BMP	Brown-Miller Parameter
Parameter Editing & Entry:	
/UTS	Ultimate Tensile Strength
/YM	Young's Modulus
/YS	Yield Strength
/SF	Fatigue Strength Coefficient
/BASQ	Fatigue Strength Exponent
/EF	Fatigue Ductility Coefficient
/COFF	Fatigue Ductility Exponent
/NP	Cyclic Hardening Exponent
/KP	Cyclic Hardening Coefficient
/CUTOFF	Endurance Limit Cut-off
/SERRE	Elastic Standard Error
/SERRP	Plastic Standard Error
/SERRC	Cyclic Standard Error
/RATIO	R - Ratio
Parameter Generation:	
/GENTYP	Generic Material Type. S,A,T,O
/UTS	Ultimate Tensile Strength
/SERR	Standard Error of Log strain
/RAREA	Reduction in Area
/SURFace	Surface Finish - POLished, GRound, GOod Machined, Average Machined, POOrly Machined, HOt Rolled, Forged, Cast, Water Corroded, Sea Corroded, User-Defined
/TREATment	Surface treatment - None, Nitrided, Cold rolled, Shot peened, All
Calculation Parameters:	
/DESign	Design Criterion Percent
/METHod	Damage accumulation method. 1-7
/MINERs	Value of Miner's Constant
/ANSTArt	Start point for stress-strain tracking. S,M

/TYP	Rosette Type. R,T,D
/CON	Rosette Construction. S,P,U
/KT1,/KT2,/KT3	Transverse sensitivity
/POIS	Poisson's Ratio of Gauge
/OUTput	Generic Name for Output Files
/OPTion	Postprocessing Options. L,C,M,S,O,D, J,P,B,R,X
/RESOPTion	Results Display Options. S,C,D,T,O,R,J
/JOBOPTion	Job File Options. N,S,L
/PRFOPTion	Preference Options. M,S,B,A
/UNIOPTion	Units Options. MP,P,K,N,MN
/NEWJOB	Start Another Job. Y,N
/LIFE	Life required for Back Calculation
/MATCHK	Material Parameter Checking. Y,N
/TOLER	Sensitivity of Back Calculation
/OVerwrite	Overwrite Existing Output Files. Y,N
/AUTOVerwrite	Overwrite Without Confirmation. Y,N
/PLOt	Whether hardcopy is required. Y,N
/PLTNAM	Hardcopy file name

Multiaxial Fatigue Theory

Before attempting to read and understand this section, it is highly recommended that you be intimately familiar with basic fatigue theory as described in Fatigue Theory (Ch. 16).

Most engineers, if they consider the possibility of fatigue failure at all, adopt the simple S-N approach developed during the last century. Modern software design tools such as MSC Fatigue incorporate fatigue concepts such as Miner's rule (1920s), the strain-life approach (1950s), and linear elastic fracture mechanics (1960s). These ideas are well tried and tested and their limitations well understood. Universities and research departments of some companies in major industries, notably power generation, ground vehicle and aerospace, are engaged in active research into new areas. Theories in these areas are much less well-established and understood. Therefore, predictions based on these ideas should be treated with some caution. Included in this category are elastic-plastic fracture mechanics, fatigue life in the short crack regime, and multiaxial fatigue.

Multiaxial Stress-Strain State

Up until recently, uniaxial cyclic loading has been implicit in the discussion of both high- and low-cycle fatigue. However, many real design situations, including rotating shafts, connecting links, automotive and aircraft components and many others involve a multiaxial state of cyclic stress. This often means that, at any

point, the directions of the principal stresses can vary during the loading cycle and are therefore a function of time. Furthermore, the magnitudes of the principal stresses themselves may no longer be proportional to each other. Both these effects complicate the analysis required for the prediction of fatigue behavior. To some extent, the situation can be clarified through an understanding of the differences in the stress-strain states that prevail under simple uniaxial loading and nonproportional multiaxial loading.

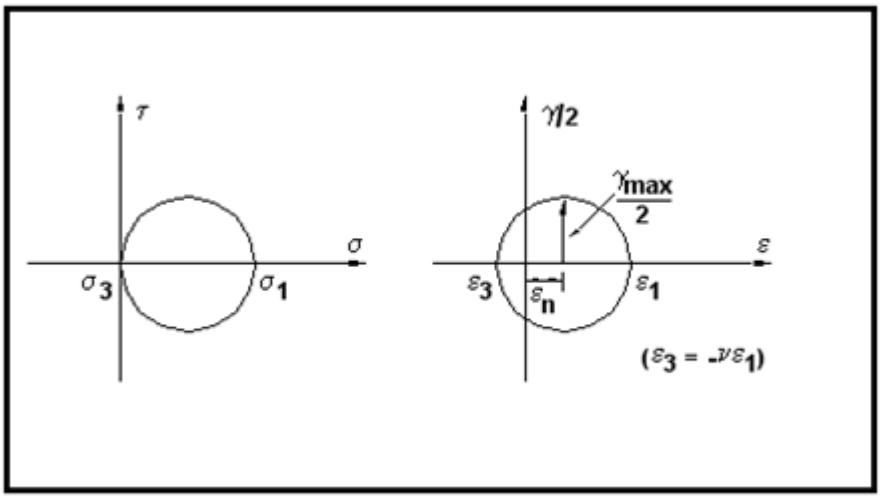


Figure 7-33 The Stress-Strain State in Uniaxial Tension

Separate Tensile or Torsional Loading

A snapshot, in terms of the Mohr's circle representation, of the stress-strain state prevailing under a static load in both tension or torsion loading is presented in Figure 7-33 and Figure 7-34. For uniaxial tension, $\sigma = \sigma_1, \sigma_2 = \sigma_3 = 0$ and $\varepsilon = \varepsilon_1, \varepsilon_2 = \varepsilon_3 = 0$ the Mohr's circles for stress and strain appear as in Figure 7-35.

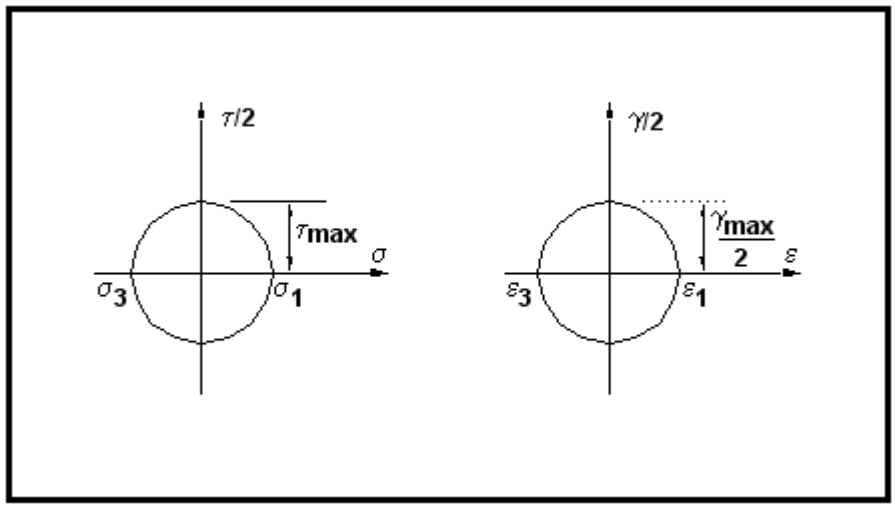


Figure 7-34 The Stress-Strain State in Torsion

For pure shear as in the case of a torsion test, $\sigma = \sigma_1, -\sigma_3, \sigma_2=0$ and $\varepsilon = \varepsilon_1, -\varepsilon_3, \varepsilon_2=0$ (the Mohr's circles for stress and strain appear as in Figure 7-34). During cyclic loading, the diameter of the Mohr's circle varies depending on the magnitude of the cyclic stress or strain. Figure 7-35 illustrates this effect for bar subjected to a constant amplitude uniaxial strain cycle.

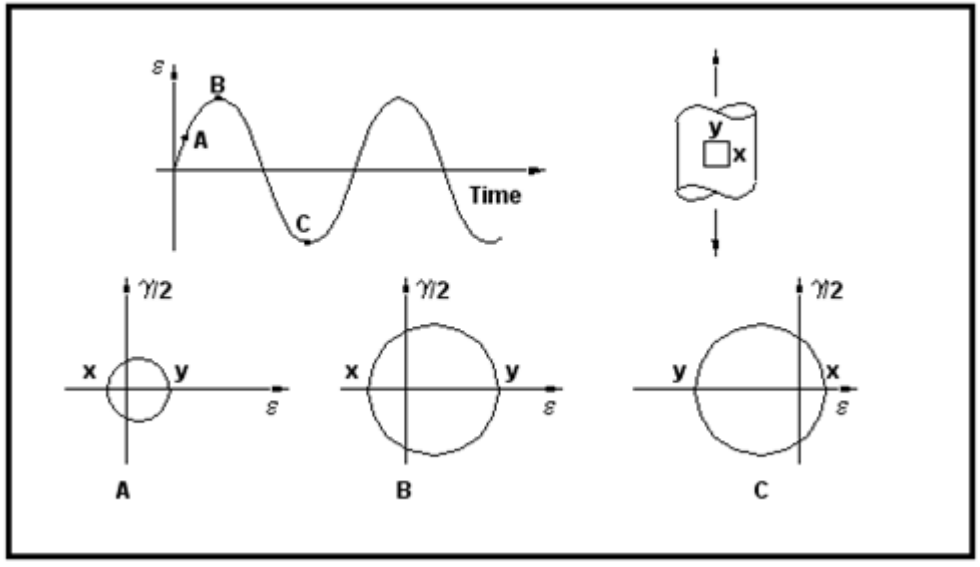


Figure 7-35 Mohr's Circles for a Uniaxial Strain Cycle

Combined Tensile and Torsional Loading

It often happens in practice that separately-induced load cases can result in *non-proportional* loading environments. Figure 7-36 can be used to illustrate the point by detailing the variation of axial and torsional strain with respect to time during the course of a complete fatigue cycle. Note that the loading path is no longer a straight line between normal and shear strain but rather a rectangular function. Note also that the direction of the principal strain is no longer fixed (i.e., the line x-y subtends an angle α with the strain axis).

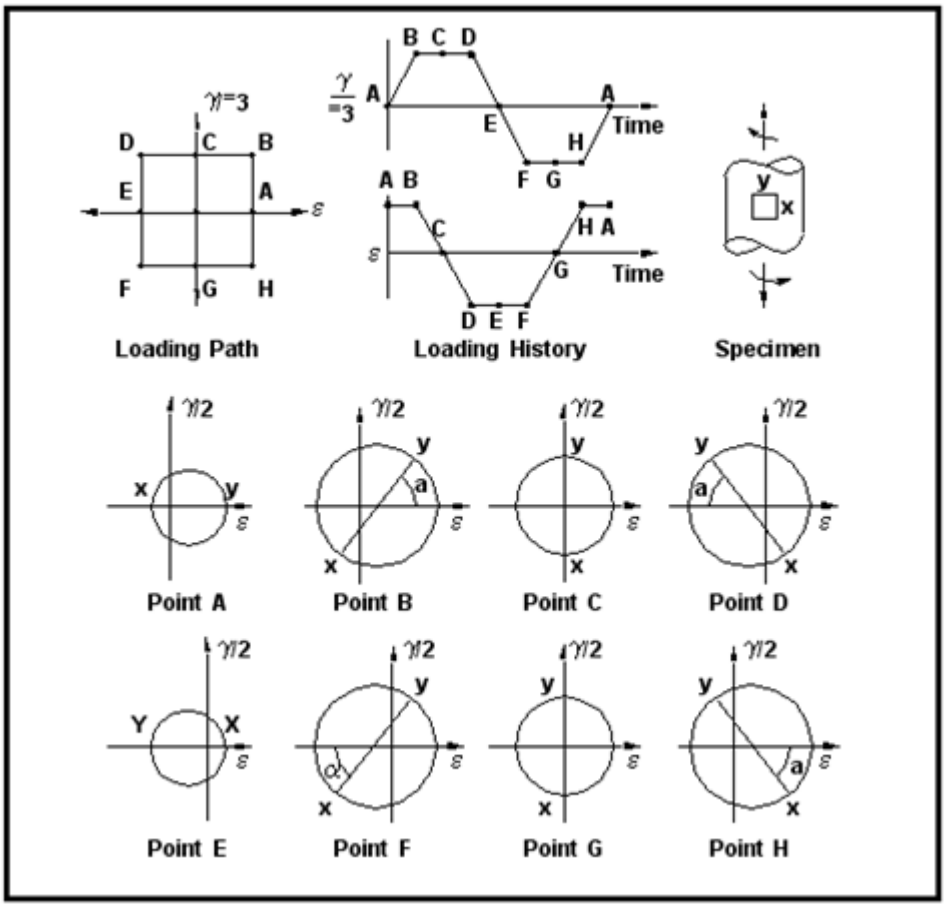


Figure 7-36 Non-proportional Loading Sequences

Mechanisms of Fatigue Damage Accumulation

Fatigue damage can be understood to be a process in which cracks initiate and then propagate until final failure. Crack initiation has been demonstrated to be the result of reversed plastic flow (i.e., the forward and backward motions of dislocations along the slip planes of metallic crystals). Slip occurs along planes of

maximum shear, and will first appear in those grains which are most favorably oriented with respect to the maximum applied shear stress. In 1961, Forsyth proposed a two stage model for early crack growth.

Stage I is a period of nucleation and crystallographically orientated growth following immediately after initiation and is confined to shear planes. In this phase, both the shear stresses and strains and the normal stresses and strains are the moduli which control the rate of crack extension.

Stage II growth is growth which occurs on planes which are orientated perpendicular to the maximum principal stress range. In this phase, the magnitude of the maximum principal stresses and strains dominate the crack growth process.

The proportion of life spent within each of these modes has been shown to depend on material type, ductile versus brittle, loading mode and amplitude. It is interesting to note that even in the case of non-proportional loading, Stage I fatigue cracks nucleate and grow on the plane of maximum shear strain range (which is subjected to the largest amplitude of normal strain).

Since *initiation life* refers to the time taken to develop an *engineering crack*, the initiation phase may include both stages of crack development. In practice, Stage I or Stage II may dominate lifetime. In uniaxial fatigue, this is not so much of a problem as the controlling parameters in both cases are directly related to the uniaxial stress or strain. In multiaxial fatigue this is no longer the case. Furthermore, the Stage I to Stage II transition is sensitive to the material and the precise nature of the loading. It follows that it is very difficult to apply general rules, and a model that produces good predictions for a given material and loading may no longer hold true when applied to a different situation. To add further confusion, there may be interaction between damage on different planes. That is to say, cracks growing on a particular plane may actually impede the progress of those growing on a different plane and lead to an increase in fatigue life.

Any theory of multiaxial fatigue must attempt, to a greater or lesser extent, to incorporate some of these physical observations if it is to have any chance of being successfully applied to real situations.

Theories of Multiaxial Fatigue

Traditionally, the approach to the design of components subjected to multiaxial loading is to make the following fundamental assertion:

Failure under a multiaxial loading is predicted to occur, according to the theory associated with a particular *modulus*, if and when the cyclically induced magnitude of that modulus, is sufficiently large that failure would occur in the uniaxial state for an identical magnitude of the same modulus.

The mechanical modulus referred to above, is a measurable quantity such as principal stress, principal shear stress or distortion energy. The range of the modulus may be defined by any two appropriate and independent values of it such as σ_{max}, and σ_m or σ_{max} and σ_{min}.

This philosophy has lead to what is usually referred to as *equivalent stress-strain* approach, where an equivalent stress or strain modulus is calculated under multiaxial loading and then applied to uniaxial data.

A problem with all of the methods is that they do not take into account the fact that fatigue is essentially a directional process, with damage (i.e., cracking) taking place on particular planes. In addition, there are problems in applying any of these methods to situations of non-proportional loading (i.e., where the number of load inputs is greater than one, and these loads have a non-constant ratio or phase relationship).

This has led to a much greater research emphasis being placed on understanding the underlying mechanisms of fatigue damage accumulation under multiaxial loading. Out of this work has arisen a somewhat different approach to fatigue life estimation based on predicting the extent of damage in specific directions and planes within the component. This methodology is usually referred to as the *critical plane* approach.

Equivalent Stress-Strain Approaches

These approaches are based on extensions to static yield theories. They assume that lifetimes for fatigue under multiaxial loading can be predicted by substituting combined stress or strain parameters in the uniaxial stress-life or strain-life equations (i.e., by calculating an equivalent uniaxial stress or strain for a given multiaxial loading). The main stress and strain parameters used are the maximum principal, the maximum shear (related to the Tresca criterion), and the von Mises or octahedral. The big advantage of this kind of approach is that it enables the large amount of uniaxial fatigue test data available in the literature or in data banks to be applied to multiaxial situations.

The von Mises method has gained widest acceptance, but all these methods have drawbacks. In the Tresca criterion, the median principal stress does not affect the equivalent stress or strain, and neither of the von Mises or Tresca criteria varies with the application of a hydrostatic stress, contrary to experimental evidence.

Before proceeding further with a more detailed account of the various equivalent stress-strain theories evolved to account for *dynamic* multiaxial stresses, it is worth examining the basis of the stress-strain combination procedures developed to account for *static* failure under multiaxial stresses.

Combined Stress Theories of Static Failure

Predicting failure and establishing a geometry that will avoid failure is relatively straightforward if a component is subjected to a static uniaxial stress. It is only necessary to refer to a uniaxial stress-strain curve for the material of interest and ensure that the service loading is less than some critical value. For example, if yielding is the failure mode to be avoided, then failure would be predicted to occur when the maximum principal stress exceeds the yield stress. However, if the component under consideration is subjected to a multiaxial loading, failure prediction is much more complicated. It is no longer possible, for example, to predict when yielding will occur through a consideration of the maximum principal stress alone because the other principal stresses will also have an influence.

A large number of multiaxial tests would be required in which all of the stress components would have to be varied over their entire range of values in all possible combinations in order to adequately describe the above situation. Such a testing program would be prohibitively expensive and perhaps not even possible for certain stress states. When faced with such complex situations engineers always attempt to develop a theory that relates behavior in the complex situation to behavior in a simple easily evaluated test through some characteristic *modulus*. Specifically, when it is required to predict the failure of a component subjected to multiaxial loading, it is usually to utilize a theory that relates failure in the multiaxial state to failure in a tension test through a well chosen modulus such as stress, strain or energy. To be useful such moduli must be calculable in the multiaxial state of stress and readily measurable in the uniaxial evaluation test. Therefore, a combined stress failure theory must contain three essential ingredients:

1. It must provide a model that relates external loading to the stresses and strains at the critical point in the multiaxial state of stress.
2. It must be based on physical properties of the material that are measurable.

3. It must relate the calculable mechanical modulus in the multiaxial stress state to a measurable criterion of failure based on the critical physical properties determined in a simple tension test.

A number of combined stress failure theories that contain these ingredients have been postulated. Numbered amongst these are the following: the maximum principal stress theory, the maximum principal strain theory, the maximum shear stress theory, the total strain energy theory, the distortion energy theory, Mohr's failure theory. Two of these theories, the distortion energy and the maximum shear stress, will be developed in more detail with failure being considered to be yielding.

Yielding Criteria for Ductile Metals

As mentioned in the example above, yielding under uniaxial loading begins at the yield stress, σ_y. Under combined loading, it is expected that yielding will occur at some particular combination of principal stresses. At present, there is no theoretical way of deducing the relationship between the stress components to correlate yielding for a multiaxial stress state with yielding in the uniaxial tension test. Therefore, yield criteria are empirical relationships. However, a yield criterion must be consistent with a number of experimental observations, the main one being the fact that pure hydrostatic pressure does not cause yielding in a continuous solid. This for example, rules out the use of the maximum principal theory since it would predict yielding failure when the magnitude of the maximum principal stress $\sigma = \sigma_1 = \sigma_2 = \sigma_3$ becomes equal to the yield stress σ_y.

Since the hydrostatic stress component of a complex stress state does not influence the stress at which a material yields, the yield criterion must be based on the stress deviator. Moreover, for an isotropic material the yield criterion must be independent of the choice of axes (i.e., it must be an invariant function). These considerations lead to the conclusion that the yield criterion must be some function of the invariants of the stress deviator. At present, there are two generally accepted criteria for predicting the onset of yielding in ductile metals.

von Mises' or Distortion-Energy Theory

In 1913, von Mises proposed that yielding will occur when the second invariant of the stress deviator, J_2 exceeds a critical value, i.e.,

$$J_2 = K^2 \tag{7-24}$$

where $$J_2 = 1/6[(\sigma_1 - \sigma_2)^2 + (\sigma_2 - \sigma_3)^2 + (\sigma_3 - \sigma_1)^2]$$

The constant k can be evaluated and related to yielding in a tension test by realizing that $\sigma_1 = \sigma_o$, and $\sigma_2 = \sigma_3 = 0$, and by substituting this for J_2,

$$k^2 = \frac{\sigma_o^2}{3} \tag{7-25}$$

and so the von Mises' prediction of yield in terms of the principal stresses becomes,

$$\sigma_o = 0.7071[(\sigma_1 - \sigma_2)^2 + (\sigma_2 - \sigma_3)^2 + (\sigma_3 - \sigma_1)^2]^{0.5} \tag{7-26}$$

and yielding will occur when σ_o exceeds the monotonic yield stress. In terms of the (x,y,z) component stress, it may be written as:

$$\sigma_o = 0.7071[(\sigma_x - \sigma_y)^2 + (\sigma_y - \sigma_z)^2 + (\sigma_z - \sigma_x)^2 + 6(\tau_{xy}^2 + \tau_{yz}^2 + \quad)]^{0.5} \tag{7-27}$$

By considering the state of stress for pure shear as in a torsion test, it is possible to identify the constant k. In torsion,

and

$$\sigma_2 = 0$$

at yield

$$\sigma_1 = -\sigma_3 = \tau$$

$$\sigma_1^2 + \sigma_1^2 + 4\sigma_1^2 = 6k^2 \tag{7-28}$$

$$\sigma_1 = k$$

This means that k represents the yield stress in pure shear, torsion. Therefore, the von Mises' criterion predicts that the yield stress in torsion will be less than that for uniaxial tension according to:

$$k^2 = \frac{\sigma_o^2}{3} \tag{7-29}$$

$$k = 0.577\sigma_o$$

In 1924, Hencky derived the von Mises' yield criterion through a consideration of distortion energy and so it is sometimes also referred to as the distortion energy criterion.

Maximum Shear Stress or Tresca Theory

The Tresca yield criterion suggests that yielding will occur when the maximum shear stress under multiaxial loading reaches the value of the shear stress under uniaxial tension test. Since according to convention σ_1 is the algebraically greatest principal stress and σ_3 the smallest, the maximum shear stress, τ_{max} is given by:

$$\tau_{max} = \frac{(\sigma_1 - \sigma_3)}{2} \tag{7-30}$$

For uniaxial tension, $\sigma_1 = \sigma_o$, and $\sigma_2 = \sigma_3 = 0$, and the shearing yield stress τ_o is equal to $\sigma_o/2$ and so

$$\tau_{max} = \frac{(\sigma_1 - \sigma_3)}{2} = \tau_o = \frac{\sigma_o}{2} \tag{7-31}$$

and

$$\sigma_1 - \sigma_3 = \sigma_o \tag{7-32}$$

For the state of pure shear, $\sigma 1 = -\sigma 3 = k$ and $\sigma 2 = 0$, the maximum shear stress criterion would predict yielding will occur when

$$\sigma_1 - \sigma_3 = 2k = \sigma_o \tag{7-33}$$

and so the maximum shear stress criterion predicts that yield in torsion will be less than in uniaxial tension according to:

$$k = \frac{\sigma_o}{2} \tag{7-34}$$

It is worth noting that the Tresca criterion appears much simpler than that proposed by von Mises and for this reason it is often used in engineering design. However, the maximum shear criterion does not take into account the intermediate principal stress. Furthermore, it suffers from the major difficulty that it is necessary to know in advance which are the maximum and minimum principal stresses.

Combined Stress States

The conditions for yielding or fatigue damage accumulation for that matter under stress states other than uniaxial or torsion loading can be studied experimentally with thin walled tubes. Axial stress can be combined with torsional loading to produce various combinations of shear stress to normal stress intermediate between the values obtained separately in tension or torsion. In addition, a hydrostatic pressure can also be introduced within the tube to induce circumferential hoop stresses. Figure 7-37 illustrates such a thin-walled tube.

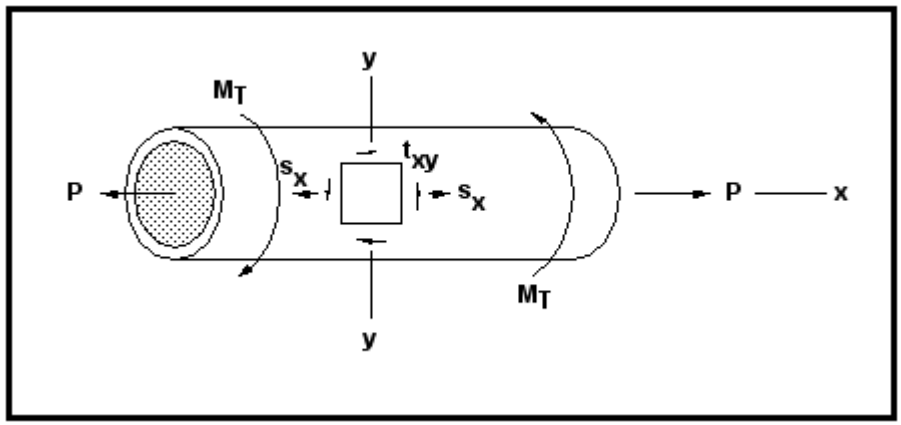


Figure 7-37 The State of Stress in a Thin-Walled Tubular Specimen

For the thin-walled specimen shown above, the principal stresses can be calculated from:

$$\sigma_1 = \frac{\sigma_x}{2} + \left(\frac{\sigma_x^2}{4} + \tau_{xy}^2 \right)^{0.5} \tag{7-35}$$

$$\sigma_2 = 0 \tag{7-36}$$

$$\sigma_3 = \frac{\sigma_x}{2} - \left(\frac{\sigma_x^2}{4} + \tau_{xy}^2\right)^{0.5} \tag{7-37}$$

Under these conditions, the von Mises' criterion would predict yielding when:

$$\left(\frac{\sigma_x}{\sigma_o}\right)^2 + 3\left(\frac{\tau_{xy}}{\sigma_o}\right)^2 = 1 \tag{7-38}$$

and the Tresca criterion would predict yielding when:

$$\left(\frac{\sigma_x}{\sigma_o}\right)^2 + 4\left(\frac{\tau_{xy}}{\sigma_o}\right)^2 = 1 \tag{7-39}$$

Both equations define an ellipse and Figure 7-38 shows that experimental results agree best with the von Mises' approach.

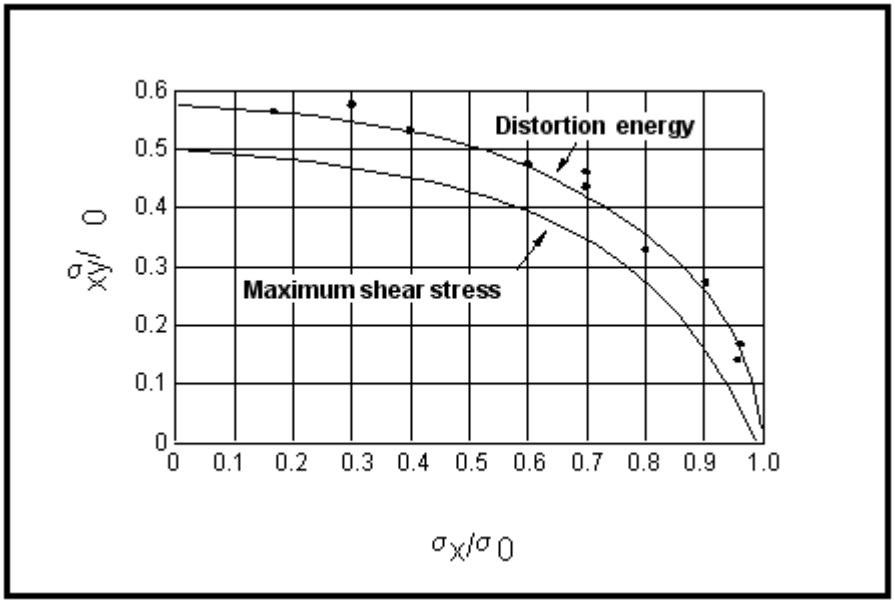


Figure 7-38 Comparison Between the Tresca and von Mises' Theories

Critical Plane Approaches

As the name suggests, critical plane approaches recognize that fatigue damage (cracking) is essentially directional and so consider the accumulation of damage on particular planes. This is in contrast to the equivalent stress-strain approaches which may be summing damage that is occurring on different planes. A range of methods for calculating damage on a particular plane is used. These methods include:

1. Simply resolving the normal strain on the plane and applying what is essentially a uniaxial analysis.
2. As above, but allowing for the multiaxial nature of the stresses through a consideration of the out-of-plane hardening.

3. Variations on the Brown-Miller approach, which consider the shear strain amplitude in the plane and the normal stress or strain.

Typically, damage is calculated for all possible planes (at, say 10 degree intervals) and the worst or critical plane selected.

Fatigue cracks normally initiate on planes of maximum shear and grow initially in mode II (shear mode), changing to mode I (opening mode) where the maximum principal may be important. In brittle materials or where initiation occurs at inclusions, growth may be in mode I from the outset. Since *initiation life* refers to the time taken to develop an *engineering* crack, this so-called *initiation* phase may include both stages of crack development. In practice, Stage I or Stage II may dominate lifetime. In uniaxial fatigue, this is not so much of a problem, as the controlling parameters in both cases are directly related to the uniaxial stress or strain. In multiaxial fatigue, this is no longer the case. Furthermore, the Stage I to Stage II transition is sensitive to the material and the precise nature of the loading. It follows that it is very difficult to apply general rules, and a model that produces good predictions for a given material and loading may no longer hold true when applied to a different situation. To add further confusion, there may be interaction between damage on different planes; cracks growing on a particular plane may actually impede the progress of those growing on a different plane and lead to an increase in fatigue life.

The Miller-Brown Parameter

In addition to the importance of the plane of maximum shear, the stress and strain normal to this plane has also been recognized to strongly influence the development of fatigue cracks. This point can be illustrated by referring to Figure 7-39.

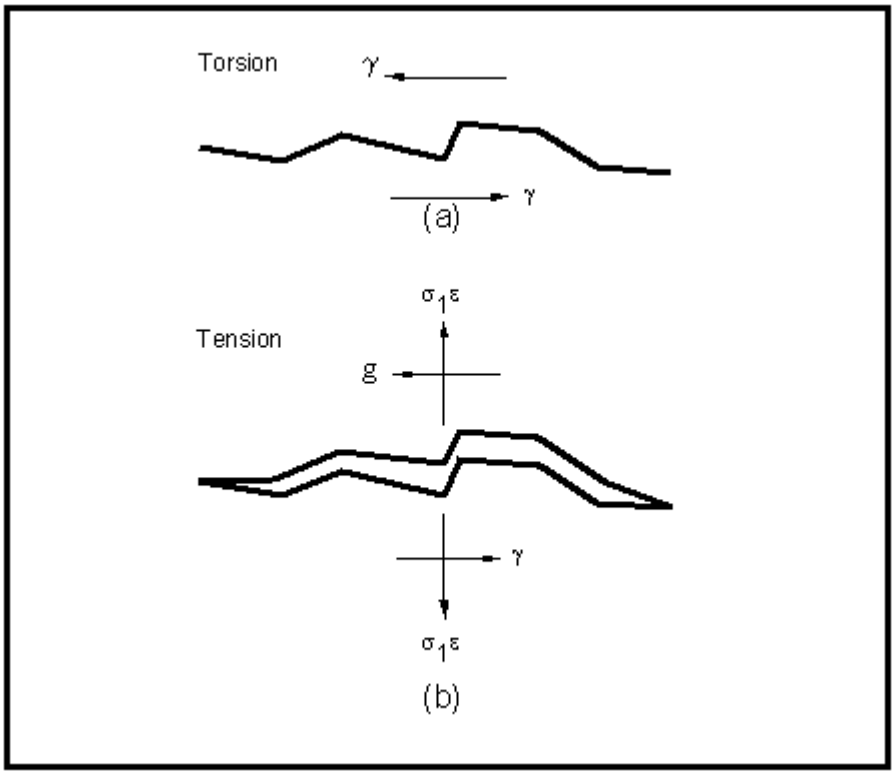


Figure 7-39 (a) Shear Strain (b) Shear Strain and Normal Stress-Strain

It is clear from Figure 7-39 (a) that the shear strain, γ, acting on the plane of maximum shear will extend the fatigue crack and, amongst other things, progress will be opposed by the friction between the crack faces. If a tensile stress or strain is applied normal to the plane, then the surfaces are separated and the frictional opposition reduced and more damage can be accumulated at the crack tips. Under these circumstances, fatigue damage can be expected to accumulate more rapidly and life to be reduced.

These ideas, whilst recognized by Mohr and others, were first formulated by Miller and Brown who presented a two parameter formulation as:

$$\gamma_{max} = f(\varepsilon_n) \tag{7-40}$$

γ_{max} is the maximum shear strain

ε_n is the strain normal to the above shear

They proposed that the maximum shear strain is the primary force in crack initiation and that the strain normal to the plane of maximum shear is a modifying factor. The above equation has been modified by Kandil and later by Fash to develop it into a damage law based on uniaxial cyclic properties:

$$\gamma^{a}_{max} + S\varepsilon^{a}_{n} = \frac{\alpha\sigma'_{f}}{E}(2N_f)^{b} + \beta\varepsilon f'(2Nf)^{c} \tag{7-41}$$

where

γ_{max} is the maximum shear strain amplitude

ε_n is the strain amplitude normal to the plane of maximum shear

The constants α and β can be calculated from:

$$\alpha = (1 + \mu_e) + S\frac{(1 - \mu_e)}{2} = 1.65 \tag{7-42}$$

$$\beta = (1 + \mu_p) + S\frac{(1 - \mu_p)}{2} = 1.75 \tag{7-43}$$

where μ_e and μ_p are respectively, the elastic and plastic Poisson's ratios and the constant S is taken to be unity. The damage law now becomes:

$$\left(\gamma^{a}max + \varepsilon^{a}_{n}\right) = \frac{1.65\sigma'_{f}}{E}(2N_f)^{b} + 1.75\varepsilon'_{f}(2N_f)^{c} \tag{7-44}$$

Note that this is similar to the Tresca formulation but with the normal strain component added, and as it stands, takes no account of the fact that the plane of maximum shear, like the principal stresses and strains, could rotate.

Non-Proportional Loadings and the Bannantine-Socie Approach

When loadings are multiaxial and non-proportional, it is possible that the directions of the principal strains (and hence also the shear strains) and their ratios may change with time. Under these circumstances, the Miller-Brown parameter is not effective, because it has no way of dealing with the shear plane rotation.

The Socie-Bannantine method has a different approach. Instead of defining the critical plane as the maximum shear plane, and calculating damage on that plane, it defines a critical plane damage parameter and searches for the plane on which the damage is a maximum. This plane is said to be the critical plane. Plane orientations can be described by means of two angles as shown in the following diagram.

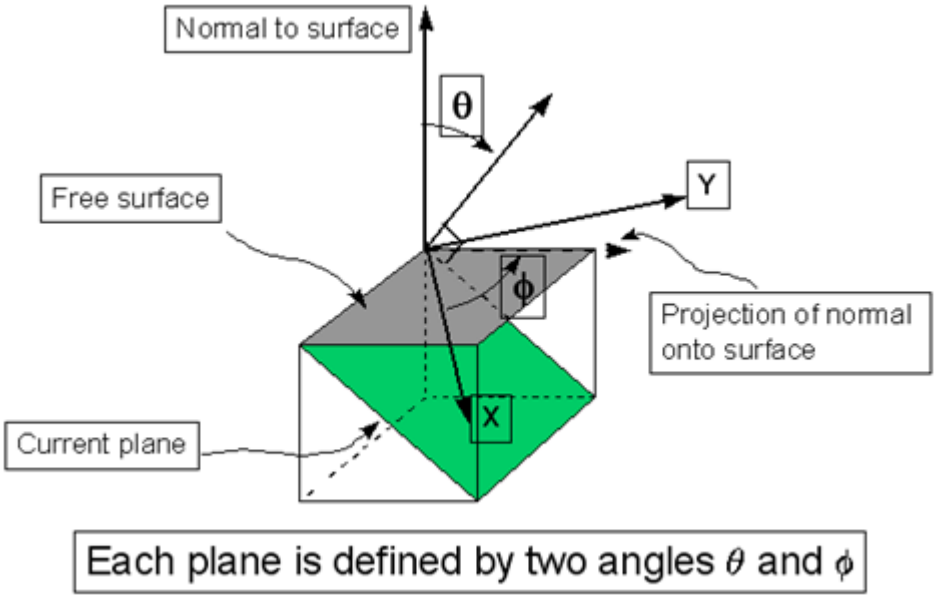


Figure 7-40 Plane Orientations Defined by θ and ϕ

In order to understand this better, it is necessary to review the behavior of short cracks. It has been noted earlier that there are two stages to crack growth, Stage I normally being shear cracking and Stage II opening mode. This is illustrated in Figure 7-41.

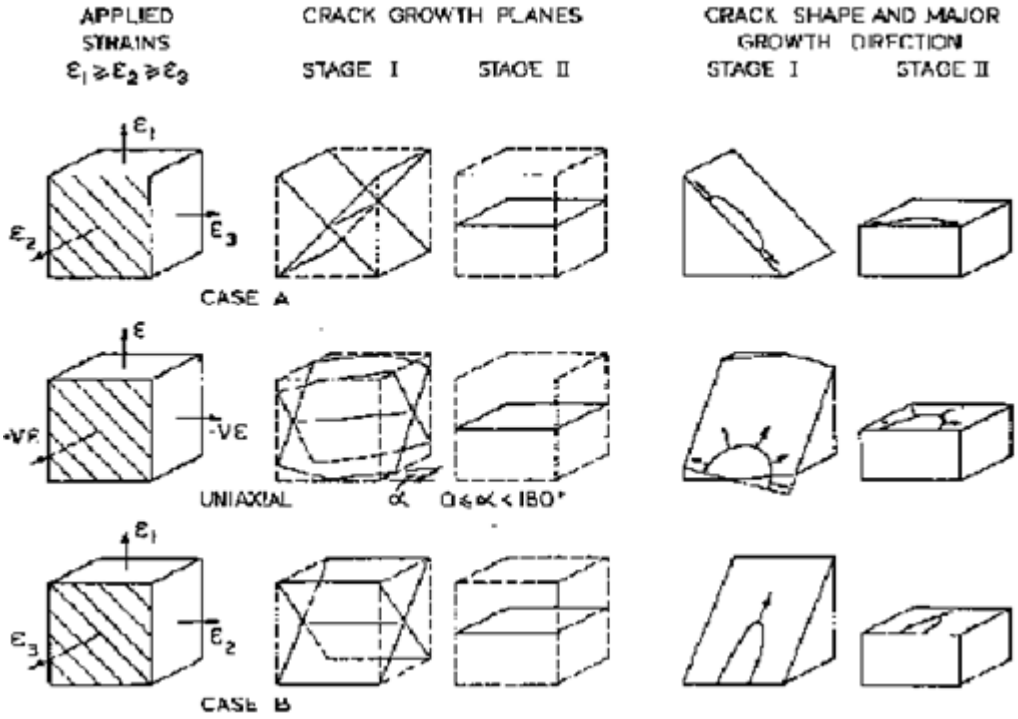


Figure 7-41 Applied Strains and Crack Growth Planes

This figure illustrates how Stage II cracking is typically normal to the maximum principal stress or strain, and how the orientation of the maximum shear planes on which the Stage I cracks grow depends on the state of stress as described by the biaxiality ratio which is the ration of the smaller to the larger in-plane principal stresses (this discussion is restricted to free surfaces). However the relative importance of shear cracking and opening mode cracking is a function of the stress state, the loading level, mean stress, and the material. A basic premise of the Socie-Bannantine approach is that the life prediction method needs to reflect the damage mechanism as seen from the crack morphology. Based on this idea, they developed a number of damage "maps" for different materials which illustrate the dominant cracking mechanism under different loading conditions.

The following figures are damage maps (schematic) for two materials under two different stress states.

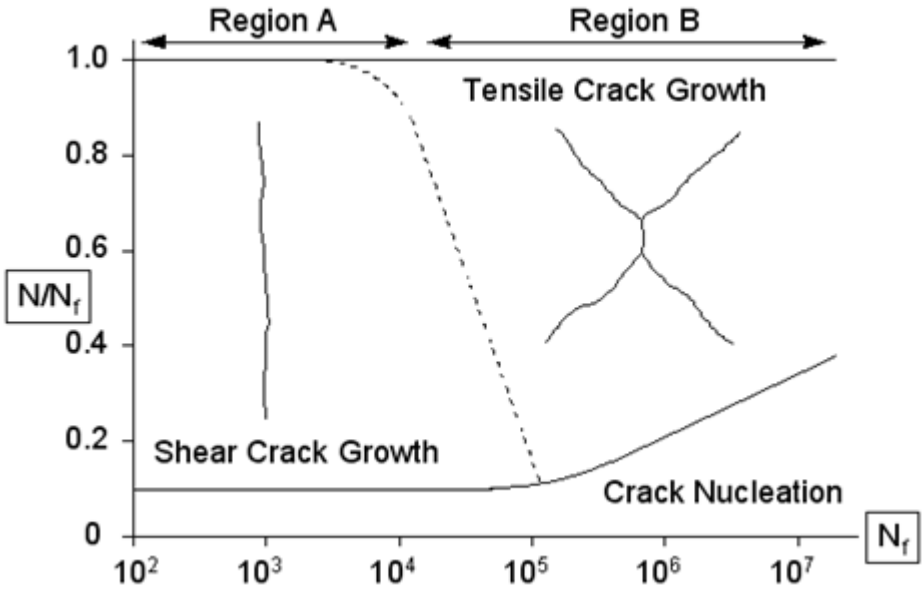


Figure 7-42 Damage Map for 304 Stainless Steel in Torsion

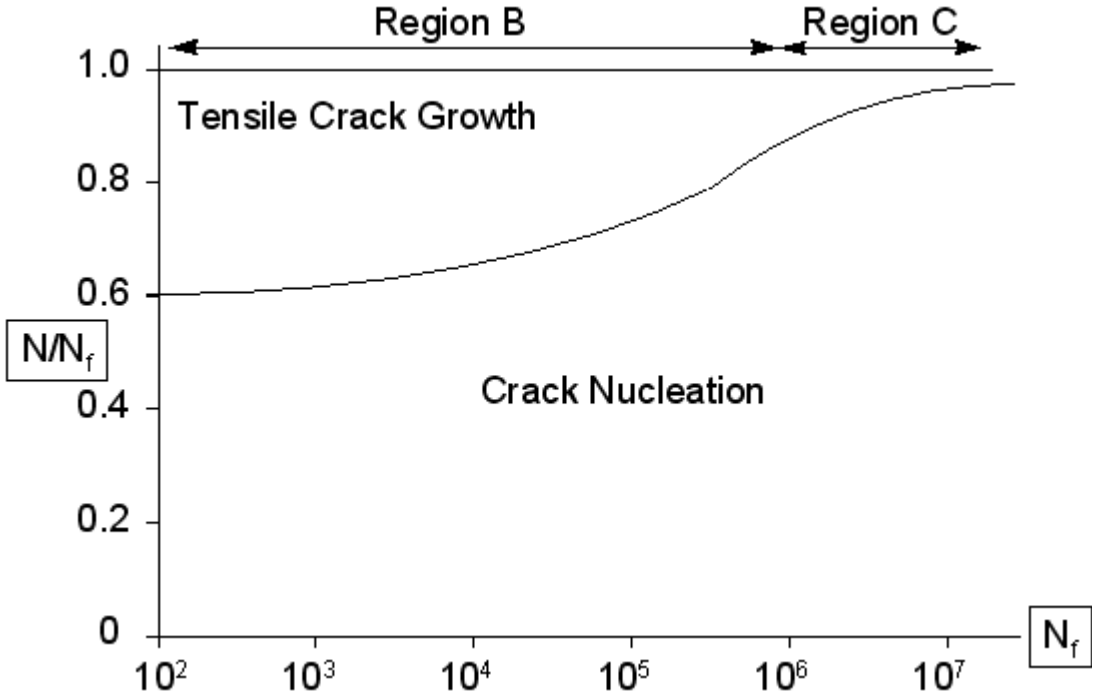


Figure 7-43 Damage Map for 304 Stainless Steel in Tension-Compression

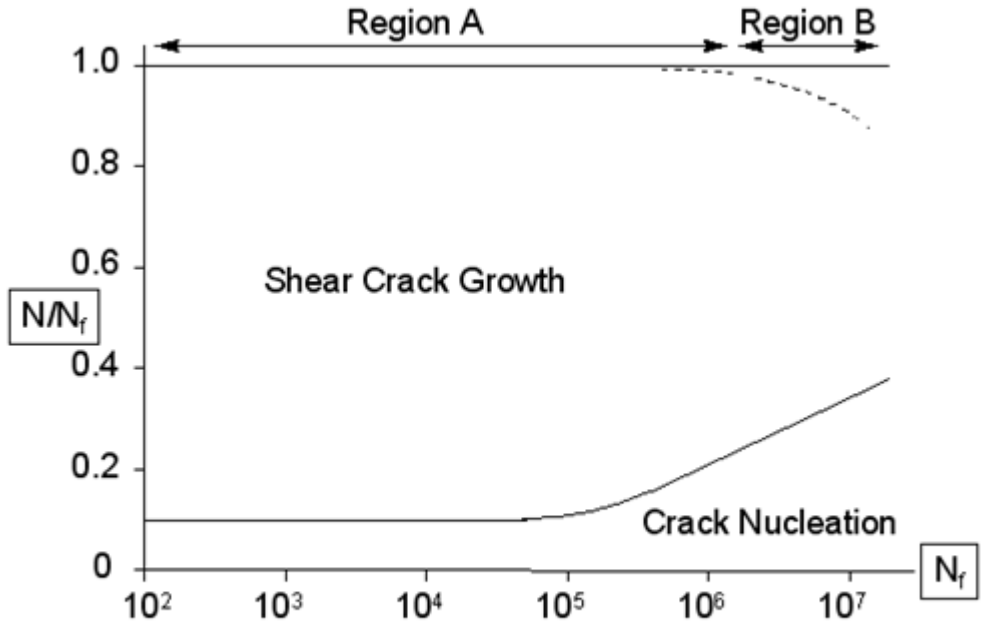


Figure 6-44 Damage Map for Inconel 718 in Torsion

Figure 7-44 Damage Map for Inconel 718 in Torsion

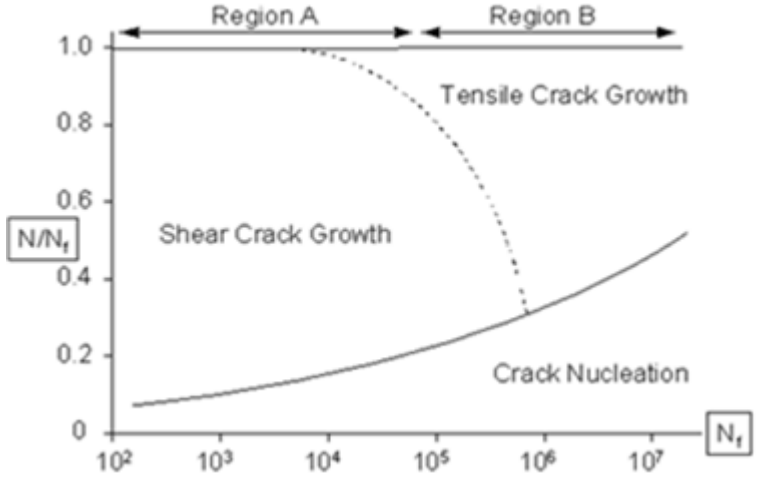


Figure 7-45 Damage Map for Inconel 718 in Tension-Compression

The Socie-Bannantine method has two damage models, one for situations where tensile cracking dominates and one for shear cracking. The equations are:

For the tensile model:

$$\frac{\Delta\varepsilon_n}{2}\sigma_{n,\,max} = \frac{\sigma_f'^2}{E}(2N_f)^{2b} + \sigma_f'\varepsilon_f'(2N_f)^{b+c} \tag{7-45}$$

being very similar to the Smith-Watson-Topper approach, and for the shear model:

$$\frac{\Delta\gamma}{2}\left(1 + n\frac{\sigma_{n,\,max}}{\sigma_y}\right) = \frac{\tau_f'}{G}(2N_f)^b + \gamma_f'(2N_f)^c \tag{7-46}$$

The shear model requires some torsional strain-life parameters. If these are not available, the above equation can be rewritten using uniaxial parameters. This formulation is devised to give the correct result under uniaxial conditions. It is this formulation that is used in MSC Fatigue:

$$\frac{\Delta\gamma}{2}\left(1 + n\frac{\sigma_{n,\,max}}{\sigma_y}\right) = \frac{(1+\nu_e)}{E}\sigma_f'(2N_f)^b\left(1 + \frac{n\sigma_f'(2N_f)^b}{2\sigma_y}\right) + (1+\nu_p)\varepsilon_f'(2N_f)^c\left(1 + \frac{n\sigma_f'(2N_f)^b}{2\sigma_y}\right) \tag{7-47}$$

In theory these models should be used to search for the plane with the greatest fatigue damage. In practice, it is adequate for the tensile model to make calculations only on planes inclined at 90 degrees to the free surface, i.e. $\theta = 90$ degrees, calculating at 10 degree intervals of ϕ, making 18 planes in all. For the tensile model, it is necessary to search both Case A $(\theta = 90)$ and Case B $(\theta = 45)$ planes making 36 planes in all. The loss of accuracy is not significant.

The Wang-Brown Cycle Counting and Damage Approach.

The Socie-Bannantine approach is a good one, but has some shortcomings.

1. The shear model often predicts the greatest damage on planes which are a few degrees away from the maximum shear plane, contrary to the physical evidence. This is a minor philosophical objection.
2. The rainflow counting method it uses is essentially a uniaxial one, and as such, when the loading is non-proportional, it can identify a different set of turning points on every plane. This is not really in keeping with the philosophy of the rainflow method. A true multiaxial rainflow counting method should correspond to the hysteresis hardening process, i.e. it should identify reversals as elastic unloading, reloading and strain hardening in a multiaxial sense.
3. It tends to make somewhat non-conservative predictions for non-proportional loadings, which are more damaging.

Wang and Brown (Ref. 62)proposed a multiaxial cycle counting method on the basis of strain hardening behavior under non-proportional variable amplitude loading. Relative stresses and strains were introduced so that a pair of turning points define the start and end points of a reversal, where the equivalent relative strain rises monotonically to a peak value. Since plastic deformation generates the driving force for small fatigue cracks, hysteresis hardening provides a physical parameter for cycle counting, analogous to rainflow counting in the uniaxial case.

Each reversal commences with elastic unloading, which is followed by reloading and plastic strain hardening up to the next turning point. The most significant turning point occurs at the highest value of equivalent strain. This is illustrated at time 0 in Figure 7-46, which shows a repeating block of a combined tension/torsion non-proportional load history. The equivalent strain is defined as the von Mises strain.

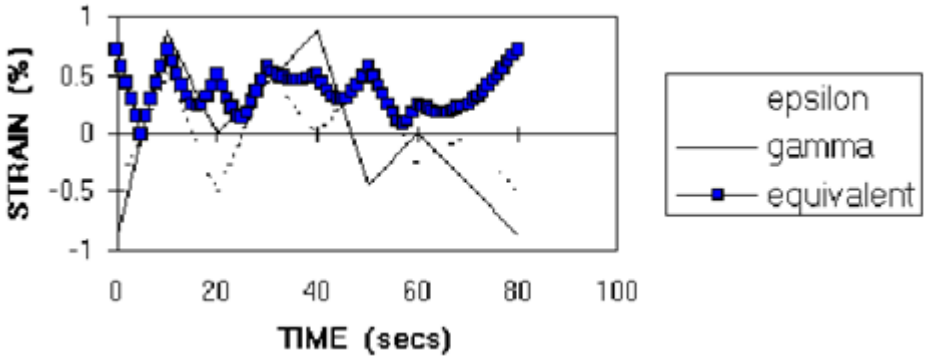


Figure 7-46 Variable Amplitude Non-proportional Strain History, Showing Applied Tensile (epsilon) and Torsional (gamma) Strains with the Absolute Equivalent Strain

The cycle counting method is illustrated by the following example. Starting from the most significant turning point, a graph is drawn for the loading block of relative equivalent strain, where relative strain represents the change of strain since time A. Figure 7-47 shows the relative equivalent strain, with respect to times 0, 10 and 20 seconds. Using the relative strain, a reversal can be defined starting from 0, up to the maximum value 10 seconds. To obtain the second reversal, the relative strain is replotted starting from the next turning point where unloading commences (at 10 seconds), and the portions of the strain hardening curve for the reversal are selected by a traditional rainflow procedure (Ref. 56). The region of unloading and reloading within that reversal is counted in the next step.

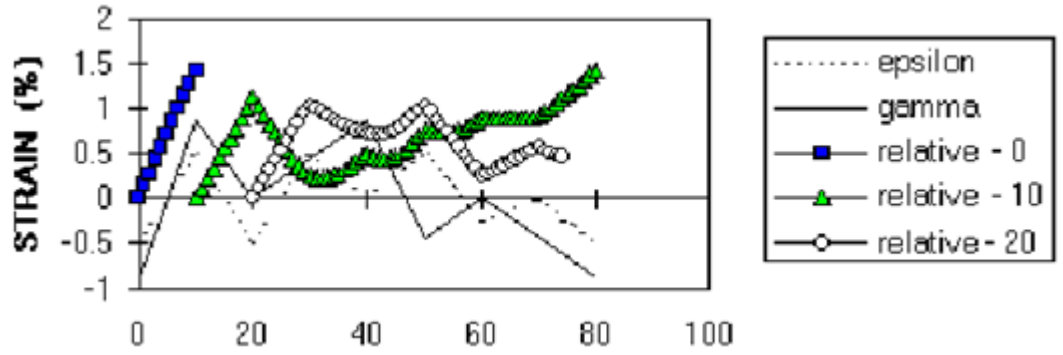


Figure 7-47 The Variable Amplitude History, Showing Relative Equivalent Strains Plotted with Respect to Times 0, 10 and 20 Seconds Respectively

Using the next turning point, relative strain is replotted with respect to 20 seconds for the subsequent continuous fragment of strain history, yielding the third reversal in Figure 7-47. This procedure is repeated for each turning point in chronological order, until every fragment of strain history has been counted.

Fatigue Damage Calculation

The counting method described above is independent of fatigue damage parameters, being based on hysteresis deformation behavior. Being unrelated to material properties, it can be integrated with any multiaxial fatigue damage model. If the counted reversals are non-proportional, a fatigue damage parameter that accounts for non-proportional straining effects is required. The path-independent damage parameter proposed by Wang and Brown (Ref. 63) has been shown to provide good correlation for several materials under proportional and non-proportional loading,

$$\hat{\varepsilon} \equiv \frac{\gamma_{max} + S\delta\varepsilon_n}{1 + \nu' + S(1 - \nu')} = \frac{\sigma_f'}{E}(2N_f)^b + \varepsilon_f'(2N_f)^c \tag{7-48}$$

where γ_{max} is the maximum shear strain amplitude on a critical plane (proportional or non-proportional), $\delta\varepsilon_n$ is the normal strain excursion between the two turning points of the maximum shear strain (that is the range of normal strain experienced on the maximum shear plane over the interval from start to end of the reversal), and $\sigma_{n,mean}$ is the mean stress normal to the maximum shear plane. The term S is a material constant (the Brown-Miller parameter) determined from a multiaxial test (typically between 1 and 2 for Case A and around 0 for Case B) and ν' is the effective Poisson's ratio. The right hand side of the equation is the same as the uniaxial strain life equation, with a Morrow mean stress correction. Mean stress is measured as the average of the maximum and minimum stress values over the reversal. The total damage induced by a loading history is calculated using Miner's rule.

Figure 7-48 is a plot of predicted life against experimental life for a variety of proportional and non-proportional tests on laboratory specimens, from (Ref. 48).

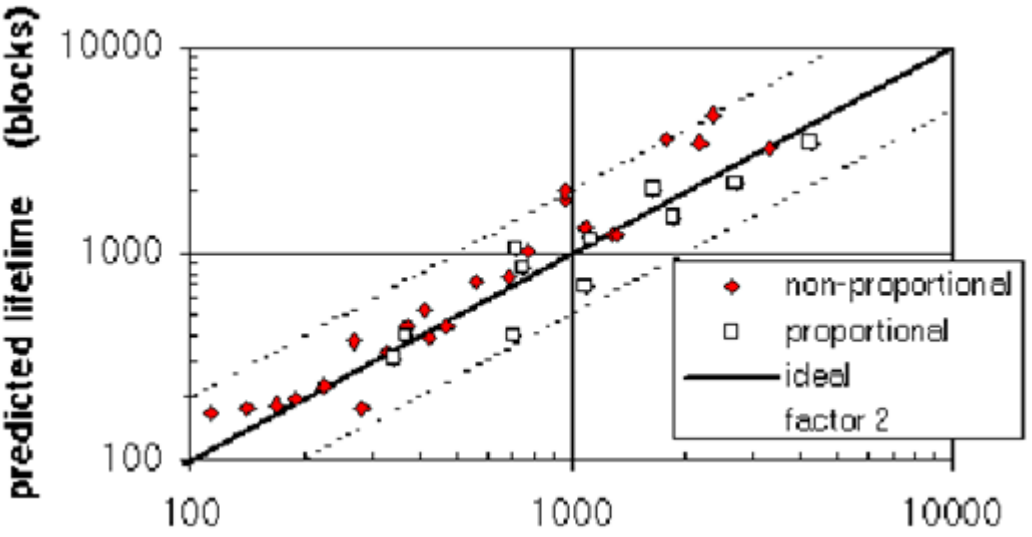


Figure 7-48 Comparison of Experimental and Predicted Results for the Wang-Brown Method

Figure 7-49 below, illustrates a comparison of experimental and predicted results for four non-proportional tension-torsion loading paths, using the Wang-Brown method, with and without the mean stress term included, the Fatemi-Socie method (shear model) and the Bannantine (tensile) model. It is clear from this picture that for most of the results, the Wang-Brown methods give the best (and also the most conservative) results. The Fatemi-Socie model is next best and the Bannantine tensile model is very non-conservative, except for the longer lasting specimens, where the cracking behavior is more likely to be dominated by opening mode.

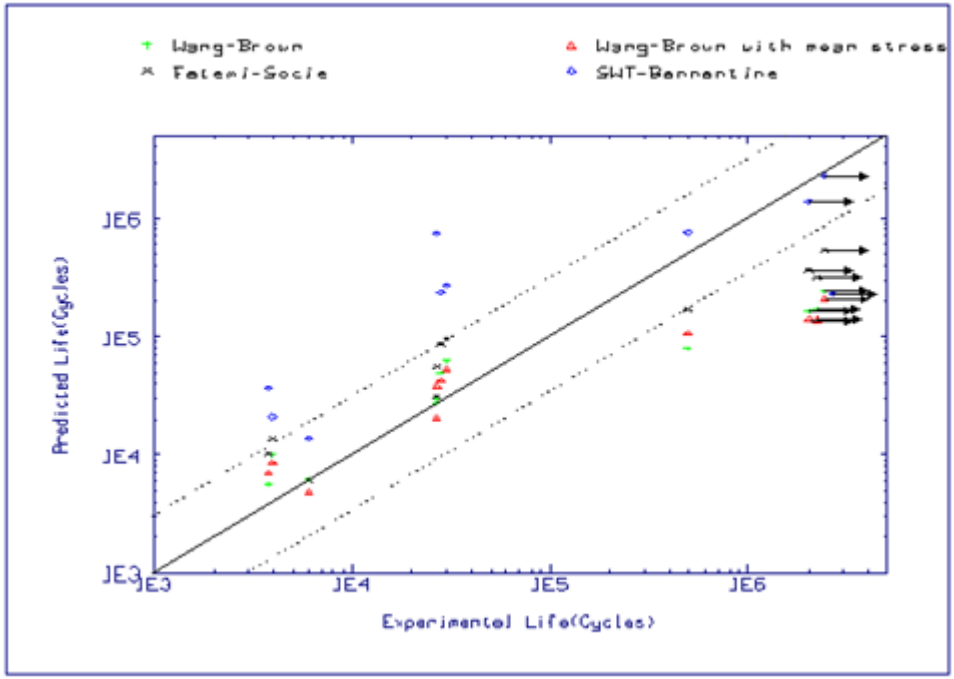


Figure 7-49 Predicted vs. Measured Lives for Various Multiaxial Methods

Multiaxial Low Cycle Fatigue Analysis

The Concept of Similitude

Analytical fatigue life estimation procedures make it possible to replace some specific fatigue testing of components and structures by analysis based on measured or predicted loadings and cyclic tests on laboratory specimens. This process depends on the assumption that the damage (fractional damage or crack extension) to the component or structure is the same as the damage to the test specimens when subjected to the same loading and environment. This assumption of similitude can be described in more detail for each of the main fatigue life estimation procedures as follows:

1. In the strain-life or crack initiation method, the damage to a particular location in a component is assumed to be the same as the damage to a smooth fatigue specimen subject to the same strains. This means that the measured or predicted strains in a component can be related to the strain-life behavior of specimens in laboratory tests to predict lifetimes. The test specimens are normally smooth and cylindrical.
2. In the material S-N method, the damage to a particular location is the same as that at the critical location of a laboratory specimen subject to the same local stresses.

3. In the component S-N method, the damage to a component is assumed to be the same as that to a test component subject to the same nominal stresses. The geometry is assumed to be the same as the test pieces which will therefore have similar stress states.
4. In the crack propagation method, the crack growth rate in a component or structure is assumed to be the same as that in a specimen (usually a compact tension specimen) which is subject to the same stress intensity factors.

Taking the case of the local strain approach, polished cylindrical specimens are normally tested in strain controlled, fully reversed, constant amplitude, axial loading (push-pull). Real components are usually not polished and experience variable amplitude loading in environments other than laboratory air. There are well established ways of correcting for these differences, using rainflow cycle counting and surface finish and environmental corrections. However, there is another problem which is that the stress state of the specimens is uniaxial (i.e., there is only one non-zero principal stress and that is always aligned with the specimen axis), this will not in general be true of real components and structures, leading to a breakdown in similitude. If we are to carry out local strain fatigue analyses on components and structures subject to complex loadings we need means of:

1. assessing the stress states prevalent in the component or structure being analyzed, and
2. relating the damage caused by the stress and strain variations in the component or structure to that in the simple uniaxial test specimens.

Stress State Characterization

Any stress state can be completely described by the principal stresses (i.e., by their size and orientation). Now, the local strain approach models crack initiation. Cracks in smooth fatigue specimens normally initiate at free surfaces, and if they initiate elsewhere in the component being analyzed (e.g., from a defect within a weld), there is a serious problem with similitude. Now the stress states at free surfaces are relatively simple because the boundary conditions at the surface dictate that there can be no direct stresses on the plane of the surface or shear stresses normal to it. It follows that one of the principal axes of stress must be normal to the surface and the other two principal stresses must lie in the plane of the surface. The same applies to principal strains, except that the direct strain normal to the surface may be non-zero. Analysis and assessment can be greatly simplified because of these conditions, particularly if the FE stress and strain results are presented in an appropriate way. The state of plane stress existing at free surfaces means that any study of surface stress states can be reduced to a 2D problem if the results are presented in a suitable coordinate system, i.e. one whose x- and y-axes lie in the plane of the surface. In such coordinate systems, the z-normal stress and the x-z and y-z shear stresses and strains should all be zero. The deviation of these results from zero is a good indicator of the quality of the local stress information provided by the FE analysis. MSC Fatigue includes a utility which can be used to transform the surface stress and strain results to local coordinate systems where the z axes represents a surface normal. The surface stress state can now be investigated by looking at the x, y, and x-y stresses only. It follows that the surface stress state can be completely described by the two in-plane principal stresses and the orientation of one of them, the other being at right angles to the first.

For clarity, the conventional definition of the principal stresses, in which $\sigma_1, \sigma_2,$ and σ_3 are ordered with σ_1 being most positive, is replaced (unless otherwise stated) by the following convention:

1. The principal stress normal to the surface is designated $\sigma_3 = \sigma_Z = 0$.

2. The other two principal stresses, σ_1 and σ_2, are ordered in magnitude, so σ_1 is now the absolute maximum principal stress, and σ_2 is the other in-plane principal stress.
3. Similarly $\varepsilon_3 = \varepsilon_Z$ and ε_1 and ε_2 are ordered in magnitude.

The stress state at any surface point and instant can now be completely described by the value of the absolute maximum principal stress σ_1, the biaxiality ratio $a_e = \sigma_2/\sigma_1$ and the angle φ_p that σ_1 makes with the local x-axis. At the surface of laboratory test specimens the stress state is uniaxial (i.e., $a_e = 0$ and φ_p is constant). The stress state variations in real components or structures can deviate from uniaxial conditions in two respects:

1. The ratio of the principal stresses a_e can be nonzero.
2. The orientation of the principal stresses φ_p can vary with time.

MSC Fatigue is able to track these quantities throughout the time history and present the user with some summary statistics. These enable the most appropriate analysis to be carried out, and highlight cases where there may be problems with the validity of the analysis. The meaning of the statistics can be explained with the help of an example based on a motor vehicle chassis component subject to multiple multiaxial load histories derived from a track test. The absolute maximum principal stress σ_1, the biaxiality ratio a_e (which can vary between –1 and +1) and the angle φ_p (which can vary between 0 and 180 degrees) were tracked and recorded for each element in the form of scatter plots of σ_1 against φ_p and σ_1 against a_e. Figure 7-50 shows that, at the element in question, there is some variation in the orientation of σ_1, but this is only when the stresses are very low. At this location, the stresses remain essentially aligned with the local x-axis. Figure 7-51 shows that the stress state is not uniaxial, because a_e is not zero or constant. However, most of the variation is again at low stresses where the stresses are least important in fatigue terms. Overall, we can say that the loading at this location is essentially proportional (i.e., the principal stresses vary in a more or less fixed ratio and orientation). There is negative biaxiality.

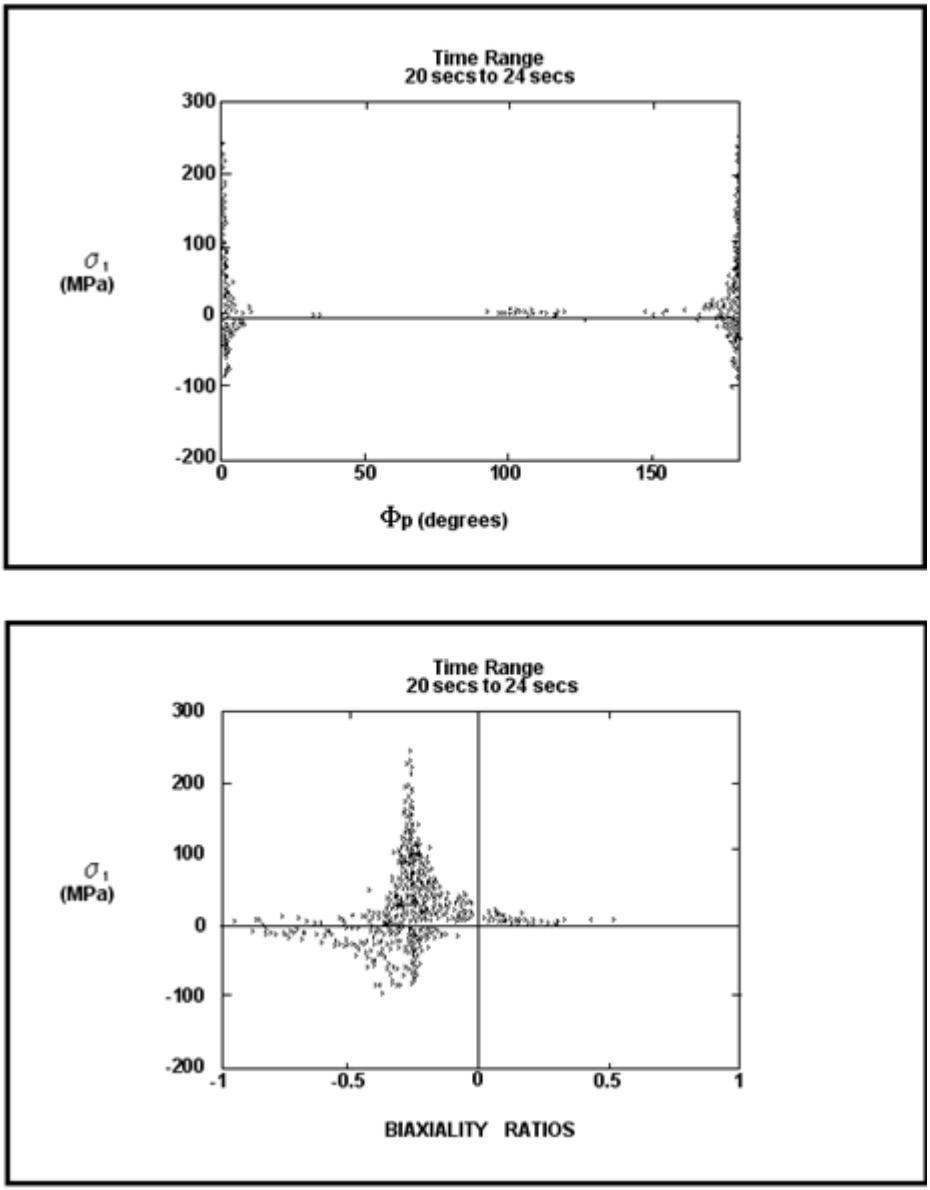


Figure 7-50 Variation in Orientation of σ_1

The next two plots once again illustrate essentially proportional loading, but this time with a positive biaxiality ratio.

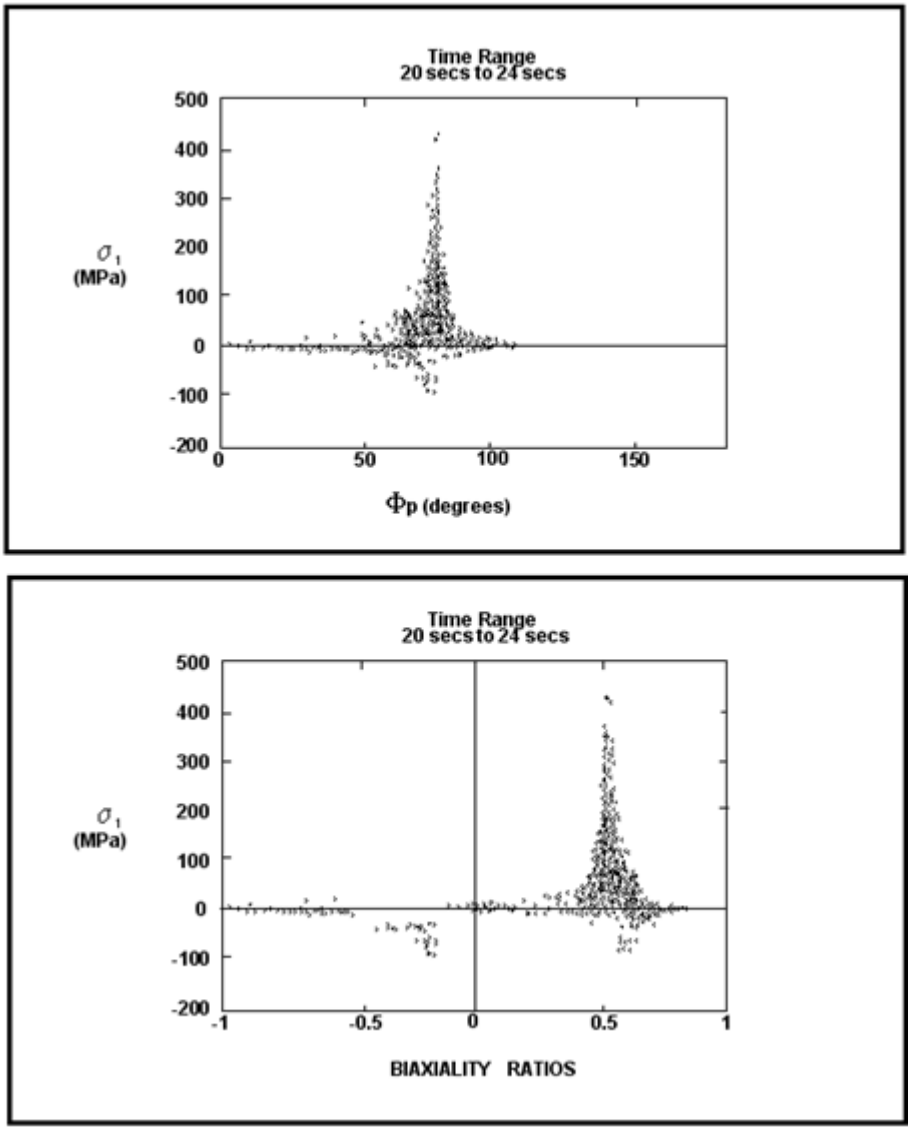


Figure 7-51 Multiaxial Stress State, Proportional Loading, Positive Biaxiality

Figure 7-52 shows a location where there is more variation in the principal stress directions. The biaxiality ratio plot shows that the dominant stress state is close to uniaxial. Note that the overall stresses are much lower here than in the previous examples.

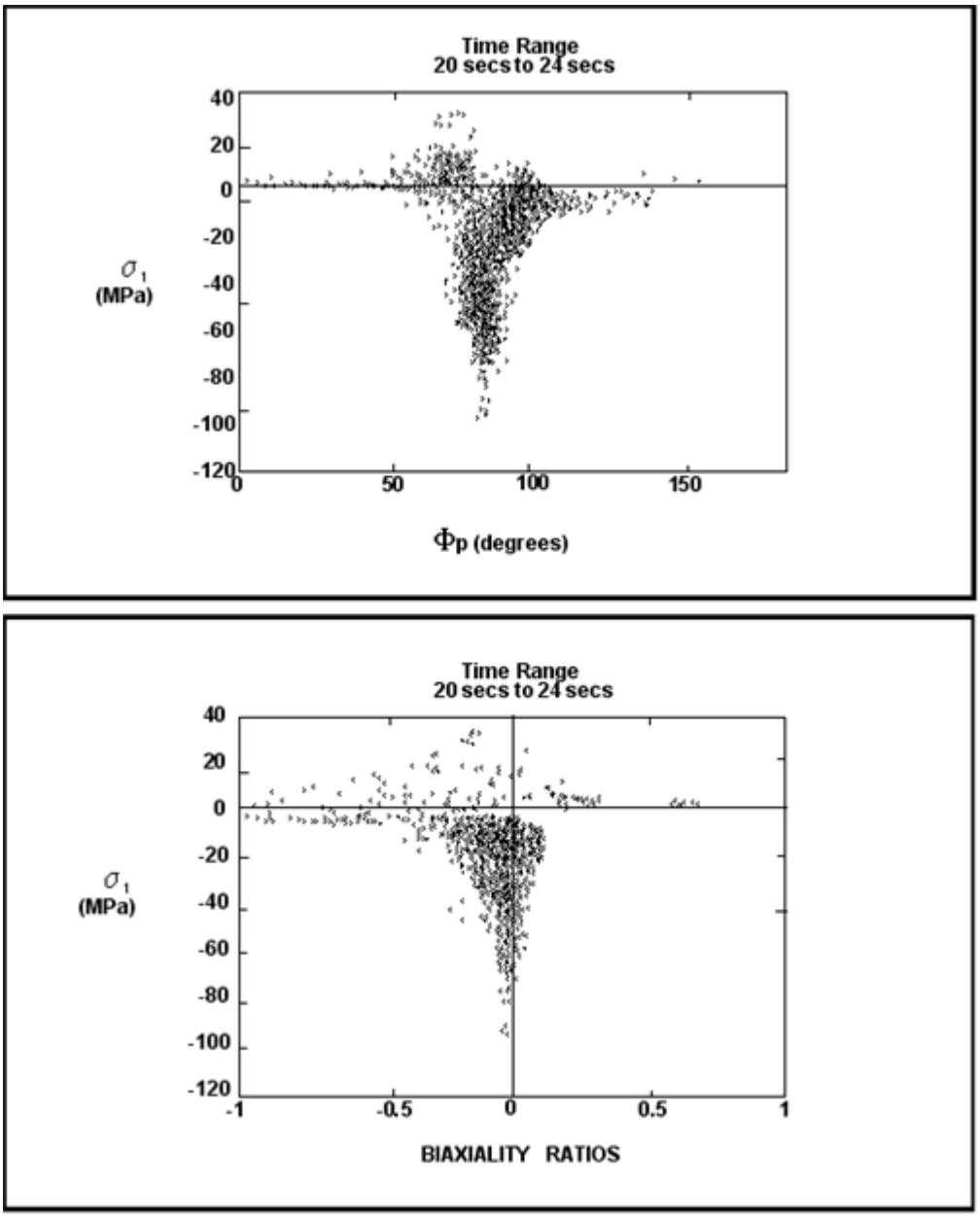


Figure 7-52 Variation in Principal Stress Directives

Figure 7-53 illustrates the stress state variations at another location with comparatively small stresses. The loading here is essentially nonproportional because the orientation of the principal stresses exhibits

considerable variation, and because the ratio of the principal stresses is not constant. Note that the stresses here are the elastic results from the FE analysis, with no correction for plasticity.

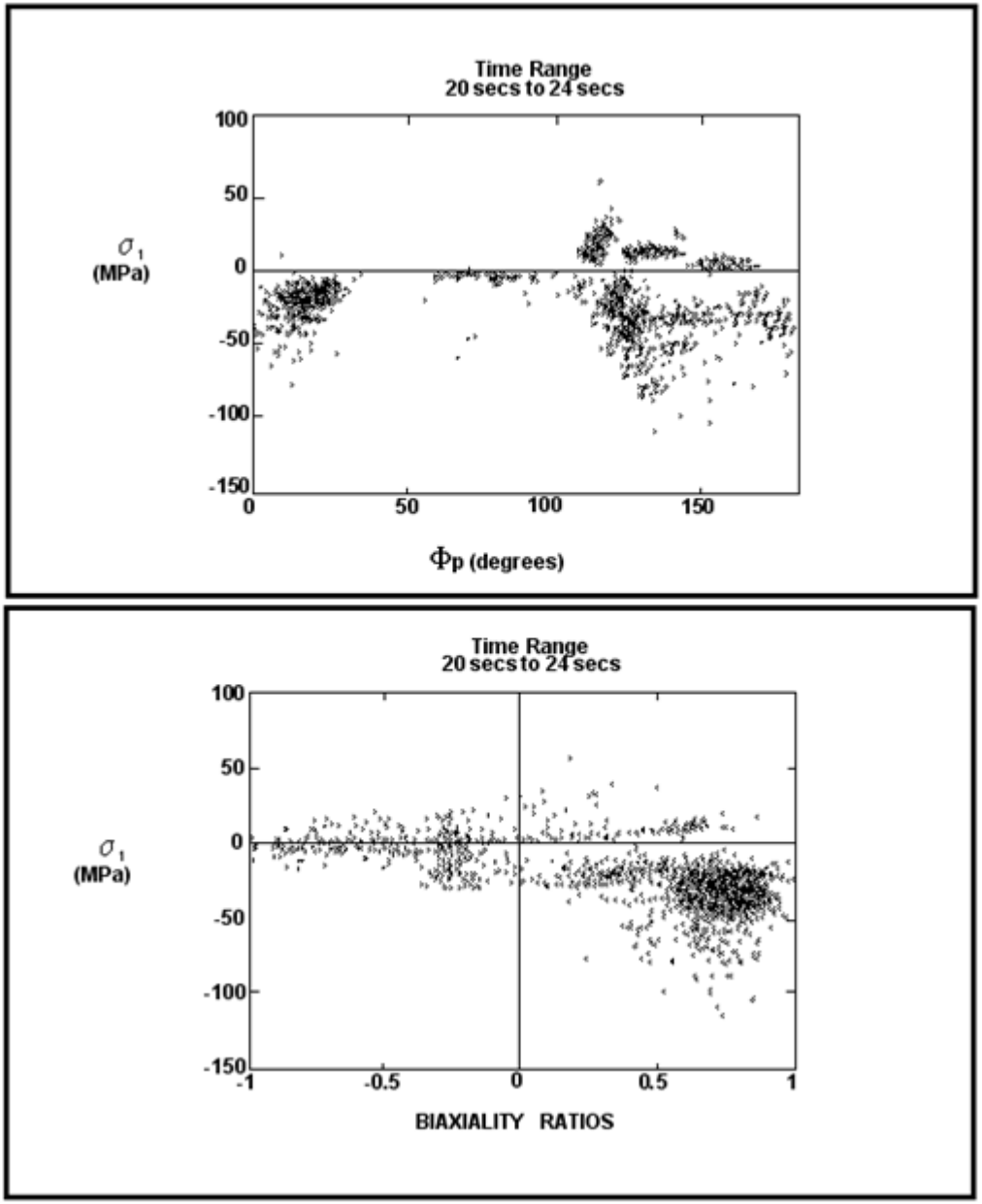


Figure 7-53 Non-proportional Loading Stress State Variations

MSC Fatigue provides the following summary information about the stress state variations, in addition to the fatigue damage results for each node or element:

1. The mean biaxiality ratio.
2. The standard deviation of the biaxiality ratio.
3. The most popular angle.
4. The spread of angles for which the magnitude of the absolute maximum principal stress σ_1 exceeds a gate value. The effect of this gate is to eliminate small stresses which are of less significance in fatigue terms. The default value of this gate is 0.2 times the UTS of the material. This is alterable by running FEFAT from the system prompt.
5. The range of the absolute maximum principal stress σ_1 .

It is important to note that complex loading on a component does not necessarily lead to complex stress state variations at the critical locations. It is in the nature of fatigue that cracks tend to initiate at geometric features such as holes fillets and radii. These features tend to have 2 effects; they act as stress raisers, concentrating the stress, but they also often have the effect of lining the stresses up in a particular direction. Also, in many situations the geometry dictates that the stress state can only be uniaxial, for instance at the edge of a thin sheet.

Choosing an Appropriate Analysis Method

Fatigue consists of the initiation and growth of a crack, or growth from a pre-existing defect. Cracking is a directional process, and is driven by irreversible (plastic) strains. It follows that the nature and orientation of fatigue damage is a function of the stress and strain state variations. When the loading is not uniaxial there is a breakdown in similitude, and strictly uniaxial methods may be inappropriate. MSC Fatigue offers three main methods of dealing with multiaxial loading which are based on extensions to static yield theory. These are known as equivalent strain methods. When loading is uniaxial, it is normal to base fatigue life estimations on the values of the axial strains. When there is multiaxiality of loading the axial strain in the component or structure is replaced by an equivalent strain, which is considered to cause damage equivalent to the same axial strain in a uniaxial test specimen. The three equivalent strains used in MSC Fatigue are as follows:

1. The absolute maximum principal strain ε_1
2. The signed Tresca parameter $\frac{\gamma_{max}}{1+\nu}\left(\frac{\sigma_1}{|\sigma_1|}\right)$

The signed von Mises strain (i.e., the von Mises strain with the sign of the absolute maximum principal stress), i.e.,

$$\frac{\sigma_1}{|\sigma_1|(1+\nu)\sqrt{2}}[(\varepsilon_1-\varepsilon_2)^2+(\varepsilon_2-\varepsilon_3)^2+(\varepsilon_3-\varepsilon_1)^2]^{\frac{1}{2}}$$

In addition, the user can select any component strain. In these equivalent strain methods, the selected parameter is cycle counted, then the Neuber correction is applied and damage assigned as if the strains were uniaxial. It is necessary to use the absolute maximum principal and the signed Tresca and von Mises parameters rather than their unsigned values because the latter give rise to truncation of cycles. This is illustrated in Figure 7-54. The example concerns a cylindrical notched specimen in fully reversed axial loading. Here, the principal stresses are ordered in the normal way. Mohr's circles of stress at the notch root are shown for tension and compression loading. The first time history illustrated is the load time history, and the axial stress and strain will be similar in form. Figure 7-54 shows the maximum principal stress. This is always positive or zero, and the bottom half of the cycles is truncated. Figure 7-54 is a schematic plot of the von Mises stress. This is clearly of no use for the purposes of fatigue analysis, having too many cycles of the wrong size. We need a parameter which reflects the reversed nature of the loading and gives us a good representation of the reversed loading and number of cycles. This is provided by the last selection, the absolute maximum principal stress. The absolute maximum principal strain and the signed Tresca and signed von Mises will be similar in form.

When a "critical plane" method is used, instead of cycle counting an equivalent stress or strain parameter, MSC Fatigue calculates the direct strain on planes whose normals lie in the X-Y plane, at 10 degree intervals. It therefore makes 18 separate fatigue calculations and only reports the result for the plane with the shortest life. It is only suitable for surface resolved stress states, but overcomes some of the problems associated with the mobility of principal stresses. The down side of this method is the increased run-time.

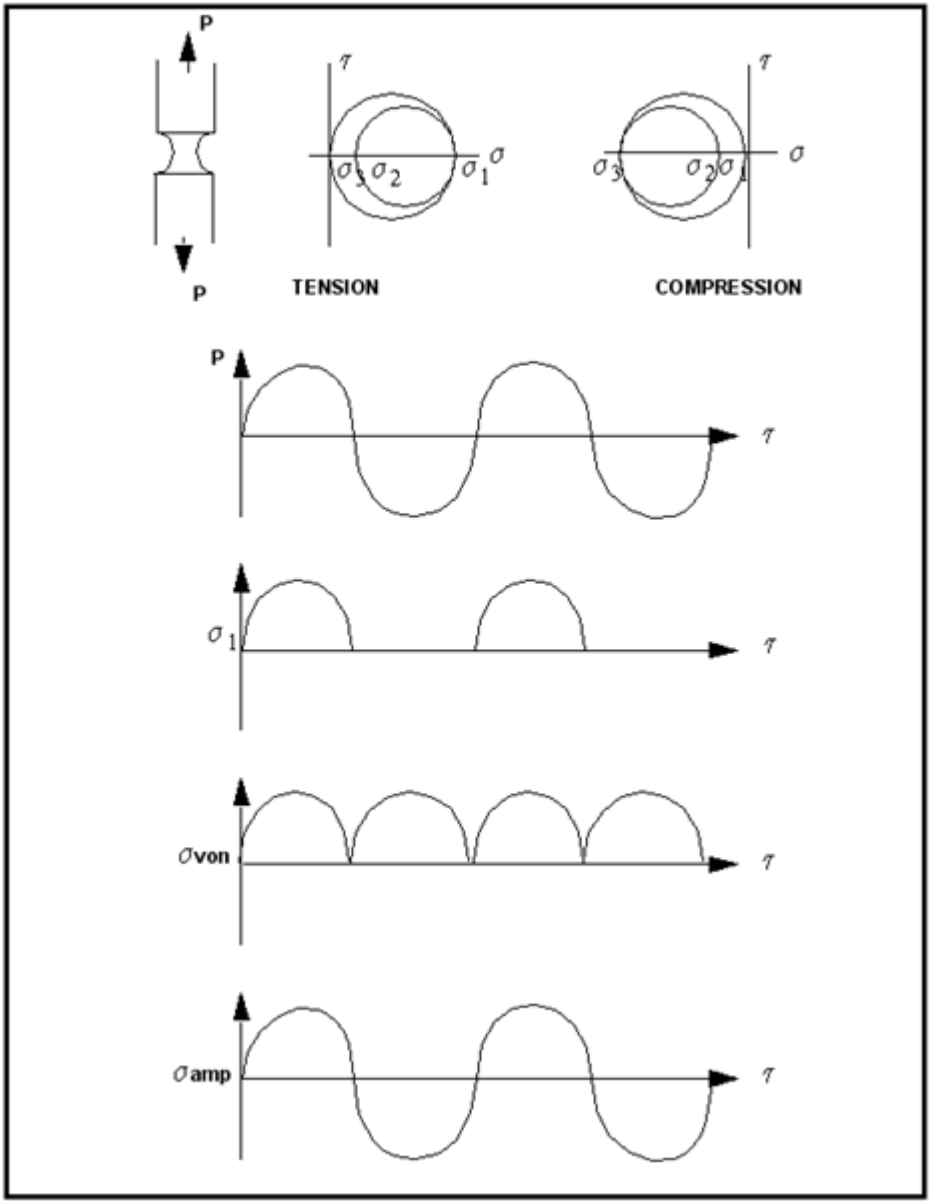


Figure 7-54 Example of Truncation of Cycles

The most appropriate method for crack initiation life predictions is a function of the prevalent stress state. Except where they start from pre-existing defects, cracks tend to initiate on planes of maximum shear, with crack growth being driven by shear strain initially before the crack turns into opening mode. Now biaxial loadings can be divided into two cases. These cases give rise to different patterns of early crack growth.

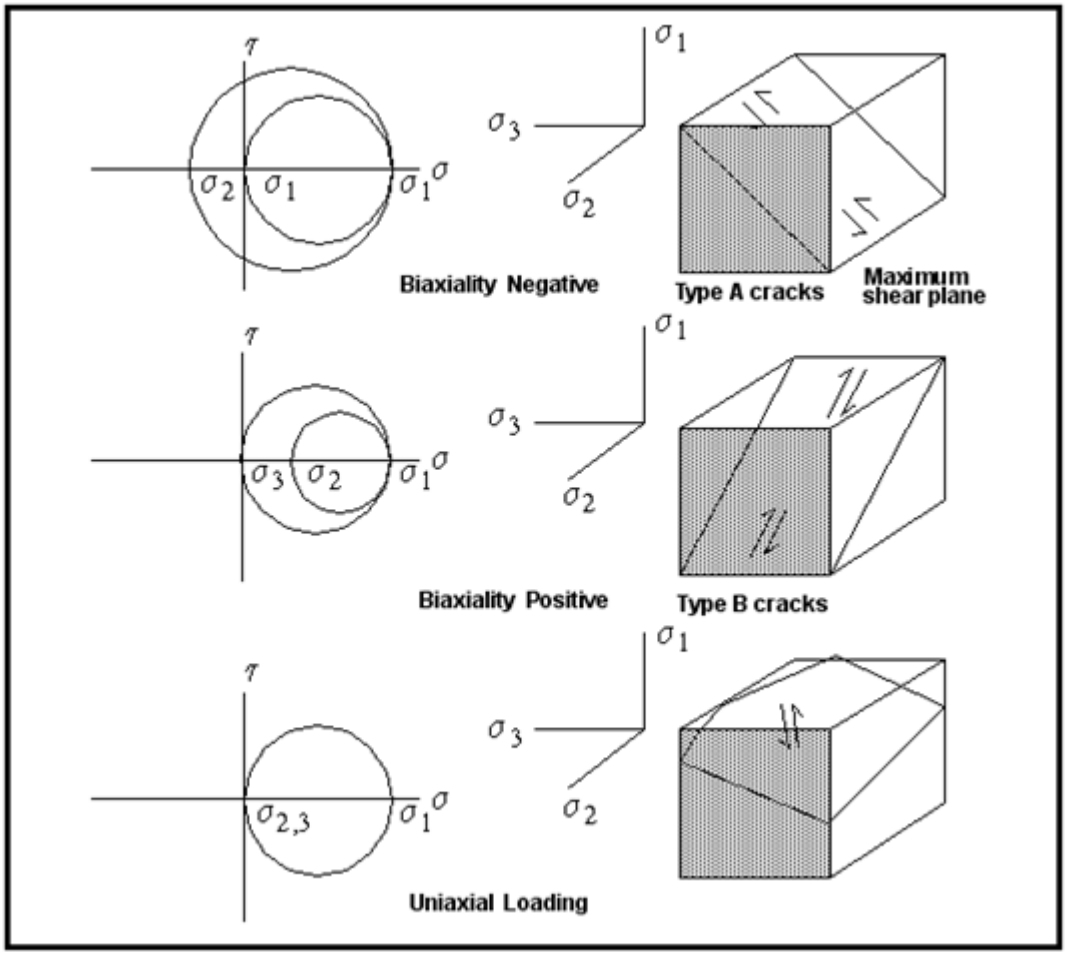


Figure 7-55 Biaxiality Ratios

Figure 7-55 (part a), the biaxiality ratio a_e lies somewhere between 0 and -1. A value of 0 represents the special case of uniaxial loading, and -1 represents pure shear. In this range, the maximum shear strain occurs on a plane intersecting the free surface (shaded) at right angles, and cracks in the initiation phase will tend to be driven along the surface in mode II. These are Type A cracks. In Figure 7-55 (part b), a_e lies between 0 (uniaxial loading) and 1 (equibiaxial loading). The plane of maximum shear now intersects the surface at 45 degrees, and cracks growing in mode II will be driven through the thickness of the material. These Type B cracks are therefore more damaging for a given range of maximum shear strain. Figure 7-55 (part c) above depicts the special case of uniaxial loading where the maximum shear plane is any plane at 45 degrees to the loading axis. Note that what we call "crack initiation" may actually include crack growth in shear and opening modes on more than one plane. Which type of crack behavior dominates "initiation" is material and stress-state dependent. The results of fatigue tests conducted under proportional loading at various biaxiality ratios suggest that the best parameter to use for loadings with $0>a_e>-1$ (i.e., between uniaxial and pure shear) is the

absolute maximum principal strain. For $0<a_e\leq 1$ (i.e., between uniaxial and equibiaxial) it is better, and more conservative, to use the signed Tresca parameter. This reflects that we are dealing with the more damaging type B cracks. The signed von Mises stress tends to give results that lie between the absolute maximum principal and the signed Tresca, except under uniaxial conditions, where all three should give the same result, and under equibiaxial conditions where the signed von Mises and signed Tresca should give the same result. There is a problem with loading where the value of a_e is around –1. Here the in-plane principal stresses are almost equal in magnitude but opposite in sign. The absolute maximum principal may "flip" between one and the other, and this may make it difficult to carry out a valid fatigue calculation. Under these circumstances, the MSC Fatigue angle spread results will indicate that the angle is moving through 90 degrees or more. The use may be able to get around this problem by using one of the strain components in conjunction with suitable fatigue data. These guidelines are summarized in the following table.

	Abs Max Principal Strain	Signed von Mises	Signed Tresca
$a_e=-1$ (pure shear)	Possible problems with mobility of principals Use Critical Plane Option		
$1<a_e<0$	OK	Conservative	Very Conservative
$a_e=0$ (uniaxial)	OK	OK	OK
$0<a_e<1$	Non-Conservative	Non-Conservative	OK
$a_e=1$ (equibiaxial)	Non-Conservative	OK	OK

The comments above refer to situations where the local loading is proportional (i.e., the principal stresses are not directionally mobile and are proportional to one another). When this is not true, two problems are introduced:

1. It is no longer valid to assume that hardening is isotropic and that simple extensions of yield criteria can be extended to calculate elastic-plastic strains and stresses. A three-dimensional kinematic hardening model becomes necessary to model the material's deformation behavior.
2. Fatigue cracking is directional. When the orientation of the principal stresses varies a great deal, linear damage summation will sum damage that actually represents cracking on different planes. In addition, in extreme cases it is possible to miss or truncate important fatigue cycles.

The further the situation deviates from proportional loading, the more caution the results should be treated with. MSC Fatigue's biaxiality indicators — the angle spread and biaxiality standard deviation provide a means of quantifying the extent to which the local loading is nonproportional.

Biaxiality Correction Options

The uniaxial cyclic stress-strain curve is frequently used in fatigue analysis of stress concentrations, or notches, under unidirectional loading (tension or bending). However, the stress states at critical locations in many components subject to fatigue loadings are not uniaxial. For instance, even in a simple case such as a notched cylindrical bar in tension, the constraint on transverse strain often causes considerable notch root multiaxiality. In general then, the use of the Neuber method in conjunction with the uniaxial cyclic stress-strain curve is likely to generate values of stress and strain that are misleading, and if used as inputs to a fatigue

analysis, potentially dangerous. Clearly, some means of taking into account the multiaxiality of stress and strains in necessary.

The following discussion addresses the description of material stress-strain responses in what will loosely be described as "proportional loading." Strictly speaking, proportional loading means that the principal stresses remain in a constant ratio and the principal stress axes remain fixed in orientation. In practice, because multiaxial loadings are very often due to constraints (for instance at the root of a circumferential notch) the stress-strain state can be described more accurately by saying that the principal strains are in a constant ratio. For the purpose of this discussion, this condition will also be described as proportional loading, and because it is this condition that is closer to the majority of non-uniaxial loadings that occur in practice, the methods describes model deformation under constant strain ratio rather than constant stress ratio.

The context for this discussion is the need in MSC Fatigue to calculate elastic-plastic stresses and strains on the basis of linear elastic FE results, using methods based on the work of Neuber (Ref. 26). Two methods are described. Both are based on the extension of the von Mises yield criterion to predict post-yield stress-strain behavior. The first, due to Hoffmann and Seeger (Ref. 27, 28), involves carrying out the Neuber correction on equivalent (von Mises) stresses and strains, and then calculating the required principal stresses and strains for the fatigue calculation using Hencky's flow rules. The second, due to Klann, Tipton and Cordes (Ref. 29), takes the uniaxial cyclic stress-strain curve and calculates a new curve relating σ_1 and ε_1 which is a function of the principal strain ratio.

The following discussion assumes that we are dealing with stresses and strains at a traction free surface. This means that there are no non-zero direct or shear stresses normal to the free surface. The free surface is therefore in a state of plane stress. There are two non-zero principal stresses and they lie in the plane of the surface. For clarity, the following definitions are adopted:

σ_1, ε_1	Principal stress, strain greatest in magnitude.
σ_2, ε_2	Other in-plane principal stress, strain.
σ_3, ε_3	Surface normal principal stress, strain.
a	Biaxiality ratio s2/s1.
$\mathbf{a_e}$	Elastic value of biaxiality ratio.

Hoffmann and Seeger method - The treatment here just describes the implementation in MSC Fatigue. The Hoffmann-Seeger method can reasonably be applied when the selected Strain Combination is Absolute Maximum Principal or Signed Tresca. Application when using the signed von Mises is identical to the normal Neuber method.

The method assumes that the Neuber method can be applied using equivalent stresses and strains based on the von Mises values. The first step then is to take whatever parameter has been cycle counted and converted to equivalent strain. In order to deal with a random variable amplitude loading it is necessary to use a signed form of the von Mises strain. The subscript e denotes elastic values:

$$\varepsilon_{q,e} = \frac{1}{1+\nu_e\sqrt{2}}\sqrt{(\varepsilon_1-\varepsilon_2)^2+(\varepsilon_2-\varepsilon_3)^2+(\varepsilon_3-\varepsilon_1)^2}\,\frac{\varepsilon_1}{|\varepsilon_1|} \tag{7-49}$$

If the loading is proportional, then this value can be calculated from ε_1 as follows:

$$\varepsilon_q = \varepsilon_1 \frac{\sqrt{1 - a_e + a_e^2}}{1 - a_e \nu_e} \tag{7-50}$$

or from the Tresca parameter $\gamma_{max}/(1+\nu_e)$ as follows, if $a_e >= 0$:

$$\varepsilon_q = \frac{\gamma_{max}}{1 + \nu_e} \sqrt{1 - a_e + a_e^2} \tag{7-51}$$

or if $a_e < 0$,

$$\varepsilon_q = \frac{\gamma_{max}}{1 + \nu_e} \frac{\sqrt{1 - a_e + a_e^2}}{1 - a_e} \tag{7-52}$$

Of course

$$\sigma_q = E\varepsilon_q \tag{7-53}$$

So far, all the stress and strain values are elastic. The Neuber correction is then carried out in the normal way, using the equivalent strain and stress values and the uniaxial cyclic stress-strain curve, i.e. assuming that:

$$\varepsilon_q = \frac{\sigma_q}{E} + \left(\frac{\sigma_q}{K'}\right)^{\frac{1}{n'}} \tag{7-54}$$

and that

$$\sigma_q \varepsilon_q = \sigma_{q,e} \varepsilon_{q,e} \tag{7-55}$$

The effective (elastic-plastic) Poisson's ratio for each cycle can now be calculated based on:

$$\nu' = \frac{1}{2} - \left(\frac{1}{2} - \nu_e\right) \frac{\sigma_q}{E\varepsilon_q} \tag{7-56}$$

The elastic-plastic biaxiality ratio is calculated from:

$$a = \frac{\frac{\varepsilon_2}{\varepsilon_1} + \nu'}{1 + \nu' \frac{\varepsilon_2}{\varepsilon_1}} \tag{7-57}$$

where the strain ratio is assumed constant

$$\frac{\varepsilon_1}{\varepsilon_2} = \frac{a_e - \nu_e}{1 - a_e \nu_e} \tag{7-58}$$

All the required principal stresses and strains can then be calculated:

$$\varepsilon_1 = \varepsilon_q \frac{1 - \nu' a}{\sqrt{1 - a + a^2}} \tag{7-59}$$

$$\varepsilon_2 = \varepsilon_1 \left(\frac{\varepsilon_2}{\varepsilon_1}\right) \tag{7-60}$$

$$\varepsilon_3 = -\varepsilon_q \frac{\nu'(1+a)}{\sqrt{1-a+a^2}} \tag{7-61}$$

$$\sigma_1 = \sigma_q \frac{1}{\sqrt{1-a_e+a_e^2}} \tag{7-62}$$

$$\sigma_2 = a\sigma_1 \tag{7-63}$$

enabling the elastic-plastic value of any desired combined stress-strain parameter to be calculated.

Parameter Modification Method (Klann, Tipton and Cordes) - The treatment here concentrates on the implementation in MSC Fatigue. Essentially, the equation

$$\varepsilon_q = \frac{\sigma_q}{E} + \left(\frac{\sigma_q}{K'}\right)^{\frac{1}{n'}} \tag{7-64}$$

is used to calculate a series of corresponding values of σ_q and ε_q . For each pair of ν' values, is calculated from:

$$\nu' = \frac{1}{2} - \left(\frac{1}{2} - \nu_e\right)\frac{\sigma_q}{E\varepsilon_q} \tag{7-65}$$

and a from

$$a = \frac{\frac{\varepsilon_2}{\varepsilon_1} + \nu'}{1 + \nu'\frac{\varepsilon_2}{\varepsilon_1}} \tag{7-66}$$

body

$$\frac{\varepsilon_2}{\varepsilon_1} = \frac{a_e - \nu_e}{1 - a_e\nu_e} \tag{7-67}$$

Pairs of values of ε_1 and σ_1 are then computed as follows:

$$\varepsilon_1 = \varepsilon_q \frac{1-\nu' a}{\sqrt{1-a_e+a_e^2}} \tag{7-68}$$

$$\sigma_1 = \sigma_q \frac{1}{\sqrt{1-a_e+a_e^2}} \tag{7-69}$$

These pairs of values represent a cyclic stress strain curve for a particular biaxiality ratio a_e. A new set of Ramberg Osgood parameters K" and n" is obtained by fitting the equation:

$$\varepsilon_1 = \frac{\sigma_1}{E"} + \left(\frac{\sigma_1}{K"}\right)^{\frac{1}{n"}} \tag{7-70}$$

to the calculated points. E" is calculated explicitly from:

$$E'' = \frac{E}{1 - \nu_e a} \tag{7-71}$$

The modified parameters are used to carry out the Neuber correction in the normal way.

Multiaxial Cyclic Plasticity

When the loadings are non-proportional, the theories required to handle multiaxial cyclic plasticity become more complex.

Plasticity theory is a complex area. The treatment here is intended to provide an overview of the methods incorporated in the MULTIAXIAL option to MSC Fatigue without getting excessively mathematical.

There are three essential components of a plasticity model

1. A yield function
2. A flow rule, and
3. A hardening rule

The Yield Function: The yield function determines the limits of elastic behavior. The most popular yield function, which gives the best representation of the behavior of real materials, the von Mises or octahedral shear stress criterion. This can be expressed:

$$S_1^2 + S_2^2 + S_3^2 = \frac{3\sigma_y^2}{2} \tag{7-72}$$

in terms of deviatoric stresses, or

$$\frac{1}{\sqrt{2}}\sqrt{(\sigma_1 - \sigma_2)^2 + (\sigma_2 - \sigma_3)^2 + (\sigma_3 - \sigma_1)^2} = \sigma_y \tag{7-73}$$

in terms of principal stresses. The parameter σ_y is the uniaxial yield stress. Graphically, the von Mises yield function appears as a cylinder in principal stress space, the axis of which represents the hydrostatic condition. The hydrostatic stress is assumed to have no effect on plasticity.

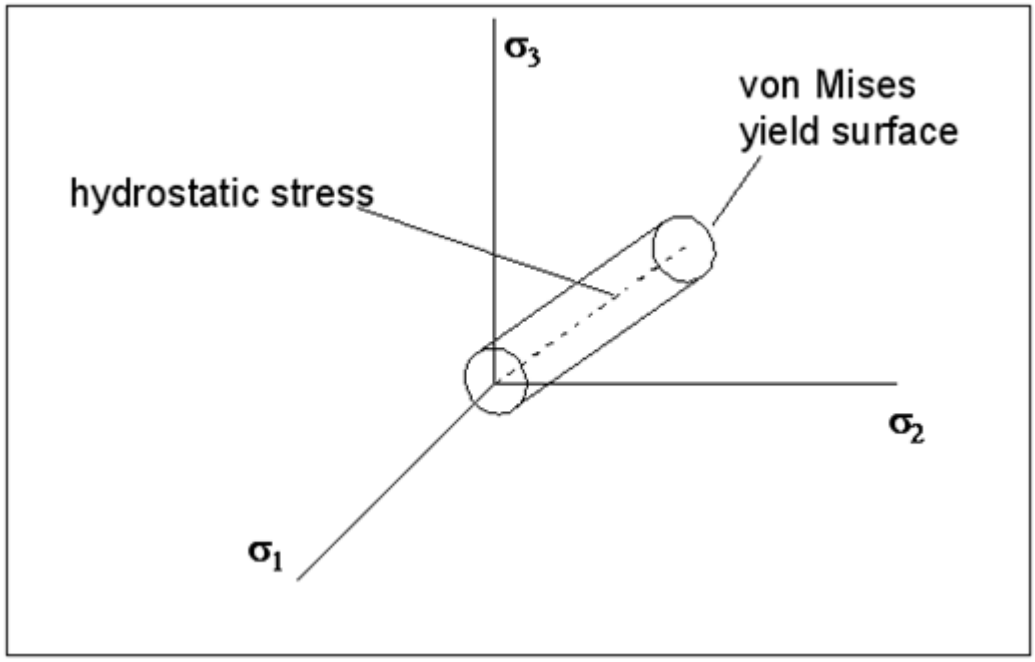


Figure 7-56 von Mises Yield Function in Principal Stress Space

On a free surface, only two principal stresses are of concern, and the von Mises yield criterion appears as an ellipse.

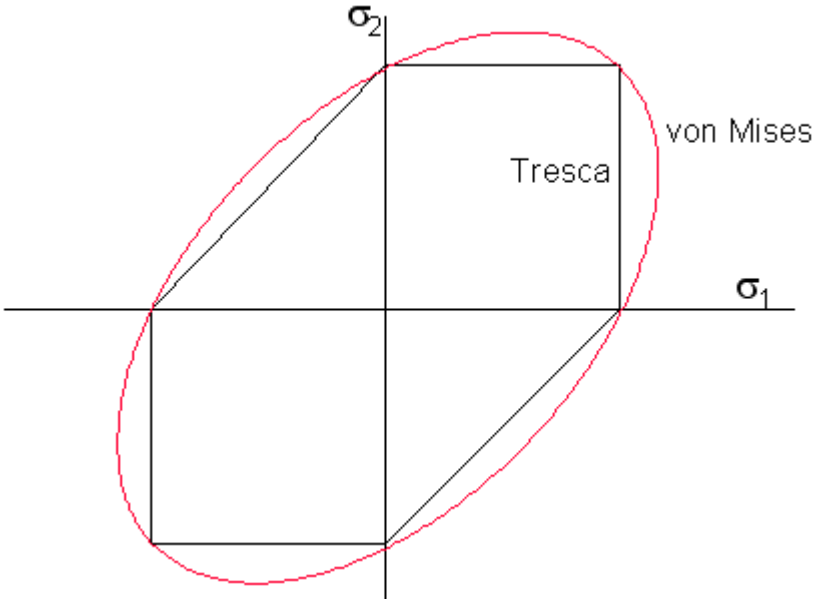


Figure 7-57 von Mises Yield Criterion on Free Surface

As long as the stress state remains within the yield surface or ellipse, there is no plastic deformation.

The Flow Rule: A basic assumption of plasticity theory is that the total strain can be partitioned into elastic components which can be calculated from Hooke's law, and a plastic component.

$$\varepsilon_{ij}^{t} = \varepsilon_{ij}^{e} + \varepsilon_{ij}^{p} \tag{7-74}$$

In incremental plasticity theory, the flow rule determines the direction of the plastic strain increment in relation to a given stress increment. Most plasticity models use the normality rule. This is based on potential flow, where the flow is determined by the partial derivatives of the potential function. If the potential function is assumed to be the yield function, then the direction of plastic flow must be normal to the yield surface. Mathematically this is expressed:

$$d\varepsilon_{ij}^{\ p} = \lambda \frac{\partial F}{\partial \sigma_{ij}} \tag{7-75}$$

Where F is the yield function and λ is a scalar quantity. The value of λ is given by:

$$\lambda = g\left(\frac{\partial F}{\partial \sigma_{ij}} d\sigma_{ij}\right) \tag{7-76}$$

where g is related to the current slope of the stress-strain curve. The term in brackets is not allowed to take negative values. The value of g at any instant is determined by the hardening rule.

The Hardening Rule: The hardening rule determines how the yield point changes, taking into account the stress-strain curve and the loading history. For uniaxial loadings, the hardening rule is defined by the memory

rule and Massing's hypothesis, which together dictate the current position of the yield point, and the current slope of the stress-strain curve.

When the loadings are multiaxial, this is not adequate - something is needed which tells how the yield surfaces will behave when the yield point is exceeded. There are two main classes of hardening rule - isotropic and kinematic.

In isotropic hardening, the yield surfaces grow when the initial yield point is exceeded. This is illustrated graphically here:

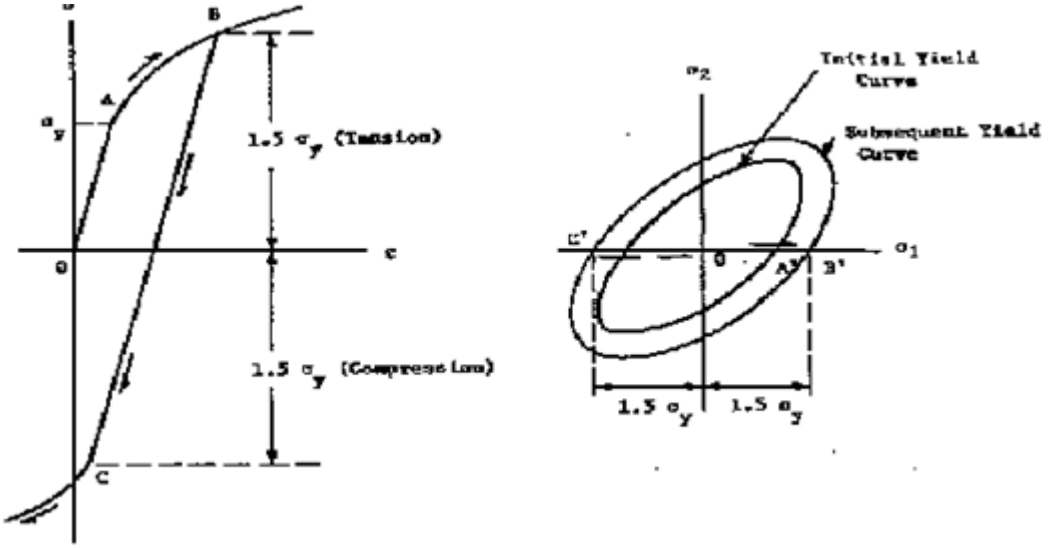


Figure 7-58 Isotropic Hardening

Isotropic hardening is OK for predicting plasticity under monotonically increasing loading, but does not properly describe cyclic plasticity. For that we need a kinematic hardening rule.

In kinematic hardening rules, the yield surface translates when the yield is exceeded.

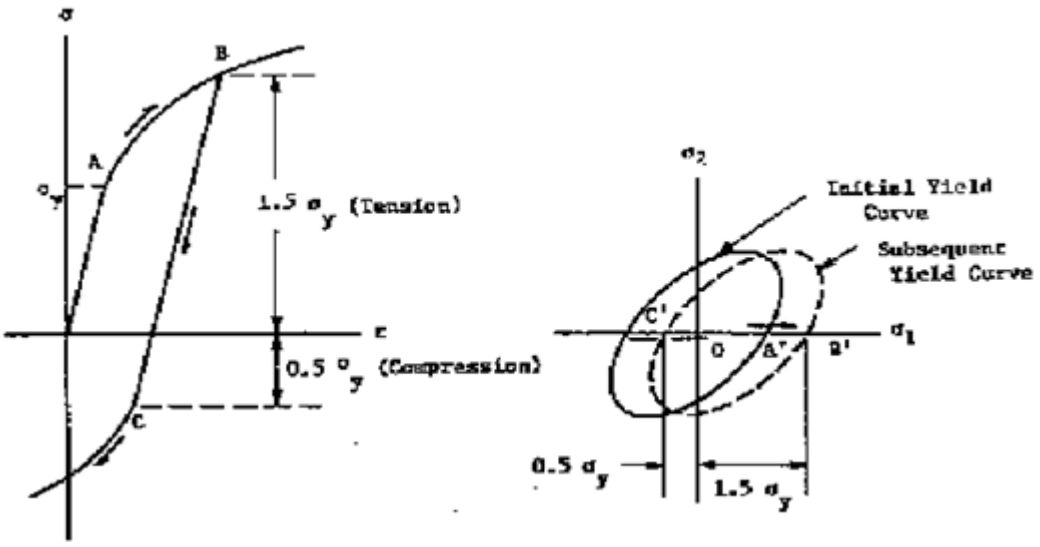


Figure 7-59 Kinematic Hardening

There are a number of possible kinematic hardening rules, and the hardening rule is the essence of most plasticity models. The hardening rule used in MSC Fatigue, both in the test-based module MMLF and the FE based module FEMLF, is the one originally suggested by Mroz (Ref. 45) and subsequently modified by Garud (Ref. 46). This is a multi-surface model, where there are a number of initially concentric yield surfaces. The inner surface corresponds to the elastic limit. Within this surface, only Hooke's law (generalized to 3 dimensions) is used. The other surfaces correspond to the changes of slope in a linearized cyclic stress strain curve.

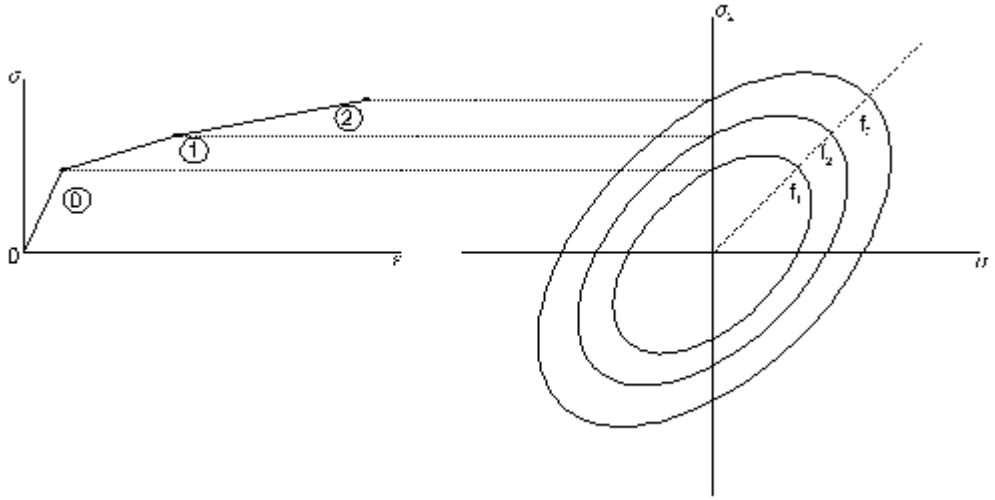


Figure 7-60 Linearized Cyclic Stress-Strain Curve

In the software, 40 surfaces are used, and the result is quite a smooth cyclic stress-strain curve. The curve is determined automatically from the Ramberg-Osgood equation.

When a loading is applied, the loading point can be traced within this diagram. So long as the loading point remains within the inner ellipse, there is no plasticity. When the loading point touches this ellipse, it must move. The direction in which it translates is described graphically in the following diagram:

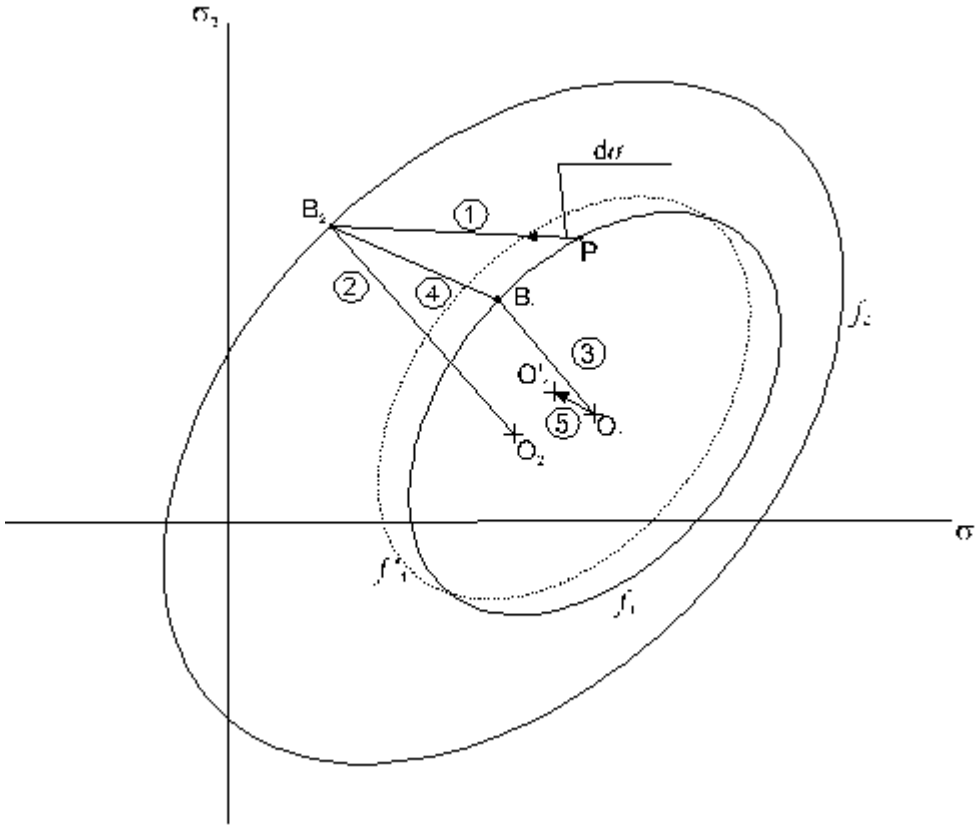


Figure 7-61 Translation of Ellipse when Plasticity Occurs

In this diagram, only two surfaces are drawn. The stress increment is extended from the surface to find the point B2 where it intersects the next surface. The center of the larger surface is joined to this point. A line is drawn parallel to this line from the center of the smaller surface to find where it intersects the smaller surface at B1. The ellipse then translates in the direction B1 B2 until the end of the current loading increment lies on the surface of the inner ellipse. This rule may sound complicated, but it ensures that when the ellipses meet, they do so tangentially. When two or more ellipses meet, they move together. The outermost active ellipse determines the current slope of the stress-strain curve.

Multiaxial Non-Proportional Notch Correction Procedure

When making local strain calculations on the basis of elastic pseudo stresses or strain, a notch correction procedure is required in order to estimate the local elastic-plastic stresses and strains that are required. When the loading is uniaxial or proportional, the Neuber method, or a variation such as the Hoffmann-Seeger method, works quite well. These methods essentially apply the Neuber equation:

$$\sigma\varepsilon = \sigma_e\varepsilon_e \tag{7-77}$$

together with the equation for the stress-strain curve for each cycle or reversal. However, when the loading is non-proportional, this is no longer adequate, because the stress response to a given strain reversal (and vice versa) is path dependent. We therefore need an incremental formulation. An incremental version of the Neuber rule for non-proportional loadings might look like this:

$$\sigma_{ij}^{e}\Delta\varepsilon_{ij}^{e} + \varepsilon_{ij}^{e}\Delta\sigma_{ij}^{e} = \sigma_{ij}^{N}\Delta\varepsilon_{ij}^{N} + \varepsilon_{ij}^{N}\Delta\sigma_{ij}^{N} \tag{7-78}$$

This must then be combined with a non-proportional cyclic plasticity model such as the Mroz-Garud model.

Additionally some assumptions are required in order to reach a solution, e.g. that the ratios of the increments of strains, stresses or total strain energy in certain directions are the same for the elastic as the elastic-plastic case. The method used in MSC Fatigue is based on the total strain energy (strain energy plus complementary strain energy). This method is described in more detail in (Ref. 69).

Multiaxial Safety Factor Analysis

Multiaxial fatigue is a term referring to fatigue of components under complex stress states. High-cycle multiaxial fatigue deals with components which during their life time are expected to experience very large number of cycles. Due to the nature of the fatigue process, it is not possible to make realistic finite life predictions for components that may experience very large numbers of major loading cycles during their service. For instance, a typical car engine running at 3000 rpm only has to run for 21 days before many engine components such as connecting rods or pistons have experienced 10^8 cycles. This number could easily be achieved within 30,000 miles. Such components are therefore designed for infinite life. Other examples include crank shafts, propeller shafts, and rear axles. In these components, it is not necessary to quantify the amount of fatigue damage, but just to consider if any fatigue damage will occur during the load history.

For high-cycle multiaxial fatigue, actual quantities of fatigue damage are not considered. Instead, the loading history is examined to determine only if damage occurs at any moment. If damage does occur, then the component does not have infinite life. The high-cycle criteria considered can also be used to calculate a safety factor of a component if infinite life is determined.

In cases of high-cycle fatigue, the complex stress states in the loading history are important to consider in order to accurately predict fatigue life. A simple uniaxial fatigue method is not sufficient and can produce non-conservative results. A multiaxial fatigue limit or endurance limit criterion is required.

High-Cycle Multiaxial Fatigue Theories

Several methods have been developed to calculate a fatigue limit in the high-cycle fatigue (HCF) regime with significant multiaxial loadings. Diboine (Ref. 70) classifies these methods according to the basis of calculation:

- Empirical
- von Mises
- Tresca
- Micro-structural

The empirically-based fatigue criteria require testing of a specific material in a loading configuration similar to that of interest in order to produce a numerical fit to the data and make further predictions. This defeats the purpose of using fatigue predictions, and therefore this type of method will not be considered further.

Von Mises-based criteria were prevalent early in the studies of high-cycle fatigue, the Sines and Crossland criteria being quite well-known. However, von Mises is a deformation criterion which does not consider the plane of maximum shear stress range. Fatigue cracks have been observed to initiate and grow initially along these planes of maximum shear stress.

The above observations lead to the more intuitive Tresca-based high-cycle fatigue criteria. Findley reported early 1959 on Tresca-based predictions, but McDiarmid's Tresca-based criterion introduced in 1987 appears to be widely accepted and well-known.

Microstructural approaches to high-cycle fatigue have also become more common in recent years. These approaches consider the stabilized, microscopic stresses in calculations of fatigue damage. Dang Van first introduced this type of approach in 1973, but his modified approach proposed in 1987 is easier to use and provides better correlation.

McDiarmid Criterion

The McDiarmid criterion (Ref. 71)(Ref. 72) has several significant features. It is a critical plane approach, where the critical plane is defined as the plane of maximum shear stress. The fatigue criterion is based on two loading parameters:

- Shear stress amplitude
- Maximum normal stress for the entire loading history on the plane on maximum shear stress

Another feature of the McDiarmid criterion is that it considers Case A and Case B cracking separately in calculating for fatigue damage. Case A cracking is along the surface, as in torsion; while Case B cracking is into the surface, as in bending.

Fatigue Limit: The fatigue limit according to McDiarmid can be expressed in the following form:

$$\frac{\tau}{t_{A,B}} + \frac{\sigma_{n,max}}{2\sigma_T} = 1 \tag{7-79}$$

τ_a	shear stress amplitude (half the maximum range of shear stress)
$t_{A,B}$	reversed shear fatigue strengths for Case A and B crack growth
$\sigma_{n,max}$	maximum normal stress for the entire loading history on the plane of maximum shear stress amplitude
σ_T	tensile strength

Any value of the left-hand side of this equation exceeding 1 indicates that damage occurs. This fatigue limit can be displayed graphically in a plot of $\tau_a/t_{A,B}$ vs. $\sigma_{n,\,max}/2\sigma_T$:

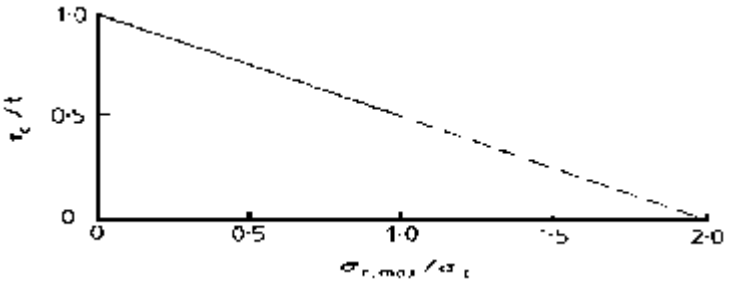


Figure 7-62 Fatigue Limit, τ_a / $t_{A,\,B}$ vs. $\sigma_{n,\,max}$ / $2\sigma_T$

Any loading that goes above the line is considered to cause damage, and the component does not have infinite life.

The reversed shear fatigue strength for Case A cracking, t_A, is the reversed torsion fatigue strength of the material. The reversed shear fatigue strength for Case B cracking, t_B, is found from a Case B cracking test case (3).

If you have only uniaxial data, one way of estimating these values is to consider the case of the uniaxial fatigue limit where the stress amplitude is σ_{fl} and assuming that the uniaxial case is the point where Case A or Case B can occur to write:

$$\frac{0.5\sigma_{fl}}{t_{A,\,B}} + \frac{0.5\sigma_{fl}}{2\sigma_T} = 1 \tag{7-80}$$

so that $t_{A,\,B}$ are given by:

$$t_{A,\,B} = \frac{\sigma_{fl}}{2 - \sigma_{fl}/2\sigma_T} \tag{7-81}$$

Implementation: Both Case A and Case B planes are searched to find the plane with the maximum value of shear stress τ_a . Both Case A (θ = 90 degrees) and Case B (θ = 45 degrees) are considered, and the orientation θ of the planes is varied by 10 degree increments over a 180 degree range. The maximum shear range for each plane is calculated by the difference of the maximum and minimum values of shear stress for the loading history. The McDiarmid criterion calculations are then performed for the planes of maximum shear stress range. Because the fatigue limit criterion is expressed:

$$\tau_a/t_{A,\,B} + \sigma_{n,\,max}/2\sigma_T = 1 \tag{7-82}$$

a safety factor can be defined:

$$\text{Safety Factor} = 1/(\tau_a/t_{A,\,B} + \sigma_{n,\,max}/2\sigma_T) \tag{7-83}$$

This safety factor is calculated by the MSC Fatigue program FEMLF and reported in the `.fos` file.

Note: The McDiarmid method is restricted to free surface stress conditions.

Dang Van Criterion

The Dang Van Criterion is classified as a 'micro-structural' method because it considers stresses and crack initiation at the local level. The stabilized local residual stresses are calculated by means of an iteration in which convergence assumes a stabilized state, a state of elastic shakedown. These stresses are used to examine microscopic stresses in the material from given macroscopic stresses.

Dang Van (Ref. 73)(Ref. 74) describes crack initiation (damage) to occur when some function of microscopic stress, f, is greater than or equal to zero at any time in a stabilized cycle. The following notation is used:

$$f\{\sigma(P,t)\} \geq 0 \quad \text{for } P \in V(M) \tag{7-84}$$

M	a point of interest/stress concentration on a component (notch, fillet)
V(M)	An elementary representative volume around M
P	A point in V(M) where a grain(s) is critically oriented
σ(P,t)	A microscopic stress tensor at location P at time t

Fatigue Limit: Dang Van chooses this function of stress to be linear, and utilizes the shear stress and hydrostatic tension at any given time:

$$f(\sigma) = \tau + a \cdot ph - b \quad \text{and} \quad f(\sigma) = \tau - a \cdot ph + b \tag{7-85}$$

t	microscopic shear stress at a given time
ph	microscopic hydrostatic tension at a given time
a	sensitivity to hydrostatic tension

The shear fatigue limit, t, and bending fatigue limit, f, may be used to estimate a and define b in the following way:

$$a = 3(t - 0.5f) / f \tag{7-86}$$

$$b = t \tag{7-87}$$

This formulation ensures that the Dang Van criterion reduces to the uniaxial fatigue limit under uniaxial conditions. If you only know the uniaxial fatigue limit then try b=t=0.5774f which means a=0.231.

The Dang Van method can be used to define a safety factor:

Safety Factor = 1/d where

$$d = \text{Max}\,[\tau(t) / (b - a*ph(t))] \tag{7-88}$$

This function establishes the Dang Van Endurance Domain shown below:

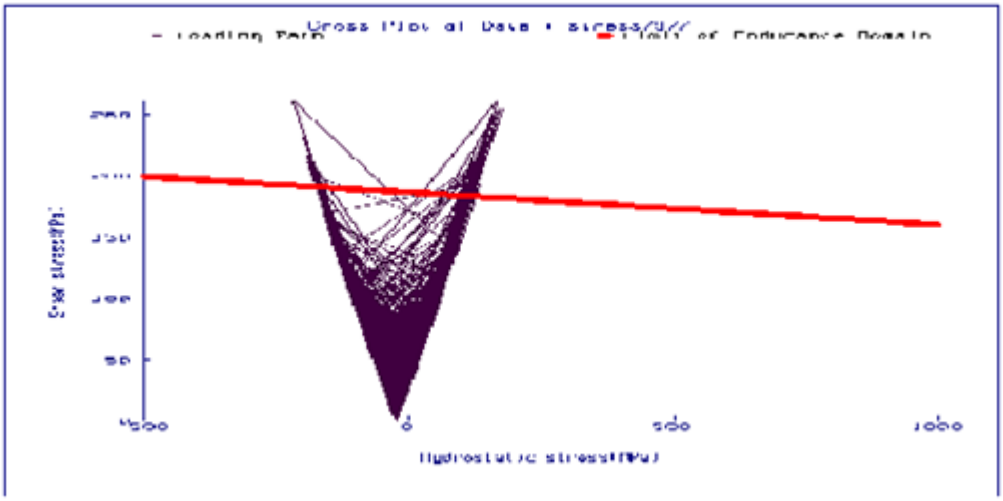


Figure 7-63 Dang Van Endurance Domain

A loading path (defined by the microscopic shear stress and hydrostatic tension) is shown on the above plot. Damage occurs when a loading line crosses the endurance limit line (safety factor =1). In this case the loading path does cross the line, so the safety factor will be less than 1. If a loading line does not cross the endurance limit, failure will not occur for any number of cycles on this loading line.

Implementation: Shown below are the steps of the Dang Van method:

For a given point M,

1. Evaluate the macroscopic stresses (Σ_{ij} , a stress tensor) at point M for each time t where $0 \le t \le \theta$. θ is the smallest interval such that $\Sigma(t, \theta) = \Sigma(t)$, i.e. θ is one block.
2. Split macroscopic stresses into hydrostatic and deviatoric components:

   Hydrostatic component, PH(t):

$$PH(t)=1/3*\text{trace}[\Sigma_{ij}(t)] \tag{7-89}$$

   Deviatoric component, $S_{ij}(t)$:

$$S_{ij}(t) = \Sigma_{ij}(t) - PH(t)*\delta_{ij} \text{ for trace elements only} \tag{7-90}$$

3. Determine the microscopic deviatoric residual stress tensor, dev ρ_{ij}^*:

   The element values of the tensor dev ρ_{ij}^* are the same throughout the cycles. These values are determined through an iterative process.
4. Determine deviatoric parts of localized stresses:

$$s_{ij}(t) = S_{ij}(t) + \text{dev } \rho_{ij}^* \tag{7-91}$$

Determine microscopic shear stress, $\tau(t)$: (7-92)

$$\tau(t) = 1/2 * \text{Tresca}\{s_{ij}(t)\} \quad (7\text{-}93)$$

where the Tresca is just the difference between the two largest principal stresses at this location and time.

5. Determine Damage Parameter, d:

$$d=\text{Max}[\tau(t)/(b - a * ph(t))] \quad (7\text{-}94)$$

The maximum is taken over all times.

6. Failure Criterion:

If d < 1, fatigue failure will not occur. Safety factor = 1/d

This process is all relatively straightforward, with the exception of the determination of the microscopic deviatoric residual stress tensor dev ρ_{ij}. This is explained as follows.

Determination of Microscopic Deviatoric Residual Stress

The determination of the so-called "Microscopic Residual Stress" ρ_{ij}* is somewhat complex in nature. It is based on the supposition that on a microscopic level, in critical grains there will be some plastic deformation even below the fatigue limit and there will be shakedown leading to the development of a residual stress. The way this residual stress is calculated is by a combination of isotropic and kinematic hardening. This process starts with the definition of an initial yield surface, centered on zero macroscopic stress. Successive loading points from the stress history are applied. When the loading point exceeds the yield stress, the yield surface moves to ensure it still contains the current loading point (kinematic hardening) and also grows (isotropic hardening). The rate at which the yield surface grows is determined by the hardening parameter. As the loading history is repeated, the surface grows until it reaches a stabilized condition in which it encompasses the entire loading history and there is no further movement or growth. The displacement of the center of the yield surface from the center of the original yield surface defines the microscopic residual stress. This process is illustrated in the following diagram.

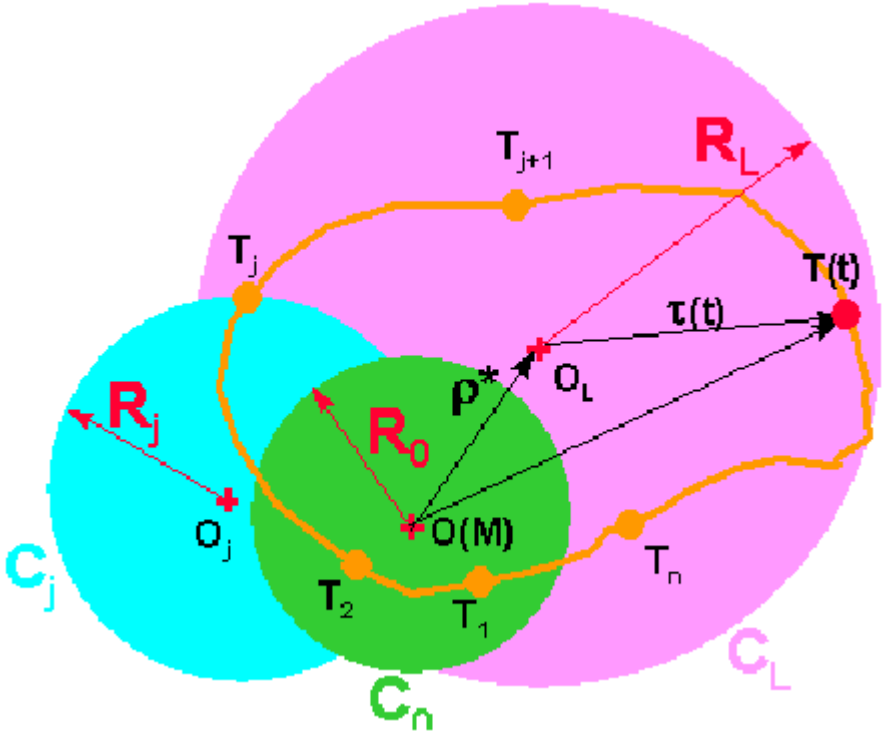


Figure 7-64 Definition of Microscopic Residual Stress

Notice that this is a 2-dimensional representation of the process. In practice the process can be thought of as an iterative method for finding the center of a 9-dimensional hypersphere which will encompass the whole loading history. The “hardening parameter” is simply a numerical parameter which controls the speed at which convergence is approached. Larger values of this parameter will give more rapid convergence but may result in some overshoot.

Note:	The Dang Van method is a full 3-D method and is applicable to 2-D and 3-D stress states irrespective of coordinate system.

8 Crack Growth

- Introduction 532
- K Solution Library (PKSOL) 535
- Crack Growth Prediction (PCRACK) 556
- Reviewing Results (PCPOST) 571
- Batch Operations 578
- Crack Growth Data Analysis (MFCG) 580
- NASA/FLAGRO 589
- Crack Growth Analysis Theory 593

Introduction

This chapter on crack growth consists of descriptions of various MSC Fatigue modules. These modules can be accessed from subordinate forms of the MSC Fatigue main form within MSC Fatigue or Patran. With the analysis set to Crack Growth, either the MSC Fatigue module PKSOL(K-solutions), PCRACK (crack growth), or PCPOST (postprocessor) will initiate.

Please familiarize yourself with Introduction (Ch. 1) before reading this chapter. Please also read Introduction, 290 in Chapter 5 for more information and Figure 6-2 for a schematic of the submittal process for crack growth analysis.

PCRACK is a cycle-by-cycle fatigue crack growth prediction module which calculates the crack extension, da, in each cycle of a sequence of cycles and adds it to the current crack size, a. This process proceeds until the failure condition is reached or a preset crack size is achieved. The driving force for crack growth in each cycle is the effective range of stress intensity, delta K (ΔK). In each cycle, the apparent or applied ΔK is calculated from the stress range, the crack size and the geometry of the component or structure under consideration. This is then modified to effective ΔK to account for the possible occurrence of:

- crack closure
- history effects
- notch effects
- environmental influence
- static fracture mode contribution

Taking these effects into account, the crack propagation life may be modeled accurately and this can only be achieved on a cycle-by-cycle basis.

There are many applications of this approach including: design analysis, pre-prediction of test programs, defect tolerance assessment, formulation of an inspection related strategy, failure investigation, and decision support. These are discussed in detail in Fatigue Theory (Ch. 16).

In addition to the crack growth module PCRACK, the K Solution library module, PKSOL is also described. It contains the means to calculate the fracture mechanics stress intensity factor, K, for over 35 geometric cases. The module also generates compliance function tables for use in the MSC Fatigue fracture mechanics prediction module, PCRACK, where K is calculated as a function of crack size, stress history and geometry.

PCPOST allows for viewing of final crack growth information.

All of these modules can be run in stand alone mode also by typing the appropriate symbol at the system prompt (i.e. **pksol**, **pcrack**, **pcpost**).

Terminal Definition

MSC Fatigue runs on a wide range of computers and graphics devices. The parameters used by each graphics device must be defined by using the MENM module, prior to the first use of MSC Fatigue. For details, please see Module Operations (Ch. 18). This is automatically accomplished when running MSC Fatigue from Patran and is transparent to the user and defaults to the Motif driver.

Analysis Route

To carry out a crack growth analysis from within MSC Fatigue, the following programs are required:

PKSOL	Enables the generation of a compliance function for the crack geometry under consideration. This is done via the PKSOL module which can be accessed from the Solution Parameters form from MSC Fatigue's main form or may be run separately by typing **pksol** at the system prompt. In any case, to submit a job from MSC Fatigue, the compliance function must be selected from the Solution Parameter form.
FatigueSubmit	Shell script (necessary for submittal from MSC Fatigue or Patran). When submitting a full analysis this way, the next three programs are run automatically via this script.
PAT3FAT	Translator; creates the fatigue input file, `jobname.fes`. This file contains all the information necessary to submit to the fatigue preprocessor.
FATTRANS	New translator that can be used in place of PAT3FAT.
PCRACK	Crack growth preprocessor. The difference between this preprocessing and that of the pre-processing done for a crack initiation or an total life analysis is that the cycles are re-ordered into their original sequence after rainflow counting is done.
PCRACK	Computes the cycle-by-cycle crack growth life.
PCPOST	The results may be reviewed using this module.

All programs in the MSC Fatigue system may be executed by typing the name of the program or its symbol. These programs may ask questions which are not normally presented to you since they are executed as batch jobs when called from the pre-/post-processing environment. The programs normally used in a typical or basic fatigue analysis are listed below.

1. Data Preparation

PFMAT	Materials Database Manager and BS5400 Weld Classification Advisor
PTIME	Time History Database Manager and ASCII Time History File Convertor
PVXMUL	A Peak-Valley Extraction Program for Reducing Lengthy Time Histories
MMFD	A Multi-file Display Program
PKSOL	Compliance Function Library/K Solution Generator

2. Crack Growth Analysis

PCRACK	Crack Growth Preprocessor (cycle counter/reorderer)
PCRACK	Crack Growth Fatigue Analyzer
PCPOST	Crack Growth Postprocessor

3. General Utilities

PCRACK	FES File ASCII/Binary Convertor
PFTRM	Terminal Driver
MCONFIL	Binary to Binary File Convertor

Necessary Files

The crack growth analysis requires the definition of a compliance function. Using the stress information selected, it then performs the crack growth calculation, using the crack growth setup forms in MSC Fatigue. For this reason, it is not a good idea to attempt a crack growth analysis outside the pre-/post- processor unless the job has already been defined at least once from within.

To carry out a crack growth calculation, the following files must already be present in the user's directory.

jobname.fin	This file contains all the analysis parameters that were defined in the main and subordinate MSC Fatigue forms (e.g., loading time history data file names). In addition, the analysis type and job titles are also defined in this file. A full description of this file is contained in The Job Information File (jobname.fin), 369.
Database	Other pertinent information such as the nodes or elements to use in determining the nominal stress (far field stress) is contained in the Patran database in the form of a group. The component stresses from these locations will be used, superimposed, averaged, and resolved (dependent on various parameter requests) for use as the far field stress in the crack growth equations.
Results Files	The actual stress results used in the crack growth analysis can be stored in the database or can come from external results files such as Patran results files (`.nod, .els`) or a Patran FEA (`jobname.res`) results file.
filename.ksn	A compliance function file produced by PKSOL. The use of PKSOL is defined in K Solution Library (PKSOL), 535.
Additional Files	Other files that are necessary to complete a successful crack growth analysis are the time history files (`ptime.adb`, `ptime.tdb` and `*.dac`), and the materials database (`nmats.mbd`) which is generally held in a central location and not necessary to be located in the user's local directory.

After the translator has been run (described in The Translator (PAT3FAT or FATTRANS), 294) and the fatigue input file (`jobname.fes`) has been created, the crack growth preprocessor, PCRACK, is run. A `jobname.tcy` file is created which is a scaled, time-ordered, rainflow cycle count for nominal stresses of the loading time history. The crack growth analysis is performed by the program PCRACK. When run in interactive mode,

this program asks for a number of input parameters which are passed in through the `jobname.fin`, `jobname.fes` and `jobname.tcy` files when run from the MSC Fatigue menus. A full description of file content is provided in Description of Files, 369.

K Solution Library (PKSOL)

The K Solution library module, PKSOL, contains the means to calculate the fracture mechanics stress intensity factor, K, for over 35 geometric cases. The module provides a means to generate a compliance function table for use in the MSC Fatigue fracture mechanics prediction module PCRACK where K is calculated as a function of crack size, stress history, and geometry.

Background Information

The fracture mechanics stress intensity factor, universally known as K (or ΔK in fatigue), relates stress and crack size into a single parameter which is the driving force for the behavior of cracks in engineering materials in terms of fracture, stress corrosion or fatigue. Its use relies on the similitude concept which implies that, independent of geometry, if K is the same then the response is the same. The relationship may be viewed as a triangle in Figure 8-1 in which if two are known the third can be derived. In this way, important decisions can be made regarding, for example, specified material toughness, NDT limits and inspection strategy, and operating stresses in engineering applications.

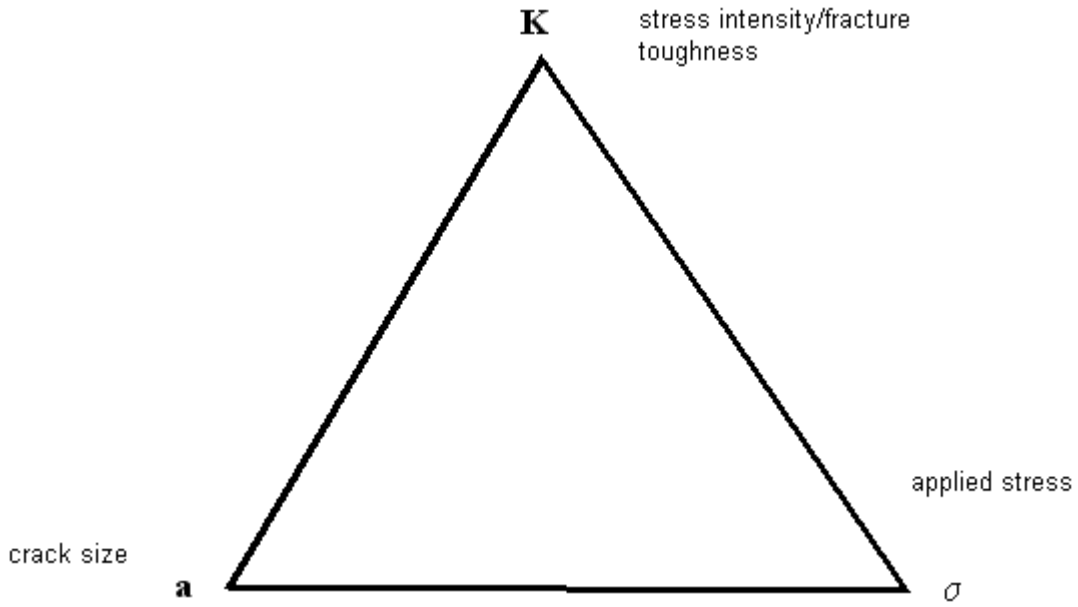


Figure 8-1 The Fracture Mechanics Triangle

The three parameters are related by the general expression:

$$K = Y\sigma\sqrt{\pi a} \tag{8-1}$$

where Y, the compliance function embodies both the global geometry of the component or structure and the crack size, shape, and location. Consequently, many cases, or K solutions, are available in standard references ((Ref. 3.) through (Ref. 8.)) and more than 35 of the most significant solutions for engineering applications are coded into the PKSOL library module. They may be used to generate a compliance function table for a specific geometry and case, for all crack sizes.

Example

As a specific example of using PKSOL to solve for one corner of the fracture mechanics triangle and its application to a crack growth analysis:

Suppose you want to be able to stop the analysis at a crack length that would cause a failure if the limit load occurred in the next cycle. Imagine you have a time history of random cycles varying between 1 and 3kips that represents a repeatable sequence, say one flight, but does NOT include the once per life 10kips limit load. The analysis would proceed to model the crack growth cycle by cycle including all the retardation, overshoot, and crack closure effects, but there are several ways of ending the process. One of them is to specify the final crack length.

The analyst would use the K solution library module (PKSOL), as a single shot solver, to calculate the crack size when the fracture toughness is exceeded at the maximum stress corresponding to the limit load. (It is also possible to calculate a final crack size when plastic collapse of the remaining section occurs rather than a KIC fracture). This crack size is then used to limit the crack growth process and effectively checks on a cycle by cycle basis "have I reached a crack size where the residual strength compromises the design limit strength?"

Module Operation

PKSOL may be accessed from the MSC Fatigue Solution Parameter form available from the main menu within MSC Fatigue or Patran when the Analysis Type is set to Crack Growth, or it may be run outside of Patran by typing the symbol **pksol**. A general description of the steps involved in a PKSOL session is shown in Table 8-1 to calculate a missing corner of the triangle.

Table 8-1 Usage of PKSOL

Step	Description	Examples
1	Select units	mm
2	Select calculation type selection	Solve for stress intensity factor, K
3	Select geometry type selection (which general shape)	Cracks at holes
4	Select specific geometry shape selection	Singe crack in hole in tension
5	Input data to define specimen and crack geometry	User enters specimen dimensions (via define and crack size, stress range)
6	Select post-processing options	Plot K against crack sizes for the above case(s)

Once initiated, the PKSOL program will ask the user to select the crack length units to be used as in Figure 8-3.

When first invoked PKSOL appears displaying two forms.

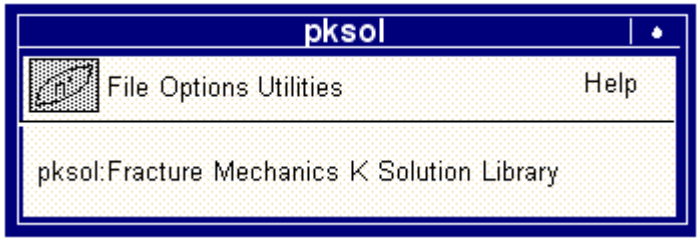


Figure 8-2 PKSOL Utility Form

The top, small form is a generic form and allows for general program control. This is discussed in detail in Module Operations (Ch. 18) for the Motif driver.

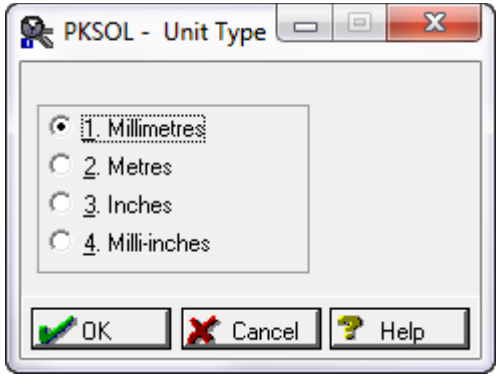


Figure 8-3 PKSOL Units Selections

Having defined the length units, the user will then be asked to define the type of calculation that he wishes to carry out. See Figure 8-4.

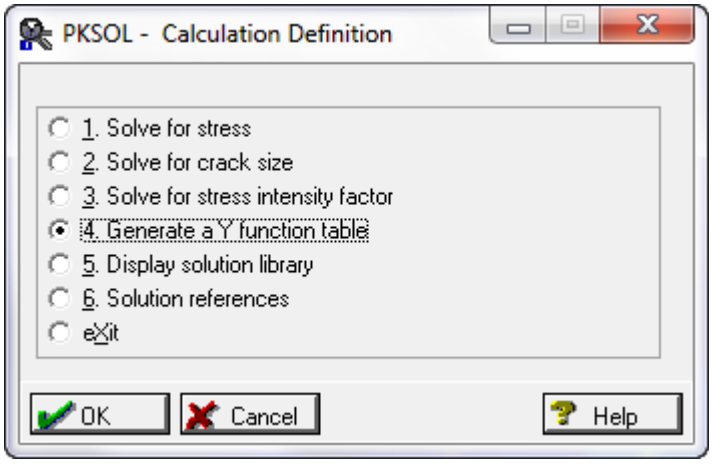


Figure 8-4 PKSOL Calculation Selections

The user must nominate one unknown on the fracture mechanics triangle (options 1-3), and over the subsequent few screens, supply the known data and define the specimens geometry.

Option	Description
Options 1-3	The first three options solve a different corner of the triangle BUT their operations are essentially the same. The user will be asked for a file name to store the output, however this file name is purely optional. Press the <RETURN> key if it is not necessary to store the information in a file. If a file name is supplied, it will be called `filename.lst` and will contain the results of the analysis in tabular format for printing.
Option 4	This generates a lookup table of the compliance function, Y, and is used in (8-1). The data in the Y function table is used in the crack growth calculation module PCRACK. If the user picks this option he will be asked for a file name which is mandatory. The result file will be called `filename.ksn`. This file if of binary format similar to the DAC files generated for load-time histories. The K solutions are grouped into nine general areas on the geometric definition menu from which you should make a selection (Figure 8-5).
Options 5 and 6	Selecting one of these options will display the K Solution Library. Selecting one will graphically display the geometries or list references.

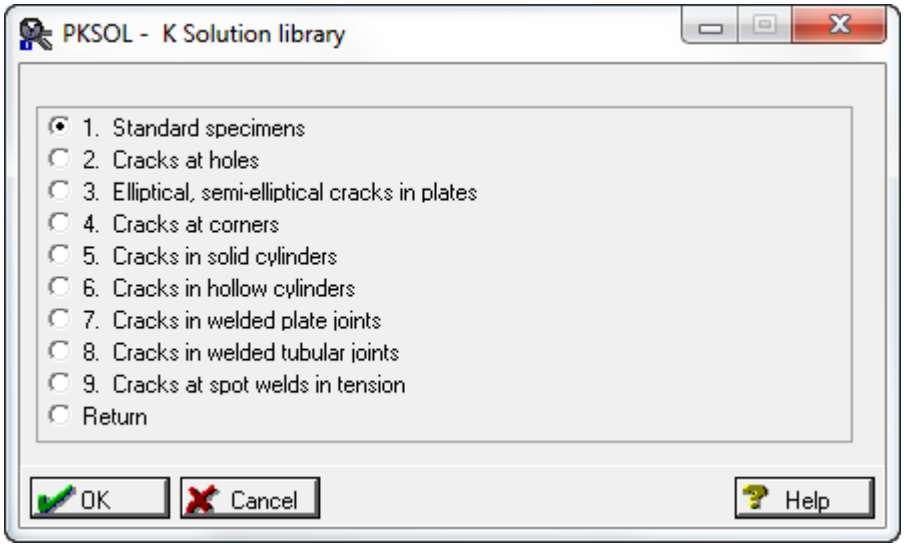


Figure 8-5 The Ten Basic Geometry Groups

For each group, another menu of specific geometric cases is given and selection from this leads to an input screen incorporating a graphic representation of the geometry highlighting unambiguously the required input dimensions. The various menus of specific cases and their geometries are shown in Figure 8-6 through Figure 8-13.

When a graphical image is presented for dimension input, the user must first use the mouse to select *Define* from the menu bar. He may then input the dimensions. Each parameter is input in turn and at the end of this sequence, alterations can be made if necessary using the syntax:

```
<parameter>=<value>
```

the default being <none> indicating no changes or no further changes. When all dimensions have been input, the user must use the mouse to select *Calculate* from the menu bar. To return to the non-graphic portion of PKSOL, click on *Return*.

To see graphical images of the different geometry options, press the F1 key while one of the options is highlighted (holding the mouse button down). A graphical display will appear. If all geometries do not show up in the graphical screen, click on *Next* or *Previous* to see the rest. Click on *Return* to return or *Hardcopy* if you wish to have a hard copy plot.

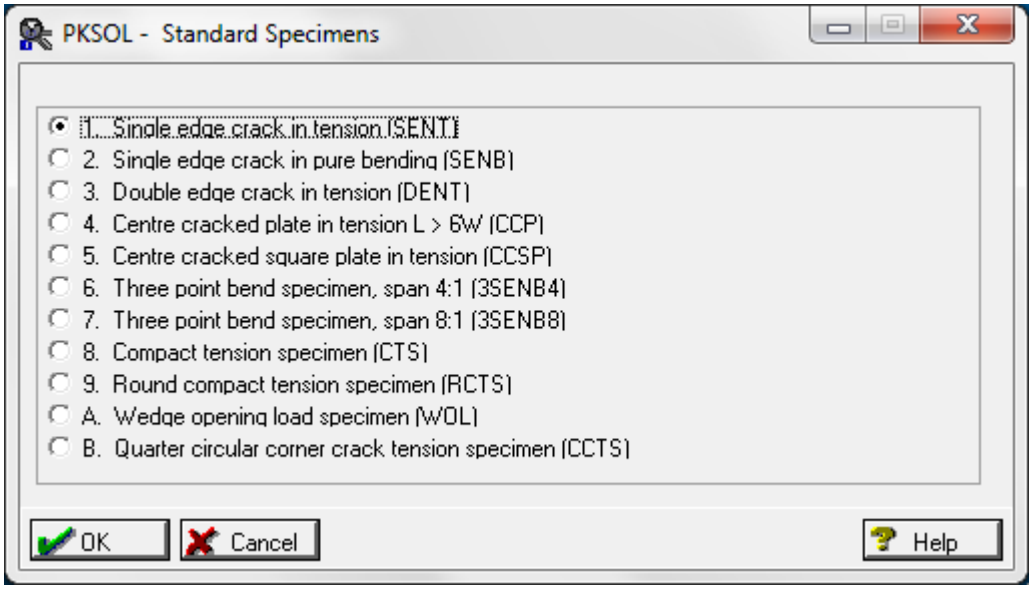


Figure 8-6 Standard Specimen Options

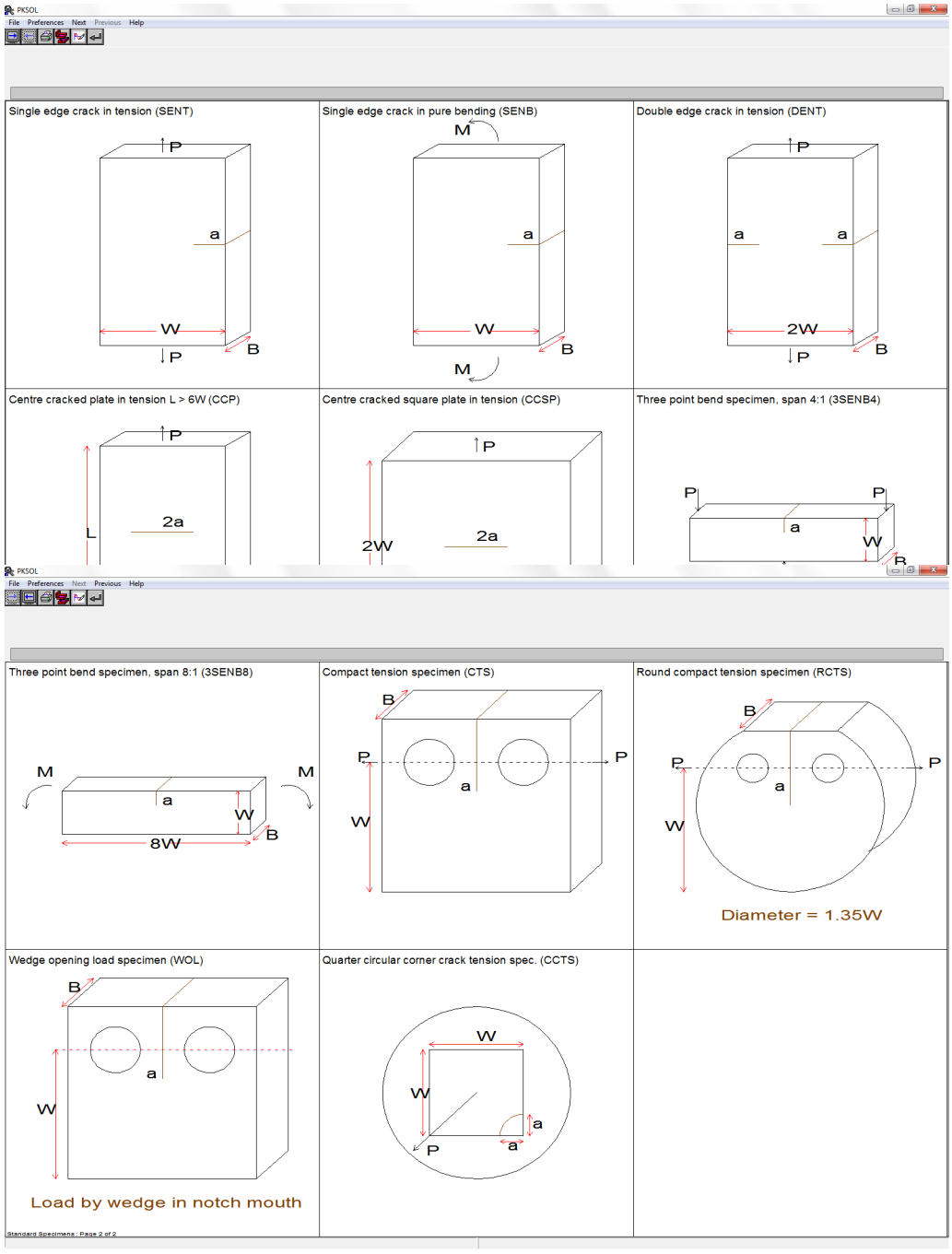


Figure 8-7 Standard Specimen Geometries

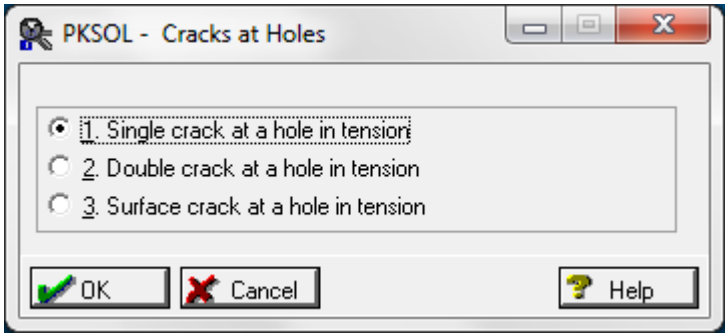


Figure 8-8 Cracks at Holes Options

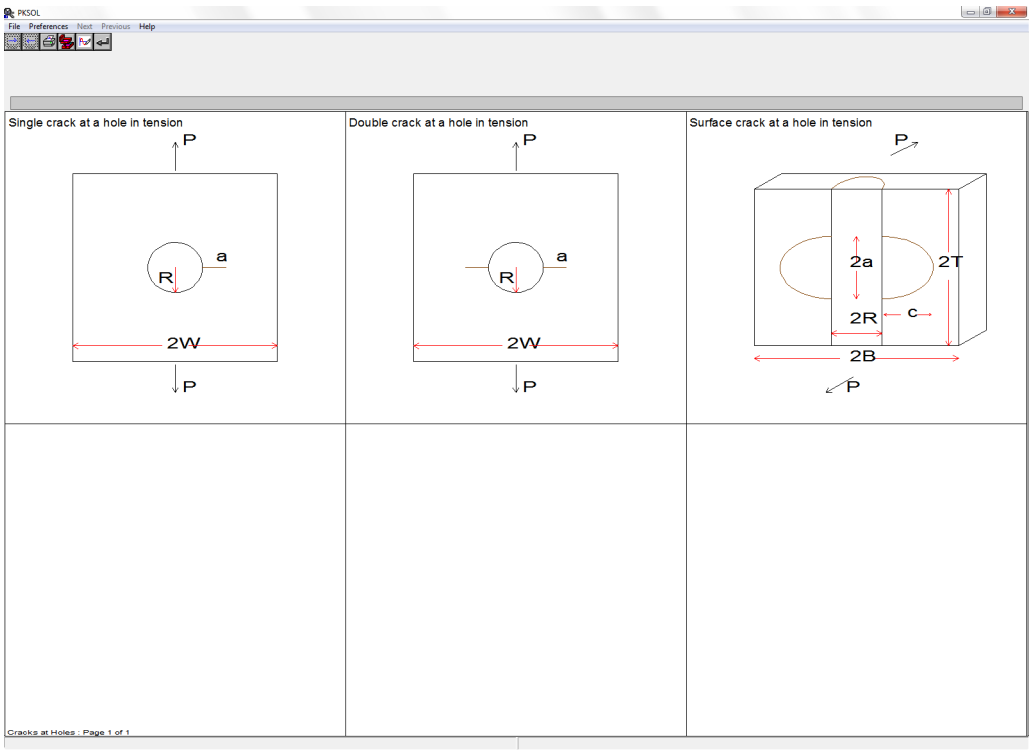


Figure 8-9 Cracks at Holes Geometries

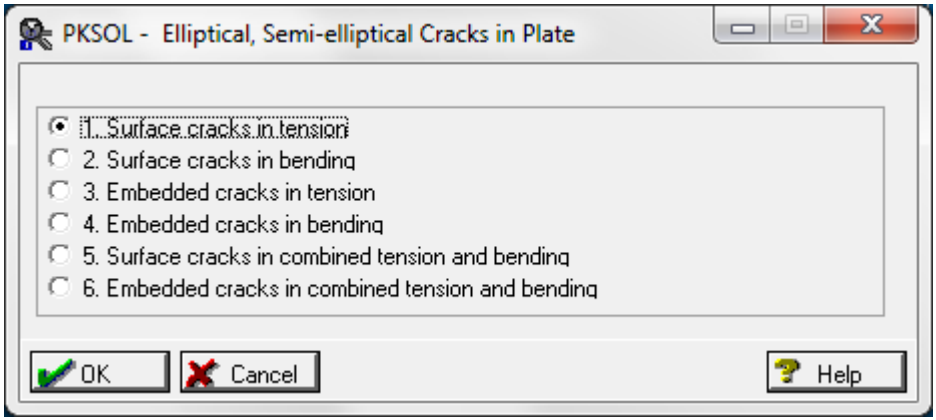


Figure 8-10 Elliptical, Semi-elliptical Cracks in Plates Options

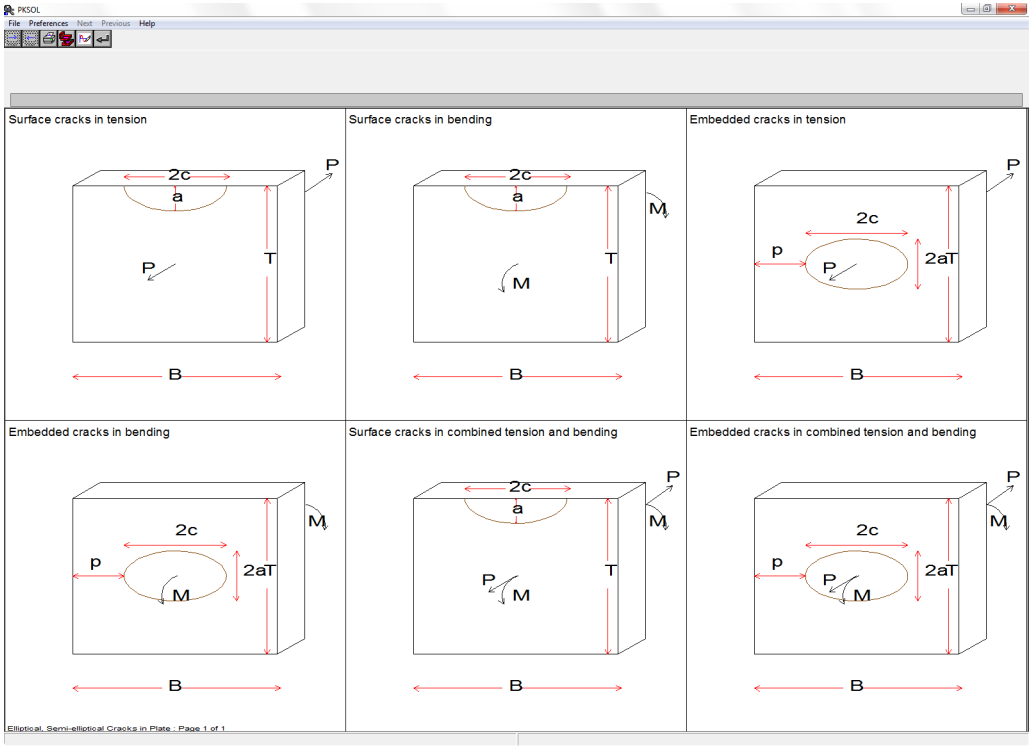


Figure 8-11 Elliptical, Semi-elliptical Cracks in Plates Geometries

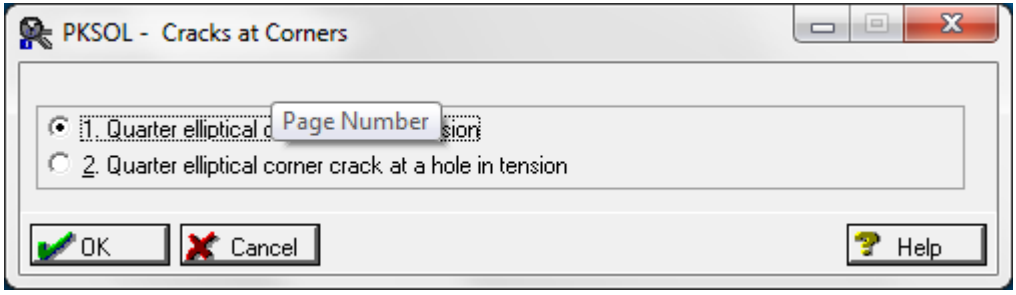


Figure 8-12 Cracks at Corners Options

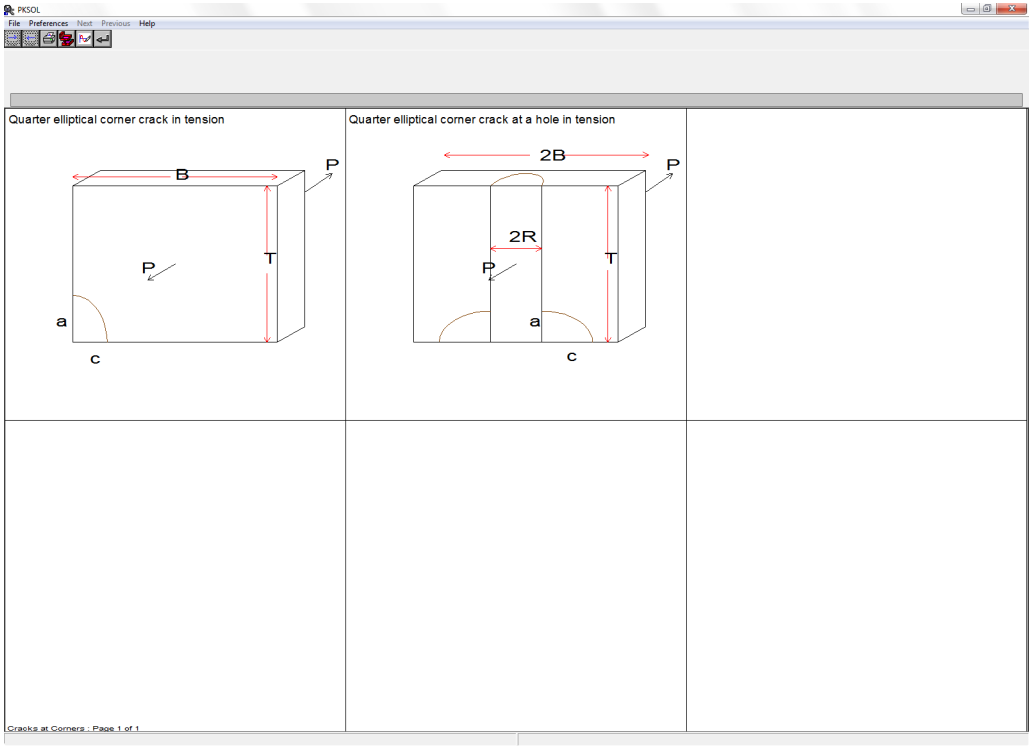


Figure 8-13 Cracks at Corners Geometries

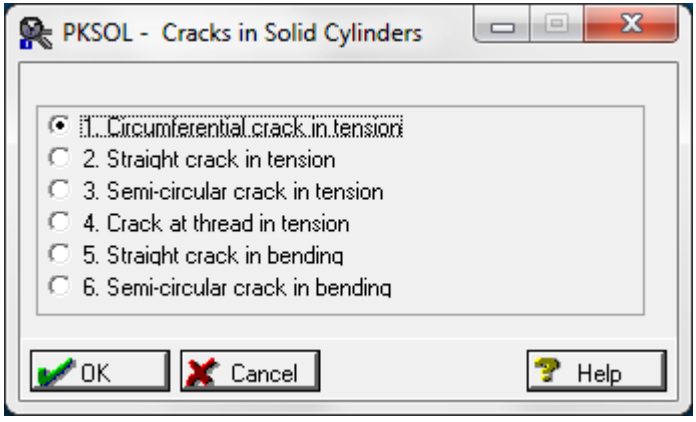


Figure 8-14 Cracks in Solid Cylinders Options

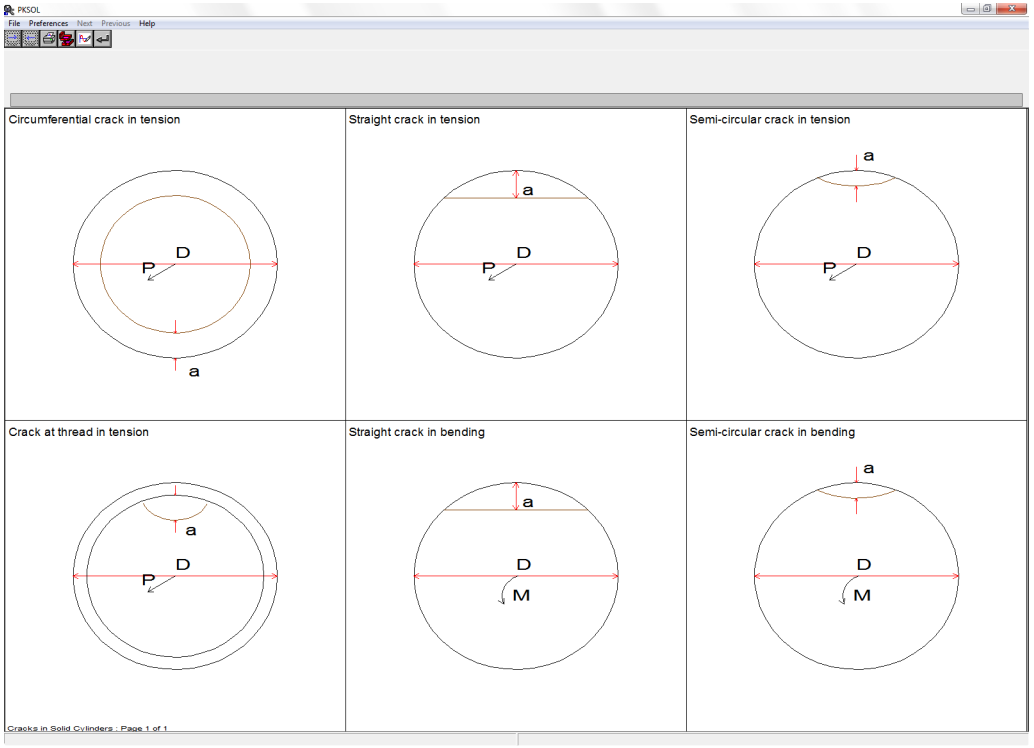


Figure 8-15 Cracks in Solid Cylinders Geometries

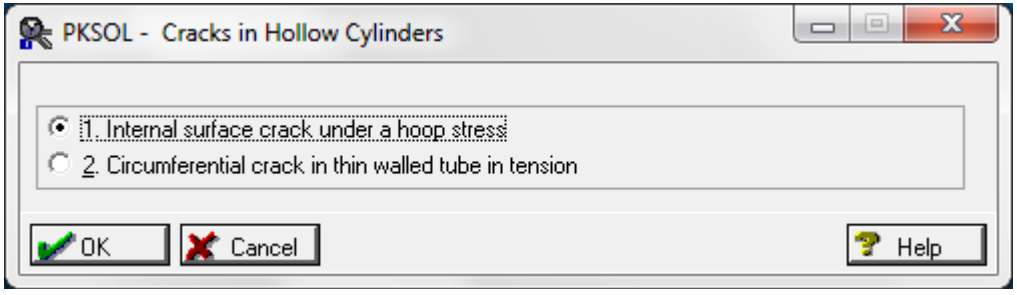


Figure 8-16 Cracks in Hollow Cylinders Options

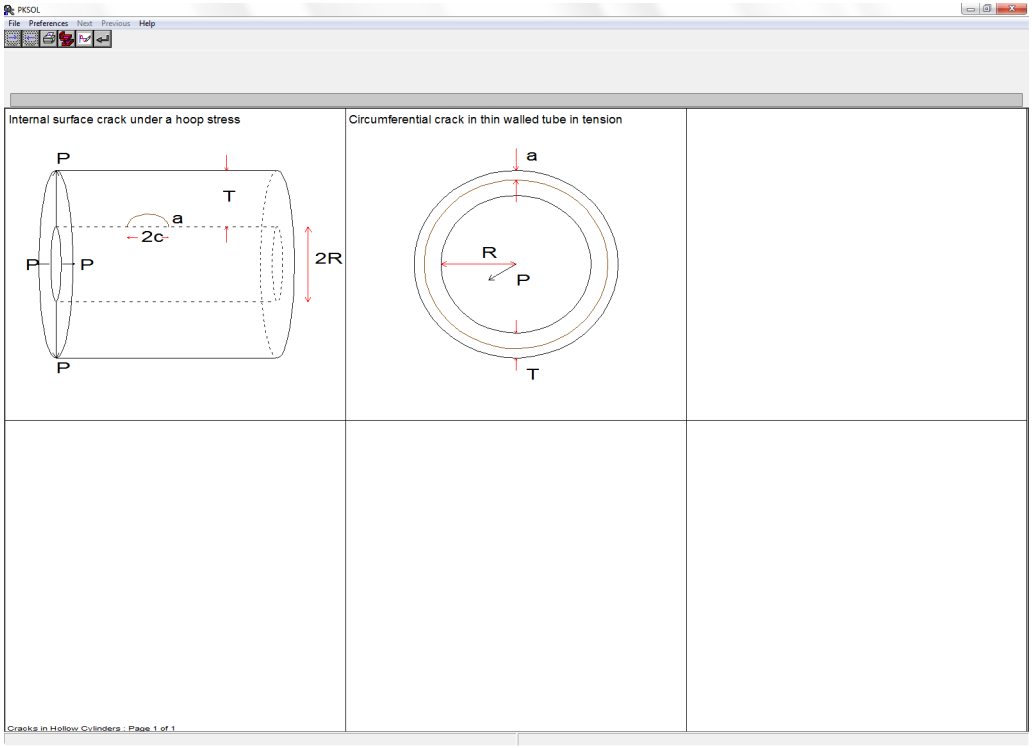


Figure 8-17 Cracks in Hollow Cylinders Geometries

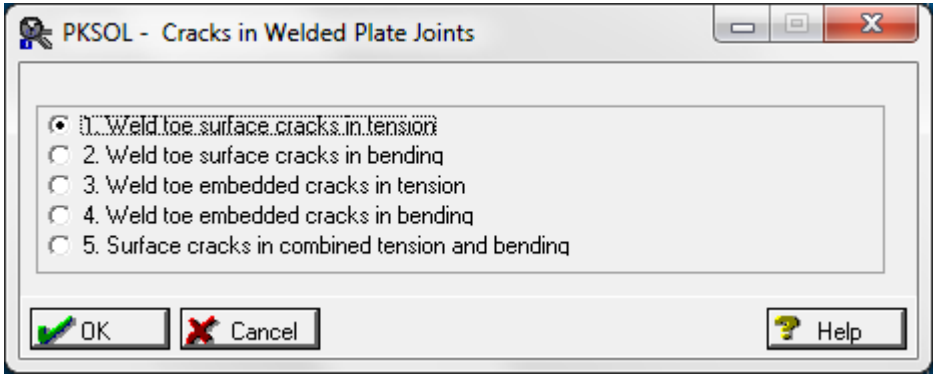


Figure 8-18 Cracks in Welded Joints Plate Options

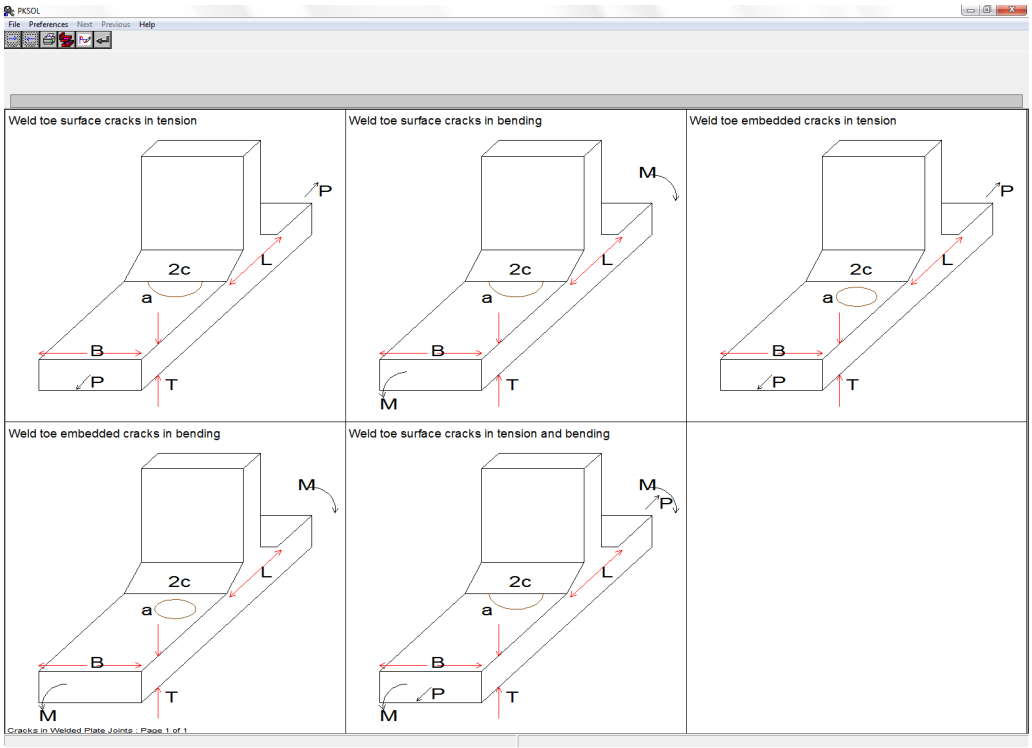


Figure 8-19 Cracks in Welded Plate Joints Geometries

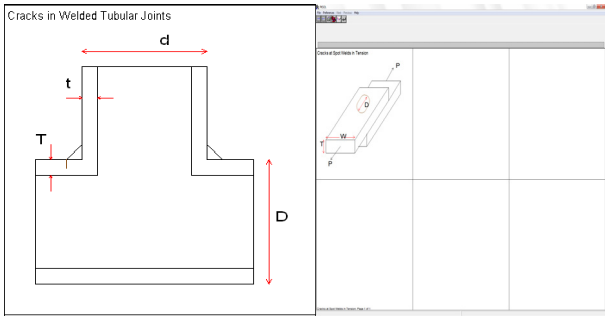


Figure 8-20 Cracks in Welded Tubular Joints and Cracks at Spot Welds in Tension Geometries

Once one of these geometries is selected, a graphical screen is displayed which allows for the definition of the local geometry dimensions. The main form (general options) is explained in Module Operations (Ch. 18). However, the Define, Calculate, and Return options on the main form are specific to this application.

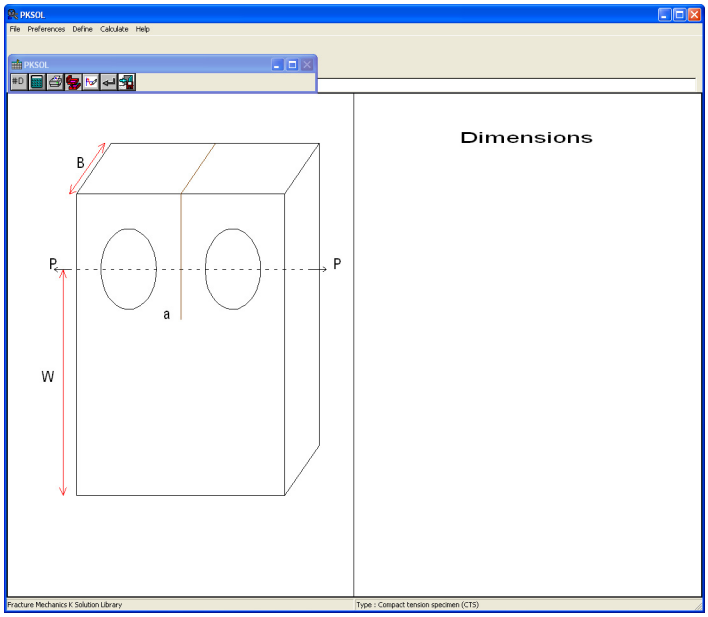


Figure 8-21 Graphical Definition of a Compliance Function

Option	Description
Define	Allows for input of the dimensions through the Command line. Dimensions appear to the right as they are defined. This option must be used first.
Calculate	Calculates the compliance function and puts the user into the post-analysis options.
File Return	Returns the user to the previous menu without calculating the compliance function. This option is actually found under the File pull-down menu.

If the user picks option A (user parametric definition) from Figure 8-5, a form will be presented that asks for the constants in the equation:

$$Y = C_1 + C_2\left(\frac{a}{T}\right) + C_3\left(\frac{a}{T}\right)^2 + C_4\left(\frac{a}{T}\right)^3 + C_5\left(\frac{a}{T}\right)^4 \quad (8\text{-}2)$$

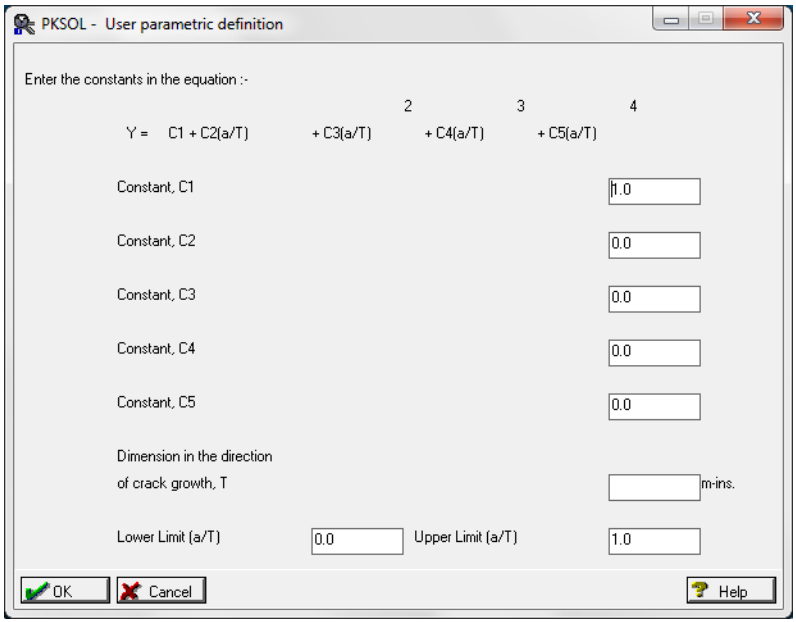


Figure 8-22 User Parametric Definition

This form allows the user to define a compliance function using a 5th order polynomial based on results from test or FEA, or from standard handbooks. The polynomial uses a non-dimensional crack size derived by dividing crack size (a) by total possible crack size (T).

By following the steps described in Table 8-1 and the on-screen instructions, a results screen is reached. For example, when calculating a critical crack size, a result display, such as in Figure 8-23, will result.

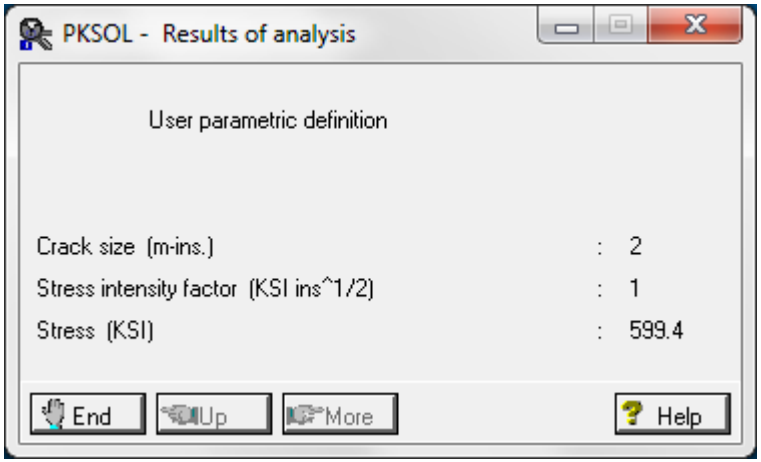


Figure 8-23 Execution Results for Critical Crack Size

Post Analysis Options

After successful completion of any of the options from Figure 8-4, the user will be presented with some post-processing options as shown in Figure 8-24. If options 1- 3 on the calculation menu (Figure 8-4) are chosen, the postprocessing options in Figure 8-24 will be presented. If option 4 is chosen, then the post-processing options in Figure 8-25 will be presented.

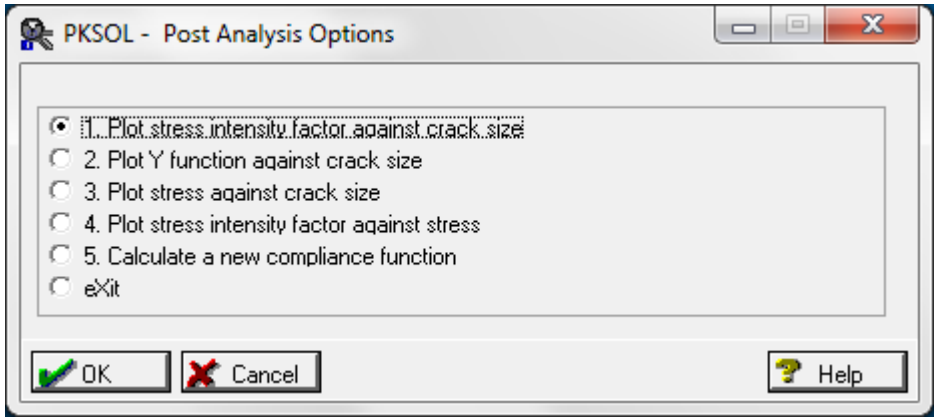


Figure 8-24 Postprocessing Options

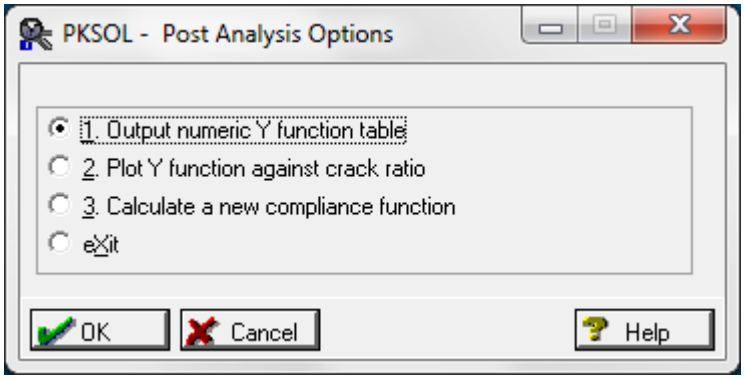


Figure 8-25 More Postprocessing Options

Field	Description
Plot (any)	Plots the requested data for the compliance function. It also writes an x-y data file with the name `filename.xyd` and a `filename.tem` (template) file in the form of a stress intensity factor vs. crack size when a Y-function is calculated. An example of this type of display is shown in Figure 8-27. With these files, the user has access to the data for plotting within the pre/postprocessor from the MSC Fatigue solution parameter form when the analysis type is set to crack growth.
Output Y Table	The Y-function table may be displayed to the screen or saved as an ASCII file for use in subsequent report preparation. If an output file is specified when this option is invoked, a file called `filename.lst` will be saved. If only the display to the screen is required press <RETURN>. An example of part of a Y function table listing is shown in Figure 8-26. It shows the value of a compliance tabulated as a function of the normalized crack size (a/W). The five columns provide up to 3 decimal places resolution for the a/W parameter.
Calculate a new compliance function	This option will return the user to the general options screen as shown in Figure 8-4.
eXit	This option will exit PKSOL and return the user to either the system level or back into the pre-/postprocessor if that is where the user invoked PKSOL.

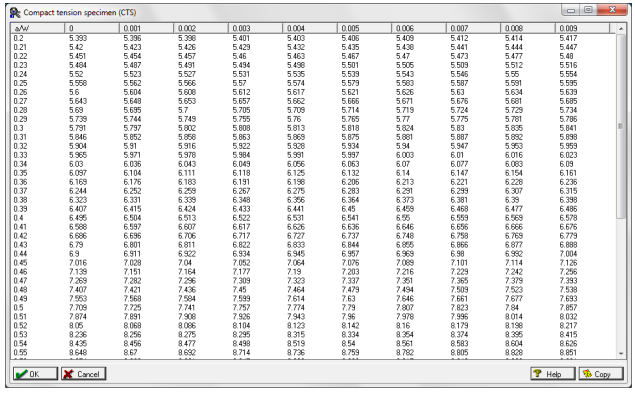


Figure 8-26 Function Tabular Display

When a plot is requested within PKSOL, a screen similar to the one below appears. Various utilities are available from the main form. These are explained fully in Module Operations (Ch. 18).

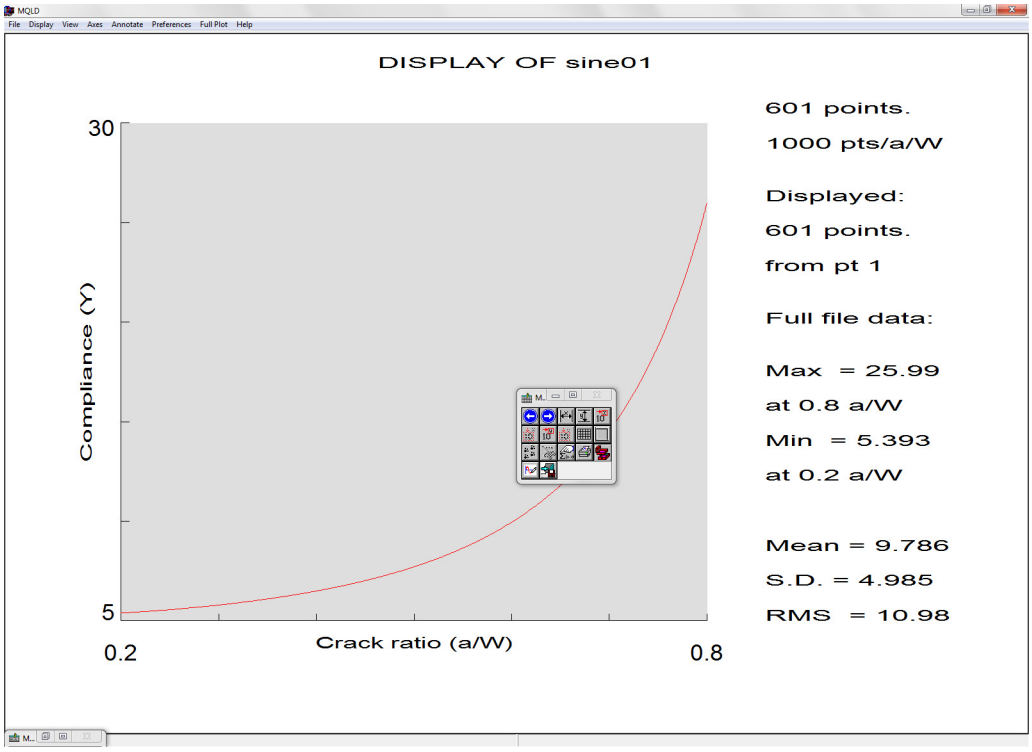


Figure 8-27 Y Function Plotted as a Function of Crack Size for Constant Stress

PKSOL K Solution References

Table 8-2 PKSOL K Solution References

Name	Reference
SENT	Murakame
SENB	Murakame
DENT	Murakame
CCP (L>6W)	Murakame
CCP (square)	Murakame
3SENB4	Murakame
3SENB8	Murakame
CTS	Murakame
Round CTS	Murakame

Table 8-2 PKSOL K Solution References

Name	Reference
WOL	Walker and May
Single crack at hole	Gray
Double crack at hole	Gray
Surface crack at hole	Newman and Raju
Surface crack in tension	Newman and Raju and PD6493
Surface crack in bending	Newman and Raju and PD6493
Embedded crack in tension	Newman and Raju and PD6493
Embedded crack in bending	Newman and Raju and PD6493
Surface crack in combined tension and bending	Newman and Raju and PD6493
Embedded crack in combined tension and bending	Newman and Raju and PD6493
Quarter elliptical corner crack in tension	Newman and Raju
Quarter elliptical corner crack at a hole	Newman and Raju
Circumferential crack in tension	Gray
Straight crack in tension	James
Semi circular crack in tension	James
Crack at thread in tension	James
Straight crack in tension	James
Semi circular crack in bending	James
Internal surface crack under hoop stress	Newman and Raju
Circumferential crack in thin-walled tube	Murakame
Weld toe surface cracks in tension	PD6493
Weld toe surface cracks in bending	PD6493
Weld toe embedded cracks in tension	PD6493
Weld toe embedded cracks in bending	PD6493
Surface cracks in combined tension and bending	PD6493
Cracks in welded tubular joints	Kam
Cracks at Spot Welds	Smith

Murakame "Stress Intensity Factors Handbook" (Ed. Y. Murakame) (Pergamon) 1990 (ISBN 0-08-034809-2)

Walker and May "Compliance Functions for Various Types of Test Specimen Geometry" E. F. Walker and M. J. May, BISRA Open Report MG/E/307/67, 1967

Gray	"Convenient Closed Form Stress Intensity Factors for Common Crack Configurations" T.G.F. Gray, Int. Jnl. Fracture 13, 1977, p65
Newman and Raju	"Stress Intensity Factor Equations for Cracks in Three-Dimensional Finite Bodies" J.C. Newman, Jr. and I.S. Raju ASTM STP 791, 1983, pI-238 "An Empirical Stress Intensity Factor Equation for the Surface Crack" J. C. Newman, Jr. and I. S. Raju, Eng. Fract. Mechs. 15 1981, p185
PD6493	"Guidance on Methods for Assessing the Acceptability of Flaws in Fusion Welded Structures" (BSI) 1991 (ISBN 0-580-19634-8)
James	"Review and Synthesis of Stress Intensity Factor Solutions Applicable to Cracks in Bolts" L.A. James and W. J. Mills, Eng. Fract. Mech. 30, 1988, p641
Kam	Dr. Jacob Kam, University College London, priv. comm.
Smith	Professor R. F. Smith, University of Sheffield priv. comm.

Crack Growth Prediction (PCRACK)

PCRACK is a cycle-by-cycle fatigue crack growth prediction module which calculates the crack extension, da, in each cycle of a sequence of cycles and adds it to the current crack size, a. This process proceeds until the failure condition is reached or a preset crack size is achieved. The driving force for crack growth in each cycle is the effective range of stress intensity, delta K (ΔK). In each cycle, the apparent or applied ΔK is calculated from the stress range, the crack size, and the geometry of the component or structure under consideration. This is then modified to effective ΔK to account for the possible occurrence of:

- crack closure
- history effects
- notch effects
- environmental influence
- static fracture mode contribution

Consequently, the crack propagation life is modeled accurately and this can only be achieved on a cycle-by-cycle basis.

There are many applications of this approach including: design analysis, pre-prediction of test programs, defect tolerance assessment, inspection related strategy, failure investigation and decision support. These are discussed in detail in Fatigue Theory (Ch. 16).

Module Operation

The operation of PCRACK can be in two modes: within the MSC Fatigue and Patran environments or in stand alone mode. The only difference is that in stand alone mode, the user must supply the jobname. (In direct mode from a preprocessor, these are passed to PCRACK automatically.) PCRACK can be accessed directly from the operating system prompt by typing the symbol **pcrack**. Once PCRACK has been initiated

in either of these modes, two windows will be presented. The top, small form is a generic form and allows for general program control. This is discussed in detail in Module Operations (Ch. 18) for the Motif driver.

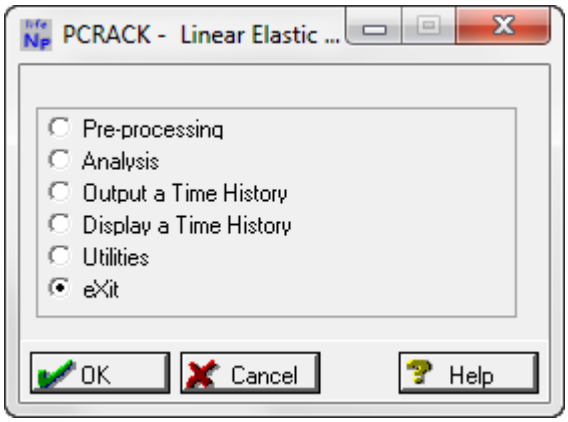


Figure 8-28 PCRACK Utility Form

Six items are available from the main menu of PCRACK which are discussed in the table below:

Option	Description
Preprocessing	Perform rainflow cycle counting and then re-orders the cycles in the order in which the component experiences them. The operation of this option is identical to that described in Fatigue Preprocessing, 318.
Analysis	Performs the actual crack growth analysis. This section describes this in detail.
Output a Time History	The combined response time history can be output to a file. See Output Time Histories, 353.
Display a Time History	The output response can be plotted. See Graphical Display of Time Histories, 355.
Utilities	These utilities are identical to those described in Utilities, 360.
eXit	Exits the program.

Results of a crack growth analysis can be viewed using a utility called PCPOST. Detailed operations of PCPOST are in Reviewing Results (PCPOST), 571.

Once an analysis is initiated in interactive mode, five input screens containing various questions appear sequentially in the order shown below.

General Information

Figure 8-29 shows the first screen that appears. If the user accesses PCRACK from the system level, he will have to supply a jobname. PCRACK already knows the jobname when entered from MSC Fatigue or Patran.

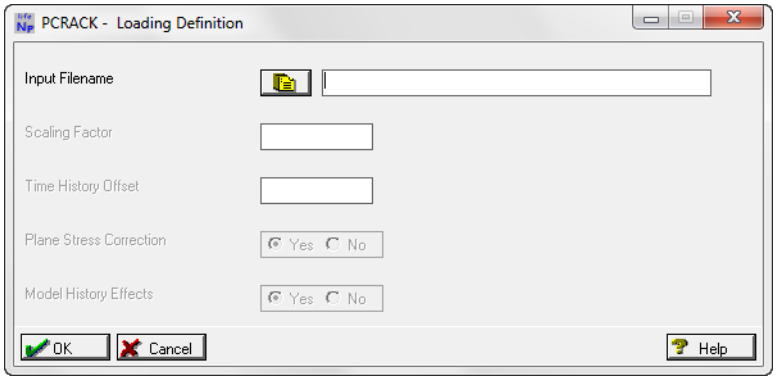


Figure 8-29 The PCRACK General Info Page

The fields to this screen are as follows:

Field	Description
Input File Name	The default will be the last jobname used in conjunction with a MSC Fatigue crack growth analysis. The jobname will pick up an existing `jobname.tcy` file which contains the local stress information. The `jobname.tcy` file is produced by the program FEFAT, the crack growth fatigue preprocessor. See Fatigue Preprocessing (Ch. 6).
Scaling Factor	This factor allows for scaling the stresses linearly to simulate the effect of a higher level of loading. The default answer is 1 (i.e., leave the stresses unscaled).
Time History Offset	The mean offset of the stresses may be altered here to simulate the effect of a further mean load or residual stresses. Only positive offsets are allowed. The default answer is 0 (i.e., no mean offset will be added). Remember that crack growth is highly sensitive to mean stress since the stress is raised to a power of m in the calculation of crack increment where m is the Paris law slope.
Plane Stress Correction	The materials data describing the crack growth rate relationship are obtained from test coupons which are stresses in a plane strain condition. A correction may be applied to modify these data to simulate a plane stress condition. The default answer to this question is N, or NO (i.e., no plane stress correction will be applied).
Model History Effects	If history effects are to be modeled, answer Yes to this question. Modelling the history effect means that the growth in one cycle is affected by the previous cycle history leading to retardations, accelerations, overshoots and other effects on growth rate and hence propagation life. It may be desirable to "turn off" the history effect to compare life predictions based on linear damage accumulation (no history effect) and those with history effects modeled.

At this point, the screen will clear by clicking **OK**, and the next page of input questions (Output Parameters) will be presented.

Output Parameters

The output parameters show up on the next screen shown in Figure 8-30.

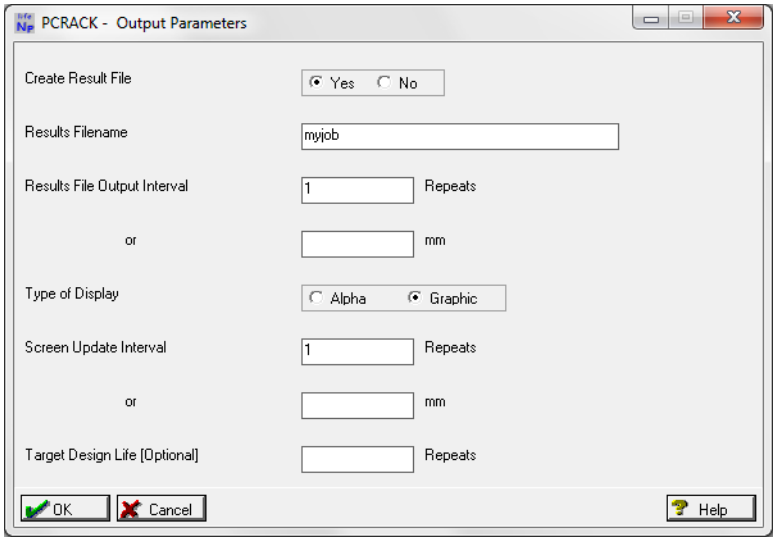


Figure 8-30 The PCRACK Output Parameters Page

The fields to this form are as follows:

Field	Description
Create Result FIle	If a results file is desired answer YES.
Results File Name	The PCRACK analysis generates a results file which will have the default name `jobname.crg`. An alternative name may be supplied in response to this question. If an output file already exists with the same name as the one supplied, overwrite permission will be requested.
Results File Output Interval	A snapshot of the calculation parameters will be taken at regular intervals and stored in the output results file (`jobname.crg`). The user may select how often the snapshot is taken by defining the number of life units between snapshots. The default will cause the gap to be between 100 and 1000 cycles, converted into user life units. If the snapshots are too frequent, the crack growth results postprocessor (PCPOST) allows the user to decimate or compact the results file to save disk space.

Field	Description
Type of Display	The graphical status display shows the rate of crack advance using an a-N graph and various computational parameters. If the user requests not to have a graphical display, a tabular display will be produced indicating crack length, elapsed life in repeats of the loading history and elapsed life in user defined units.
Screen Update Interval	The PCRACK graphical screen display will be updated at regular intervals. The update frequency should be chosen to reflect the speed of the computer on which the analysis is being carried out. An update interval of between 100 and 1000 cycles will be chosen using the default. Faster computers will require longer update intervals to prevent the computations becoming bound by the screen update rate.
Target Design Life	The crack growth analysis may be terminated after a set number of user life units as specified here. If the number of life units is left undefined as for the default answer, the analysis will proceed until another failure criterion is satisfied.

At this point, the screen will clear by clicking **OK**, and the next page of input questions (Local Geometry) will be presented.

Local Geometry

The next screen displayed is shown in Figure 8-31. The user will first be prompted for a file name containing the compliance function for the crack geometry of interest. This file was generated by PKSOL (see the previous section). Once the file name has been selected by clicking the **OK** button, the screen updates and shows the following fields which can be modified.

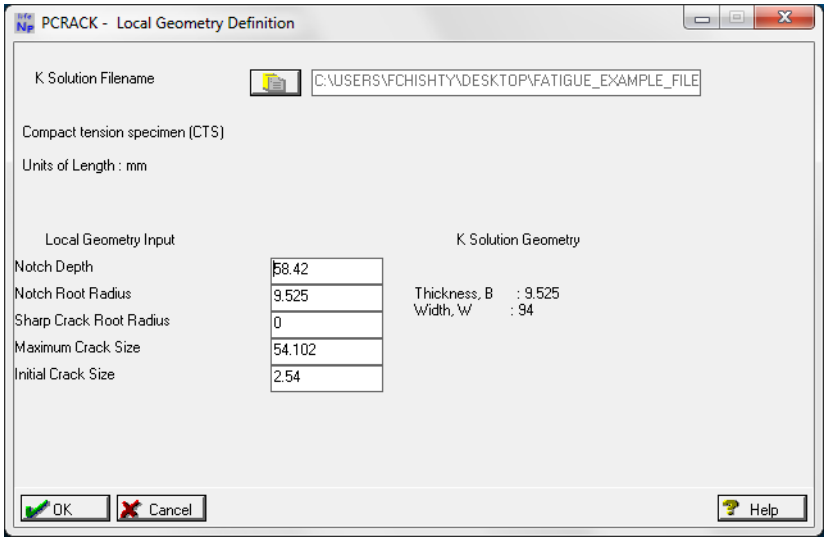


Figure 8-31 The Local Geometry Definition Screen

The fields of this form are described below:

Field	Description
K Solution File Name	The K solution is generated by the program PKSOL and is stored in a file with file type `filename.ksn`. The K solution relates crack size and compliance. For more detail, refer to K Solution Library (PKSOL), 535. The default K solution file name offered will be the last K solution used or generated. The local geometry will now be defined. Note that the local geometry fields do not appear until the K solution file name is supplied and accepted. The length units will be reported to you as already defined in the K solution file. For example: Units of length: inches The user will then be asked to define the physical size of any additional notch not modeled using FEA and not accounted for in the compliance function.
Notch Depth	If the crack under consideration is a surface crack, then a surface notch can be introduced into the global geometry from which the crack can grow. The crack will be influenced by the presence of the notch and its associated stress field. To correct the K solution for this, the local geometry of the notch has to be defined. The notch depth must be defined in the same length units as used for the crack definition when generating the K solution. Valid units will be either mm, m, milliinches or inches. If a notch is not present, (i.e., notch depth is zero), then the next two questions will not be asked and their defaults will be used.
Notch Root Radius	This input is the notch root radius which defaults to 0.000039in if the Notch Depth above is zero. It must be defined in the same length units as used for the crack definition when generating the K solution.

Field	Description
Sharp Crack Root Radius	This is the user-specified value of the root radius of a "sharp notch" (i.e., a real fatigue crack, which defaults to 0.000039in if the Notch Depth above is zero). The sharp crack root radius must be defined in the same length units as used for the crack definition when generating the K solution. Valid units are as for the previous question.
Maximum Crack Size	The default maximum crack size (n) is the component dimension in the cracking direction. This field enables you to specify a smaller value to take into account plastic collapse, yielding, etc. Note that if fracture toughness is exceeded at a crack size less than the minimum, then this condition will define the end of fatigue crack growth. All crack lengths MUST be defined in the same units as used for crack geometry when creating the K solution and you will be reminded of these units at the top of the input screen.
Initial Crack Size	This is a critical value. It should be either: 1.The size of a discovered defect. 2.A postulated defect. 3.The NDT limit (i.e., the largest defect that cannot reliably be detected by nondestructive testing). The initial crack size may be chosen in the range 0 to n where n is the maximum possible physical length of the crack based on the component geometry. A zero initial crack size may only be specified if a notch has been defined in the local geometry definition (see above). If no notch has been defined and you enter zero for the initial crack size, then PCRACK will compute the short crack parameter, l_o, and substitute this for the initial crack size. See Cycle-by-Cycle Approach, 600, for details on l_o.

Material and Environment

The next form enables the user to specify the material type and the test environment (Figure 8-32).

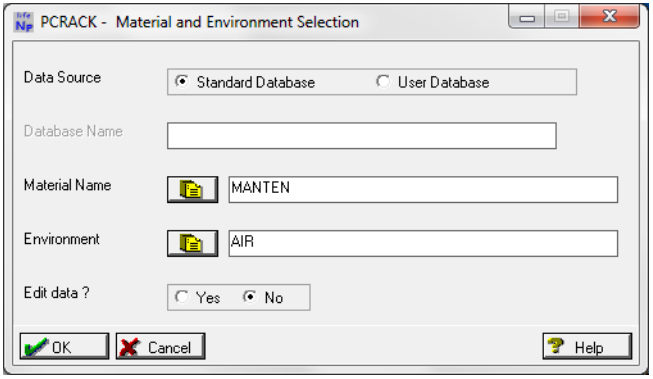


Figure 8-32 The Material and Environment Selection Screen

Field	Description
Data Source	This option allows users to select materials data from a user-defined database. The default is to search the standard database, which will exist in either the local or central directory.
Database Name	The name of the user-defined database to use. This database may exist anywhere on the system, but the default is to look in the local directory.
Material Name	The material name is the name used in the materials database under which are stored the parameters defining the crack growth properties of the material. The default name offered will be the last materials data set used. If the user wishes to specify another material, he may either type in a name that he knows exists or may use the List capability to select a material from a list. The screen that appears when a materials listing is shown allows the user to view not only all the available materials but also to view the material parameters associated with each material. This eliminates the need to run PFMAT to view these parameters during an interactive run.
Environment	There may be crack growth materials data sets stored in the database for a range of different environments. The user may chose which environment to use in the analysis. If an environment is specified for which there are no data in the database, the parameters for AIR will be assumed and an environment definition will be requested (see Environment Definition below). If the user wishes to specify another environment, he/she may either type in a name that he knows exists or may use the List capability to select an environment from a list.
Edit Data	This option allows the user to edit the materials data for the selected environment.

Material Parameter Editing

At this point, the form will clear and the materials parameters retrieved from the database presented either for air or for the chosen environment (Figure 8-33). The user may change any of these parameters for the duration of the PCRACK session. See Rules for Changing Young's Modulus, 171.

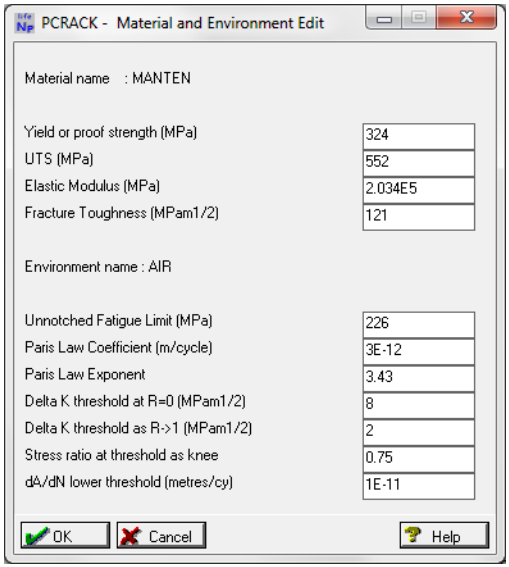


Figure 8-33 The Material and Environment Edit Screen

The analysis will now commence. If the status has been requested to appear graphically, then the screen will clear, and a graphics screen display will be updated regularly. The parameters on the top of this screen display are: elapsed life, accumulated crack size, current apparent ΔK , current effective ΔK and current growth rate da/dN.

Also, highlighted on the right-hand side, are the complexing effects currently modifying the apparent to effective ΔK (i.e., closure, history, notch, environment and static fracture modes).

Within the central rectangular area is a graphical display which updates and rescales automatically at regular intervals as specified. The graph shows crack size plotted against life, as shown in Figure 8-34.

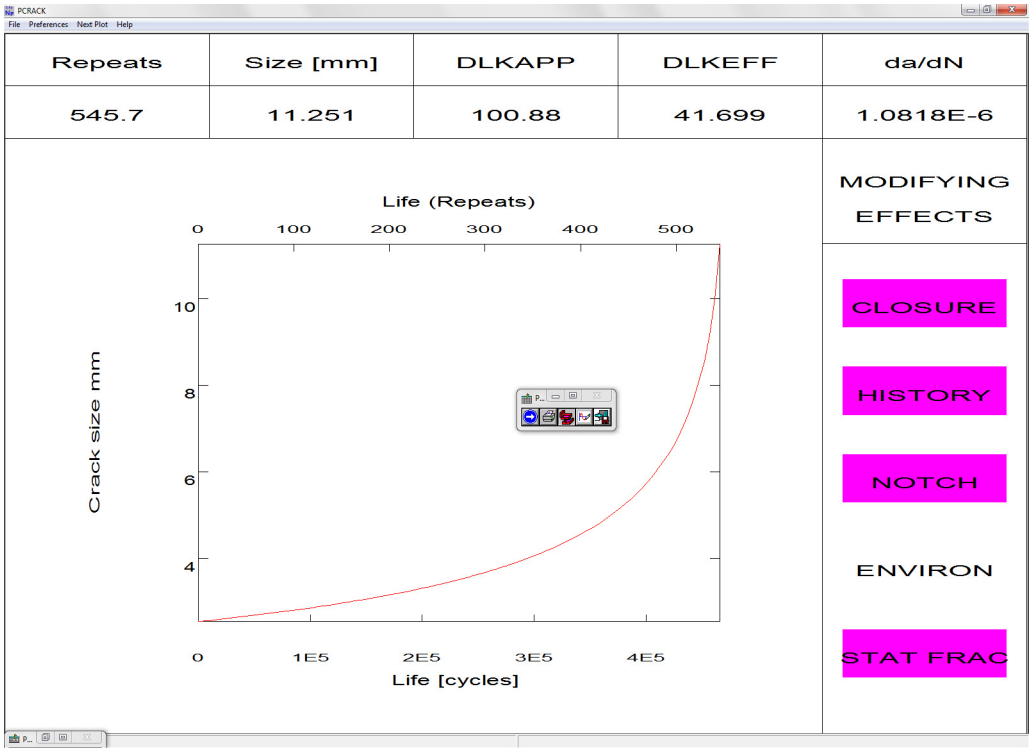


Figure 8-34 On-Line Graphics Display of Crack Size — Cycles

If the crack growth is negligibly small, then the process will cease after one repeat of the history or the design life, whichever is greater; this is to avoid infinite or exceptionally long runs. The crack growth analysis may be aborted at any time by clicking on the *Abort* option on the main form. The other options, Alpha and Replot, will convert the graphical display to an alphanumeric display and replot the screen respectively. If an alphanumeric display is requested it may be converted back to graphics by selecting the Pause button. A window will appear giving the options of *Resuming*, *Quitting*, or returning to *Graphical* display.

When the process stops, whether it is due to the final crack size or fracture toughness being exceeded or due to negligible growth or to a user abort, the final situation will be presented to you as shown in Figure 8-35.

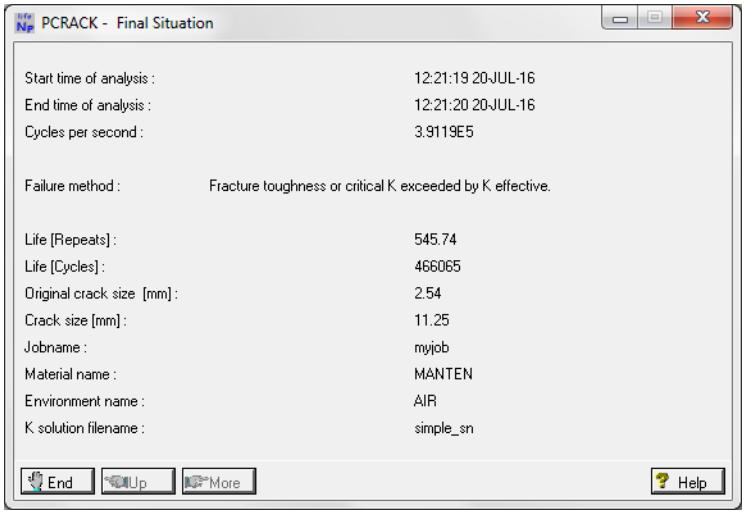


Figure 8-35 Final Situation Display

The start time and end time can be used to calculate the run time and processing speed in user-defined events per second. The list of parameters gives the current values at the point of termination. The final situation results page is also written to the `pfatigue.prt` file for later reference.

From this screen, the user may then proceed to the post analysis menu which follows on the next screen display.

Post Analysis Menu

After an initial crack growth job has been run, the Post Analysis menu appears as in Figure 8-36. The options are described below.

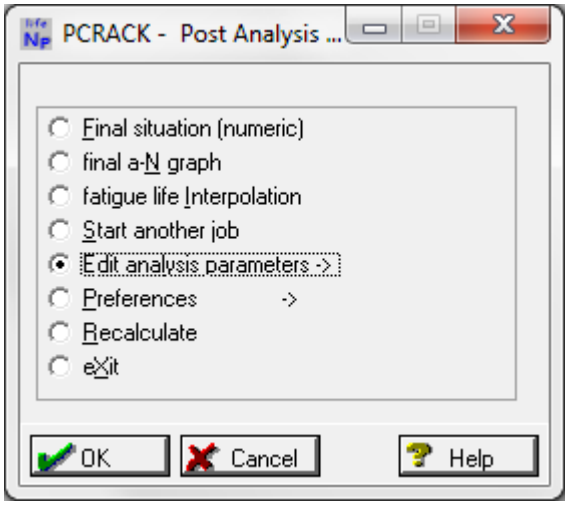


Figure 8-36 The Post Analysis Menu

Field	Description
Final Situation	This option gives a screen of information as shown in Figure 8-35. The results are saved in a `jobname.crg` type file.
Final a-N Graph	This gives the autoscaled final graphics plot (Figure 8-34).
Fatigue Life Interpolation	This option gives the user the opportunity to interpolate the crack size and elapsed life data for different values of either the initial crack size or the final crack size or both. The defaults offered in the following example (Figure 8-37) are the original initial and final crack sizes. It is possible to interpolate life beyond the crack size limits of the initial calculation. However, such interpolation is only approximate as the crack size-life relationship is both nonlinear and bounded by certain physical constraints. **Note:** The initial crack size used when the user enters a value of zero is in fact the short crack parameter, lo. The value of this parameter for the current job is the default value shown in the initial crack size field. See Cycle-by-Cycle Approach, 600.
Start Another Job	As the title suggests, this loops the user back to screen 1 of this program and allows him to go through the whole process again.
Edit Analysis Parameters	This will bring up a submenu (Figure 8-38) which will allow for editing parameters associated with the initial job setup each of which has been discussed in detail previously.
Preferences	This option allows for turning material checking on or off, setting a failure criterion, or doing an auto-overwrite. The material parameters generally have certain valid ranges which the program checks for. If the user enters a value out of the valid range he will be told and requested to enter a valid number. He may see what the valid range is for a given parameter by using the Help on Question option from the help pull-down menu on the top bar when the appropriate field is active. If the user does not want to be bothered with this, he must set the material checking preference off (Figure 8-39). The failure criterion option determines whether residual k is included in the calculation of k to exceed klc. The auto-overwrite option sets a flag for this job. If the answer is Yes, then all output files are automatically deleted without conformation. If No, the program will request confirmation prior to deleting any output files.
Recalculate	After editing parameters, changing materials, etc., the user will need to select this option to see the effect of any of these changes. This reruns the analysis.
Exit	Allows for exiting the program and puts the user back either into the MSC Fatigue forms or back to the system prompt from which PCRACK was invoked.

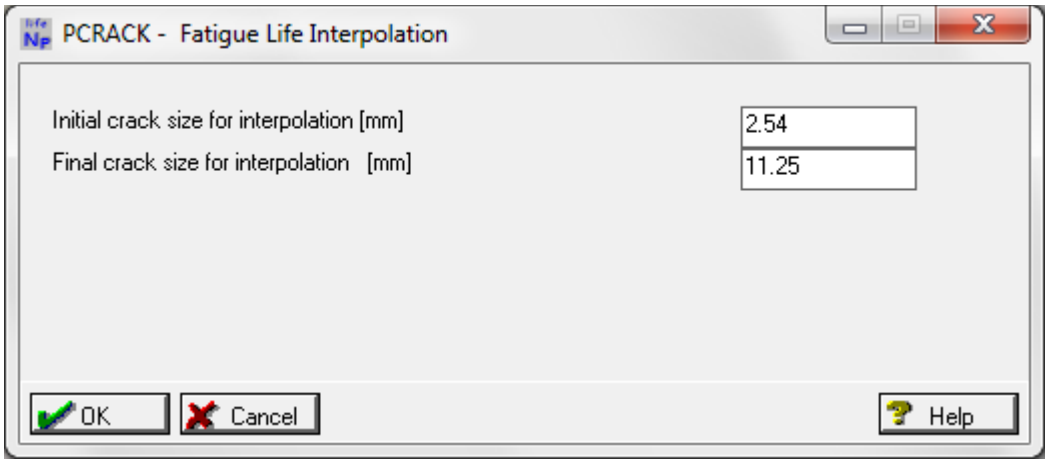


Figure 8-37 The Interpolate Fatigue Life Screen

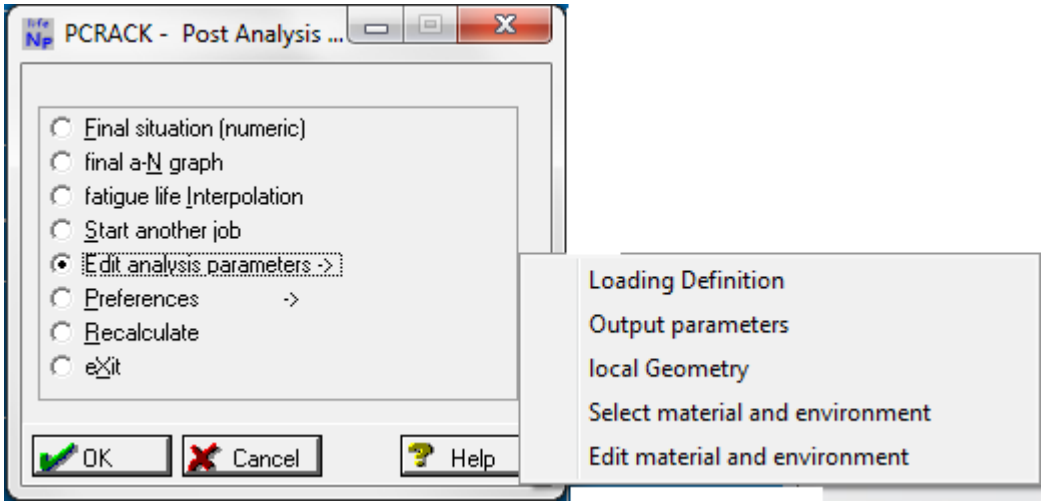


Figure 8-38 The Edit Parameters Submenu

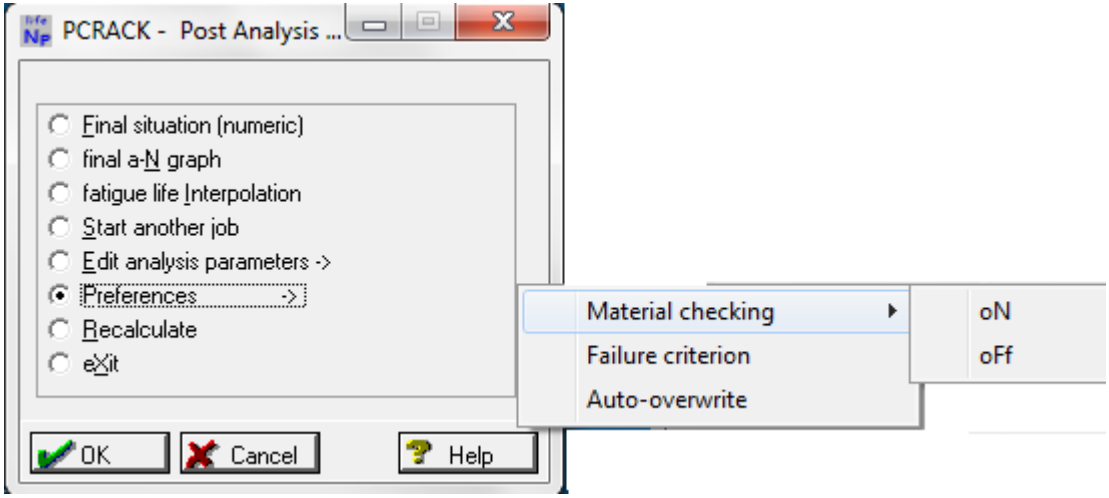


Figure 8-39 The Material Checking Preference

Reviewing Results (PCPOST)

When performing a crack growth analysis, this option executes the external MSC Fatigue module PCPOST (a Tabular and Graphical Postprocessor for the crack growth analysis) which allows various listing options and final graphical displays for the specified results file. PCPOST may also be run from the operating system prompt by typing the symbol **pcpost**.

PCPOST is designed specifically for postprocessing the results of cycle by cycle fatigue crack growth analysis carried out by MSC Fatigue. The crack growth analysis produces a results file which contains a "snapshot" of the situation at equally spaced intervals of cycles throughout the predicted life. This file may be large due to the possibly lengthy operation of the crack growth modeling. The results file preserves the full description of the crack growth analysis run in the form of 14 different parameters recorded at each interval. The program operates interactively in motif or mask mode and menu selections are made according to the rules defined in Module Operations (Ch. 18). The PCPOST module has four functions:

1. To list the data on screen and the session file for subsequent report preparation.
2. To display the two main crack growth analysis graphs and produce hard copies of the same for inclusion in a report.
3. To interpolate the crack size-cycles data for sensitivity studies, inspection strategy or defect assessment purposes.
4. To compress unnecessarily large crack growth results files.

The 14 parameters saved at each "snapshot" are indicative of the complexing effects being accounted for in the crack growth analysis. These parameters are listed below:

Life in user units	Life in cycles
Crack size	Growth rate
Stress range	Apparent delta K
Effective delta K	Current overload K
K at crack closure	Residual K
K from static fracture	Effective stress ratio
Notch correction factor	Growth corrosion enhancement

A full description of the significance and mathematical derivation of these parameters is given in Fatigue Theory (Ch. 16).

PCPOST Module Operation

When first invoked PCPOST appears displaying two forms.

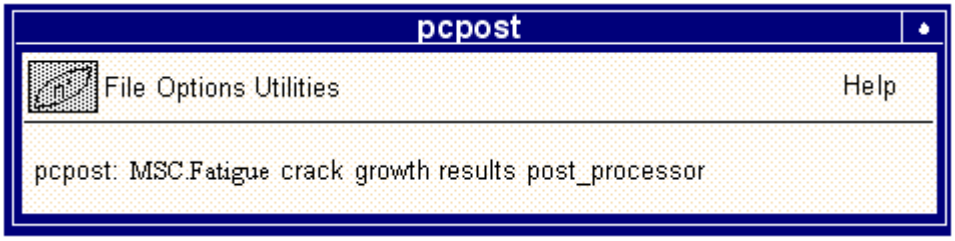


Figure 8-40 PCPOST Utility Form

The top, small form is a generic form and allows for general program control. Once initiated, PCPOST will present the user with a number of menus and questions. The second form, however, will ask for a valid jobname as shown below.

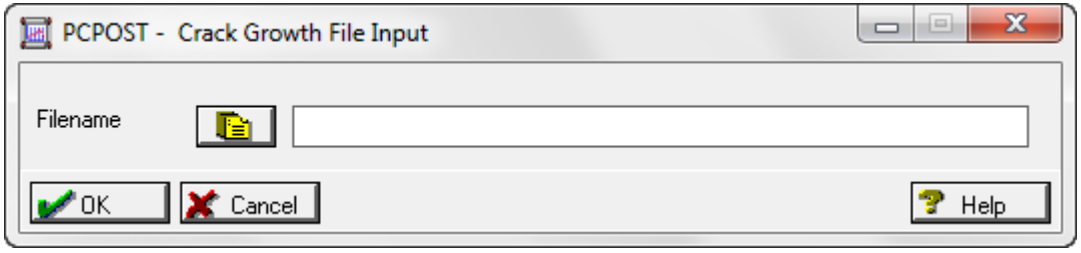


Figure 8-41 PCPOST Jobname Prompt

When PCPOST is spawned from the MSC Fatigue Results form the current jobname is automatically passed and appears as the default as in Figure 8-42.

Enter the name of a MSC Fatigue `jobname.crg` results file containing crack growth results. PCPOST reads these files which are produced by PCRACK. A jobname must be entered. If multiple jobnames exist the List button may be used to obtain a listing of available results files.

Once a file name has been supplied, and then confirmed by clicking OK, the main PCPOST form will appear. The main menu will appear as shown in Figure 8-42.

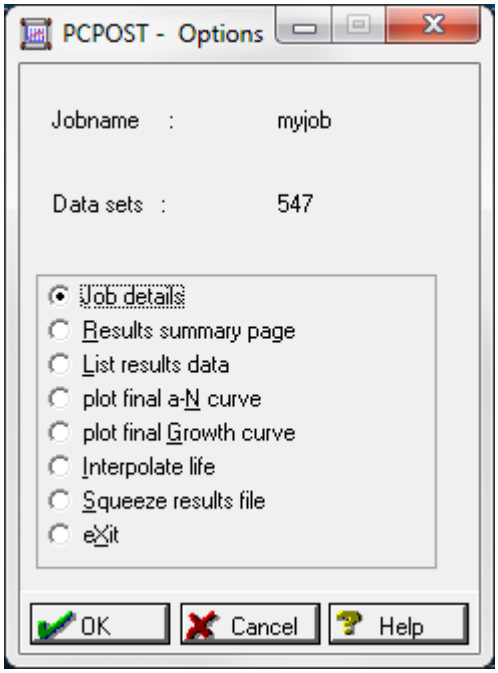


Figure 8-42 The PCPOST Main Menu Form

The menu options are described below.

Job Details

This option provides information on the file which has been highlighted in the file list. The information detailed is:

1. Jobname
2. Number of data sets
3. Job description

Results Summary Page

This option presents the summary of the final situation when the crack growth analysis was halted, (i.e., when the failure criterion was met). An example of this display is shown in Figure 8-43.

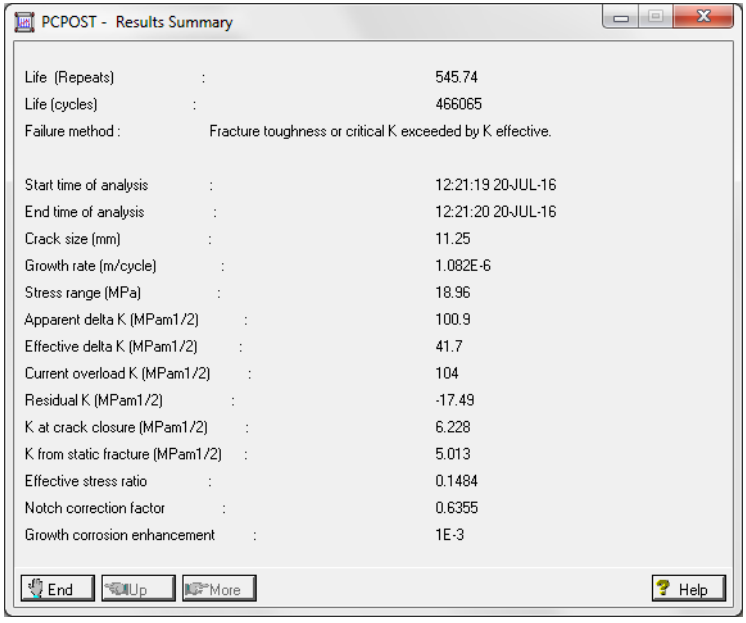


Figure 8-43 .The PCPOST Results Summary Screen

List Results Data

This option leads to a list of the columns of information saved in the results file. The user is presented with a tag submenu which allow the selection of up to 6 columns of data to be listed to the screen and the session file. The tagging submenu and selection list are shown in Figure 8-44.

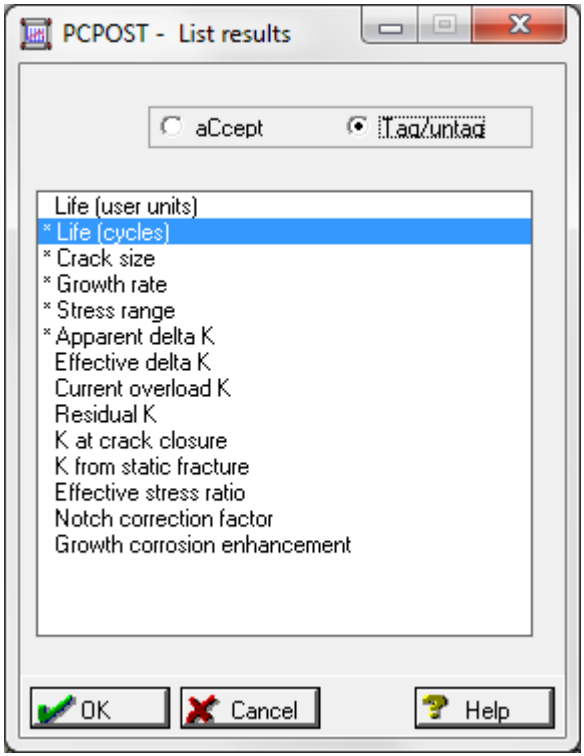


Figure 8-44 Tagging Submenu and Selection List

The * signifies that these parameters have been selected (or tagged). The first 6 items are automatically tagged when this option is selected. They may be untagged or tagged by highlighting the relevant item and pressing the Tag/untag toggle. The table is produced by pressing the Accept button.

Plot Final a-N Curve

The final crack length (a) versus number of cycles (N) curve is presented on the screen together with the additional parameters which indicate the status at the point when the analysis terminated. This curve may be saved in the `.plt` file by invoking the hardcopy option under the file menu of this window. An example of this final A-N screen display is shown in Figure 8-45.

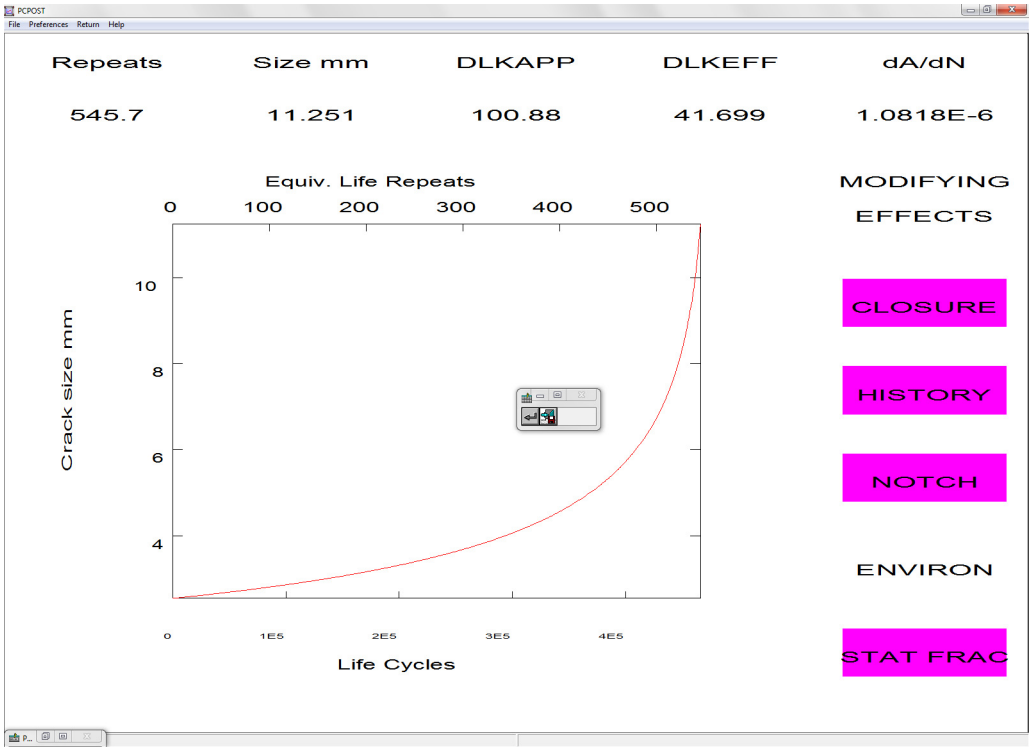


Figure 8-45 A PCPOST Graphical Screen

The pull-down menus at the top of this form provide for some graphical preferences. These as well as the commands that may be entered in the Command line are detailed in Module Operations (Ch. 18) for the Motif interface.

Plot Final Growth Curve

This is identical to the previous option except that by selecting this option, the final crack growth rate (da/dN) versus driving force (ΔK) curve is presented on the screen together with the additional parameters which indicate the status at the point when the analysis terminated.

This curve may be saved in the `.plt` file also by invoking the hardcopy option under the File menu of this window.

Interpolate Life

This option allows for the specification of an initial and final crack length, and the life for these two parameters will be reported. Multiple initial or final crack sizes may also be entered, separated by commas, spaces, or other separators, and the resulting lives listed. The default crack sizes will be the initial and final crack sizes used in the original crack growth analysis. The interpolate form is shown in Figure 8-46.

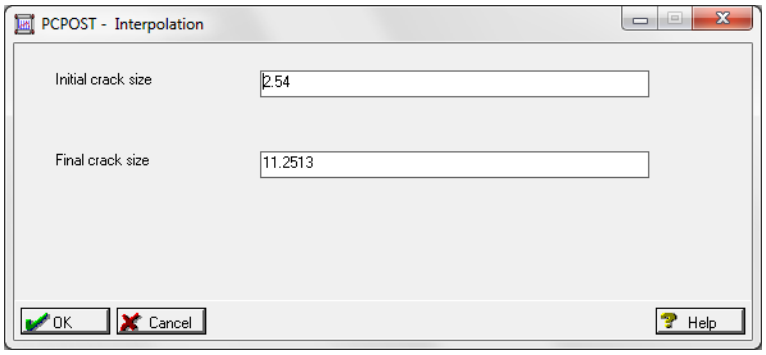


Figure 8-46 The Interpolate Screen

It is possible to interpolate life beyond the crack size limits of the initial calculation. However, such interpolation is only approximate as the crack size-life relationship is both nonlinear and bounded by certain physical constraints.

Squeeze Results File

The results file from a crack growth analysis may be unnecessarily large due to the fact that the update frequency for this file is set before the length of the analysis or component life is known. This option allows the decimation or reduction of this file for archiving purposes or simply to save space.

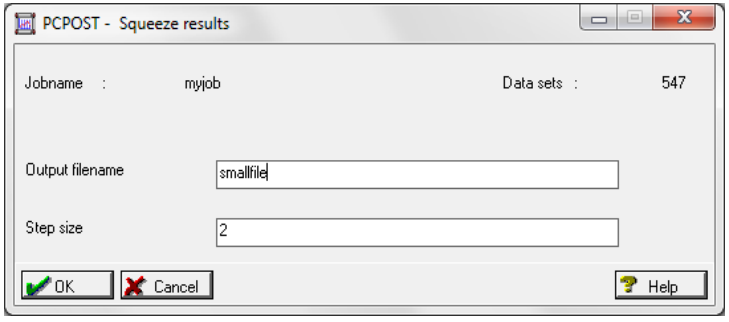


Figure 8-47 The Squeeze Results Screen

A new output file name will be requested. (This cannot be the same as the input file for security reasons.) The number of values to select is then determined by asking for a step size.

If the answer here was 10, every tenth set of values (i.e., every 10th entry) in the `jobname.crg` file would be saved in the new file and the file would be about 10% of the size of the original file.

eXit

This option will terminate the PCPOST program.

Batch Operations

PKSOL and PCRACK can be executed in batch mode similar to other MSC Fatigue modules, although it is not recommended and certainly not easier or more intuitive than interactive mode. However, it may be useful in some cases to set up a batch job execution if the run time may be very long and it is not desirable to resetup the job through the MSC Fatigue menus within MSC Fatigue or Patran.

Most of the batch keywords are presented here. However, not all of them are described. To find the appropriate batch keyword for the operation(s) that is necessary, enter PKSOL or PCRACK and use the Help on Question from the Help pull-down menu with the area or field of interest active. At the end of each help page (that is displayed), is the keyword for that option. In the graphics mode of PKSOL, the keywords for each of the geometry dimensions can be determined also by using the Help on Window option from the Help pull-down menu when entering dimensions.

PKSOL Batch Operation

Related keywords for PKSOL are:

/UNIts=	This is the Units option (1,2,3, or 4 for millimeters, meters, inches, or milli-inches, respectively). (/UNI=1)
/CALulation=	This is the calculation option as in Figure 8-4. (/CAL=4)
/OUTput=	This is the output file name. (/OUT=keyhole)
/GEOmetric def=	This is the main geometric definition as in Figure 8-5. (/GEO=1)
/SELection=	This is the sub-geometric definition of the crack geometry as specified in0 Figure 8-6 through Figure 8-12. (/SEL=8)
/OPTion=	The option can be Define, Calculate, or Exit (/OPT=df, /OPT=cl, /OPT=ex)
/B=	This is an example dimension for one of the geometries. (/B=1)
/W=	This is another dimension from one of the geometries (width). (/W=3)
/A=	This is the crack size, or sizes. (/A=1)
/K=	This is the stress intensity, or intensities. (/K=50)
/STR=	This is the stress, or stresses. (/STR=50)
/C1= - /C5=	These are the constants to equation 5-2. (/C1=1.0)

/T=	This is the thickness of the specimen for use with equation 5-2. (T=1)
/OV=	Overwrite flag if you wish to overwrite a file name. (/OV=Y)

Example:

```
pksol/uni=1/cal=4/out=keyhole/opt=df/geo=1/sel=8/b=1/w=3
/opt=cl/opt=ex/ov=y
```

This example will run PKSOL in batch and will create a compact tension specimen with dimensions B=1mm and W=3 mm. The resulting file will be called `keyhole.ksn` and if one already exists by that name, it will be overwritten. If there are problems running PKSOL in batch, check the `batlog.lst` file for clues as to why it may not be working.

PCRACK Batch Operation

Related keywords for PCRACK are:

/MSCOPTion=	The PCRACK main menu option (P, A).
/ANALysis=	Do a full analysis after preprocessing (Y, N).
/FACTor=	Time History Scale Factor of loading.
/TIME=	Time history
/OFFset=	Time history offset. Not to be confused with the other /OFF below.
/LOAD=	Load magnitude
/LCASE=	Load case number
/EDILOD=	Edit load case information (Y, N). (/EDILOD=N)
/INPut=	This is the name of the `jobname.tcy` file. (/INP=myjob)
/SCAling=	This is the scale factor. (/SCA=1)
/OFFset=	This is the stress offset in MPa. (/OFF=100)
/PLAne=	This is the plane stress correction flag (Y/N). (/PLA=N)
/HIStory=	This is the model history effects flag. (/HIS=Y)
/OUTput=	This is the output file name. (/OUT=myjob)
/OV=	Overwrite flag if you wish to overwrite a file name. (/OV=Y)
/INCrement=	This is the increment of cycles at which you wish to output parameters to the output file `jobname.crg`. (/INC=10)
/UPDate=	This determines how often the graphics screen is updated. This is important if you wish to look at the final situation plots graphically. (/UPD=10)
/MAXimum=	This is the target life in user units. (/MAX=50)
/KSOlution=	This is the name of the K solution file. (/KSO=keyhole)
/DEPth of notch=	This is the Notch Depth in the appropriate units as defined by the K solution. (/DEP=2.3)

/RADius of notch=	This is the Notch Root Radius in the same units. (RAD=1)
/SHArp radius=	This is the Sharp Crack Root Radius in the same units. (/SHA=1E-4)
/AF=	This is the Final Crack Size in the same units. (/AF=2)
/AI=	This is the Initial Crack Size in the same units. (/AI=0)
/MATerial=	This is the material to be used. (/MAT=MANTEN)
/ENVironment=	This is the environment to be used. (/ENV=AIR)
/YIEld=	This is the Yield Strength in MPa. (/YIE=324)
/UTS=	This is the Ultimate Strength in MPa. (/UTS=552)
/E=	This is the Young's Modulus in MPa. (/E=2.034E5)
/KCR=	This is the Fracture Toughness in MPa m1/2. (/KCR=121)
/FL=	This is the Unnotched Fatigue Limit in MPa. (/FL=226)
/C=	This is the Paris Law Coefficient in m/cycle. (/C=3E-12)
/M=	This is the Paris Law Exponent. (/M=3.43)
/DKR0=	This is the Delta K threshold at R=0 in MPa m1/2. (/DKR0=8)
/DKR1=	This is the Delta K threshold at RŠ0 in MPa m1/2. (/DKR1=2)
/RCR=	This is the Stress ratio at threshold knee. (/RCR=0.75)

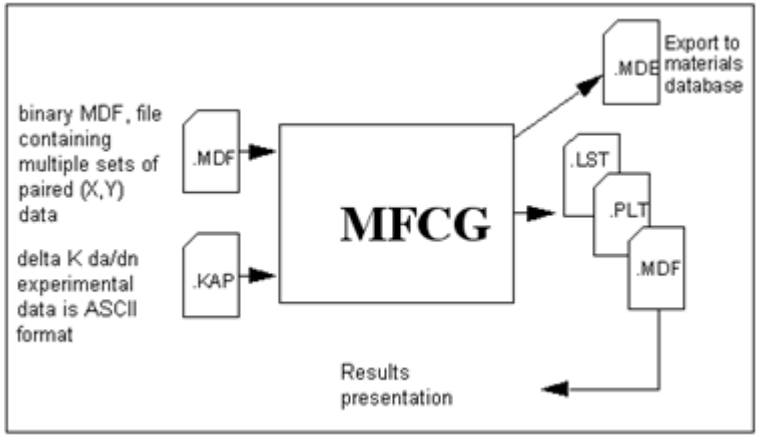
Example:

```
pcrack
/inp=keycg/out=keycg/ov=y/ksol=keyhole/mat=manten/env=air
```

This example will run PCRACK in batch and will use the file `keycg.tcy` as the input file and create a file called `keycg.crg` with overwrite on. The material used is MANTEN and the environment is AIR. The K solution file name is `keyhole.ksn`. All other parameters are takes as default from the initial job setup. If there are problems running PCRACK in batch, check the `batlog.lst` file for clues as to why it may not be working.

Crack Growth Data Analysis (MFCG)

The MFCG module calculates the values of the Paris Law coefficient, C, and exponent, m, from actual fatigue crack growth test data obtained under constant amplitude loading conditions. The data is corrected for the influence of crack closure and static fracture modes so that the whole data set may be used to obtain C and m by least squares linear regression.



The input may be from either:

- An ASCII file of paired data points of the form <delta K> <da/dN>, previously prepared using an editor. The default file type is .kap.
- A binary, .mdf, file containing multiple sets of paired (X,Y) data in the form X=<delta K>, Y=<da/dN>. An R value must be entered for each data set used from the file.

The MFCG module calculates the Paris Law coefficient, C, and exponent, m, in the expression:

$$\frac{da}{dN} = C(\Delta K)^m \tag{8-3}$$

The crack growth data, when plotted in log-log format, usually appears as a sigmoidal shape (Figure 8-48). The upward swing is due to the increasing contribution of static fracture modes as K_{max} approaches the fracture toughness, K_{IC}, leading to rapid unstable fracture. The downward swing is due to the increasing influence of crack closure leading to a threshold ΔK for zero growth rate. It is often difficult to determine C and m from a very limited linear central portion of the data, which may often be really only a single inflection point.

By correcting the "apparent" ΔK values, calculated from measured crack length, load and geometry, to "effective" ΔK values to account for these two influences, it is possible to straighten out the S shape and use the whole data set to determine C and m by least squares regression fitting of a straight line (Figure 8-49).

The mathematical formulae for the corrections are derived as follows:

$$\Delta K_{app} = K_{max} - K_{min} \tag{8-4}$$

$$\Delta K_{eff} = K^{eff}_{max} - K^{eff}_{min} \tag{8-5}$$

$$K_{eff} = K_{max} + K_{max}/(K_{1c} - K_{cl})^{eff} \tag{8-6}$$

$$K_{eff} = \text{greater of } (K_{min} K_{cl}) \tag{8-7}$$

where K_{cl}, the crack closure level, is obtained from:

$$K_{cl} = \Delta K_{th}/(1-R) \tag{8-8}$$

where R is the stress ratio (K_{min}/K_{max}) used in the test and ΔKth is the threshold ΔK at that R value.

This leads to the correct version of the Paris Law:

$$\frac{da}{dN} = C(\Delta Keff)^m \tag{8-9}$$

or

$$log\left(\frac{da}{dN}\right) = m log(\Delta Keff) + log C \tag{8-10}$$

which is a straight line.

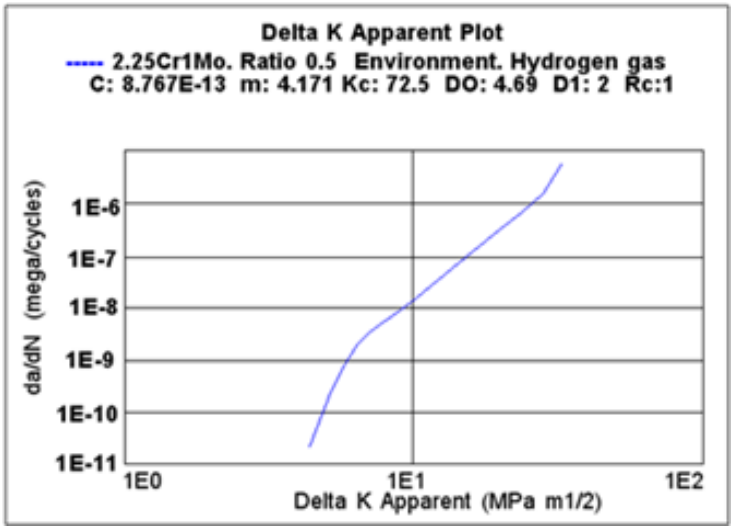


Figure 8-48 "Apparent" ΔK-da/dN Data

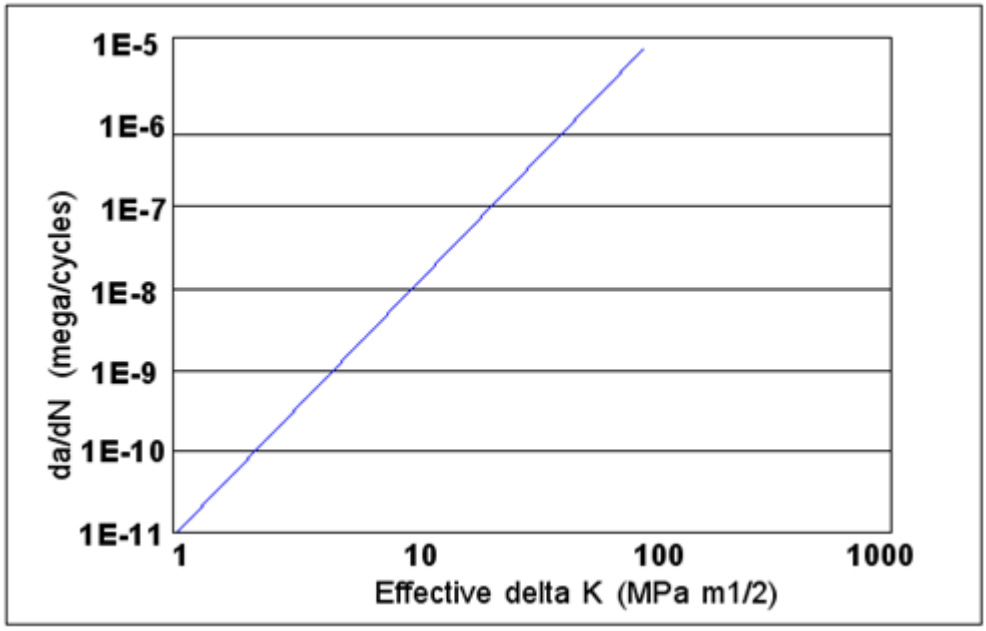


Figure 8-49 "Effective" ΔK-da/dN Data

To achieve the above calculations MFCG requires the user to enter the following data.

- The critical K or KIC in MPa m1/2
- The test R value
- The delta K threshold at R=0 in MPa m1/2
- The delta K threshold as R->1 in MPa m1/2
- The R value at threshold knee

The critical K or K_{IC} is the material fracture toughness; the test R value is the stress ratio (minimum/maximum load) at which the laboratory test was carried out; the ΔK threshold at R=0, the ΔK threshold as R tends to unity and the R value at the threshold knee are the 3 parameters which describe the threshold response of a material as a function of stress ratio as per Figure 8-50.

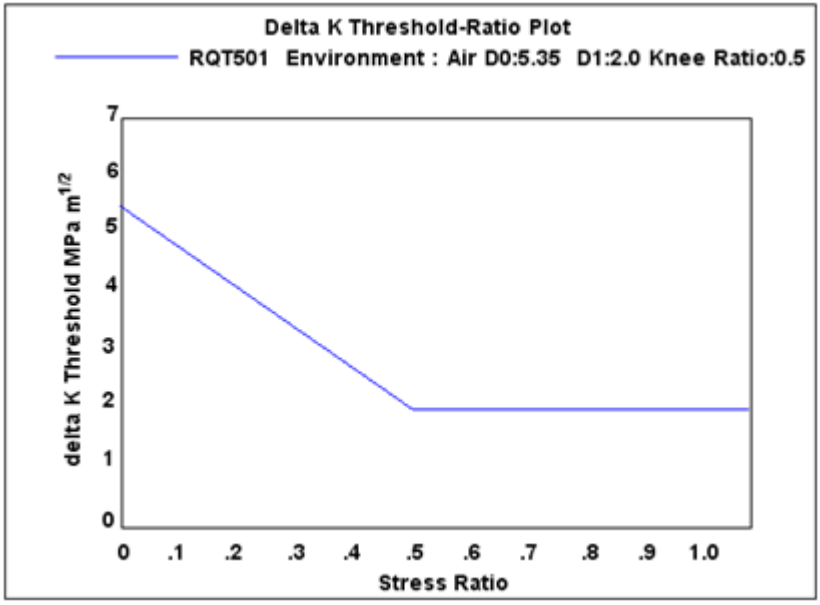


Figure 8-50 Description of ΔK_{th} Response as a Function of Stress Ratio

MFCG is a module which takes an ASCII file of apparent ΔK , da/dN pairs, takes input values of K_{IC} and ΔK_{th} and carries out the corrections and linear regression. A post analysis menu allows presentation of the results and data either numerically or graphically, on screen or written to files. The filtered parameters may be exported to a materials database by MFCG for later use in fatigue crack growth calculations using the PCRACK module. It should be noted that in some circumstances, MFCG can be used to estimate the material values for fracture toughness and threshold from crack growth data by repeated calculations using the module to optimize the best fit to the test data, particularly for a family of datasets from tests conducted at different R values.

MFCG Module Operation

The MFCG module can be operated in one of the following 3 ways:

- From the MSC Fatigue Tools menu system
- In stand-alone mode by typing **mfcg** at the system prompt.
- By incorporating the MFCG commands in a batch operation.

Once running in interactive mode MFCG will display this screen.

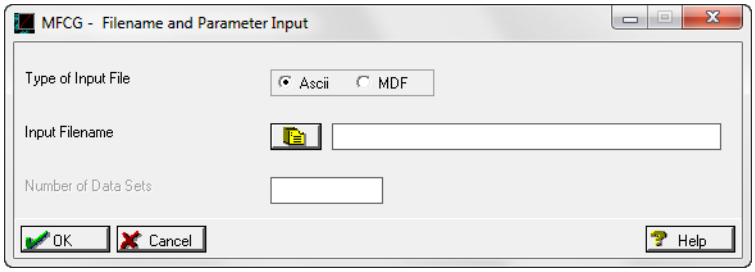


Figure 8-51 The MFCG File Input and Process Definition Screens

MFCG can process da/dN-ΔK data either as ASCII or binary XY files (.mfd files). This question asks the user to select which type of input file by choosing one of the toggle options.

Field	Description
Input File Name	MFCG will prompt the user to enter the name of the data file to analyze. This input file must be either an ASCII file of paired data points in the form of ΔK (, or space) da/dN, previously prepared either by using a standard editor or as a result of data reduction from computer controlled testing (the default extension is .kap) or a two parameter file of ΔK, da/dN (default extension .mdf).
Output File Name	The next prompt requests the name of an output file to save the results of the **MFCG** analysis. This will have a .mdf extension by default and the generic name of the input file is offered as a default. Input .kap files are output as .mdf files. The results file is used subsequently for graphical presentation though a printout is also possible from the post analysis menu. If the output file name specified already exists, then the user will be asked to confirm that it is to be overwritten.
Number of Data Sets (MDF files only)	The input .mdf file may contain several data sets. This question is asking for the number of data sets to be included in analysis. The answer should be a positive number not exceeding the number of data sets defined in the .mdf file. An R value will later need to be specified for each data set included in the analysis. If there are 5 data sets defined in the file and 3 is entered into this field then only the data from the first 3 sets are included in the analysis.

Field	Description
Test R value	R can be any value between 0 and 1 when the maximum load is positive. R is zero for "zero to tension" loading and -1 is for "fully reversed tension/compression loading". For fatigue crack growth tests with constant load amplitude and constant R value, R is given by: R=minimum load or stress or K / maximum load or stress or K
Critical K or K_{1C}	To correct for the influence of static fracture modes (the upward bend of the log-log plot) a value of critical K at fracture, or the fracture toughness value K_{1C} is required for the material under test.
ΔK Threshold at R=0, and R nearing 1	The threshold values are used at low da/dN to determine the crack closure influence and to correct the apparent ΔK. The units are MPa $m^{1/2}$. ΔK threshold is the positive value at which fatigue cracks will not propagate due to the presence of crack closure reducing the effective ΔK to 0. This condition has been experimentally determined at a cut-off rate of 1e-11 m/cycle and measured values vary with R as; ΔK threshold = D0 - (D0-D1) * R/Rc where D0 is the threshold at R=0, D1 is the threshold as R nears 1, and R_c is the R value at the knee point. If the calculated ΔK threshold is less than D1 then D1 is automatically used.
R value at threshold knee	If the test R value is greater than R_c, MFCG will automatically use the D1 value.

Post Analysis

When the test data has been defined, clicking **OK** will process the input file and then display the post analysis screen as in below.

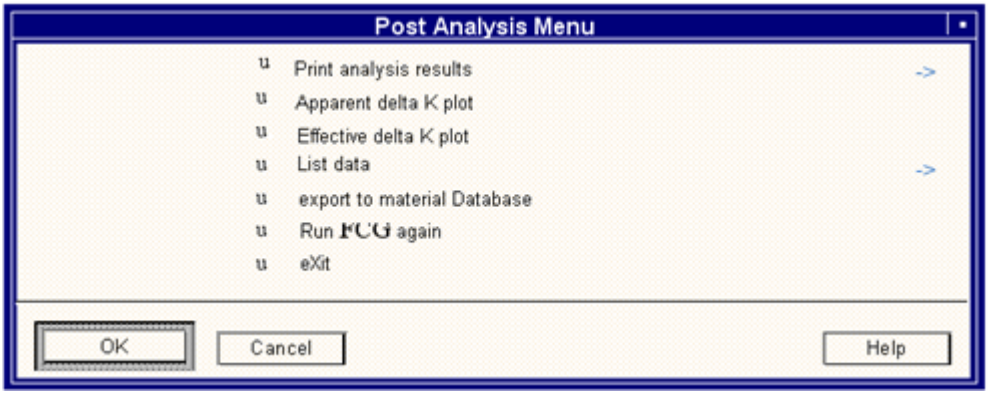


Figure 8-52 The Postprocessing Menu

Field	Description
Print Analysis Results	Prints the results as statistically fitted values of m and C, the standard deviation and the correlation coefficient. The page of results are numerical (not graphical plots) and can be seen on screen, or saved as a text file to disk.
Apparent ΔK Plot	This option plots the original unprocessed input file. It may be plotted as data points only, or a `best fit` S curve can be superimposed on the data points.
Effective ΔK plot	This option plots the processed input file (the output file). It may be plotted as data points only, or a `best fit` fitted straight line plot can be superimposed on the data points.
List Data	Lists the apparent ΔK, effective ΔK, and da/dN to either screen or disk file (where they may be imported into most commercial word processors). The number of sets of data values can be indefinitely large depending upon the number of points in the input data file, and there may be many pages of data.
Export to Material Database	This option allows the materials database to be updated to include the results from this analysis. The choice of databases are from Central, Local, or User.
Run MFCG Again	Runs **MFCG** again and returns to screen 1 for another set of experimental data.

The MFCG Graphical Display System

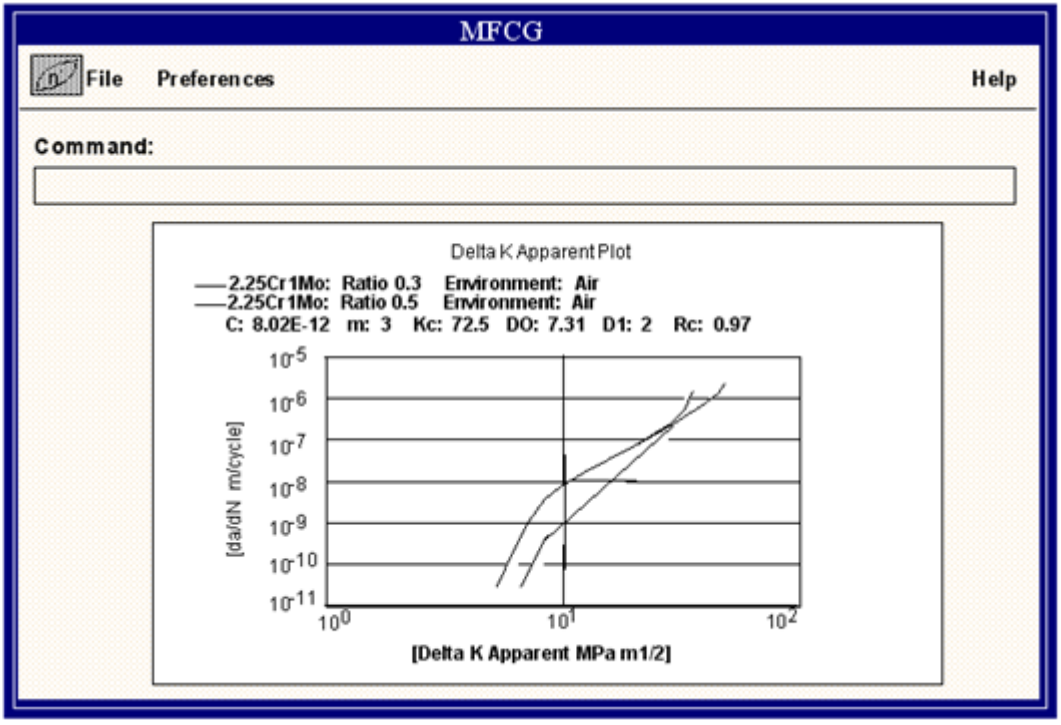


Figure 8-53 Graphical Display with Main Menu

The graphical interface is mostly explained elsewhere in this guide and is similar if not identical to other graphical displays in MSC Fatigue. However, some options are tailored to MFCG. Those options are explained below.

Command	Menu option	Meaning
LI	Line On	Display the statistically fitted regression line to either the apparent or effective ΔK plots.
NL	Line off	Remove the statistically fitted line from either the apparent or effective ΔK plots.
Return		Return to the post-processing menu
Exit		Exit to the operating or menu system

The options pop-up menu contains no unique option; it presents the more commonly used menu options in a very convenient form.

Batch Operation

A list of MFCG's batch keywords:

Example:

```
mfcg /inp=test/kic=100/r=0.1/d0=5.35/d1=2/ov=yes/opt=li/opt=hc
/men=a/pltnam=test1/ptit=test 1
```

This will cause MFCG to read in a file called `test.kap` and, using a K_{IC} of 100 MPa m1/2, D0=5.35, D1=2 and the default threshold parameters (taken from BSI document PD6493, 1988 version) for a test stress ratio of 0.1, write a `ΔK` effective plot to a plot called `test1.plt` and display C, m and statistics to the notebook. By default the `.mdf` output file will be `test.mdf` (because no /OUT was specified). Examples of graphical output from MFCG are given in Figure 8-48 and Figure 8-49.

NASA/FLAGRO

An interface to NASA/FLAGRO has been created within the MSC Fatigue environment. This interface works with NASA/FLAGRO version 2.03 as delivered by COSMIC. The PCL (PATRAN Command Language) within the MSC Fatigue forms allow you to easily set up a NASA/FLAGRO safe life analysis by specifying the crack geometry and parameters, the materials information and the loading schedule. The PCL will write out a NASA/FLAGRO batch file and then execute NASA/FLAGRO interactively in the invoking window using the generated batch file.

Currently, only safe life analysis is supported. Materials defined in the FLAGRO standard or user-defined files may be accessed. Load schedules defined in the FLAGRO schedule file may also be accessed. In certain instances, defined schedules and loading blocks may be saved to this file also.

Most error checking is done by the NASA/FLAGRO program itself. Therefore care should be taken to make sure proper geometry and other input definitions are defined correctly before creating the batch file and

executing FLAGRO. If the program fails due to improper input data, then it will be necessary to run FLAGRO interactively to find out where the error occurred.

There are three major inputs to the NASA/FLAGRO program, as there are to any fatigue or fracture program. These are geometric information (crack geometry), materials information and loading information. A familiarity with the use of MSC Fatigue is suggested in that the forms for job set up follow closely to those of other analysis types with in MSC Fatigue.

The NASA/FLAGRO executables themselves can be retrieved form the MSC Fatigue delivery media. It is suggested that you use these files. No guarantee is given if other files are used that may have been modified (this is public domain software). The following files are unloaded:

```
<install_dir>/bin/exe/flagro - NASA/FLAGRO executables
<install_dir>/bin/exe/nasfla
<install_dir>/bin/exe/nasbem
<install_dir>/bin/exe/nasmat
<install_dir>/flagro_files/NASMFM - NASA/FLAGRO material files
<install_dir>/flagro_files/NASMFC
<install_dir>/flagro_files/USRMFM
<install_dir>/flagro_files/USRMFC
<install_dir>/flagro_files/NASGLM
<install_dir>/flagro_files/NASGLC
<install_dir>/flagro_files/SCHDLE - NASA/FLAGRO block loading file
<install_dir>/flagro_files/SCHCOM
```

Module Operation

To load the FLAGRO interface into the MSC Fatigue forms within MSC Fatigue or Patran, type the following command in the command line:

fatigue_main_form.flagro()

Do this before you open the MSC Fatigue form. You may want to include this line at the end of the `init.pcl` file so that it always appears automatically.

There are three basic inputs just as for the other analysis types, those being Solution Parameters, Material Information and Loading Information. To run FLAGRO you simply supply the crack geometry information in the Solution Parameters form, the materials information in the Materials Info form and the loading schedule information in the Loading Info form. A description of these are given below. Only the Safe Life option of FLAGRO is supported through these PCL forms. Other limitations are pointed out below.

Solution Parameters

Select the crack geometry of interest and fill out the dimensions necessary. An icon showing the appropriate parameters that need defining is given for each crack geometry.

Very little error checking is done. So it is possible to create a batch file that will not run through FLAGRO properly if crack dimensions and NDE flaw sizes are not correct. Use Tables 1 & 2 in the FLAGRO user guide to input proper crack geometries. To debug a run that doesn't seem to be running properly you may, after creating a batch file, type at the system prompt:

nasfla <jobname.cmd >file

Then inspect the resulting "file" for possible problems in your input.

Data tables and polynomial series not supported. About 95% of all crack geometries and inputs are supported. For crack geometries SC02, SC04, and SC06, you must input stress distributions in the spreadsheet or the problem will not run. If a blank cell is detected starting from the top row, then input is assumed to be at an end. The exception to this is leaving the cells all blank for geometry SC04. This will cause it to use a unit pressure load.

Material Information

Select a cell from the one row spreadsheet (only one material is supported presently) and then select a material code from the list box that appears. The next cell will be selected and another list box will appear until you have filled in all the materials information. The spreadsheet reads from the NASMFM file if units are SI or from the NASMFC file if units are English. You may select a User Defined Material File (UDMF) using the button called Select User Database. Again the USRMFM or USRMFC file will be read depending on the units.

These files can be located anywhere but there is a hierarchal order in which they are searched. First they will be read if located in the local working directory, next the user's home directory and finally the <install_dir>/flagro_files directory. In all cases however, the actual file selected will be copied to the local working directory on execution of FLAGRO.

To select the standard database (NASMF*) press the button called Select Standard Database. It again will search for this file in the hierarchal order.

Note:	When reading a USRMF* file, the file must conform to the necessary format. No two letter codes, e.g. [DF], can occupy more than one line; otherwise they will not be read and therefore will not be selectable from the spreadsheet.

If you select a user database, it must exist in a directory other than the one you are working in. To use an existing database in the current directory use a standard database by pressing the Select Standard Database button. All user databases and standard databases must be called NASMFM, NASMFC, USRMFM, or USRMFC.

Loading Information

At a bare minimum, you must define at least one block. There are three spreadsheets on this form that define the schedule (the order and number of time each block is to be processed), the individual blocks themselves, and the load steps for each block. The last two are three-dimensional spread sheets where the third dimension is the block you are defining.

This is a fairly complicated form and widgets appear and disappear depending on what you are selecting at the moment. For instance, when you are entering information in the block case information spread sheet, the load step information spread sheet will disappear to allow entry of block case info. The same is true for the load step info spreadsheet. To enter data into the spread sheets, simply click on one of the cells. A widget will appear to allow data entry. To get both spreadsheets back on the screen, you only need to deselect the selected spread sheet cell. In some cases, you may need to click twice on a cell.

Stress offset information is only used when a named loading block is selected. Offset information is only used when values are entered. If the offset cells in the spreadsheet are left blank then no offset will be applied to the block.

Scale factor information is not necessary. If these cells are left blank, then unity will be used.

For both scale factor and offset information, you may select a node or element from the graphics screen, indicating that you want the stress result from that node or element to be used as the scale factor or offset value. You must have selected a proper results case from the Select Results Case button. If a tensor value is selected you must select a scalar derivation. Von Mises is the default.

The Load Step Information values that appear when a named block is read from file is purely informational. Changing these values in the spread sheet will have no effect. The values from the file will always be used.

To view the Loading Block Information that corresponds to the proper Load Step Information, make sure both spreadsheets are viewed on the same layer. This can be confusing.

The New Name specified will always be taken from the Block Case Information. It can only ever be eight characters in length and will be truncated. If a name is given, it will be saved to the local SCHDLE file. Existing names will not be overwritten or duplicated. Only a Manually Input block can be saved to file.

To delete a row from the Schedule spreadsheet, a cell from that row must be selected. The maximum number of load steps must be entered in the Max. No. of Load Steps databox to change the number of rows in the Load Step Information spreadsheet. When writing a batch file for NASA/FLAGRO execution, if a blank row in this spreadsheet is encountered, or a blank cell in a row, the load steps will be considered terminated at the previous row regardless of whether any more rows contain data.

The stress values entered into the Load Step spreadsheet must correspond to the proper stresses in the crack geometry as indicated by the icon in the Solution Parameters form. That is if S0 is necessary, then the S0 columns must be filled out. If S0 is not necessary and yet the cells are filled out, they will be ignored.

If you select a user database, it must exist in a directory other than the one you are working in. To use an existing database in the current directory, use a standard database by pressing the Select Standard Database button. All user-defined databases and standard databases must be called SCHDLE.

Reference stresses and setting of Kmax < KIscc check are not supported.

Don't forget: it is easy to define a lot of blocks then forget to include them in a schedule.

Job Control

When you wish to run the job, you simply open the Job Control form and press the Apply button with the Action set to Full Analysis. A batch file called `jobname.bat` is created along with the FLAGRO output file `jobname.out`.

If the action is set to Save Job Only, then only the batch file will be created with out actual FLAGRO execution. With the batch file you can then run FLAGRO externally and/or edit the file. You may also delete a job and all file associated with that job with the Delete Job action. The other action on this form is Interactive which will simply open an Xterm and allow you to run FLAGRO externally.

On submission of a job, various other files may be created: `jobname.bat` - this is the FLAGRO batch file, `jobname.cmd` - this is the command file that drives FLAGRO execution from the Patran interface forms, `jobname.sif` - this is a compliance function table that is generated for some of the crack geometries.

Results

No results import is supported.

Crack Growth Analysis Theory

In general engineering, the possibility of crack-like defects being present at the start of service life has to be recognized. This is particularly true when asking oneself the question "what if we can't be sure the structure is not cracked?" (i.e., the safest assumption is that the biggest crack that nondestructive testing cannot reliably find is present). Even if the component is not cracked to start with, the remaining life after initiation may be significant or possibly dominant with respect to the total life. Furthermore, for inspection related activities, the remaining life of a component discovered to be cracked during inspection needs to be determined so that decisions regarding serviceability can be supported by reliable life estimations.

In low-stress (linear elastic) situations, the growth of fatigue cracks governs useful life leading to the "defect tolerance approach" to design which necessarily also implies regular inspection and assessment. In service, fatigue loading is nearly always random and the operating environment is nearly always corrosive or aggressive in nature.

Diagnosis or life estimation can only be achieved in one of two ways: laboratory testing under accurately simulated service conditions of geometry, loading and especially environment, or life estimation from quantitative models of crack growth processes. Laboratory testing is very expensive and, because of the time dependent corrosion influence which prohibits test acceleration, it is also very time consuming. For example,

an offshore platform designed to withstand 10^8 cycles in a 30-year life would require a 30-year test at 0.2 Hz to validate the design. Computer-based estimation methodologies can perform the same validation with equal accuracy in a small fraction of the time at a small fraction of the cost.

Basic Principles of Fracture Mechanics

The branch of engineering science concerned with linear elastic fracture mechanics (LEFM) is by no means a new one. The earliest work in the UK dates back to Inglis (1913) but the major developments took place following the research by Griffith at RAE in 1920, and Irwin in the USA in 1956, and LEFM has since flourished.

The driving force for a crack to extend is not the strain or stress but the stress intensity factor, universally known as K. This is not to be confused with the stress concentration factor, K_t or with the strain hardening exponents k or k'. The stress intensity factor embodies both the stress and the crack size and uniquely describes the crack tip stress field independent of global geometry. The relationship may be viewed as a triangle:

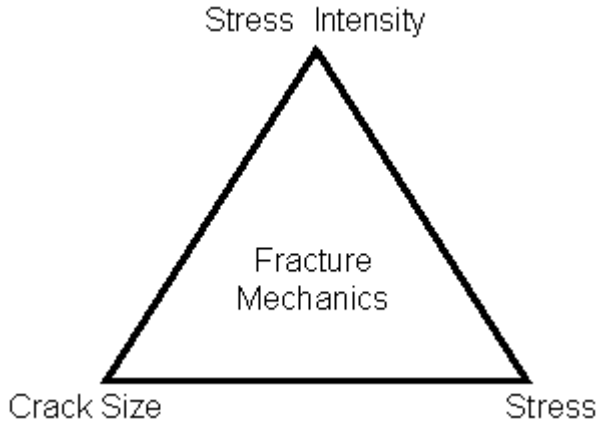


Figure 8-54 The Fracture Mechanics Triangle

The triangle may be solved by knowing two of the three corners to derive the third. For example, the material toughness, K_{IC}, and the NDT crack size limit may be known and hence the working stress may be derived to provide safety against fracture.

The relationship between K, applied stress (S) and crack size (a) is:

$$K = YS\sqrt{(\pi a)} \tag{8-11}$$

The compliance function, Y, describes the geometry of the structure, component or specimen in which the crack exists. It assumes a value 1 for a defect buried in an infinite body and 1.12 for a surface crack in a semi-infinite body. For finite-sized bodies (i.e., real structures), Y may be seen as a geometric correction which takes a general form:

$$Y = C_1 + C_2\left(\frac{a}{T}\right) + C_3\left(\frac{a}{T}\right)^2 + C_4\left(\frac{a}{T}\right)^3 + C_5\left(\frac{a}{T}\right)^4 \tag{8-12}$$

though it may take a much more complex mathematical description in some cases.

In fatigue analysis, again the stress intensity factor provides the driving force for crack growth in the sense that the rate of crack growth, da/dN, is governed by the cyclic range of K, ΔK . The two are related by a crack growth "law", the best known being the Paris Law derived by Paul C. Paris at Lehigh University in 1960:

$$\frac{da}{dN} = C(\Delta K)^m \tag{8-13}$$

In terms of fatigue crack growth and overall life, a rectangular rather than a triangular representation is used:

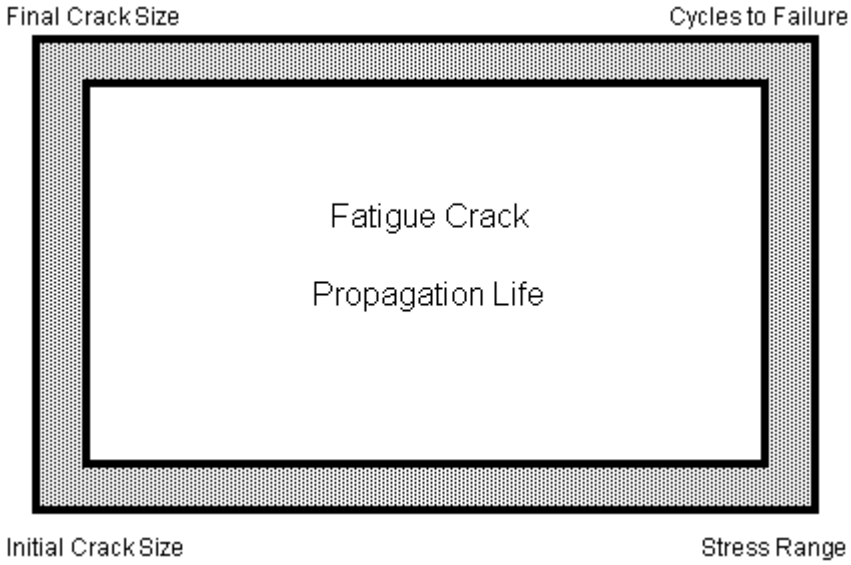


Figure 8-55 The Fatigue Crack Propagation Rectangle

Here we have a relationship between fatigue stress range and fatigue life, in the same way as in an S-N curve approach, but we now also have the initial and final crack sizes and indeed all the crack sizes in between these limits throughout the fatigue crack propagation period. In a similar way to solving the triangle, the fatigue crack propagation rectangle can be solved by declaring the three known corners to derive the unknown fourth.

This relationship may be expressed with the following equations:

$$\frac{da}{dN} = f(\Delta K_{eff}) \tag{8-14}$$

$$N_f = \int_{a_i}^{a_f} \frac{da}{f(\Delta K_{eff})} \tag{8-15}$$

The crack growth prediction model in MSC Fatigue relies on the approximating assumptions that fatigue crack growth can be represented by a simple law:

$$\frac{da}{dN} = C(\Delta K_{eff})^m \tag{8-16}$$

and that the stress intensity factor range can be derived from the solution:

$$\Delta K = Y \Delta S \sqrt{\pi a} \tag{8-17}$$

If the geometric compliance function, Y, is assumed to be constant and independent of crack length, a, then integration of (8-16) is straight forward. This latter assumption is only strictly true for a very few cases such as a surface crack in an infinite plate. However, it is a quick way of assessing the crack growth performance of a structure.

Substituting (8-16) in (8-17) and integrating leads to:

$$N_f = \frac{1}{C(Y\Delta S\sqrt{\pi})^m\left(\frac{m}{2}-1\right)}\left[\frac{1}{a_i^{\left(\frac{m}{2}-1\right)}} - \frac{1}{a_f^{\left(\frac{m}{2}-1\right)}}\right] \tag{8-18}$$

except for the special case where m equals exactly two when:

$$N_f = \frac{ln\left(\frac{a_f}{a_i}\right)}{CY^2\Delta S^2\pi} \tag{8-19}$$

(8-18) represents a relationship between fatigue life (N_f), stress range (ΔS), initial crack size (a_i) and final crack size (a_f). Note that from (8-18), a_f could be infinite and still give a finite fatigue life indicating the relative unimportance of the final crack size. The stress range can be a constant amplitude value or a CA value equivalent to a variable sequence calculated as a weighted average (known as the Weighted Average Stress Range or WASR).

MSC Fatigue Crack Growth Models

The cycle-by-cycle fatigue crack growth prediction module calculates the crack extension, da, in each cycle of a sequence of cycles and adds it to the current crack size. This process proceeds until the failure condition is reached or a preset crack size is achieved. The driving force for crack growth in each cycle is the effective range of stress intensity, ΔK . In each cycle, the apparent or applied ΔK is calculated from the stress range, the crack size and the geometry of the component or structure under consideration. This is then modified to effective ΔK to account for the possible occurrence of:

- crack closure
- history effects
- notch effects
- environmental influence
- static fracture mode contribution

Consequently, the crack propagation life is accurately modeled and this can only be achieved on a cycle-by-cycle basis.

The module may be viewed as part of a greater whole in which a complete case is analyzed using stress-time information, geometric descriptions, and materials response characteristics to produce a prediction that can be subsequently postprocessed and in particular, interpolated.

There are many applications of this approach including: design analysis, pre-prediction of test programs, defect tolerance assessment, inspection related strategy, failure investigation, and decision support.

The basic premise of the fracture mechanics approach to fatigue relies on the "similitude concept" which adopts the simple belief that if the K is the same, the response is the same. In fatigue terms, the similitude applies to the cyclic range of K, ΔK , and the "response" is the rate of fatigue crack growth. It should be clearly understood that even though the growth of macroscopic fatigue cracks is governed by linear elastic fracture mechanics, the microscopic crack extension process by fatigue still necessitates local plasticity. At the tip of a fatigue crack, there are two plastic zones, one for monotonic shear the other for reversed shear, whose sizes depend on the ratio of ($(K/\sigma y)^2$. These crack tip plastic zones are assumed to be negligibly small in relation to crack size, especially for high strength materials, but essential to the fatigue crack growth process.

To undertake a fatigue crack growth life prediction, it is necessary to adopt a cycle-by-cycle approach and to have a good knowledge of the service loading environment, the appropriate properties to describe materials response, a geometric description and the means of combining this information into a coherent model of the fatigue crack growth process.

The Service Environment

The cycle-by-cycle fatigue crack growth modeling requires a sequence of cycles to be extracted from a representative stress time history. Representative here means not only the sequence of stresses but also the durations, cyclic frequencies, and waveforms. These help to define the environmental effects in corrosion fatigue and the corrosive medium itself should also be characterized in terms of composition, temperature, etc.

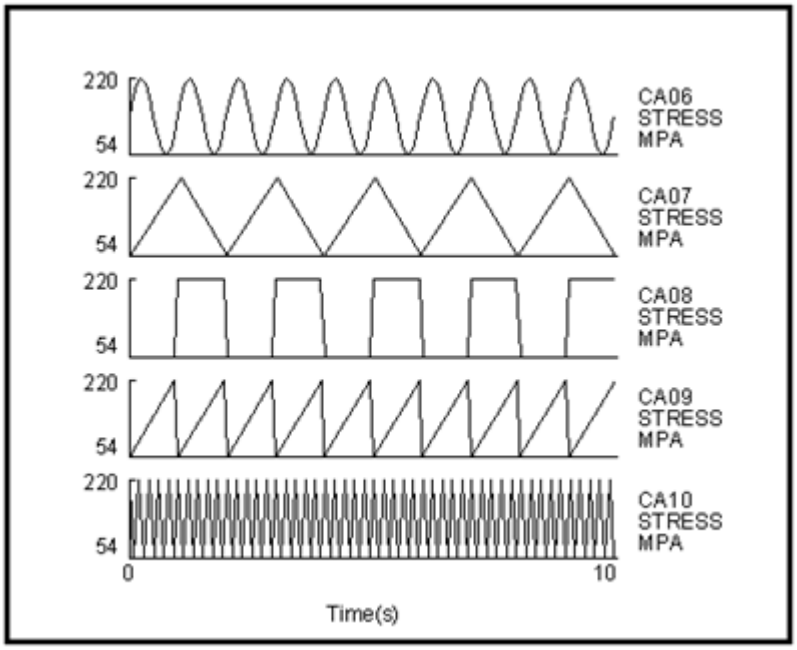


Figure 8-56 Multi-file Display of Stress Histories with Different Frequencies and Waveforms

For example, slower frequencies produce faster crack growth rates, on a da/dN basis, and sine waveforms give faster rates than square waveforms. A multi-file display of stress histories with different frequencies and waveforms is shown in Figure 8-56. A stress time history is computed at a node by applying the local scaling. The stress time history is then cycle counted, including reordering in preparation for processing by the main crack growth analyzer.

Geometric Description

The geometric description of a component or engineering structure is accomplished by means of the appropriate stress intensity solution. This is a mathematical description of the relationship between K, stress (range), and crack size for the geometry in question and many solutions have been derived and documented. The most useful of these are contained in the MSC Fatigue compliance function library module, PKSOL.

This compliance function converts the value of K for each successive loading cycle according to the current crack size. An example of a compliance function is shown in Figure 8-57.

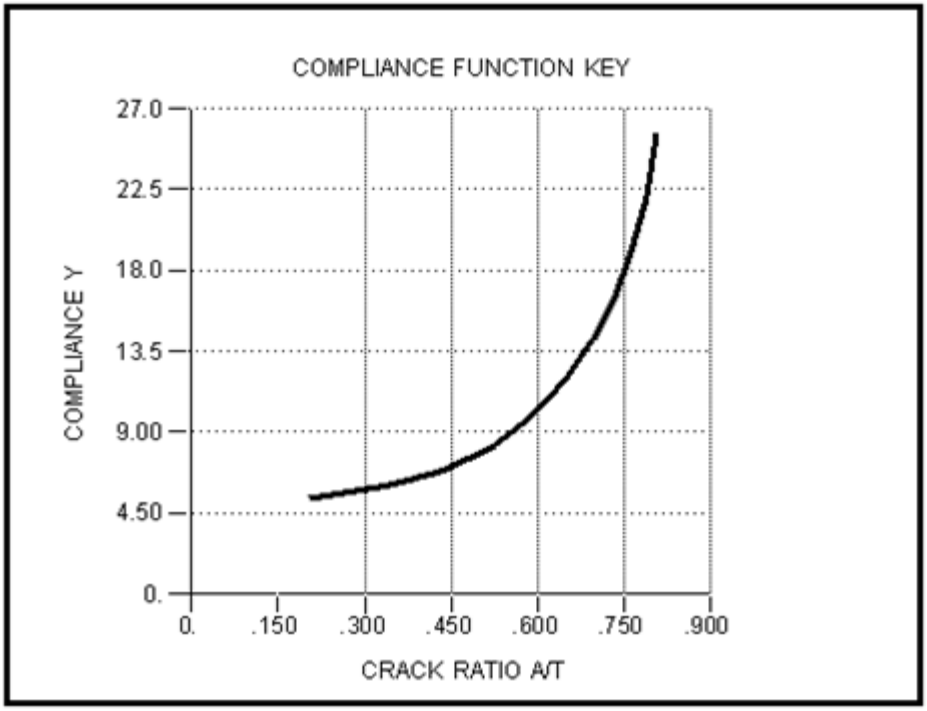


Figure 8-57 An Example of a Compliance Function

Materials Response

The material response is modeled by measuring fatigue crack growth rates as a function of ΔK in constant load amplitude laboratory tests using small scale specimens. These can be carried out under the correct

environmental conditions and then the similitude concept is invoked to apply the data to large scale components and structures. Also, the fatigue crack growth threshold characteristics and the material fracture toughness are measured using the same type of specimens and all these tests are standardized (e.g., in BSI and ASTM Standards).

The parameters which characterize the materials response in terms of fatigue crack growth have been measured for many materials, in standard tests, and reported in the literature. Until now, no materials database has been constructed in a commercial crack growth prediction software package. The materials response parameters required for crack growth prediction using MSC Fatigue are contained within the materials database and may be viewed using the database manager, PFMAT.

Cycle-by-Cycle Approach

The cycle-by-cycle approach has been adopted as the only way in which fatigue crack growth can be accurately modeled. In each fatigue cycle in turn (the correct order must be preserved), the apparent ΔK is calculated from:

$$\Delta K_{app} = Y\Delta\sigma\sqrt{\pi a} \tag{8-20}$$

where $\Delta\sigma$ is the stress range of the cycle, a is the current crack size, and Y is the compliance function obtained from the relevant lookup table generated for the geometry in question by the K solution library module, PKSOL. This value of ΔK_{app} has to be modified if necessary to account for the effects of potentially complicating features until the effective ΔK_{eff}, the driving force that the crack tip actually experiences, is derived. Only then can the crack extension, the amount of crack growth or damage, be calculated simply from the Paris Law (Ref. 10.) as:

$$da = C(\Delta K_{eff})^m \tag{8-21}$$

This is added to the current crack size and the process repeated until the crack growth life is exhausted.

Note that this approach disregards all other crack growth laws developed since the original Paris Law, to account for complexing effects, such as Forman's Law for static fracture modes (Ref. 11.) and the Klesnil and Lukas correction for threshold (Ref. 12.). The Paris Law holds if, and only if, the effective ΔK is used and all possible complicating features are modeled through effective ΔK.

The various mechanistic models leading to mathematical descriptions of complex fatigue crack growth behavior are essentially derived from extensive ECSC sponsored research work carried out at British Steel, Swinden Laboratories, involving more than 25 man-years effort and nearly $2 million. The research work is reported ((Ref. 13.) through (Ref. 22.)) in detail including the results of hundreds of tests designed to validate the predictive models.

For each of the features or mathematical models, a description is given in the following sections:

Initial Crack Size

The initial crack size, to which individual cyclic crack extensions are cumulatively added, may be preset but often is unknown. A minimum initial crack size is modeled in MSC Fatigue to ensure that:

1. the crack size is not too small to be handled by the concepts of fracture mechanics as implemented in the MSC Fatigue crack growth analyzer (i.e., it does not violate the assumption of an isotropic continuum by being smaller than a metallurgical feature such as grain size).
2. the crack size at the start of the propagation phase is equal to the crack size at the end of the initiation phase (i.e., critical location fatigue analysis can be combined with fracture mechanics crack growth analysis with ensured continuity to give total fatigue life).

The minimum crack size is derived from the so called "short crack parameter" which is calculated from the "Kitagawa Diagram". An example of a Kitagawa Diagram, which relates threshold stress and threshold ΔK to fatigue crack size is given in Figure 8-58.

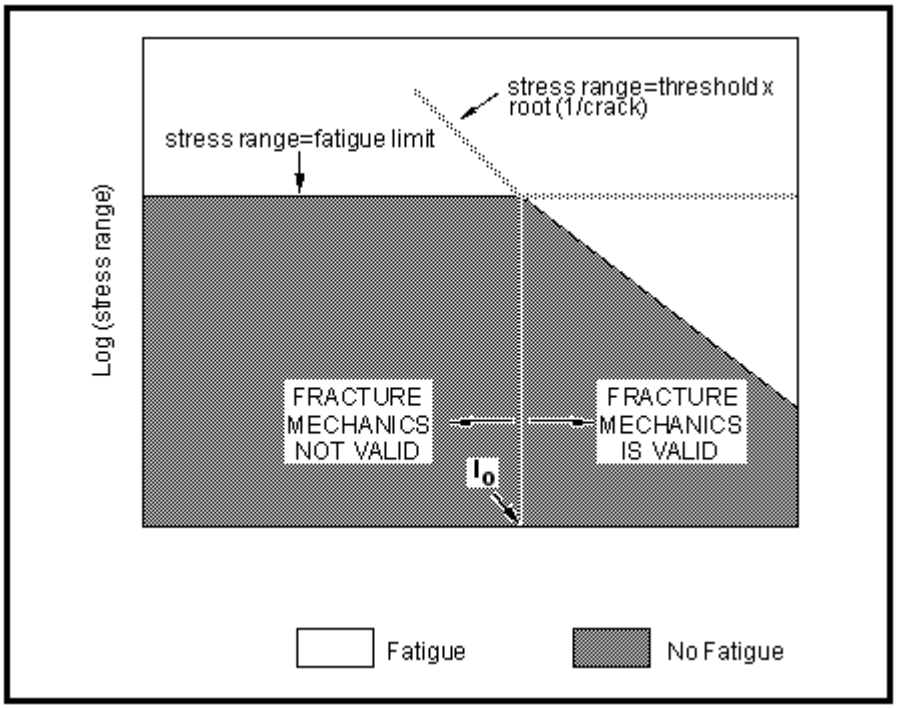


Figure 8-58 An Example of a Kitagawa Diagram for Fatigue

The short crack parameter, is given by:

$$l_o = \frac{1}{\pi}\left(\frac{\Delta K_{th}}{\Delta \sigma_o}\right)^2 \tag{8-22}$$

where ΔK_{th} is the threshold ΔK value, and l_o is taken to be the minimum initial crack size. It is in fact calculated in each cycle using the value of ΔK_{th} for the stress ratio, R, of that cycle for other purposes, but for the minimum initial crack size it is calculated for R = 0.

Crack Closure and Thresholds

Crack closure can be caused by a number of mechanisms including plasticity, oxide, roughness, corrosion product, metallurgical transformation and incompressible viscous fluids. Starting from the viewpoint that a crack cannot grow when it is closed, then if crack closure occurs at a level, K_{cl}, that is above K_{min} in the cycle, the effective range of stress intensity becomes:

$$\Delta K_{eff} = K_{max} - K_{cl} \tag{8-23}$$

The crack growth rate, da/dN, will be reduced as the driving force that the crack tip experiences is reduced from ΔK_{app} to ΔK_{eff}. In the limit, as ΔK_{app} is reduced, K_{max} reaches K_{cl} and ΔK_{eff} becomes zero; crack growth ceases but there remains a positive applied ΔK, ΔK_{app}, identified with the threshold for fatigue crack growth, ΔK_{th}.

Experimental measurements of crack closure (or opening) level are often made by use of crack mouth displacement gauges or back face strain gauges. In threshold testing, the measured closure level is subtracted from the apparent ΔK_{th} to still leave a positive result, sometimes termed the "intrinsic" threshold. In fact, this is erroneous as only the mechanical closure level is remotely measured (plasticity induced closure) but not the crack tip oxide induced closure. At high stress ratio, R (=K_{min}/K_{max}), the mechanical closure mechanisms produce K_{cl} below K_{min} but thresholds induced by oxide lead to an approximately constant value of 2MPaR(m) for steels (Ref. 20.).

Applying the logic that at threshold (zero growth rate), the effective ΔK is zero then the overall closure level can be obtained directly from:

$$K_{cl} = \frac{\Delta K_{th}^{R}}{(1-R)} \tag{8-24}$$

It must be appreciated that measured ΔK_{th} values are linked to a cut off growth rate (10^{-11} m/c) not zero, though the error is small due to the steepness of the growth rate curve in the rear threshold region.

If the closure level, K_{cl}, is a constant for a particular stress ratio, then in ΔK increasing tests (and service situations) the closure effect peters out to nothing as K_{min} rises above K_{cl} so that ΔK_{app} and ΔK_{eff} are equal.

The closure level, which defines the threshold ΔK, manifests itself as a stress ratio effect on ΔK_{th}: higher R (higher mean stress) gives lower ΔK_{th} values. It has been found from a wide ranging survey of measured threshold values (Ref. 20.) that the relationship can be described by the equation:

$$\Delta K_{th} = D_0 - \frac{(D_0 - D_1)R}{R_{crit}} \tag{8-25}$$

ΔK_{th}	Threshold ΔK at a particular R value
D_0	Threshold ΔK at R=0
D_1	Threshold ΔK as R->1

R Stress ratio for the ΔK_{th} in question

R_{crit} Value of R above which ΔK_{th} is constant and equal to D_1

The appearance of the dependence of ΔK_{th} on R is as shown in Figure 8-59, which is taken from the database module PFMAT.

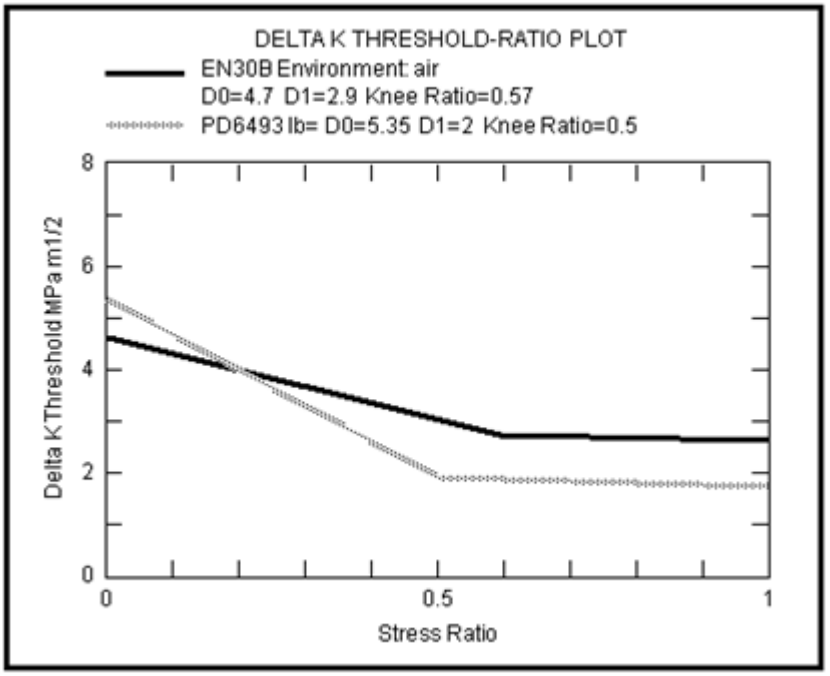


Figure 8-59 The Dependence of ΔK_{th} on Stress Ratio

Short Cracks

A logical progression from the previous section on crack closure is to consider the apparently anomalous behavior of short cracks. It is well-established that short cracks grow at faster rates and at lower ΔK values than would be projected by long crack data and thresholds. Broadly speaking, this is because short cracks have less closure (especially plasticity induced closure) associated with them and indeed may have no threshold at all.

A definition of what is "short" and what is "long" arises from the Kitagawa Diagram (see previous section on initial crack size). Clearly if the cyclic stress range is above the fatigue limit, $\Delta\sigma_o$, then fatigue failure will eventually occur implying zero threshold for growth whatever the defect size. For crack sizes smaller than l_o, conventional fracture mechanics is unsafe in its prediction of no growth and the use of K is dubious as the crack size may well be less than the microstructural features such as grain size, pearlite colony spacing, etc. The parameter l_o thus defines the boundary between short and long cracks and takes values typically in the range 0.01 to 1 mm for steels where high strength steels take the smallest values.

Obviously, because short cracks grow faster, they very quickly become long cracks. As they do so, the plastic deformation wake builds up the crack closure level to that pertaining to long cracks. Mechanistically, it could be suggested that short cracks are not anomalous; long cracks are, because of the closure phenomena associated with them.

Short crack growth is not modeled in MSC Fatigue in its present form. However, the crack length dependence of the crack closure level is taken into account. The derivation of the formula in (8-26) is given by Austen (Ref. 20.), but it describes how the closure level, K_{cl}, depends on K_{max} (because of the plasticity effect) and on crack size in relation to l_o.

$$K_{cl} = K_{max} - K_{max}\left(\frac{a + l_o}{a}\right)^{\frac{1}{2}} + \frac{\Delta K_{th}}{(1 - R)} \tag{8-26}$$

Note that it reduces to (8-24), the long crack value of K_{cl} when a is very much greater than l_o. An example of the buildup of crack closure level with increasing crack size is shown in Figure 8-60.

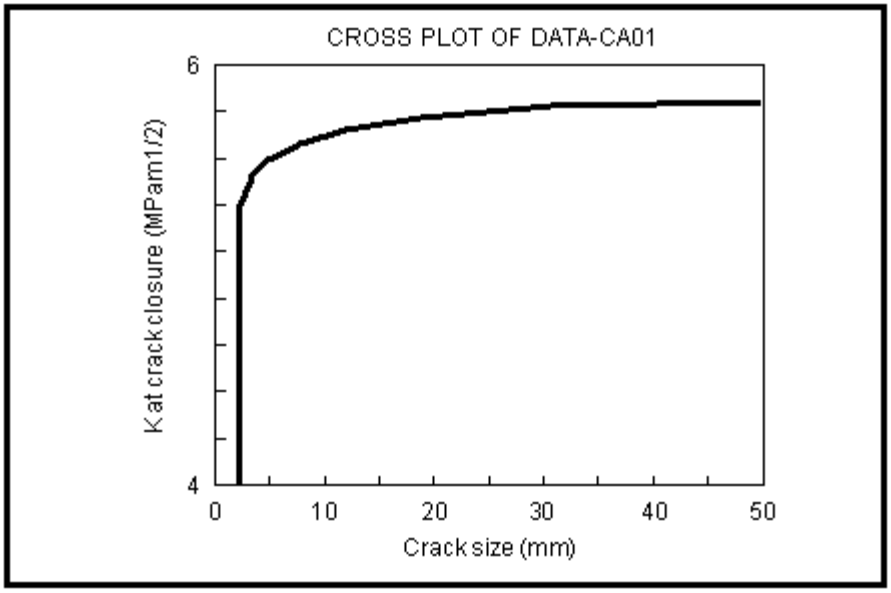


Figure 8-60 The Buildup of Crack Closure Level with Increasing Crack Size

Notches

To account for cases where fatigue cracks grow from notches, it is necessary to modify the fracture mechanics description of the crack tip stress field. The presence of the notch overrides the crack tip K solution when the crack is close to the notch but, as the crack grows by fatigue, it escapes from the notch influence. Therefore, it is inappropriate to simply multiply the stress range by the stress concentration factor and equally inappropriate to ignore the notch presence in the K solution. A mathematical model is required in which the notch correction factor varies with crack size and disappears as the crack escapes from the notch influence. Cracks near notches behave like short cracks and a model developed to meet these requirements (Ref. 20.) incorporates the short crack parameter, l_o (see previous sections).

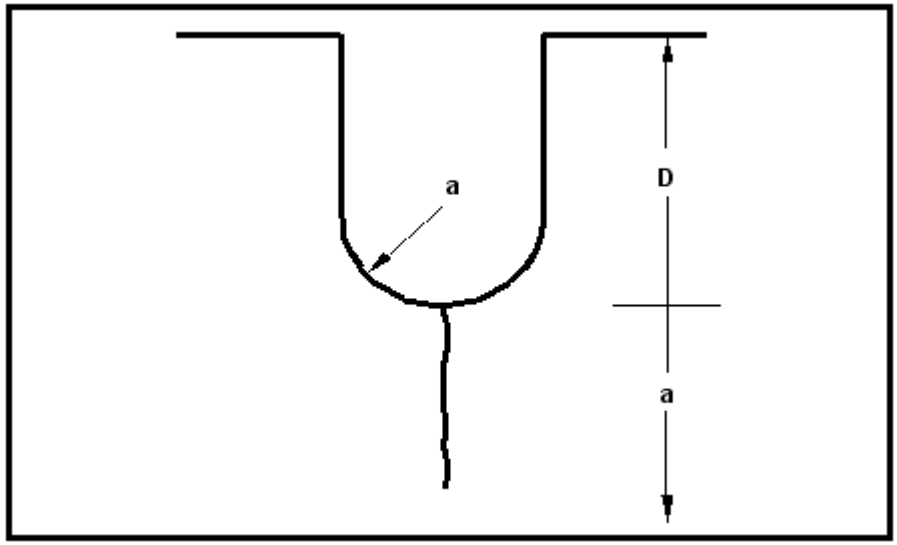


Figure 8-61 Schematic Illustration of a Crack at a Notch

Referring to Figure 8-61 for definition of terms, the procedure is to calculate ΔK_{app} assuming the notch is part of the crack, based on a notional crack size equal to a+D. Then this value is reduced by a correction factor to account for the fact that the notch is not a sharp crack but has a significant root radius, ρ. The full derivation is given elsewhere (Ref. 20.); the formula built into MSC Fatigue to account for notch effects is:

$$\text{correction factor} = \frac{\Delta K_{eff}}{\Delta K_{app}} = \left\{ \frac{a + \left(\frac{aD}{l_o}\right)\left(\frac{\rho_c}{\rho}\right)^{0.5}}{(a+d)} \right\}^{0.5}$$

ΔK_{eff}	Effective ΔK at the crack tip
ΔK_{app}	Remotely applied or apparent ΔK
a	Crack size
D	Notch depth
r	Notch root radius
ρ_c	Root radius of a sharp fatigue crack
l_o	Short crack parameter

The correction factor is less than unity and the formula correctly reduces to unity for a sharp notch $(\rho = \rho_c)$. The factor reaches unity when the crack escapes from the notch; the crack size at this point is given by:

$$a = l_o\left(\frac{\rho}{\rho_c}\right)^{0.5} \tag{8-27}$$

a cannot be less than l_o so that the minimum correction factor is:

minimum correction factor =

$$\left\{\frac{l_o + D\left(\frac{\rho_c}{\rho}\right)^{0.5}}{(a + D)}\right\}^{0.5} \tag{8-28}$$

A plot of the notch correction factor against crack size produced by the PCPOST module is shown in Figure 8-62.

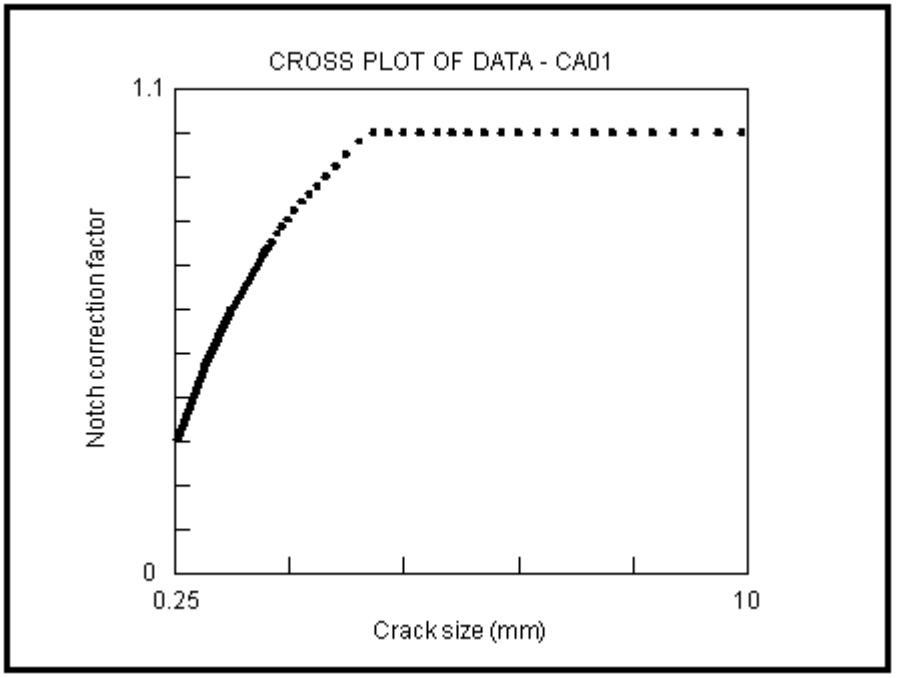


Figure 8-62 Plot of Notch Correction Factor Against Crack Size

Static Fracture

Fatigue crack growth rate increases with increasing crack size because it is driven by ΔK and ΔK depends on $(a)^{0.5}$. Nevertheless, as the critical fracture condition is approached, growth rate increases even more rapidly due to the contribution of static fracture modes to the cracking process. Eventually, the crack growth rate becomes infinite (or at least as fast as the speed of sound) when K_{max} in the cycle reaches the critical K usually identified with the material fracture toughness.

To model this increase in growth rate from static fracture modes in MSC Fatigue, an additional K, K_{fs}, is added to K_{max} so that ΔK_{eff} is increased and growth rate is increased according to (8-21), the Paris Law. This model (Ref. 20.) is based on LEFM though current research at British Steel is aimed at deriving an equivalent model based on the J integral concept to account for elasto-plastic fracture modes. The formulas are:

$$\Delta K_{eff} = K_{maxeff} - K_{mineff} \tag{8-29}$$

$$K_{maxeff} = K_{max} + K_{fs} \tag{8-30}$$

$$K_{fs} = K_{max} / (K_{IC} - K_{max}) \tag{8-31}$$

ΔK_{eff}	Effective ΔK
K_{maxeff}	Effective maximum K in the cycle
K_{mineff}	Effective minimum K in the cycle
K_{fs}	Additional K from static fracture modes
K_{max}	Apparent maximum K in the cycle
K_{IC}	Material fracture toughness

Note that K_{fs} is small at small values of K_{max} and tends to infinity as K_{max} approaches K_{IC}. The "stat frac" light in the on-line MSC Fatigue display of modifying effects is activated when K_{fs} is greater than 5% of K_{IC}.

A plot of K_{fs} with increasing ΔK_{app} is shown as an illustrative example in Figure 8-63.

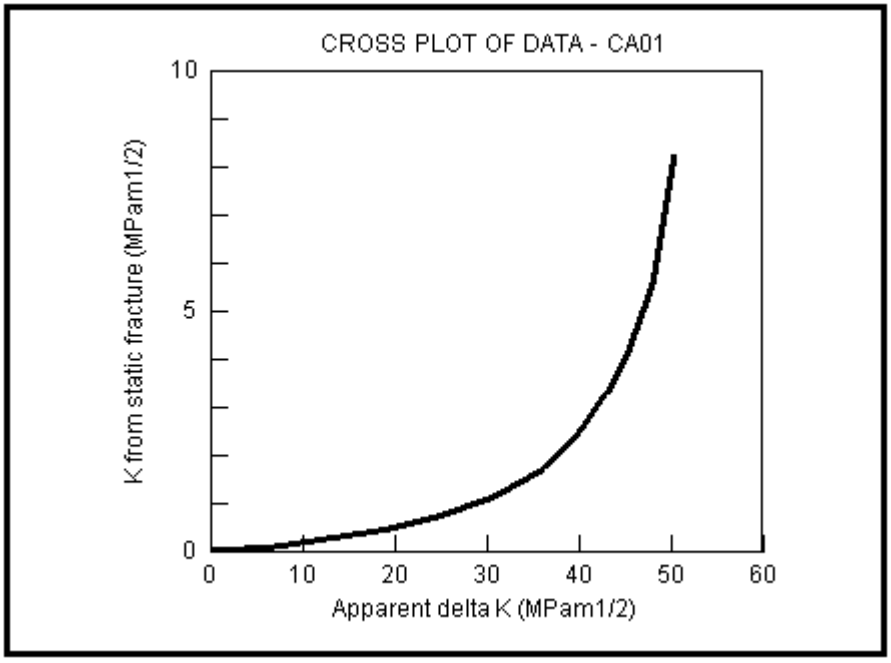


Figure 8-63 Contribution from Static Fracture Modes, K_{fs}, Increasing with ΔK_{app}

Cycles Sequence

Under constant amplitude loading, the fatigue crack growth rate response has been modeled using ΔK_{eff} so that crack closure and static fracture are modeled by:

$$\Delta K_{eff} = K_{maxeff} - K_{mineff} \tag{8-32}$$

$$K_{maxeff} = K_{max} + K_{fs} \tag{8-33}$$

$$K_{mineff} = K_{min} or K_{cl} \tag{8-34}$$

To account for the effect of cycles sequence, the so called "history effect," an additional term is incorporated into these equations as follows:

$$K_{maxeff} = K_{max} + K_{fs} - K_R \tag{8-35}$$

$$K_{mineff} = (K_{min} - K_R) or K_{cl} \tag{8-36}$$

where K_R is termed the "residual K." This residual K arises whenever a previous cycle has produced a crack tip plastic zone that extends beyond the plastic zone of the current cycle. Prior deformation therefore causes the cyclic deformation to deviate from constant amplitude behavior and a history effect results. History effects include retardation (possibly to crack arrest), acceleration, delayed retardation, and what is known as "overshoot" or "lost retardation." These invalidate linear damage accumulation concepts (Miner's Rule) and necessitate cycle-by-cycle modeling.

As an example, consider the case of a single overload in a constant amplitude sequence. The crack growing under constant amplitude has two plastic zones associated with its tip: a maximum zone based on Kmax and a reversed zone based on ΔK. When the overload occurs, it then has four plastic zones at the crack tip (see Figure 8-65) and, because of the confusion and complexity, transient response results. The measured crack size — cycles and growth rate — ΔK information appears as in Figure 8-64 with retardations occurring at each overload.

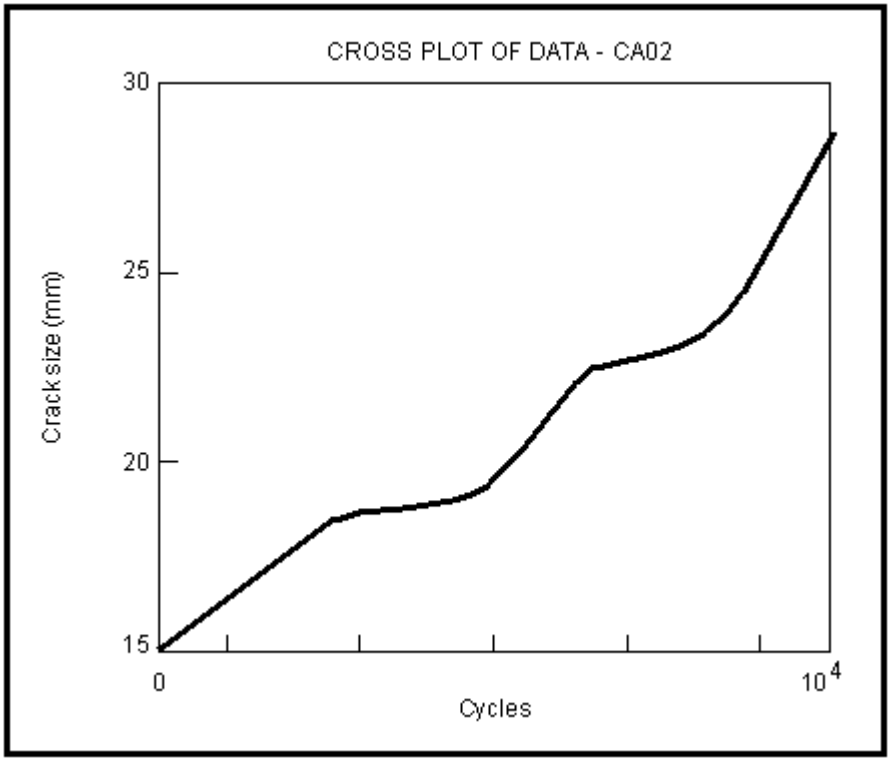


Figure 8-64 Crack Length — Cycles Response for Single Overloads

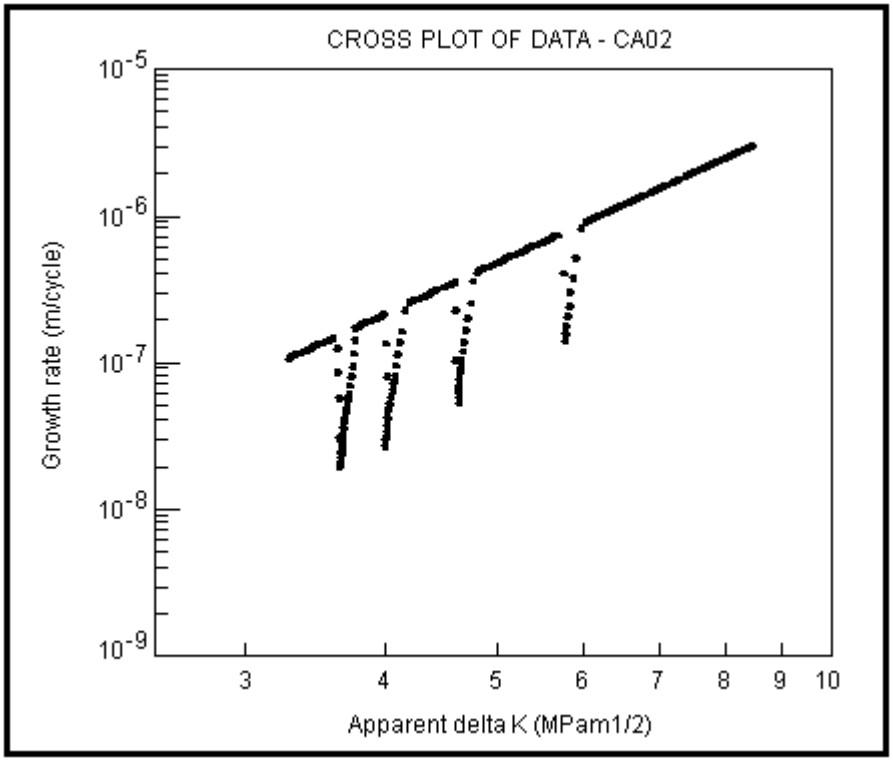


Figure 8-65 Growth Rate — Δ K Response for Single Overloads

In a random sequence of cycles, the largest current overload governs the response of the crack to each cycle in turn.

The original formulation for K_R was due to Willenborg (Ref. 23.)and was subsequently modified by Johnson at NASA (Ref. 24.) at the same time as the closure concepts of Elber (Ref. 25.) were also being introduced to explain and model history effects. The original Willenborg expression was:

$$K = K_{OL}\left(1 - \frac{\Delta a}{Z_{OL}}\right)^{0.5} - K_{max} \tag{8-37}$$

K_{OL} K due to the overload

Δa Crack extension since the overload

Z_{OL} Plastic zone size due to the overload and given by:

$$Z_{OL} = \frac{1}{3\pi}\left(\frac{K_{OL}}{\sigma_y}\right)^2 \tag{8-38}$$

This has been substantially expanded by Austen ((Ref. 13.), (Ref. 14.), (Ref. 20.), (Ref. 21.)) to account for delayed retardation and overshoot as:

$$K = K_{OL}\left(\frac{1-\Delta a}{Z_{OL}-Z}\right)^{0.5} - (K_{OL}-K_{max})\left(\frac{1-\Delta a}{Z_{rev}}\right)^{0.5} - K_{max}\left(\frac{1-\Delta a}{Z_{OL}}\right)^{0.5} \quad (8\text{-}39)$$

Z = Maximum plastic zone size for the current cycle

Z_{rev} = Reversed plastic zone size for the overload

These are given by:

$$Z = \frac{1}{3\pi}\left(\frac{K_{max}}{\sigma_y}\right)^2 \quad (8\text{-}40)$$

and

$$Z_{rev} = \frac{1}{12\pi}\left(\frac{K_{OL}-K_{min}}{\sigma_y}\right)^2 \quad (8\text{-}41)$$

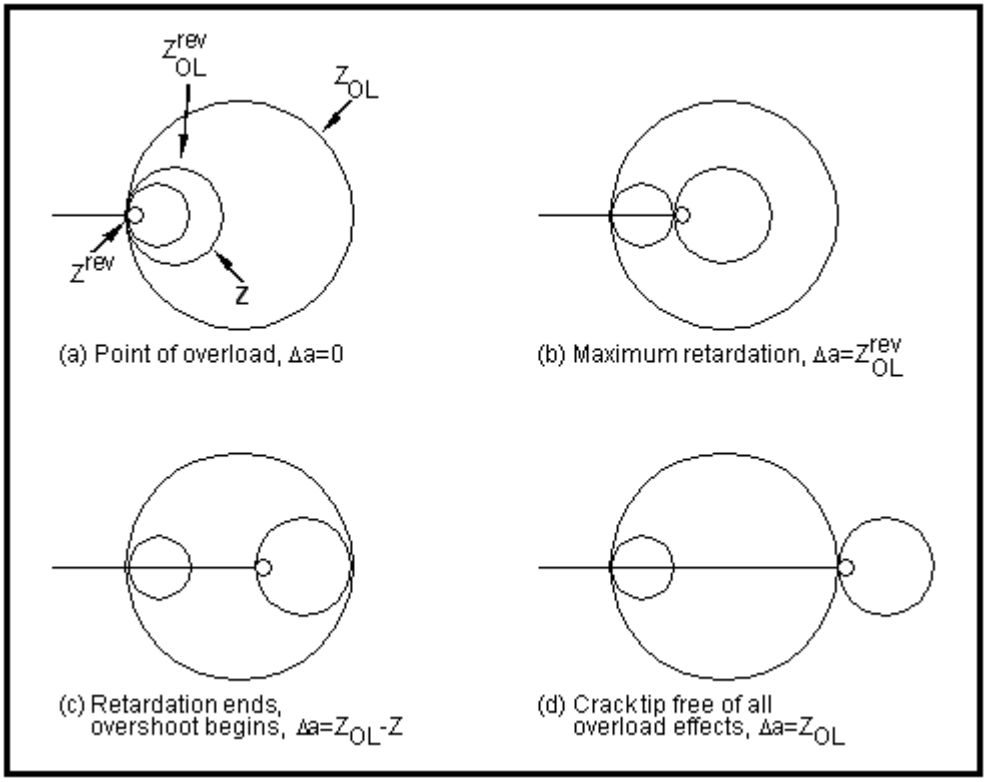


Figure 8-66 Crack Tip Plastic Zone Interactions Following an Overload

The first term in (8-39) describes the decay in K_R which becomes zero as the boundary of the current maximum plastic zone crosses that due to the overload as in Figure 8-66 (c). The second term in (8-39) describes

the delayed retardation such that K_R is in fact zero at the point of overload (Figure 8-66 (a)) and realizes its maximum effect as the moving crack tip crosses the reversed plastic zone due to the overload (Figure 8-66 (b)). The third term in (8-39) controls the overshoot phenomenon where, during the crack extension phase from Figure 8-66 (c) to Figure 8-66 (d), K_R is negative and increases both K_{maxeff} and K_{mineff}. An overshoot will occur if K_{mineff} is thus raised above the current closure level, K_{cl}. During this phase, the crack behaves almost as a closure free short crack and it is only when it is fully clear of the overload zone does "normality" return (Figure 8-66 (d)).

The passage of the crack through the overload zone is illustrated in terms of stress intensity in Figure 8-67.

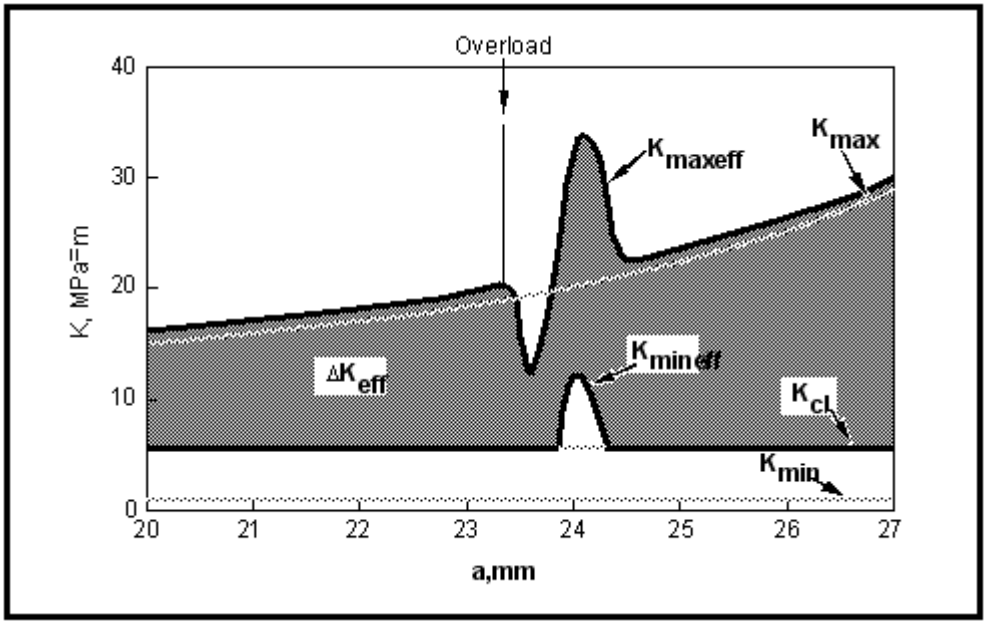


Figure 8-67 Overload Effects on Stress Intensity

This model not only incorporates delayed retardation, overshoot, and closure but also the significant crack extension caused by an overload is modeled via:

$$\Delta a = C\left(K_{OL} + \frac{K_{OL}}{(K_{OL} - K_{IC})} - K_{min}\right)^m \tag{8-42}$$

It is notable that in single overload block loading, the growth rate is significantly retarded (which should increase life), but the overall life may in fact be reduced because of premature failure at an overload.

Clearly, history effects are complex but they are physically observed as in the experimental data shown in Figure 8-67 and it is only through cycle-by-cycle modeling of the crack tip response that the actual behavior is reproduced in prediction methods and software. In fact, Figure 8-64 and Figure 8-65 are MSC Fatigue predictions.

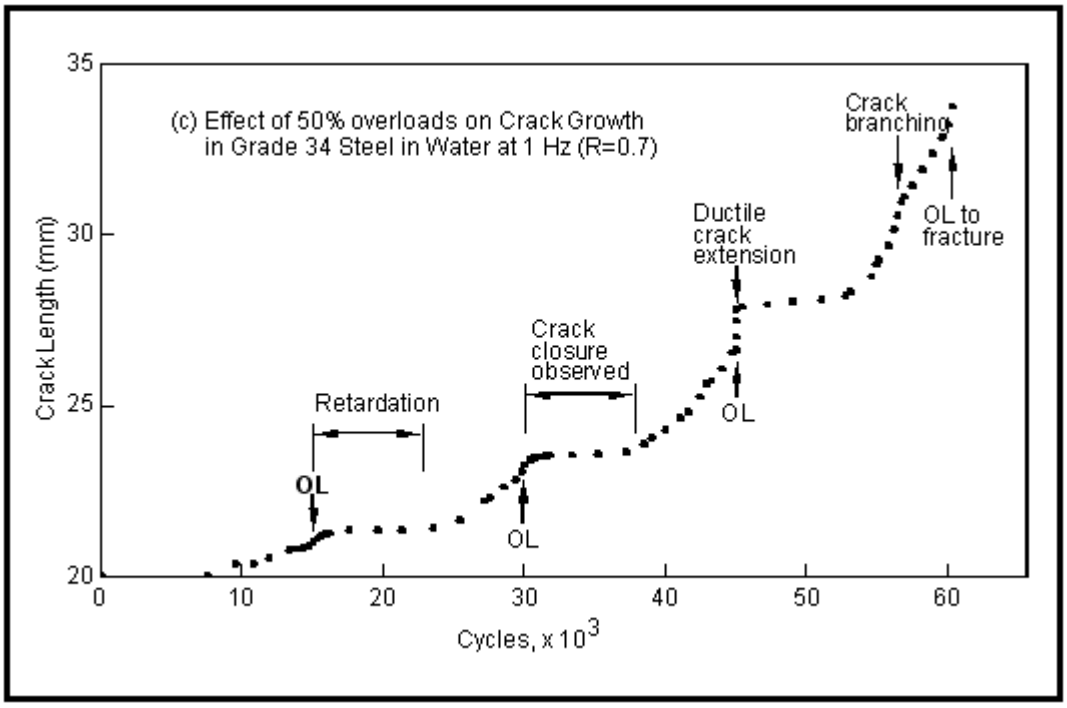


Figure 8-68 Example of Experimental Crack Growth History Showing Sequence Effects (Ref. 21.)

Environment

Fatigue crack growth rates, and hence lifetimes, are strongly influenced by the environment to the extent that corrosion fatigue is a synergistic phenomenon in which corrosion and fatigue are mutually enhanced. Cracks generally adopt that process which gives the fastest rate with the easiest or lowest energy path and this was termed "process competition" in corrosion fatigue modeling (Ref. 15.). To predict lifetimes accurately, environmental influences must be taken into consideration. If materials data is available for the precise environment concerned, then MSC Fatigue can use them for accurate predictions. However, each change of cyclic frequency, waveform, temperature, electrochemical potential or chemical composition of the environment produces a requirement for a new dataset and clearly this is impossible to accommodate within a database even if the test data were available.

The MSC Fatigue module allows materials data for air to be combined with models for the environmental effects on fatigue crack growth rate for the most common environmental mechanism, that of hydrogen embrittlement. In this way, corrosion fatigue can be modeled in the absence of appropriate corrosion fatigue crack growth data for the material/environment system.

In aqueous environments, two electrochemical processes occur: the anodic reaction dissolves metal which makes the crack less sharp, the cathodic process generates hydrogen which, in the presence of cyclic deformation, concentrates in the crack tip plastic zone. Crack tip blunting reduces ΔK_{eff} and slows down the growth rate; localized hydrogen embrittlement introduces an alternative, faster growth rate.

The equation which governs the growth rate for local hydrogen embrittlement is:

$$\frac{da}{dN} = \text{greater of } \frac{(L-s)(\delta - S)}{Z-s} \text{ and } s \tag{8-43}$$

L	Hydrogen penetration distance = $4(D_H \bullet t)^{0.5}$
s	Striation width = $C\,(\Delta K_{eff})^m$
d	Maximum crack tip opening $F(0.12,\sigma_y\,E)\,BBC(F(\Delta K,(1-R)))^2$
Z	Plastic zone size given by (8-26)
t	Cycle rise time
D_H	Hydrogen diffusion coefficient

The most important parameter is the hydrogen diffusion coefficient and published values very widely between 10^{-7} and 10^{-12} m^2/sec. The effect of the hydrogen influence on crack growth rates and life is compared to behavior in air in Figure 8-69. In nonaqueous hydrogen bearing environments or if cathodic protection suppresses anodic dissolution in aqueous ones, then only the hydrogen effect needs to be modeled leading to the often observed plateau effect in growth rates (Figure 8-69).

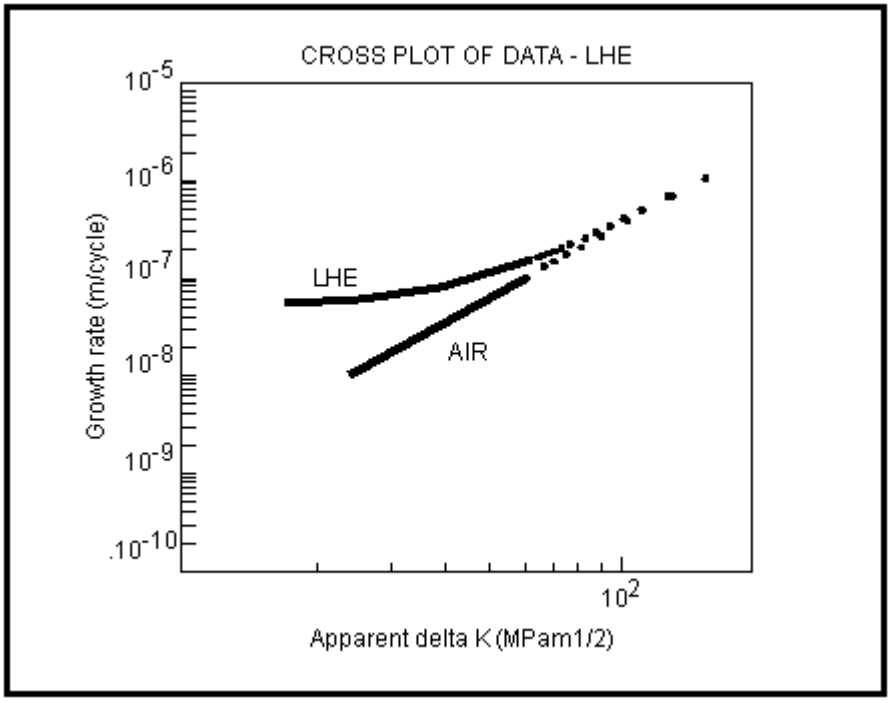


Figure 8-69 Growth Rate Behavior for Local Hydrogen Embrittlement

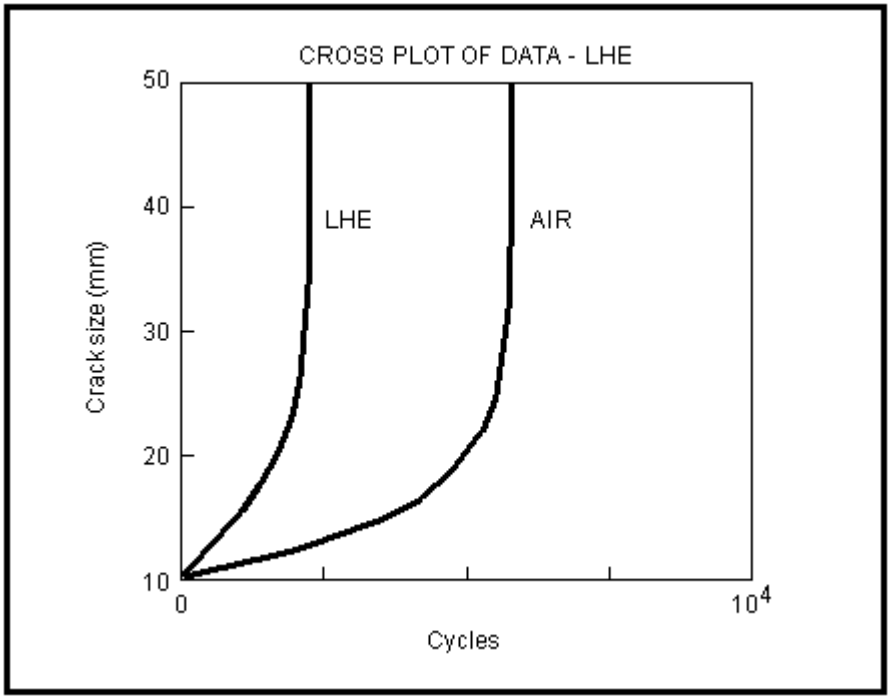


Figure 8-70 Crack Size — Life Behavior for Local Hydrogen Embrittlement

The blunting effect is accounted for by reduction in ΔK_{eff} according to:

$$\Delta K_{eff} = \Delta K\left(\frac{\rho_{sharp}}{\rho_{blunt}}\right)^{0.5} \quad (8\text{-}44)$$

ρ_{sharp} = Sharp or air crack tip radius

ρ_{blunt} = Blunt or corroded crack tip radius

A full derivation of the derivation of ρ_{blunt} from Faraday's Laws of electrochemistry is given elsewhere ((Ref. 16.), (Ref. 20.). The most important parameter here is the anodic current density and again, published values vary widely from 10^{-5} to 1 amp/sq cm. The effect of blunting alone, which may be experienced in square wave loading for example, is shown in comparison to air behavior in Figure 8-71 and Figure 8-72. The overall combined effect of crack tip blunting and local hydrogen embrittlement is shown in Figure 8-73 and Figure 8-74.

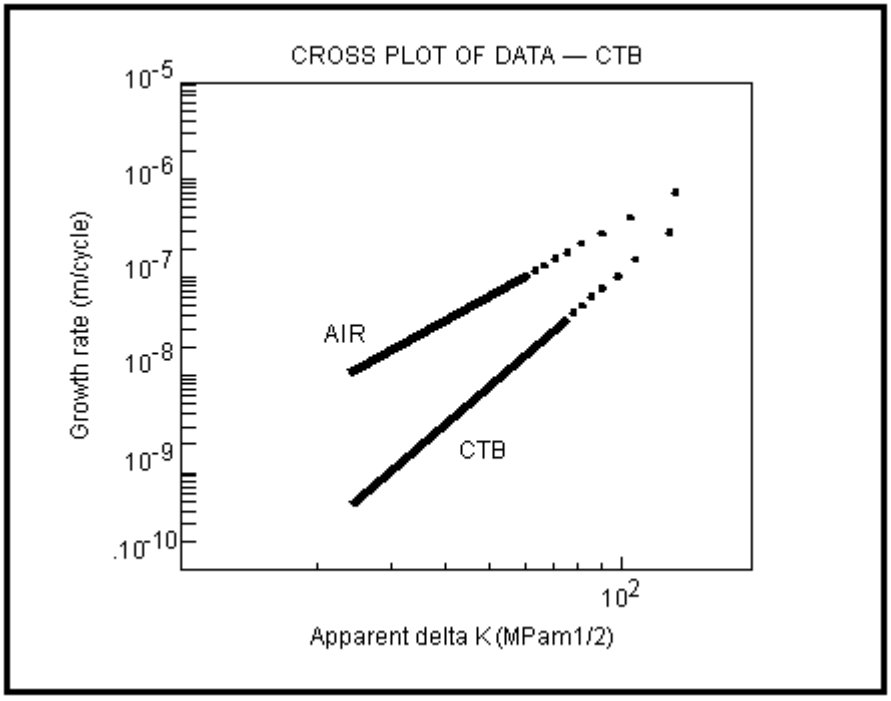


Figure 8-71 Growth Rate Behavior for Crack Tip Blunting

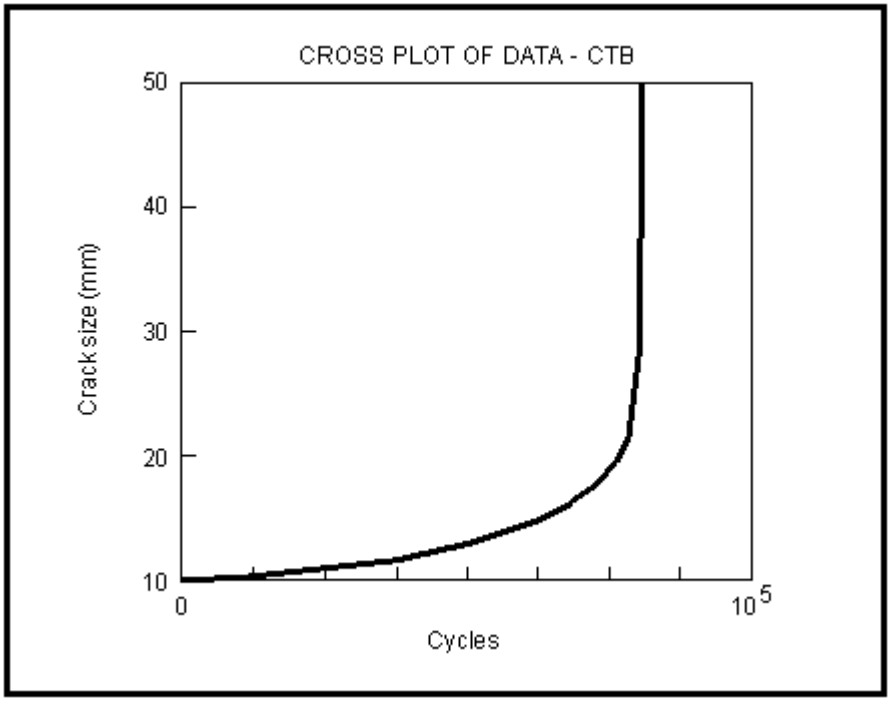


Figure 8-72 Crack Size — Life Behavior for Crack Tip Blunting

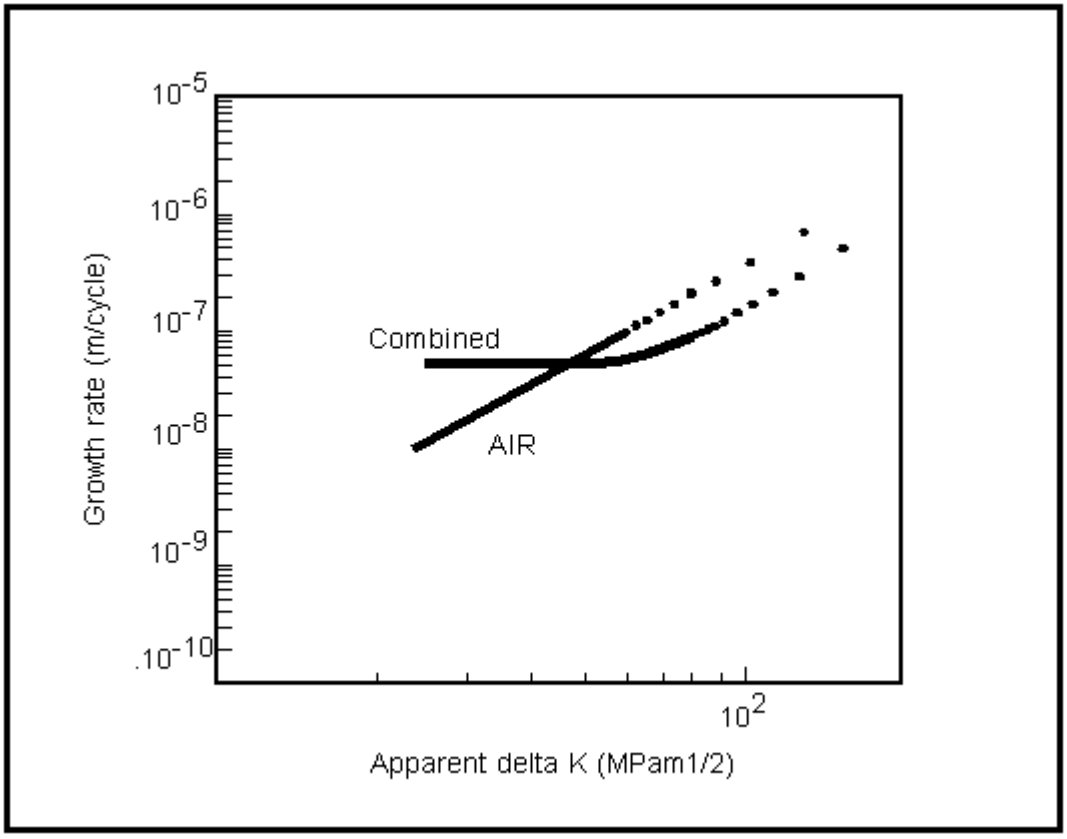


Figure 8-73 Growth Rate Behavior for Combined Corrosion Fatigue Processes

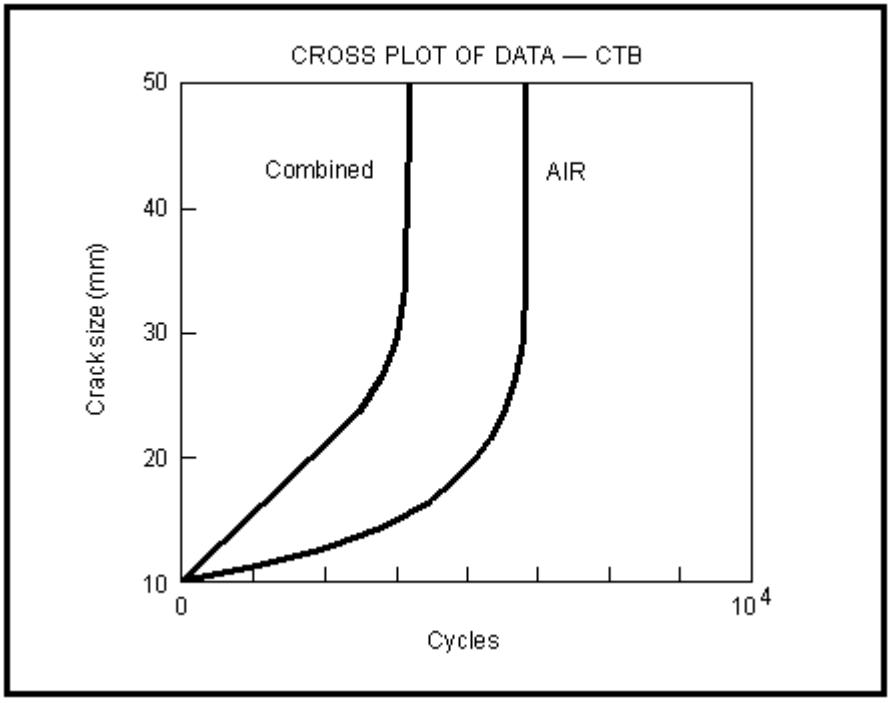


Figure 8-74 Crack Size — Life Behavior for Combined Corrosion Fatigue Processes

Applications of Fracture Mechanics

There are many potential applications for using MSC Fatigue to solve engineering problems arising from the presence and growth of fatigue cracks. A brief list of some of them is given below:

1. **Pre-Prototype Design Analysis** — An assessment can be made of the crack tolerance of a design in terms of its safety and integrity throughout its design life. The use of the software predictions may reduce the number of prototype builds by one or more, thus saving considerable time and expense.
2. **Prototype Testing** — At some stage in the design process, prototype testing may be necessary and the determination of the test program parameters can be enhanced by pre-predicting the test results using MSC Fatigue. Thus, the test program itself can be optimized in terms of time and cost.
3. **Inspection Strategy** — This is the case where a crack is discovered in a component in service and decisions are required quickly concerning actions to be taken:
   - Should the component be removed from service?
   - Can it be repaired and returned to service?
   - Can it be left until the next planned maintenance?
   - If it is safe now, when should it be inspected again?

These questions can be answered using fatigue life calculations and the process repeated at every inspection until a remnant life estimation at the end of the original design life may lead to decisions concerning possible reliving of the component.

4. **Failure Analysis** — The software can be used to recreate the service failure in the computer, to work backwards to the start point of the known life and reach some conclusions concerning the material, loading, environment and original design and manufacture. The conclusions from the life estimations can be used to determine the probable causes of the failure and to answer questions about the likelihood of similar failures in nominally identical components.
5. **Decision Support** — MSC Fatigue provides a general set of tools with which to derive evidence to support engineering decisions. Such decisions need not now rely entirely on engineering experience and good sense but can be substantiated by data obtained from "state-of-the-art" life estimation methods.

Estimation of LEFM Data

The parameters required to characterize fracture toughness and fatigue crack growth properties can be generated purely on the basis of ultimate tensile strength, UTS. The rules for this generation are empirical and come from the British Standards Document PD6493 (1991) and the Dynamic Test Agency Structural Integrity Assessment Manual (volume 9, 1994).

The same rules apply for steels, aluminium alloys and titanium alloys for which the values of elastic modulus, E, are assumed to be 210,000, 73,000 and 110,000 MPa respectively. The rules should not be used for other materials.

The parameters are calculated according to the following rules:

(Note that these parameters are in the MPa, MPam1/2 and metres/cycle units system.)

Yield or proof strength, σ_y:	UTS / 1.3
Fracture toughness, K_{IC}:	0.133 E / σ_y
Unnotched fatigue limit, FL:	0.357 * UTS
Paris Law coefficient, C:	92610 / E^3
Paris Law exponent, m:	3
Threshold at R=0, D0:	5.38* (E / 210000)
Threshold at R->1, D1:	2* (E / 210000)
Critical R value, Rc:	0.5
da/dN lower threshold:	1e-11
SCC threshold, K_{ISCC}:	K_{IC} / 2

The above parameters are for non-welded materials in air. If the material is welded then the Paris Law coefficient, C, is increased by a factor of 2. If the material is undergoing fatigue crack growth in an aqueous environment, such as water, then the Paris Law coefficient, C, is increased by a factor of 7.3. If the material

is both welded and undergoing fatigue crack growth in an aqueous environment, then the Paris Law coefficient, C, is increased by a factor of 14.6.

Multiaxial Stresses and Crack Propagation Methods

The crack propagation methods incorporated in MSC Fatigue are essentially 1-dimensional. All geometry, stress-state, and stress distribution information must be taken into account using the stress intensity calibration. The analysis will assume that the crack path, crack shape, and stress distribution (e.g., proportion of tension and bending) are as defined by the stress intensity calibration. MSC Fatigue's biaxiality indicators provide some means of placing confidence limits on these assumptions.

9 Vibration Fatigue

- Introduction 626
- Job Setup (dtfat) 630
- dtfat Batch Operation 648
- Job Setup (Classic Modules) 649
- Postprocessing Results 667
- FE Vibration Fatigue Analysis (FEVIB) 670
- Frequency Fatigue Life Estimation (MFLF) 695
- Vibration Fatigue Theory 710

Introduction

This chapter on vibration fatigue analysis using a total life approach consists of descriptions of various MSC Fatigue modules and the job setup required to perform the analysis. These modules can be accessed from subordinate forms of the MSC Fatigue main form within the MSC Patran programs.

There are two basic approaches for performing a fatigue analysis in MSC Fatigue. One approach is to use the new DTLib-based solution, dtfat, which is constantly being updated and extended. This solution integrates the latest solver technology from HBM nCode. The other approach is to use the more special-purpose Classic modules. The Classic modules may be more familiar to long-time MSC Fatigue users, and the modules also have the advantage of having a interactive GUI and an "Optimization" tool.

It is possible to carry out a MSC Fatigue analysis outside the pre- and post-processing environments such as the Patran environment by executing the analysis programs from the system prompt. The main reason for doing this is to provide an alternative and sometimes faster route for carrying out multiple computations with only one or two small changes to the analysis parameters. This can be achieved because the MSC Fatigue job parameters and the nodal or elemental fatigue data are stored in ASCII files. For completeness, a general schematic showing the analysis route is shown in Figure 9-1. The aspects of this figure are described throughout this chapter.

There are two Classic modules for vibration fatigue analysis delivered with this system. Each has its own special purpose. In summary, the module FEVIB is the vibration fatigue analyzer that accepts multi-location input files from finite element results. These results can be either a combination of transfer functions from a frequency response analysis with corresponding loading input PSDs or the output response PSDs from a random vibration analysis.

The other module is MFLF which is a more specialized single location analyzer which works directly from a response PSD which can come from anywhere such as a test measurement or the response output from a single location of a FEVIB analysis.

The job setup is described specifically for vibration fatigue analyses in Job Setup (dtfat), 630, however it is recommended that you be familiar with, at the least, Using MSC Fatigue - The Classic Modules (Ch. 3). Although it does not deal directly with vibration fatigue analysis, many of the basic principles are applicable and only discussed there.

Terminal Definition

MSC Fatigue runs on a wide range of computers and graphics devices. The parameters used by each graphics device must be defined by using the MENM module, prior to the first use of MSC Fatigue. For details, please see Module Operations (Ch. 18). This is automatically accomplished when running MSC Fatigue from a pre- and postprocessing environment such as Patran and is transparent to the user and defaults to the Motif driver.

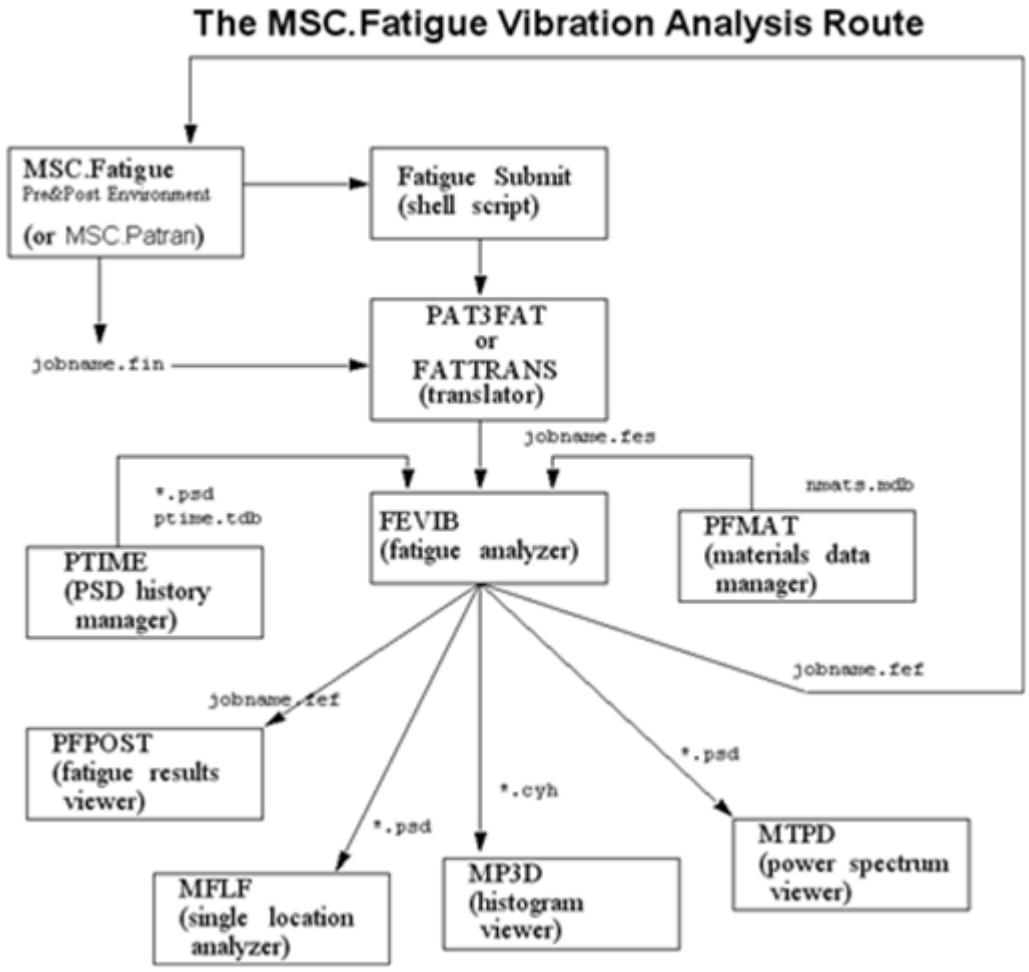


Figure 9-1 Aa MSC Fatigue Vibration Submittal Schematic

Basic Information

All programs in the MSC Fatigue system may be executed by typing the name of the program or its symbol. These programs may ask questions which are not normally presented to you since they are executed as batch jobs when called from the pre/postprocessing environment. The programs normally used in a typical or basic fatigue analysis are listed below.

Interaction with the dtfat solver is normally handled through the MSC Fatigue GUI, since it doesn't have its own GUI. Since dtfat is a multi-disciplinary solver, a different solver module is not required to perform different types of analysis. More detail is given in the following sections.

1. Data Preparation

dtmat	Materials Database Manager
PFMAT	Materials Database Manager
PTIME	Loading Input Power Spectrum and Time History Database Manager, including ASCII Time History File Convertor
MMFD	A Multi-file Display Program

2. Global Multi-Node/Element Analysis

PAT3FAT	Model database (Patran) to Fatigue Input Translator
FATTRANS	New model database (Patran) to Fatigue Input Translator
dtfat	Multi-Discipline Fatigue Analyzer (damage summation)
FEVIB	FE-Fatigue Analyzer (damage summation)
PFPOST	Global Fatigue Results Postprocessor

3. Design Optimization Analyzer

FEVIB	Single Node/Element Total Life (S-N)
MP3D	Cycle/Damage 3-Dimensional Histogram Display
MTPD	Display of output (and input) PSDs
MFLF	Single location vibration fatigue analyzer

4. General Utilities

FEVIB	FES File ASCII/Binary Convertor
PFTRM	Terminal Driver
MCONFIL	Binary to Binary File Convertor

Analysis Route

The actual programs necessary to complete a global multi-node or element vibration fatigue analysis are:

FatigueSubmit	Shell script (necessary for submittal from MSC Fatigue or Patran)
PAT3FAT	Translator (creates the fatigue input file `filename.fes`)
FATTRANS	New Translator (creates the fatigue input `filename.fes`)
dtfat	Fatigue analyzer (accepts FRF/input load PSD combinations)
FEVIB	Fatigue analyzer (accepts response PSD or FRF/input load PSD combinations)
FEVIB	Design optimization and sensitivity studies as well as access to other graphical tools are all accessible from FEVIB.

The programs and options must be used in this sequence, unless you know the critical location ahead of time, then you may skip directly to the design optimization stage. The results may be reviewed using PFPOST or by inspecting the ASCII nodal or element results file (`jobname.fef`) using a text editor or by inspecting fringe plots directly within the pre/postprocessor. PFPOST is generally not compatible with output files from dtfat.

Necessary Files

When a global multi-node fatigue analysis is set up using the MSC Fatigue pre-/post-processing menus, these are the files necessary to run the analysis and the files that are created.

jobname.fin	This file contains all the analysis parameters that were defined in the main and subordinate MSC Fatigue forms such as the analysis type and job titles. A full description of this file is contained in The Job Information File (jobname.fin), 369.
Database	Other pertinent information such as the nodes or elements and the FE results from which to calculate fatigue life is contained in the database in the form of a group or groups. The component stresses or strains from these locations will be used, scaled, superimposed and resolved (dependent on various parameter requests) and used in the fatigue calculations.
Additional Files	Other files that are necessary to complete a successful fatigue analysis are the loading power spectrum files (`ptime.adb`, `ptime.tdb` and `*.psd`), and the materials database (`nmats.mbd`) which is generally held in a central location and not necessary to be located in the user's local directory.

The first of these files is ASCII and may be edited using a standard text file editor. Although this method of defining the MSC Fatigue job parameters is not as automated as using the MSC Fatigue menus, it does offer a simple and rapid method of changing a few job parameters without the encumbrance of a menu structure.

After the translator has been run (described in The Translator (PAT3FAT or FATTRANS) and Submit Script, 629) and the fatigue input file (`jobname.fes`) has been created, the fatigue analyzer, FEVIB, is run which is controlled through a submission script called FatigueSubmit. The fatigue analysis is actually performed by the program FEVIB. When run in interactive mode, this program asks for a number of input parameters which are passed in through the `jobname.fin`, and `jobname.fes` files when run from the MSC Fatigue menus. A full description of file content is provided in Description of Files, 369.

The Translator (PAT3FAT or FATTRANS) and Submit Script

MSC Fatigue uses a translator to combine the information in the database and the job information file (`jobname.fin`) to create the fatigue input file (`jobname.fes`). For this reason, the translator must be run each time any information is changed in the database and/or the results and information files such as if the FEA analysis has been rerun.

To run the translator, the following command should be used:

```
pat3fat jobname
```

or

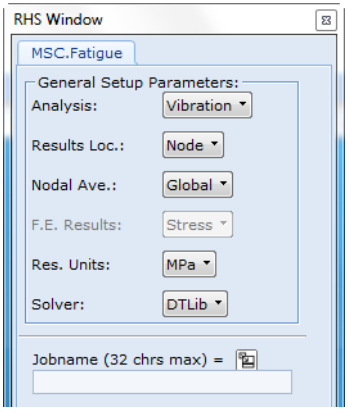
fattrans jobname

PAT3FAT or FATTRANS both produce a `jobname.fes` file which is a binary file. An ASCII to binary and binary to ASCII file convertor is provided for the `jobname.fes` file. The convertor program is part of the FEVIB module and is described in Utilities, 693. A description of the ASCII and binary versions of the fatigue input file (`jobname.asc/fes`) is provided in The Fatigue Input File (jobname.fes), 401. Remember that the fatigue executables can only read the binary version and the ASCII version must always be converted back to binary via FEVIB's utility options.

Once a job is translated and an input file has been created, the actual control of this job from Patran is done through a script file or program called FatigueSubmit. An example of this file is provided in CFG File Specific Parameters, 383.

Job Setup (dtfat)

The setup starts in the MSC Fatigue forms within Patran. You must set the analysis type to Vibration. There are three basic inputs just as for any other analysis types, those being Solution Parameters, Material and Loading Information.



Solution Parameters (Vibration)

The following table lists DTLib solution parameters that are associated with Vibration analysis as shown in the figure below:

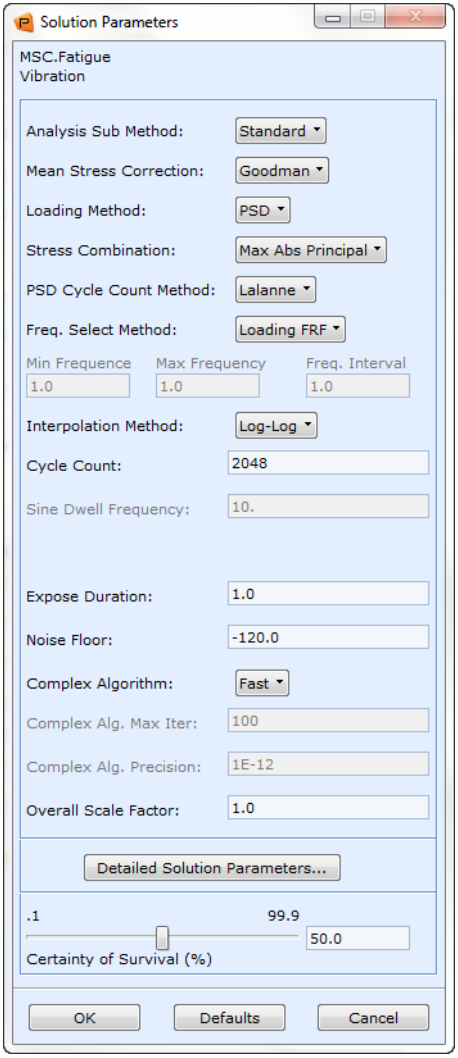

Solution Parameters
MSC.Fatigue
Vibration
Analysis Sub Method:
Standard
Mean Stress Correction:
Goodman
Loading Method:
PSD
Stress Combination:
Max Abs Principal
PSD Cycle Count Method:
Lalanne
Freq. Select Method:
Loading FRF
Min Frequence
Max Frequency
Freq. Interval
1.0
1.0
1.0
Interpolation Method:
Log-Log
Cycle Count:
2048
Sine Dwell Frequency:
10.
Expose Duration:
1.0
Noise Floor:
-120.0
Complex Algorithm:
Fast
Complex Alg. Max Iter:
100
Complex Alg. Precision:
1E-12
Overall Scale Factor:
1.0
Detailed Solution Parameters...
.1
99.9
50.0
Certainty of Survival (%)
OK
Defaults
Cancel

Table 9-1 Solution Parameters

Parameter	Options	Description	Default
Analysis Sub-Method		A number of new material models and S-N methods are available. The interpolation-type methods (or multi-curve methods) allow the user to correct for mean stress effects using the measured material properties rather than rule of thumb approaches, such as Goodman. Please note that a limited number of multi-curve data sets is included with MSC Fatigue. See the section on the **dtmat** material manager for more information on how to add user material data sets. Note that when the Haigh diagram multi-curve method is chosen, the mean stress correction is automatically set to **Interpolate**.	**Standard**
	Standard	Use the standard nCode S-N method and standard S-N material data sets	
	Multi-Mean Curve	Use Multi-Mean curves to determine S-N material properties for a given mean stress. Use Multi-Mean curve material data sets.	
	Multi R-Ratio Curve	Use Multi R-Ratio curves to determine S-N material properties for a given r-ratio of stress. Use Multi R-Ratio curve material data sets.	
	Haigh Diagram	Use Haigh Diagram constant life curves to determine S-N material properties for a given mean stress. Use Haigh Diagram curve material data sets.	
	Bastenaire	Use the Bastenaire probabilistic material data sets for S_N and E-N.	
	Multi-Temp Curve	Use the Temperature Dependent material data sets.	
Mean Stress Correction			**None**
	None	No mean stress correction applied.	
	Goodman	Goodman method of mean stress correction applied.	
	Gerber	Gerber method of mean stress correction applied.	

Table 9-1 Solution Parameters

Parameter	Options	Description	Default
	Interpolate	Use multi-curve method. Note that this only applies to Multi-Mean Curve, Multi R-Ratio Curve, and Haigh Diagram S-N analysis sub-methods. Automatically selected for these Haigh diagram S-N analysis sub-methods.	
	FKM	The FKM method as implemented is based on the method described in the FKM Guideline "Analytical Strength Assessment of Components in Mechanical Engineering", Tr. E. Haibach. 2003.	
	Goodman Tension-Only	Goodman method of mean stress correction applied in tension only.	
	Gerber Tension-Only	Gerber method of mean stress correction applied in tension only.	
Loading Method		For vibration, DTLib does not provide a time history of stress, but rather determines the rainflow count directly. The Loading Method supports the simulation of vibration shaker table tests, driven by PSD (power spectral density) multiple PSDs, swept sine, sine dwell or sine on random loading.	**PSD**
	PSD	Single PSD. Loading may be equally spaced data in time-series or histogram format, or unequally spaced multi-column data (PSD vs. Frequency).	
	Multiple PSD	Multiple PSDs and cross spectra. The PSD as above and cross spectra defined by multicolumn data of magnitude and phase values.	
	Sine Sweep	A swept-sine loading definition, which can be equally or unequally spaced (Amplitude vs. Frequency).	
	Sine Dwell	A sine dwell definition of a single frequency and amplitude.	
	Sine On Random	A sine on random definition. An input PSD as above and a number of sine tones defined by their frequency and amplitude.	
Stress Combination Method			**Max Abs Principal**
	Max Abs Principal	The absolute Maximum Principal stress is defined as the principal stress with the largest magnitude	

Table 9-1 Solution Parameters

Parameter	Options	Description	Default
	Signed Von Mises	The Signed von Mises stress is the von Mises stress, but forced to take the sign of the Absolute Maximum Principal stress.	
	Critical Plane	The Critical Plane option is available only for stress flagged as 2-D. The normal stress is calculated and rainflow counted on multiple planes.The critical plane is the plane with the most predicted fatigue damage. The planes on which the normal stress is determined have normals that lie in the plane of the physical surface, i.e., in the x-y plane of the 2-D stress results coordinate system. The orientation of each plane is defined by the angle ϕ made with the local x-axis.	
PSD Cycle Count Method		Method to predict the rainflow cycle count (or more correctly the PDF of rainflow range) from the local PSD stress response.	**Lalanne**
	Lalanne	Probably the best solution currently is the Lalanne theory. The probability density function of stress range is in effect the predicted rainflow count per second.	
	Dirlik	Dirlik proposed an empirical closed form solution to estimate the pdf of stress range N(S), following extensive computer simulations using the Monte Carlo technique and different signals.	
	Narrow Band	Based on the work of Bendat and Rice. Tends to be conservative for broad-band signals.	
	Steinberg	A narrow band solution.	
Frequency Selection Method		This property determines the frequencies at which the local stress response will be calculated.	**Loading & FRF**
	Loading & FRF	The frequencies at which the local response is calculated will include all the frequencies included in the load PSD or Sine Sweep, and the FRF. This is a good default, as further refinement will not improve the precision of the calculation.	
	Loading	The frequencies at which the local response is calculated will be the same as the frequencies included in the load PSD or Sine Sweep.	

Table 9-1 Solution Parameters

Parameter	Options	Description	Default
	FRF	The frequencies at which the local response is calculated will be the same as the frequencies at which the FRF was calculated.	**Use Max Curve**
	Evenly Spaced	The local response will be calculated at a set of frequencies to be specified by the user, using the properties **Min Frequency**, **Max Frequency**, **Frequency Interval** described below.	
Min Frequency	0 <= **Value**	If Frequency Selection Method = Equally Spaced, this setting specifies the minimum frequency at which the local response is to be calculated.	**No Default**
Max Frequency	0 <= **Value**	If Frequency Selection Method = Equally Spaced, this setting specifies the maximum frequency at which the local response is to be calculated.	**No Default**
Frequency Interval	**Value**	If Frequency Selection Method = Equally Spaced, this setting specifies the frequency interval at which the local response is to be calculated.	**No Default**
Interpolation Method		When the local response needs to be calculated at frequencies where the FRF or loading (PSD or SineSweep) is not defined, the necessary values are interpolated.	**Log-Log**
	Log-Log	Use Log-Log interpolation.	
	Lin-Lin	Use Linear-Linear interpolation.	
Cycle Count	**Value > 0**	The rainflow cycle count (based on the continuous PDF) is discretized into a number of histogram bins. Increasing the number of bins increases precision at the expense of computation time.	**64**
Sine Dwell Frequency	**Value > 0**	Frequency at which sine tone loading occurs.	**10.**
Sine Dwell Units	**Seconds, Hours**	Time units for sine dwell duration.	**Seconds**
Sine Dwell Duration	**Value > 0**	Duration of the sine dwell.	**600**
Sine on Random Frequencies	**0. <= Values**	This is a comma separated list of the frequencies of all the sine tones present in a sine on random analysis. It must be specified as a comma separated list of floats.	**No Default**
Sine on Random Amplitudes	**0. <= Values**	This is a comma separated list of the amplitudes of all the sine tones present in a sine on random analysis. It must be specified as a comma separated list of floats.	**No Default**

Table 9-1 Solution Parameters

Parameter	Options	Description	Default
Exposure Duration	**0. < Value**	By default, this is set to 1 second. This means that the rainflow count represents the likely rainflow count per second, and when we carry out a fatigue calculation, the result is the damage per second.	**1.**
Noise Floor	**-5000 <= Value <= -10**	This defines the level in db below the peaks in the PSD which will be considered noise. The default value -120 db corresponds to six orders of magnitude below the peak in the PSD. Values below this limit are set to zero.	**-120**
Complex Algorithm		Decides if we use the **Fast**, or the **Exact** method for calculating the solution of complex matrices.	**Fast**
	Fast	Use Fast method.	
	Exact	Use Exact method.	
Complex Algorithm Max Iter	**10 < Value**	This is the maximum number of iterations used, when attempting to get convergence in the calculation of complex eigenvalues.	**100**
Complex Algorithm Precision	**Value < 1E-5**	Precision to use in the complex eigenvalue algorithm.	**1E-12**
Overall Scale Factor	0 < **Value** <= 1e10	The scale factor to apply to the combined data prior to performing damage calculation.	**1.0**
Certainty of Survival	0 <= **Value** <= 100	Percentage certainty of survival	**50**

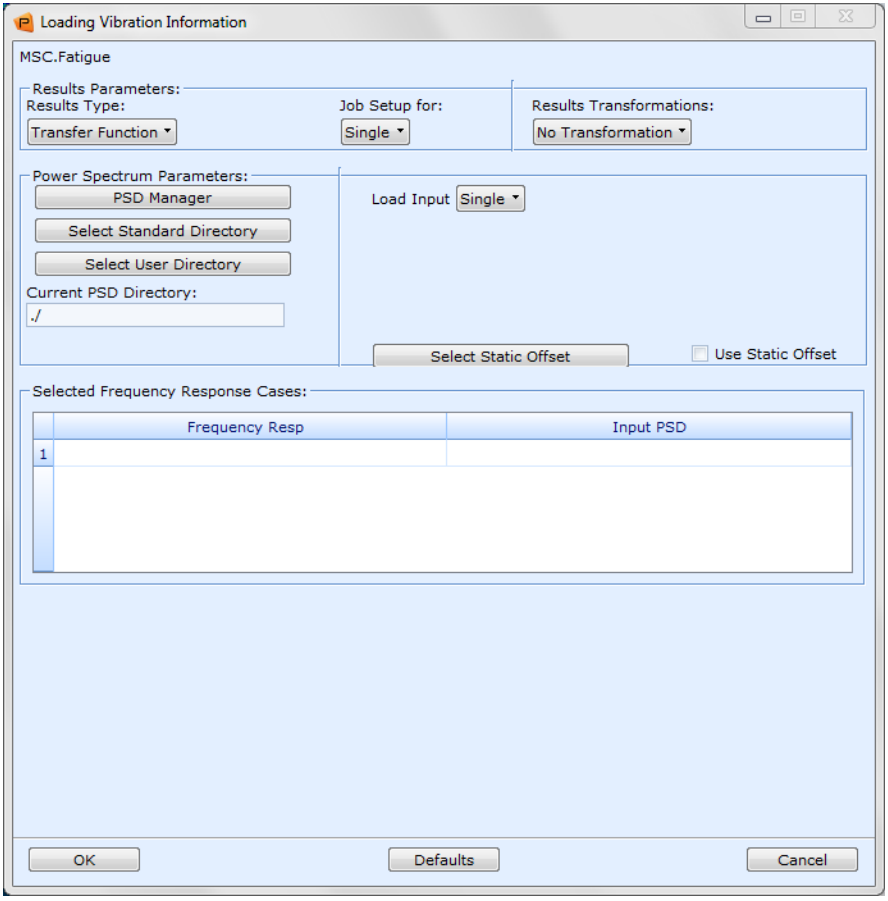
Superposition of Static Load Case

None of the various methods for creating a rainflow count (Lalanne, Dirlik, Narrow Band, Steinberg, Swept-Sine) predict mean stress values. The rainflow cycle count that these methods produce has zero mean stress for all cycles. Although vibration in a linear system cannot generate mean stresses, mean stresses may exist and may be modeled due to a variety of reasons, for example, gravity loading or assembly stresses. For this reason, DTLib allows a mean stress to be included from a static offset loadcase. The mean stress applied is either the Abs Max Principal or Critical Plane stress extracted from

the static offset loadcase stress tensor, and is used to shift the mean of all cycles in the predicted rainflow count.

Loading Information (Single PSD)

The appearance of the Loading Information form depends on the choice of Loading Method, as defined on the Solution Parameters form. This is due to the substantial difference in data needed to satisfy the selected Loading Method. The following section describes the Loading Information required for the **Loading Method = PSD**.


Loading Vibration Information
MSC.Fatigue
Results Parameters:
Results Type:
Transfer Function
Job Setup for:
Single
Results Transformations:
No Transformation
Power Spectrum Parameters:
PSD Manager
Select Standard Directory
Select User Directory
Current PSD Directory:
./
Load Input
Single
Select Static Offset
Use Static Offset
Selected Frequency Response Cases:
Frequency Resp
Input PSD
1
OK
Defaults
Cancel

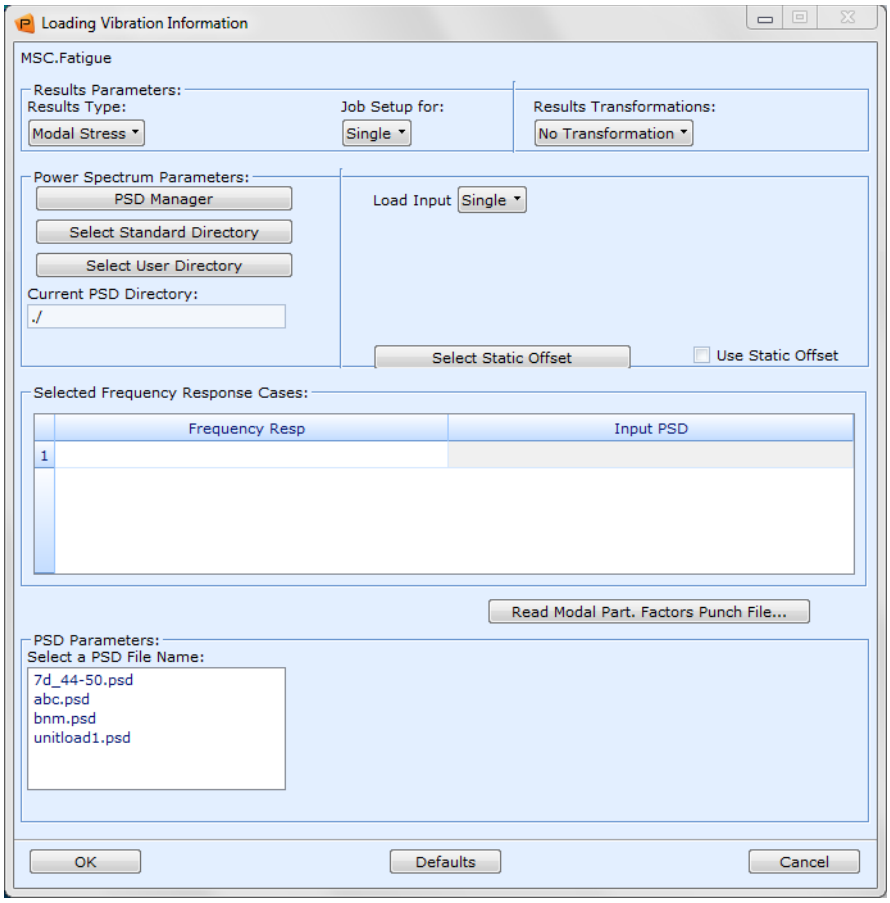
The settings on the Loading Information form are described in the table below.

Table 9-2 Loading Information Form Settings (Loading Type = Single PSD)

Table Entry	Option	Description	
Results Type		There are two basic FE result types that dtfat can accept for vibration fatigue. The Loading Information form updates itself when this parameter is changed. The additional information needed when using one of these result types is described later in this section.	**Transfer Function**
	Transfer Function	FE results used in the vibration analysis is frequency response.	
	Modal Stress	FE results used in the vibration analysis is modal stress and complex modal participation factors.	
Job Setup	**Single**	This is currently the only option, since Duty Cycle is not currently supported for vibration fatigue.	**Single**
Results Transformation		This option is generally obsolete, since FE results are transformed to a consistent coordinate system before nodal averaging.	**No Transformation**
	No Transformation	No transformation of FE results is performed. DTLib assumes that FE results for vibration fatigue are 2-D, and thus only considers the Sxx, Syy, and Sxy components of the results tensor. This option should be used for vibration fatigue analysis.	
	Transform to Basic	FE results are transformed to the basic coordinate system.	
Load Input	**Single**	This is the only option for **Loading Type** = **PSD**	**Single**
Use Static Offset	**False, True**	The FE results from a static load case are used to model fastener or other assembly-type loads. Use the **Select Static Offset** button to select the appropriate FE results case.	**False**

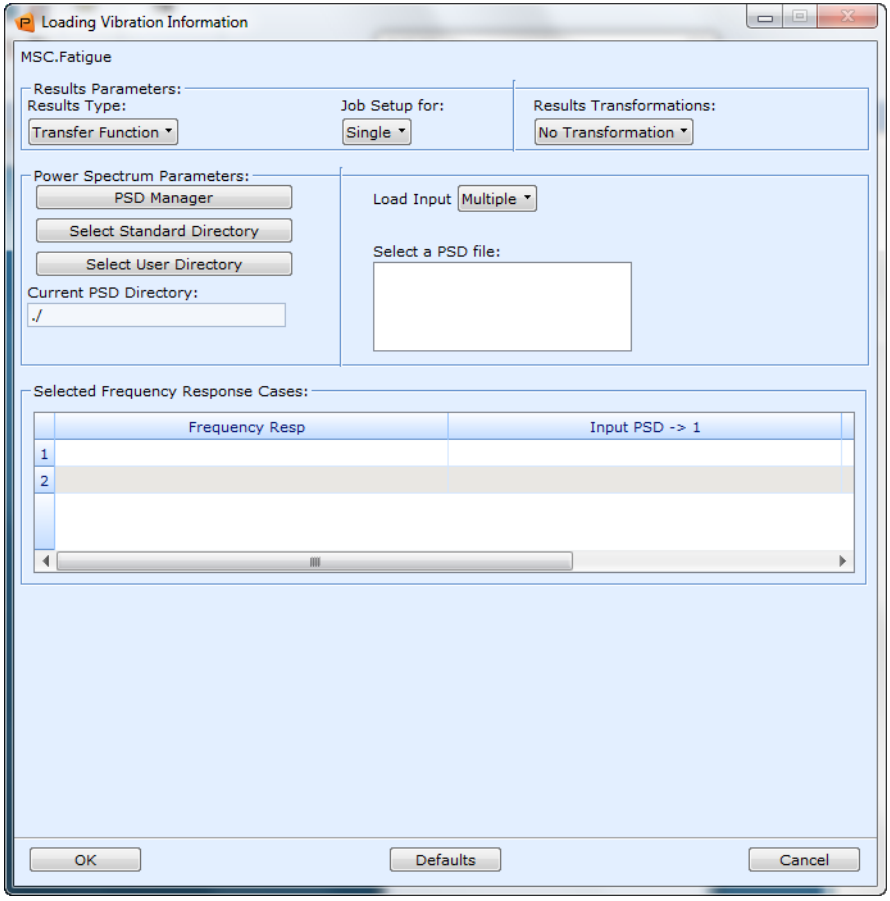
If **Results Type** = **Modal Stress** is selected, the modal stress cases can be selected in the usual way by clicking on the **Get/Filter Results** button. The complex modal participation factors must be read into MSC Fatigue and converted to the appropriate file format. This can be accomplished by reading in the Nastran punch file via the **Read Modal Part. Factors Punch File** button shown below and selecting the appropriate punch file.

The punch file read and convert step creates a number of csv files, jobname01.csv - jobname0n.csv, one file for each mode in the punch file. These files contain the frequency vs complex modal participation for each mode.



Loading Information (Multiple PSD)

The following section describes the Loading Information required for the **Loading Method** = **Multiple PSD**.



The settings on the Loading Information form are described in the table below.

Table 9-3 Loading Information Form Settings (Loading Type = Multiple PSD)

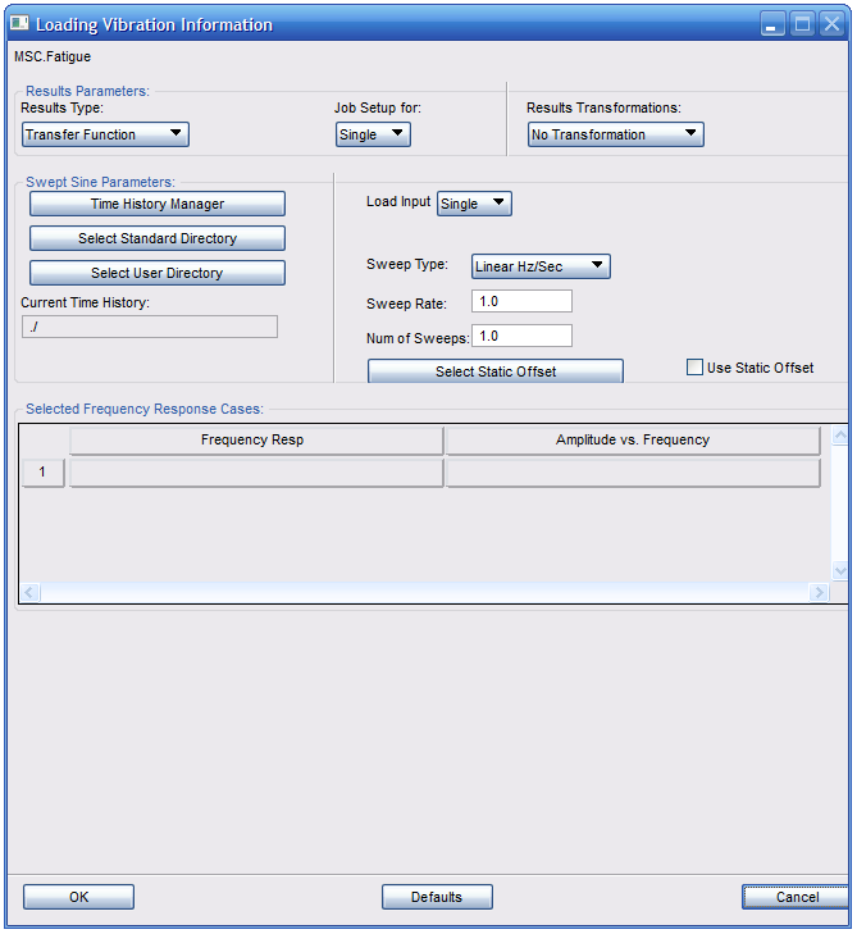
Table Entry	Option	Description	
Results Type		This is the only option for Multiple PSD.	**Transfer Function**
	Transfer Function	FE results used in the vibration analysis is frequency response.	
Job Setup	**Single**	This is currently the only option, since Duty Cycle is not currently supported for vibration fatigue.	**Single**
Results Transformation		This option is generally obsolete, since FE results are transformed to a consistent coordinate system before nodal averaging.	**No Transformation**

Table 9-3 Loading Information Form Settings (Loading Type = Multiple PSD)

Table Entry	Option	Description	
	No Transformation	No transformation of FE results is performed. DTLib assumes that FE results for vibration fatigue are 2-D, and thus only considers the Sxx, Syy, and Sxy components of the results tensor. This option should be used for vibration fatigue analysis.	
	Transform to Basic	FE results are transformed to the basic coordinate system.	
Load Input	**Multiple PSD**	This is the only option for **Loading Type** = **Multiple PSD**	**Multiple PSD**
Number of Input	**2<= Value <= 24**	This sets the number of Frequency Response cases used in the analysis. This also controls the number of PSD inputs that can be used to drive the model. Currently, 24 inputs is supported via the MSC Fatigue GUI.	**2**
Matrix Input		This controls whether user populates the PSD and Cross-Spectra portion of the Select Frequency Response Cases spreadsheet from an existing matrix file or by selecting PSD and Cross-Spectra files directly.	**From Matrix File**
	From Matrix File	Using this option, the user can load PSD and Cross-Spectra file information from a existing matrix file via the **Select a Matrix File** button.	
	From Spreadsheet	The user must select existing PSD and Cross-Spectra files to associate with a given frequency response case. This is done by clicking in the Multi Column portion of the spreadsheet and selecting from the available loading files shown in the PSD or Cross-Spectra list box.	
Selected Matrix File	**Value**	Name of the selected matrix file. The contents of the matrix file are automatically load when an existing matrix file is selected via the **Select a Matrix File** button.	**No Default**

Loading Information (Sine Sweep)

The following section describes the Loading Information required for the **Loading Method** = **Sine Sweep**.


Loading Vibration Information
MSC.Fatigue
Results Parameters:
Results Type:
Transfer Function
Job Setup for:
Single
Results Transformations:
No Transformation
Swept Sine Parameters:
Time History Manager
Select Standard Directory
Select User Directory
Current Time History:
./
Load Input
Single
Sweep Type:
Linear Hz/Sec
Sweep Rate:
1.0
Num of Sweeps:
1.0
Select Static Offset
Use Static Offset
Selected Frequency Response Cases:
Frequency Resp
Amplitude vs. Frequency
1
OK
Defaults
Cancel

The settings on the Loading Information form are described in the table below.

Table 9-4 Loading Information Form Settings (Loading Type = Sine Sweep)

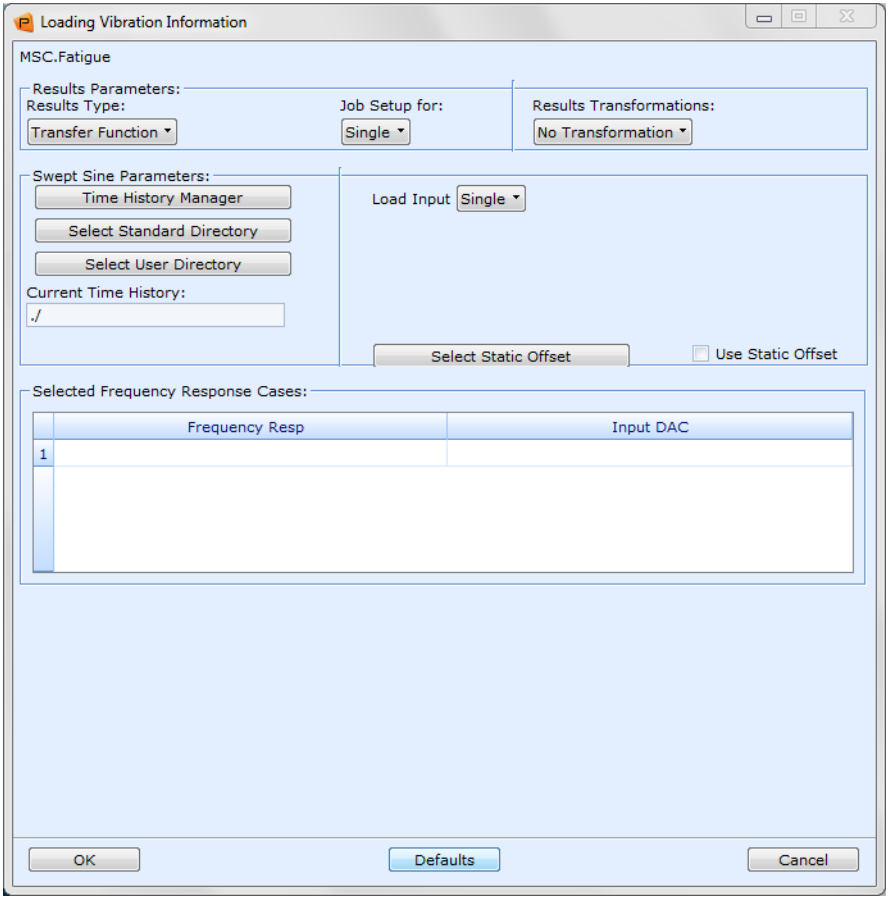
Table Entry	Option	Description	
Results Type		There are two basic FE result types that dtfat can accept for vibration fatigue. The Loading Information form updates itself when this parameter is changed. The additional information needed when using one of these result types is described later in this section.	**Transfer Function**
	Transfer Function	FE results used in the vibration analysis is frequency response.	
	Modal Stress	FE results used in the vibration analysis is modal stress and complex modal participation factors.	
Job Setup	**Single**	This is currently the only option, since Duty Cycle is not currently supported for sine sweep fatigue.	**Single**
Results Transformation		This option is generally obsolete, since FE results are transformed to a consistent coordinate system before nodal averaging.	**No Transformation**
	No Transformation	No transformation of FE results is performed. DTLib assumes that FE results for vibration fatigue are 2-D, and thus only considers the Sxx, Syy, and Sxy components of the results tensor. This option should be used for vibration fatigue analysis.	
	Transform to Basic	FE results are transformed to the basic coordinate system.	
Load Input	**Single**	This is the only option for **Loading Type** = **Sine Sweep**	**Single**
Sweep Type		Each sine sweep definition defines a relationship between amplitude of excitation and frequency. The Sweep Type and Sweep Rate determine the way and the rate at which the calculation (or corresponding physical test) sweeps through the frequencies.	**Linear Hz/Sec**
	Linear Hz/Sec	The sine sweep will be linear through the frequencies at a rate defined by the property Sweep Rate in Hz per second.	
	Log Octaves / Decade	The sine sweep will be logarithmic – at a constant number of octaves per minuted defined by the property Sweep Rate.	

Table 9-4 Loading Information Form Settings (Loading Type = Sine Sweep)

Table Entry	Option	Description	
	Log Secs / Decade	The sine sweep will be logarithmic, with a constant number of seconds per decade defined by the property Sweep Rate.	
Sweep Rate	0. < **Value**	The rate at which the calculation sweeps through the frequencies	**1.0**
Num of Sweeps	0. < **Value**	Defines the number of complete sweeps (low to high frequency) to be carried out. Damage predicted will be for this number of sweeps.	**1.0**
Use Static Offset	**False, True**	The FE results from a static load case are used to model fastener or other assembly-type loads. Use the **Select Static Offset** button to select the appropriate FE results case.	**False**

Loading Information (Sine Dwell)

The following section describes the Loading Information required for the **Loading Method** = **Sine Dwell**.


Loading Vibration Information
MSC.Fatigue
Results Parameters:
Results Type:
Transfer Function
Job Setup for:
Single
Results Transformations:
No Transformation
Swept Sine Parameters:
Time History Manager
Select Standard Directory
Select User Directory
Current Time History:
./
Load Input
Single
Select Static Offset
Use Static Offset
Selected Frequency Response Cases:
Frequency Resp
Input DAC
1
OK
Defaults
Cancel

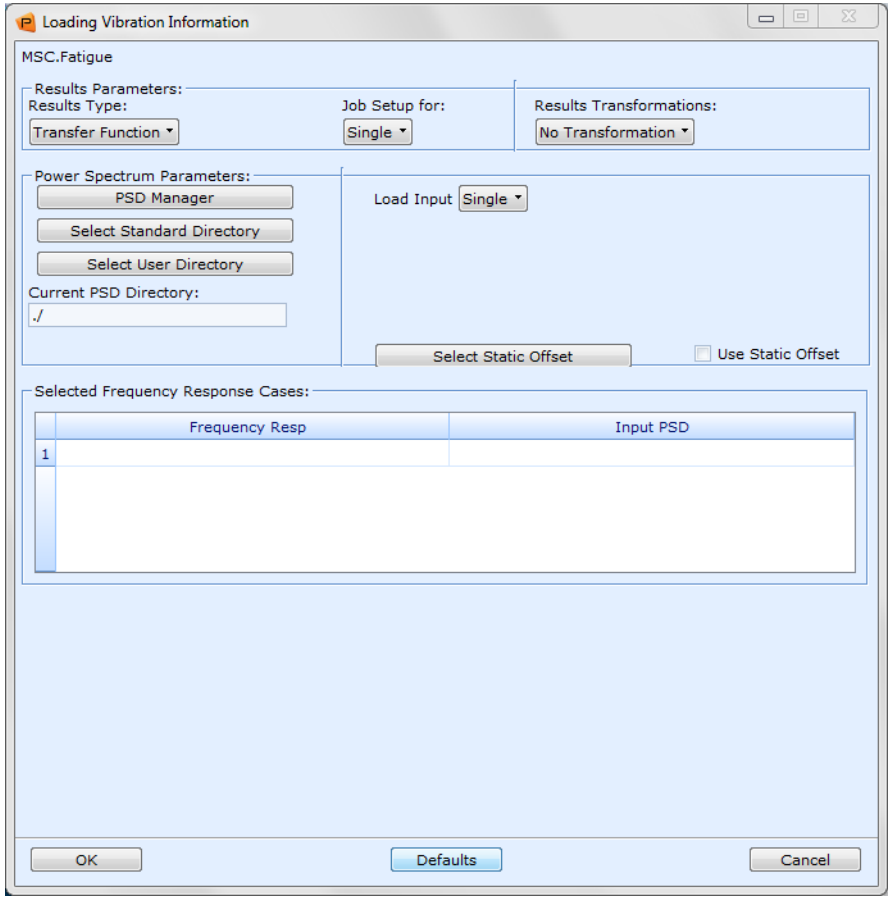
The settings on the Loading Information form are described in the table below.

Table 9-5 Loading Information Form Settings (Loading Type = Sine Dwell)

Table Entry	Option	Description	
Results Type		There are two basic FE result types that dtfat can accept for vibration fatigue. The Loading Information form updates itself when this parameter is changed. The additional information needed when using one of these result types is described later in this section.	**Transfer Function**
	Transfer Function	FE results used in the vibration analysis is frequency response.	
	Modal Stress	FE results used in the vibration analysis is modal stress and complex modal participation factors.	
Job Setup	**Single**	This is currently the only option, since Duty Cycle is not currently supported for vibration fatigue.	**Single**
Results Transformation		This option is generally obsolete, since FE results are transformed to a consistent coordinate system before nodal averaging.	**No Transformation**
	No Transformation	No transformation of FE results is performed. DTLib assumes that FE results for vibration fatigue are 2-D, and thus only considers the Sxx, Syy, and Sxy components of the results tensor. This option should be used for vibration fatigue analysis.	
	Transform to Basic	FE results are transformed to the basic coordinate system.	
Load Input	**Single**	This is the only option for **Loading Type** = **Sine Dwell**	**Single**
Use Static Offset	**False, True**	The FE results from a static load case are used to model fastener or other assembly-type loads. Use the **Select Static Offset** button to select the appropriate FE results case.	**False**

Loading Information (Sine on Random)

The following section describes the Loading Information required for the **Loading Method** = **Sine on Random**.


Loading Vibration Information
MSC.Fatigue
Results Parameters:
Results Type:
Transfer Function
Job Setup for:
Single
Results Transformations:
No Transformation
Power Spectrum Parameters:
PSD Manager
Select Standard Directory
Select User Directory
Current PSD Directory:
./
Load Input
Single
Select Static Offset
Use Static Offset
Selected Frequency Response Cases:
Frequency Resp
Input PSD
1
OK
Defaults
Cancel

The settings on the Loading Information form are described in the table below.

Table 9-6 Loading Information Form Settings (Loading Type = Sine on Random)

Table Entry	Option	Description	
Results Type		There are two basic FE result types that dtfat can accept for vibration fatigue. The Loading Information form updates itself when this parameter is changed. The additional information needed when using one of these result types is described later in this section.	**Transfer Function**
	Transfer Function	FE results used in the vibration analysis is frequency response.	
	Modal Stress	FE results used in the vibration analysis is modal stress and complex modal participation factors.	
Job Setup	**Single**	This is currently the only option, since Duty Cycle is not currently supported for vibration fatigue.	**Single**
Results Transformation		This option is generally obsolete, since FE results are transformed to a consistent coordinate system before nodal averaging.	**No Transformation**
	No Transformation	No transformation of FE results is performed. DTLib assumes that FE results for vibration fatigue are 2-D, and thus only considers the Sxx, Syy, and Sxy components of the results tensor. This option should be used for vibration fatigue analysis.	
	Transform to Basic	FE results are transformed to the basic coordinate system.	
Load Input	**Single**	This is the only option for **Loading Type** = **Sine on Random**	**Single**
Use Static Offset	**False, True**	The FE results from a static load case are used to model fastener or other assembly-type loads. Use the **Select Static Offset** button to select the appropriate FE results case.	**False**

dtfat Batch Operation

The DTLib-based solver has the capability to output Response PSD, damage histogram, and stress range histograms. This can be done by running the solver from the command line, and using the appropriate command line options. Since this can result in a large amount of data, it is recommended that this optional output is only performed for a few entities of interest. The entities can be selected via the command line option `/ENTIDS` or using hot-spot filtering techniques.

In order to produce the output data, the /VIBOUT command line option is required. The command line option /OUTTXT can be used to switch between text-based output (i.e., csv file) and binary output (i.e., hst file). Note the /FERES=PDA command line option must be used. An example command line is as follows:

```
dtfat /FERES=PDA /VIBOUT=Y /ENTIDS=71 /INP= jobname.fin
```

The following text-based response psd, damage histogram, and stress range histogram data files are created for each entity:

jobname_1_.xmh

jobname_1_.xmh

jobname_psd_1_01.csv

Using the /OUTTXT=N command line option, binary data files are produced. The hst files can be viewed using **mp3d**. MSC Fatigue currently does not have a viewer for the s3m file.

```
dtfat /FERES=PDA /VIBOUT=Y /OUTTXT=N /ENTIDS=71 /INP= jobname.fin
```

The following text-based response psd, damage histogram, and stress range histogram data files are created for each entity:

jobname_psd_1_.s3m

jobname_damage_1_01.hst

jobname_rainflow_1_01.hst

Job Setup (Classic Modules)

When the Analysis switch is selected from MSC Fatigue/Main Interface is selected from the Tools pull-down menu from Patran, the following form appears.

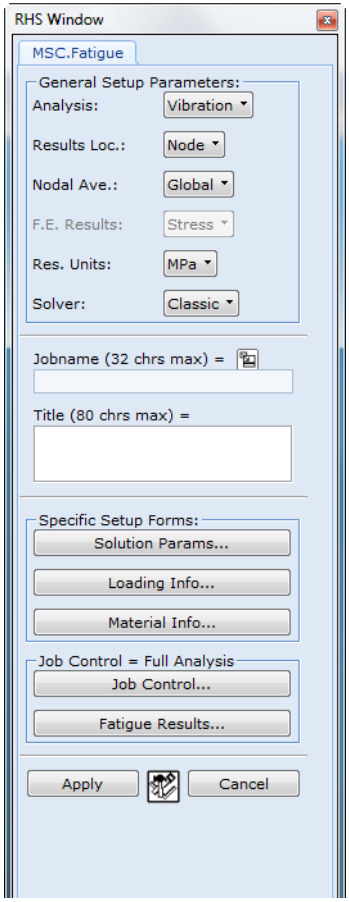
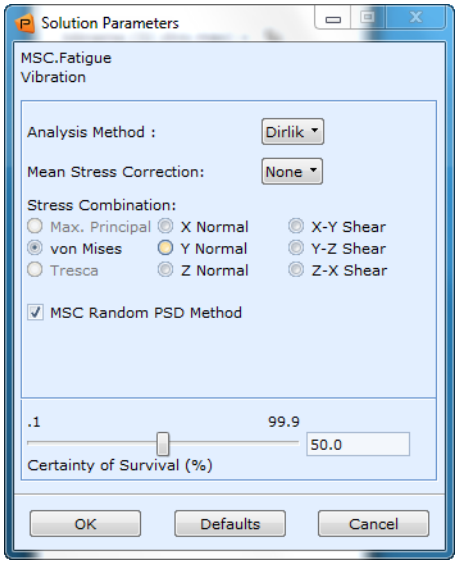


Figure 9-2 The Job Setup Form for Vibration Fatigue

Solution Parameters

The following forms appear when invoking the Solution Parameters button on the main MSC Fatigue setup form for vibration fatigue. Note that not all the Stress Combination parameters may be available for vibration fatigue.



The following table describes each parameter in detail:

Parameter	Description
Analysis Method	Dirlik is the default because it is more generally applicable to the widest range of problems, especially where frequency content is not narrow band. Other choices are Narrow Band or All. "All" will calculate lives based on all analysis methods. These analysis methods of calculating fatigue lives from response PSDs are described in Frequency Domain Approaches of Life Estimation, 749.
Mean Stress Correction	Acceptable values of mean stress are *Goodman, Gerber*, or *None.* These mean stress correction methods are described in detail in Fatigue Theory (Ch. 16). Although one of the above must be selected, sensitivity study allows the comparison of results using all of these correction methods. Goodman and Gerber are two methods of correcting S-N curves for mean stress that results for tensile mean stresses mostly fall between, with Goodman being the most conservative. (For compressive mean stresses, both methods as applied in MSC Fatigue may be non-conservative. It may be better to chose None.) **NOTE:** The mean stress corrections do not affect the analysis since there is currently no method to input an offset stress.
Stress Combination	This option menu selects the stress parameter used in the fatigue analysis. The effectiveness of the selection is dependent on the FE results being feed into the fatigue analysis. For frequency response, the six multiaxial component stresses defined by the stress tensor can be resolved into one uniaxial or combined value for fatigue calculations at each node. These stress scalar combinations can be either one of these components, *X Normal*, *Y Normal*, *Z Normal*, *X-Y Shear*, *Y-Z Shear*, *Z-X Shear*, *Max. Abs. Principal*, *von Mises*, Tresca*. These are valid when using the Classic solver (FEVIB). If response PSDs are being fed into the system, you should select one of the components only. Even though the six component values are present, no resolution into any other values can be achieved when dealing directly with a response PSD, therefore the seven combination methods (Max. Abs. Principal, Max. Principal, Min. Principal, Signed von Mises, von Mises, Signed Max. Shear, or Max. Sheared/Tresca) would be inappropriate. By default the X-component will be selected if an inappropriate selection has been made.
Certainty of Survival	This defines the *Certainty of Survival* based on the scatter of the S-N curve. For example, to be 96% certain that the life will be achieved, set the slider bar at 96. This value is used to modify the S-N curve according to the standard error scatter parameter (SE). The design criterion parameter will be meaningless if the value of SE is 0. A Design Criterion value of 50 leaves the S-N curve unmodified.

*Note that there are two mechanism for resolving to the von Mises stress parameter, which can give slightly different answers. The most accepted method is to use the MSC Random PSD Method, which resolves the transfer function streses to von Mises stresses identical tothe method used by MSC Random. This also can affect the individual components and is the recommended method.

Materials Information Form

By selecting the Material Information button located on the main MSC Fatigue setup form, a Materials form will appear. This form differs for the various fatigue analysis types. For vibration fatigue this form and its inputs are identical to that of a crack initiation or basic S-N analysis. You are referred to that discussion in S-N Material Parameters, 64.

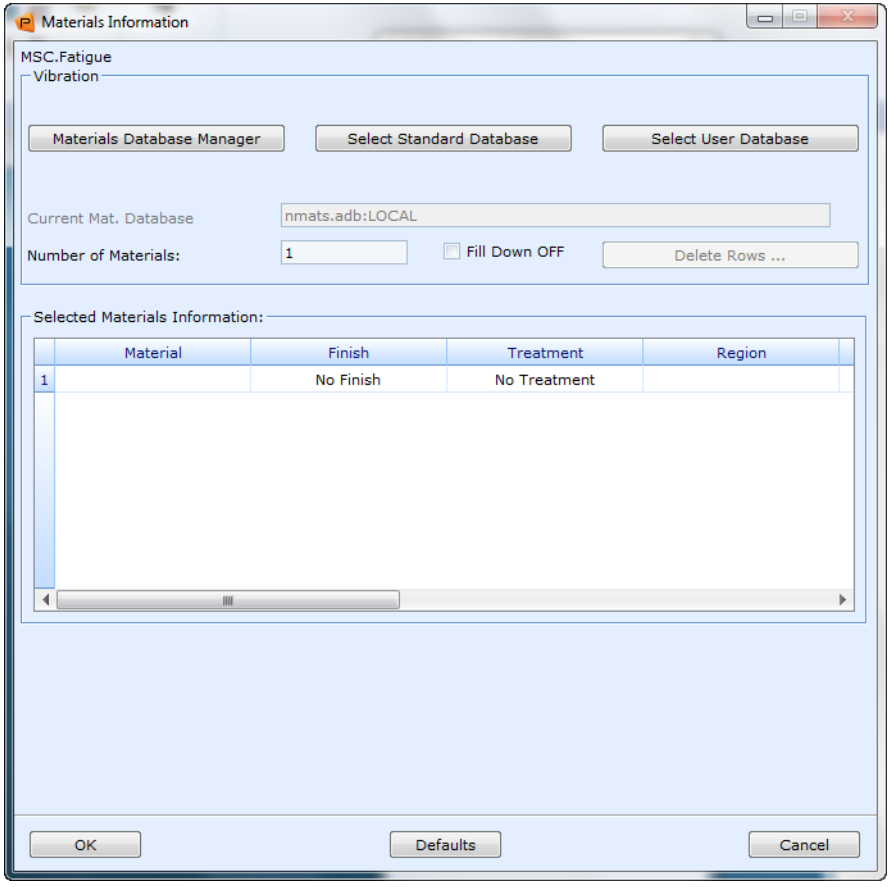


Figure 9-3 Material Parameter Form

Loading Information Form

By selecting the *Loading Information* button located on the main MSC Fatigue setup form, the Loading form will appear. Each aspect of this form is discussed in detail in this section. The form is divided into three basic parts: general results parameters, loading setup (spreadsheet), and the input selection area.

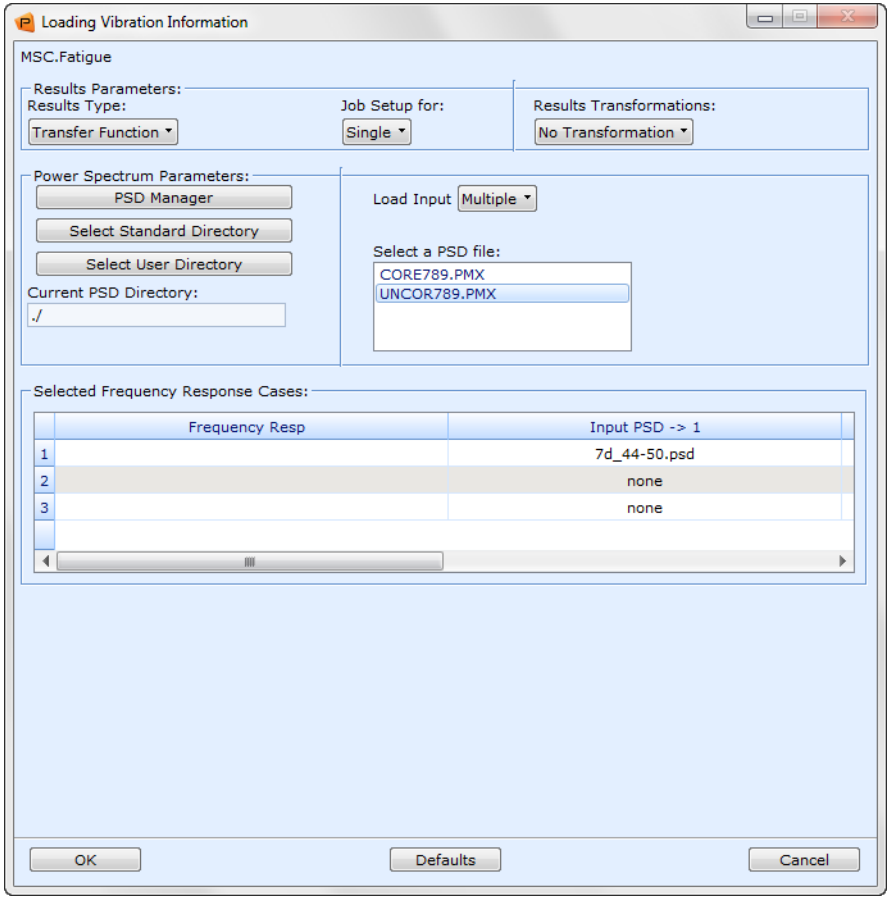


Figure 9-4 The Loading Information Form

PSD Database Manager

About a third the way down the form and if the Result Type is Transfer Function, the user may access MSC Fatigue's PSD Loading Database Manager (PTIME) by clicking on the PSD Manager button with the mouse. This option initiates a separate program in the MSC Fatigue system. The program may also be started from the operating system prompt by typing the symbol **ptime**. The program operates interactively.

The detailed operation of PTIME is described in Loading Management (Ch. 5). PTIME manages a local database containing details of the loading time histories, rainflow matrices, and input PSDs in the local directory. It enables the user to manipulate the PSDs in order to prepare them for use during fatigue analysis. PTIME creates two files in the local user directory called `ptime.tdb` and `ptime.adb`. In most cases, these must exist for MSC Fatigue to operate successfully. The one exception to this is when the user uses finite element results from a *random vibration* analysis as opposed to a *frequency response* analysis in which case no external input loading PSD data is necessary.

The two additional buttons at the same level on the form as the PSD Manager button allow for selection of the database directory. The Select Standard Directory button will select the local working directory as the location of the time history database (`ptime.adb`). The hierarchy is to first search for this standard database in the user's local directory, then the user's home directory, and then finally in the central installation directory.

The Select User Directory button will select a specified user directory where a loading database file (`ptime.adb`) may exist.

Anytime a listbox is accessed to reveal available PSD files or matrices for selection, the listbox will be filled with the contents of the loading database from the selected directory. This enables users to keep libraries of loading PSDs in separate locations and allows easy access to them without having to copy the needed files to the local working directory.

General Results Parameters

General results parameters that are set on the Loading Information form are described in the table below.

Parameter	Description
Results Type	There are two basic result types that MSC Fatigue can accept for vibration fatigue. These are Transfer Function (Frequency Response) Analysis Results, 656 and Power Spectum (Random Vibration) Analysis Results, 659. The Loading Information form updates itself when this parameter is changed. The additional information needed when using one of these result types is described later in this section.
Results Transformations	This option is available when the results are Transfer Functions only. Also, this option is only applicable when doing a nodal-based fatigue calculation using FE results which are associated with elements. Often times elemental results from FE codes are output in the element coordinate systems. In order to properly calculate averaged nodal stresses from elemental FE results, it is necessary to transform them from their elemental systems to the basic coordinate system. The default is *No Transformation* in which the user must take responsibility that the elemental results are all in the same coordinate system, otherwise the choice is to *Transform to Basic* in which it is ensured that all results are in the basic coordinate system before any nodal averaging takes place.

In all cases the FE results must be stored in the database. See the general discussion in Database Results, 75.

Results stored and subsequently extracted from the Patran database are translated and then written to the analysis input file (`jobname.fes`) in the following manner:

Transfer Function (Frequency Response) Analysis Results

The difference between frequency response and random vibration with respect to fatigue analysis is very similar to the differences between using static and transient results for a time domain approach. For static results, you must identify separate load cases and associate a time variation of the loading through external means (a time history created by PTIME) for each load from which the combined stress response is determined. This is then fed into the fatigue analysis as input. For transient analysis, the time variation of the loading is built into the FE analysis with the end result being a stress time response as input to the fatigue analysis.

For frequency response analysis, which produces transfer functions, it is necessary to associate an input loading PSD to each transfer function which is very similar to the static case associated to a time history.

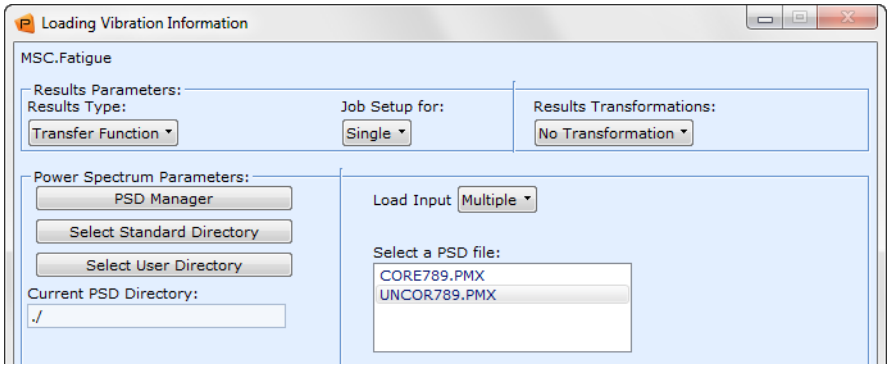


Figure 9-5 Loading Information Form for Transfer Functions

Three scenarios can exist:

- Single input loading
- Multiple input loading - uncorrelated
- Multiple input loading - correlated

Single Load Input:

- Set the Load Input pulldown to Single. A single row in the spreadsheet will appear. See Figure 9-6.
- Select the cell in the spreadsheet in the Frequency Resp column. This will activate the Results Parameters widgets at the bottom of the form.
- Select the transfer function (result case), the stress tensor, and the layer and then press the Fill Cell button. If these widgets are empty you need to retrieve the results from the database using the Get/Filter Results button. See Getting and Filtering Database Results, 659.
- The Input PSD cell will then become active and the widgets at the bottom of the form will update allowing you 0to select an input PSD created in PTIME from a listbox.
- Close the form when you are done.

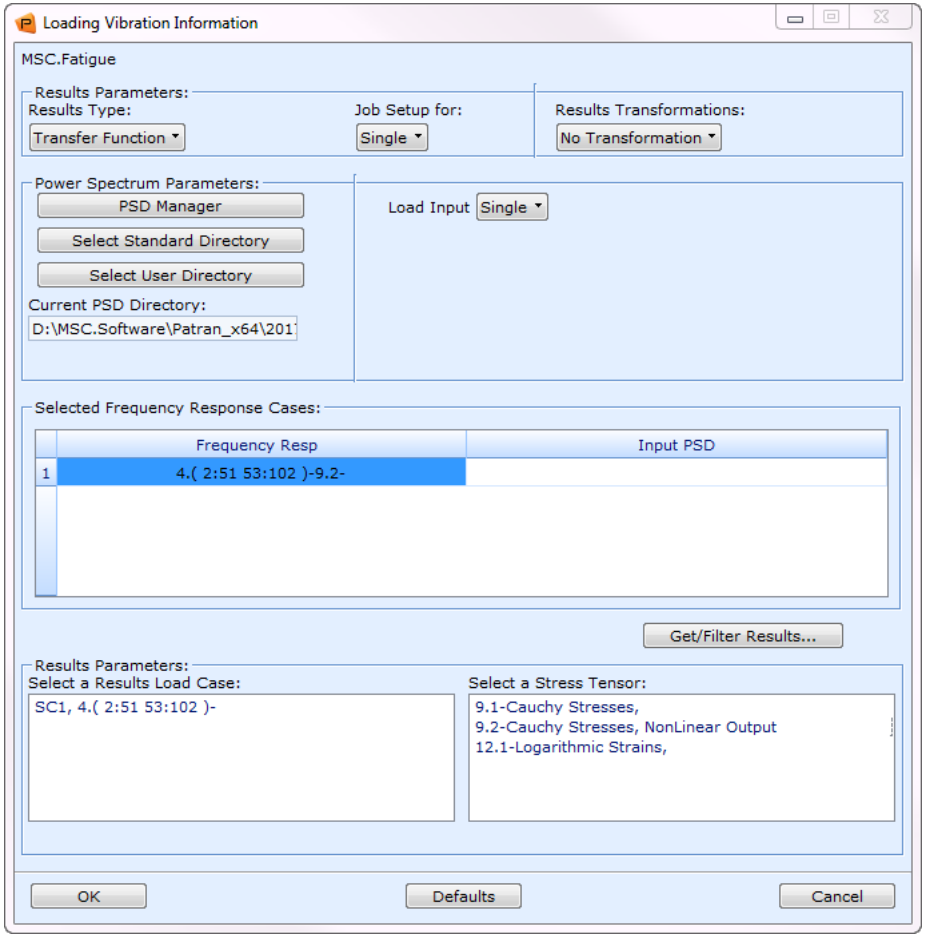


Figure 9-6 Loading Information Form for Transfer Functions (Continued)

Multiple Load Input:

- Set the Load Input pulldown to Multiple.
- Select a PSD matrix file from the listbox below the Load Input pulldown. This file contains a listing of PSD inputs (uncorrelated) and their cross terms (correlated) as defined by the PTIME program. See Rainflow Matrix, 234.
- The number of load inputs is determined by the size of the matrix. A four by four will create four rows in the spread sheet and will automatically fill in the Input PSD column with the diagonal terms of the matrix.
- You must select the cells (one at a time) under the Frequency Resp column to select the transfer functions (result cases), stress tensor and layer as described for the single load input in the same manner.
- Close the form when you are done.

The transfer functions are written to the `jobname.fes` fatigue input file as the six stress components for both the real and imaginary parts (that makes 12 total) for each frequency. This will be repeated for each loading input and at all locations.

Power Spectum (Random Vibration) Analysis Results

For a Random Vibration analysis, the form updates as shown. Only one basic piece of information is required and that is the result case IDs of the PSD as stored in the database. You must select the single cell in the spreadsheet and then select the PSD (result case), the stress tensor and a layer and press the Fill Cell button to fill in the cell. If no PSD results appear in the listboxes at the bottom of the screen then you need to retrieve the results from the database. See the next section on retrieving and filtering results.

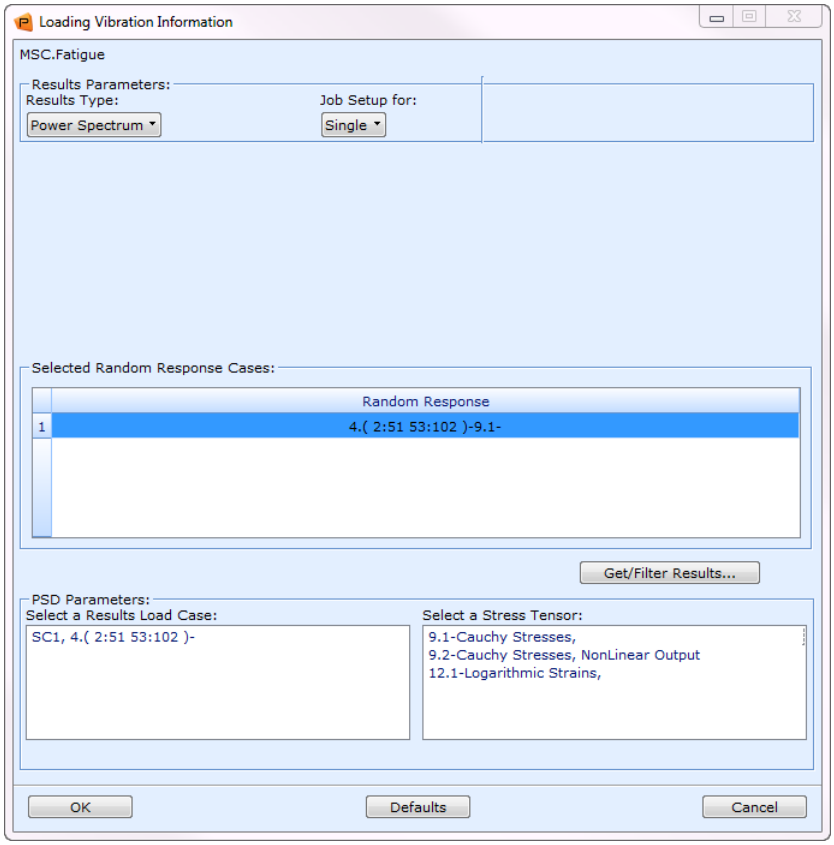


Figure 9-7 The Loading Information Form for Response PSDs

The response PSDs are written to the `jobname.fes` fatigue input file as the six stress components for each frequency. This will be repeated for all locations.

Getting and Filtering Database Results

When the Loading Information form is first presented, the listbox containing the database result cases or transfer functions appears empty. It is necessary to invoke the Get/Filter Results form in order to fill the

listbox with the relevant results. Depending on whether you are extracting transfer function results or result PSDs (power spectrum) will determine how the filtering mechanism works. In both cases the Get/Filter Results form appears as follows:

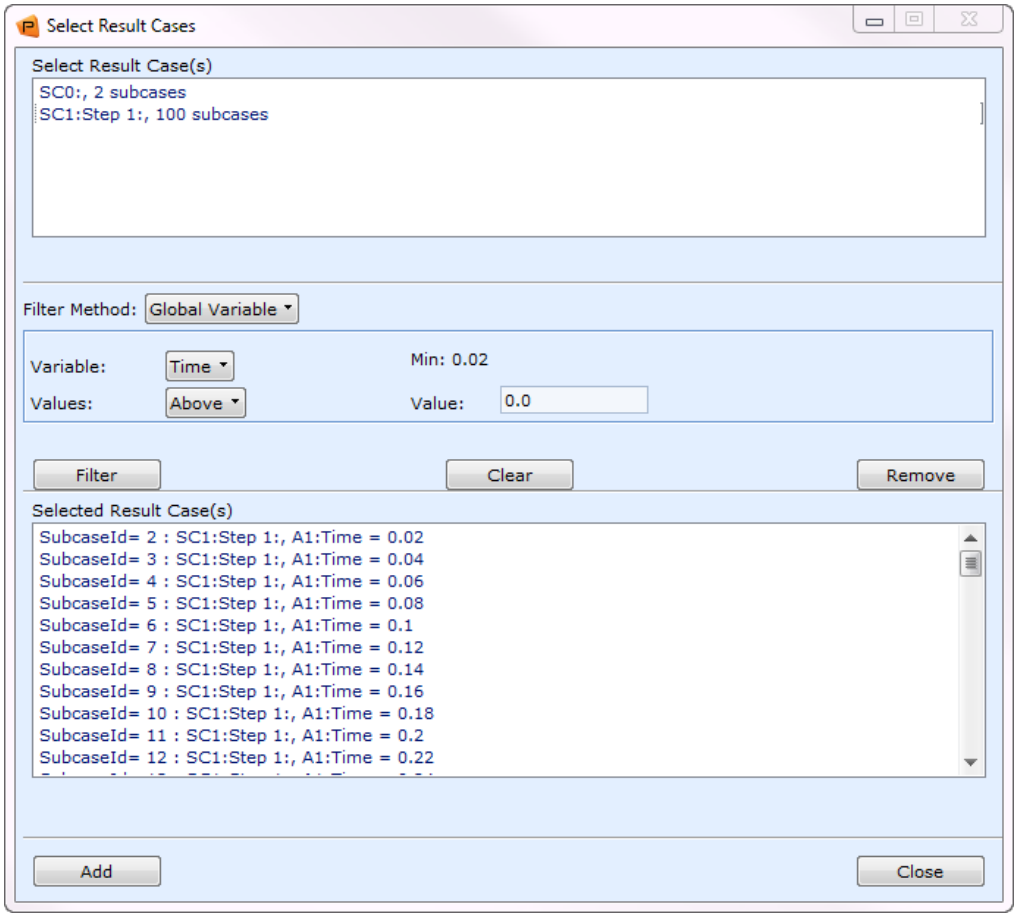


Figure 9-8 The Results Filter Form

This form is expandable (by dragging the corners or edges) to allow you to view the entire Result Case names and global variable if necessary. The filtering methods are described in detail below.

Table 9-7 Filter Methods

Method	Description
Global Variable	Any global variables associated with the selected Result Case will show up in the Variable option menu. Select the one you would like to filter with, change the criteria using the Value option menu and enter the value or range to filter by. Press the Filter button to complete the filter action. Press the Add button at the bottom of the form to activate the filtered subcase selection.
String	Enter a string and use wild cards (the * character) to filter results. For example, if you wanted all subcases with the string Time in it then you would use *Time* as the string with wild cards on each end of the word. Press the Add button at the bottom of the form to activate the filtered subcase selection.
Subcase IDs	Subcases can be filtered on Subcase IDs by entering the appropriate IDs. To select separate IDs, separate them by spaces (1 3 5). To select a range, use a colon between the numbers (1:5). To select by increments, use two colons, for example: 1:10:2, which interpreted means select subcases one through 10 by twos, or use any combination of spaces and colons between subcase IDs to select as many as you wish. Press the Add button at the bottom of the form to activate the filtered subcase selection.
All	No filter method is selected. No options are available. Simply press Filter and all subcases will be selected from whatever primary Result Case is selected. Press the Add button at the bottom of the form to activate the filtered subcase selection.

Transfer Function and Power Spectrum Result Cases

In both cases, filtering and extracting the results work in a similar fashion. The general use of this form to select the appropriate frequency steps for a fatigue analysis is as follows:

1. Select the Result Case. The top of the form lists all available Result Cases with the number of subcases (frequency steps) associated with the Result Case. These subcases can be time steps, load steps, frequency steps, design cases, or static subcases but only frequency makes sense in this case.
2. Set the Filter Method and Criteria.
3. Press the Filter button. To select all subcases of a particular Result Case, simply press the Filter button after the first step. The default filtering should allow for selection of all subcases.
4. Press the Add button to transfer the selected subcases (frequency steps) to the Loading Information form.
5. For transfer function results, repeat these steps for each transfer function load case corresponding to each loading input PSD.

Because this form treats all Result Cases and their subcases in a general way, it is up to the user to ensure that the results selected are truly from frequency response or random vibration analysis.

When the result cases appear in the listbox on the bottom right of the Loading information form, the result case IDs will appear collapsed, e.g. (2.112:205) which signifies that all the frequencies belonging to that particular transfer function are contained in the displayed result case.

Tips/Notes on Input PSDs vs Transfer Function (TF) Frequency Content

1. If the Input PSD has its first frequnecy defined as zero (0), the user is informed to fix this and the program returns to the main menu. MSC Random has the same requirement. This is so the program does not continue on and choke on the log interpolation.
2. If the Input PSD frequency range is not larger than the frequency range of the Transfer Function, the user is warned about this. The program continues and calculates fatigue life and the Output PSD, but the Output PSD is truncated above and/or below the Input PSD frequency range. To avoid this, the lowest Input PSD frequency should be less than or equal to the the TF's lowest frequency and the highest Input PSD frequency point should be greater than or equal to the TFs highest frequency point.

Job Control

By selecting the *Job Control* button located on the main MSC Fatigue setup form, a Job Control form will appear. This form is generic for any of the fatigue analysis types. The form allows for job submission and monitoring as well as other functions explained below. The form updates itself depending on the action required. In all cases except when reading an old job setup, the action is linked to the jobname entered in the main MSC Fatigue setup form. This simple form appears as:

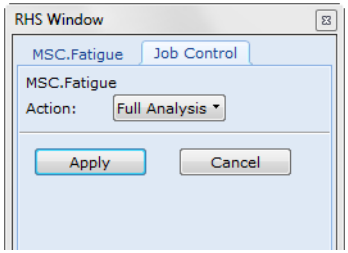


Figure 9-9 The Job Control Form

The actions that can be invoked from this form include Full Analysis, 663, Translate Only, 664, Monitor Job, 665, Abort Job, 666, Delete Job, 666, Read Saved Job, 666, Interactive, 667 and Analysis Manager, 667. (The action that is invoked from this form uses the current jobname on the main setup form. Click on Apply to invoke the action.)

Full Analysis

When the action is set to *Full Analysis* and the user clicks the Apply button the following occurs:

- The job begins the submission process by checking to see if an existing job of the same name exists. If it does, overwrite permission will be requested.
- All of the information requested in the main setup form and the subordinate solution parameter, materials, and loading information forms are written to a MSC Fatigue input file called `jobname.fin` and includes 90% of the fatigue input parameters. It is an ASCII unformatted file whose text lines consist of "Parameter = Value". The other 10% of the fatigue input information is retained in the database and consists of the region application information (nodes or elements) for the material and surface finish/treatment combinations and, in most cases, the FE results. If any information is not complete, the user will be notified and the submission process will terminate.
- Information is extracted from the database such as the region or group data and results via the PAT3FAT or FATTRANS translator. MSC Fatigue or Patran is suspended while the translation is in progress. A `jobname.fes` file results from this translation. This is the fatigue analysis input file and is binary in nature. It can be translated to ASCII form and edited if desired.
- A Linux shell script, , 383, is invoked from which the actual fatigue analysis begins. This controls the execution of the FEVIB module. This consists of reading the `jobname.fes` file, multiplication of the of load input PSDs by the transfer functions, rainflow cycle counting where applicable, and the actual fatigue calculations. The results of this operation is a file called `jobname.fef`.

- If MSC's Analysis Manager is installed and licensed, the job as described above will be submitted via the Analysis Manager as opposed to a Linux shell script (although the script is still executed). It is important that this module be configured properly for proper execution. See Analysis Manager, 667 for details.

When the job has been completed the results can be read into the database under the Results form. See Postprocessing Results, 667.

The following files are generated during a Vibration Fatigue analysis:

Filename	Description
***jobname*.fin**	Fatigue job parameter data.
***jobname*.fes**	FE stress and fatigue input file.
***jobname*.fef**	Fatigue results file.
***jobname*.msg**	Message file.
***jobname*.sta**	Job status file.

Translate Only

This option operates as described in Full Analysis, but stops after the creation of the `jobname.fes` file.

Save Job Only

This option operates as describe in Full Analysis, but stops after the creation of the `jobname.fin` file.

Monitor Job

When the action is *Monitor Job*, the current job status will be reported in the status box. This is achieved by examining the contents of the `jobname.sta` file which is updated at various points during the analysis. The `jobname.msg` file will contain a history of the job (i.e., all the messages generated while running the current MSC Fatigue session). Continue clicking the Apply button for an updated current report.

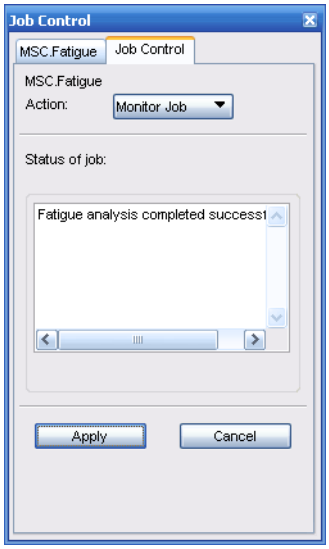


Figure 9-10 Monitoring a Job

When a job is submitted it will pass through a number of phases. The user will be informed through the status option of the progress of the job. Both success and error messages are displayed. Normal operation messages which the user may experience include the preprocessing % completed, the successful completion of the preprocessor, the % completed of the analysis, and the successful completion of the analysis. Not all the messages will be displayed since the status file is updated very quickly in some cases. In certain cases, the status file may not be available in which case a "Try again" message will appear. When execution is through MSC's Analysis Manager, these messages appear in the Analysis Manager message window.

In addition, there may be other messages giving status of other aspects of the job such as Factor of Safety analysis or Crack Growth analysis. Error messages are also displayed via these status messages.

If the status message does not appear to be updating, it is possible that the job has halted due to an error. In many cases, that error message will be reported through the status facility. However, if it is not reported, you can investigate the problem by opening another window and examining the following file:

jobname.msg: This file will contain all the status messages for the job including any error messages. See Error Messages (App. 19) for a description of error messages and possible solutions.

Some hints on determining why a job has failed:

1. If the `jobname.fin` files exist in your directory, try running the job interactively by typing:

   `pat3fat` `jobname`

then checking the message file.

2. If the `jobname.fes` file exists, run the FEVIB program interactively and watch for error messages. Type **fevib** at the system prompt.

Important:	If a job inadvertently quits, sometimes a `jobname.fpr` file is left in the directory. This file is created during submission to detect a running job so that inadvertent submissions while a job is in progress of the same jobname are detected. In some cases, it may be necessary to remove this file before re-submitting the job.

Abort Job

If the action is set to *Abort Job*, the job will be aborted by clicking the Apply button. This is achieved by creating an empty `jobname.abo` file in the current directory. The MSC Fatigue modules periodically check for this file and when detected, will abort.

Likewise, a `jobname.abo` file can be created or the `jobname.sta` file renamed to `jobname.abo` from the operating system prompt and the same result will be achieved when a job is running.

If execution is via MSC's Analysis Manager, then the Analysis Manager will handle the abortion of the job. All files will automatically be cleaned up by the Analysis Manager.

Delete Job

If the action is set to *Delete Job*, the various files associated with the job will be deleted by selecting the Apply button. The files that will be deleted if encountered are:

Filename	Description
***jobname*.fin**	Fatigue job parameter data.
***jobname*.fes**	FE stress and fatigue input file.
***jobname*.fef**	Fatigue results file.
***jobname*.msg/log**	Message and log files.
***jobname*.sta**	Job status file.
***jobname*.abo**	Abort detection file.

Other files may also be deleted if associated with the particular job. Use this option with caution.

Read Saved Job

When the action is *Read Saved Job*, a databox will appear. The user can enter the jobname here or select the "Select .fin file" button. This button will bring up a file select form that lists the available jobs from the local directory. By selecting one of the existing jobs and clicking on the **OK** button, the jobname databox will be automatically filled in. When the Apply button is selected, the reading of the `jobname.fin` file will initiate. If this file is successfully read, the widgets and parameters in the main MSC Fatigue setup form and the subordinate solution parameter, materials, and loading forms will be updated.

If the read is unsuccessful or if the `jobname.fin` file is not complete, some of the parameters will not be correct. It is always advisable to check to see if the parameters have been updated properly before attempting to submit the analysis.

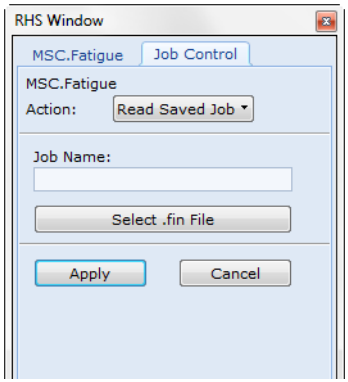


Figure 9-11 Retrieving an Old Job

If you read an old job in, it is assumed that the material/surface treatment-finish combination groups that may be called out in `.fin` file actually exist in the database. If the groups do not exist, they will have to be recreated before a submission can be successful.

Important:	If the name of the job is known beforehand, it is possible to type the name of the job in the Jobname databox on the main MSC Fatigue form and press the <RETURN> key to read a saved job. If the job exists, permission to read will be asked.

Interactive

By clicking Apply with the Action set to *Interactive* will cause a separate MSC Fatigue module to be invoked called FEVIB. This module can be invoked at any time. However, to perform any fatigue analysis, the analysis steps must have been performed up through the translation stage. With the exception of the basic utilities in FEVIB, the existence of a `jobname.fes` file is a minimum requirement.

For operation of this MSC Fatigue module, see FE Vibration Fatigue Analysis (FEVIB), 670.

Analysis Manager

This option is explained in Analysis Manager, 96.

Postprocessing Results

By selecting the *Results* button located on the main MSC Fatigue setup form, a Results form will appear. Each of the actions on the Results form is explained in detail in the following sections.

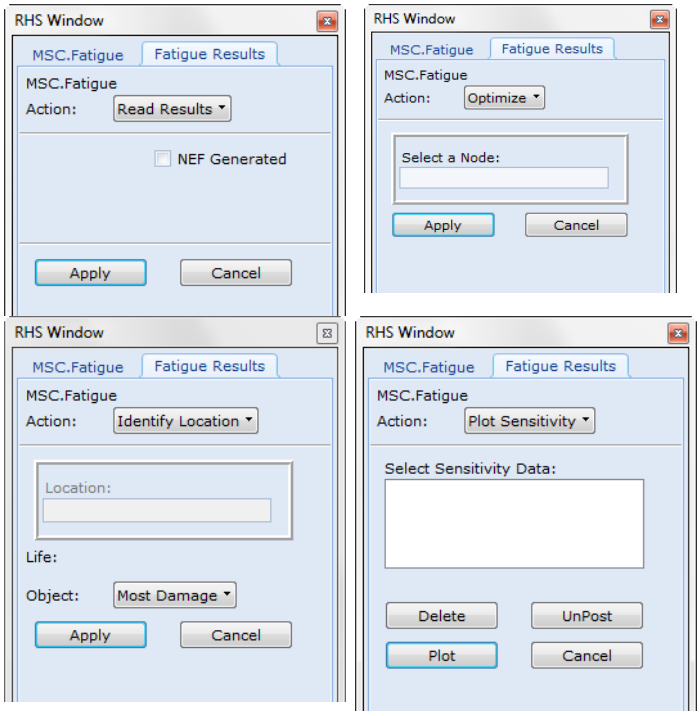


Figure 9-12 Postprocessing Options

Read Results

With the Action set to *Read Results* on the Results form, the MSC Fatigue analysis results can be read into the database. If the same database being used contains finite element results, then the database will now have fatigue results and finite element results. Each fatigue job is known as a Results Load Case under the Results application. The only things that differentiate them from a FEM load case are their results titles. This section will explain how to effectively postprocess fatigue results including:

- Displaying Fringe plots, writing reports, and X-Y plots (see Fringe Plots, Text Reports, XY Plots, 102).
- Setting color Spectrums (see Using Spectrums and Ranges, 105). When results are imported, a new spectrum is created for you called `fatigue_spectrum`. This spectrum is simply the `standard_spectrum` color scheme in reverse so that the short fatigue lives can be displayed in red (hot) and long fatigue lives in blue (cold). This is opposite of the way stress results are displayed with the `standard_spectrum`. It is your responsibility to set the spectrum to the one that you want.
- Displaying result Values. When results are imported, a new range is also created for you in log increments called log_range. Some values such as Damage of Life are hard to visualize with a standard range of evenly spaced ranges. A log range helps spread the color bands out on results that may range from a small number to a very large (infinite life) number. Again you are responsible for selecting the range that best fits the displayed results.

Explanation of results that can be processed:

Parameter	Description
Irregularity Factor	This is the irregularity factor parameter which is defined as the number of upward crossings over the number of peaks. The value may take on any number between zero and one.
Root Mean Square	The rms value.
Damage (per Second)	Damage - the reciprocal of life.
Life (seconds)	Life value in seconds.
Log of Damage	Log value of damage. The interpretation of which is 10^{value}.
Log of Life	Log value of life.
Angle Range	This is the maximum range of the angle (in radians) between the principal and the local x-axis as used for a stationarity check.

List Results

If the Action is set to *List Results* on the Results form, the PFPOST will be spawned. A discussion of PFPOST can be found in Reviewing Results (PFPOST), 363.

This module allows easy access to important tabular results such as the most damaged nodes or elements, irregularity factors and other damage parameters. The tabular listing of the results is explained in Results Postprocessing, 692.

Re-Analyze

If the Action is set to *Re-Analyze* on the Results form, the option executes the external MSC Fatigue module, FEVIB and puts the user in interactive mode for a global vibration fatigue analysis. This program may also be started from the operating system prompt by typing the symbol **fevib**. The detailed operations of FEVIB for reanalyze is described in Global Vibration Fatigue Analysis, 671.

Design Optimization

If the Action is set to *Optimize* on the Results form, the option executes the external MSC Fatigue module, FEVIB and places the user directly in the Design Optimization option. This program may also be started from the operating system prompt by typing the symbol **fevib**. The detailed operations of FEVIB for optimize is described in Design Optimization, 674.

Sensitivity Plot

If the action is set to *Plot Sensitivity* on the Results form, the user can select the XY Plot they wish to view.

Use the Delete, UnPost, and Post buttons to delete, unpost, or plot the xy data, respectively. The difference between delete and unpost is that delete removes all curve information and the plot from the viewport and the database, while unpost just hides the viewport but the information in the database is retained.

Extract PSD

If the Action is set to *Extract PSD* on the Results form, the option executes the external MSC Fatigue module, FEVIB and places the user directly in the Output Power Spectrum option. This program may also be started from the operating system prompt by typing the symbol **fevib**. The detailed operations of FEVIB and how to extract the combined stress response PSDs are described in Output Power Spectrum, 690.

Identify Location

If the Action is set to *Identify Location* on the Results form, the user has the choice of identifying the location with the Most Damage, the Worst Safety Factor, the Most Biaxiality, the Least Prop. Loading, or the Largest Mobility.

When the Apply button is clicked, an arrow pointing to the location and a circle will be drawn to indicate the node of interest. If the location is an element it will be highlighted. The corresponding value will be place on the graphics screen and also in the databox on the form.

If no results exist for the particular quantity, a message to this effect will be displayed. Keep in mind that the results must be imported into the database and the jobname corresponding to the results must be in the Jobname databox on the main form. Also, to find the correct results, the program keys off of the Analysis Type. So be sure that the Analysis Type is set correctly on the main form, otherwise it will look for the wrong analysis type results. If the same results have been imported more than once, only the first will be detected. It is a good idea to delete previous results datasets from the database before importing new results if this feature is to be used. that is to say, results of the same type with the same jobname.

FE Vibration Fatigue Analysis (FEVIB)

The MSC Fatigue analysis module FEVIB performs many different tasks from simple file conversions to full blown vibration fatigue analysis. Module operation of each of these tasks is described in detail in this section.

FEVIB handles all the processing, data file import and export or conversion, the actual S-N fatigue analysis and all post-processing and design optimization activities. In many ways, it is very similar to the standard fatigue analysis module FEFAT described in Total Life and Crack Initiation (Ch. 6).

The operation of FEVIB can be in one of two modes: within the MSC Fatigue and Patran environments or in stand-alone mode from the system prompt. The only difference is that in stand alone mode, the user must supply the jobname when asked to perform the analysis. (In direct mode from a preprocessor such as Patran, these are passed to FEVIB automatically.) FEVIB can be accessed directly from the operating system prompt by typing the symbol **fevib**. Once FEVIB has been initiated in either of these modes, two windows will be presented. The top, small form is a generic form and allows for general program control. This is discussed in detail in Module Operations (Ch. 18) for the Motif driver.

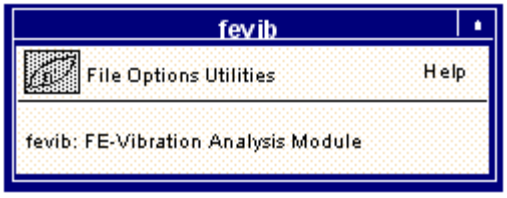


Figure 9-13 FEVIB Utility Form

The main menu appears as follows. Each item is discussed in this section.

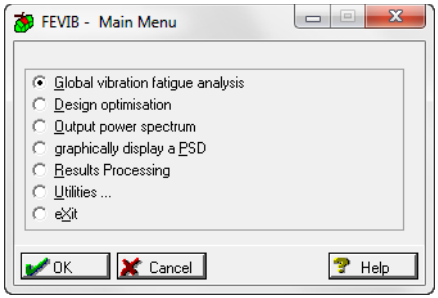


Figure 9-14 FEVIB Main Menu Form

Global Vibration Fatigue Analysis

A standard S-N or crack initiation fatigue analysis is split into two parts: the pre-processing (cycle counting) and the analysis. For vibration fatigue analysis, these processes are integral. A rainflow cycle count and the actual fatigue life are the results of a vibration fatigue analysis only when the Dirlik + mean method is used. Other methods only produce a fatigue life. Therefore, when running the global multi-node/element analysis, FEVIB will always run the complete analysis without stopping after a preprocessing stage, unlike FEFAT.

Important:	Remember that the fatigue analysis may take some time so it may be beneficial to ensure that the terminal is available for a long interactive session. It may be worth considering operating FEVIB in batch. Batch operation of these programs is discussed in FEVIB Batch Operation, 693. Typically though, vibration fatigue analyses are faster than standard time-based fatigue analyses.

When the Global option has been selected, the user will be presented with a number of questions. The first question asks for the input file name. Click the **OK** button once a file name (`jobname.fes`) has been selected. Use the List button to list all available fatigue input files. These files have been created by the PAT3FAT or FATTRANS translator. The default will be the last `jobname.fes` created. Once a valid file name has been entered, the user will be presented with a summary of the `jobname.fes` file that has been opened. Each of these parameters can be changed or edited.

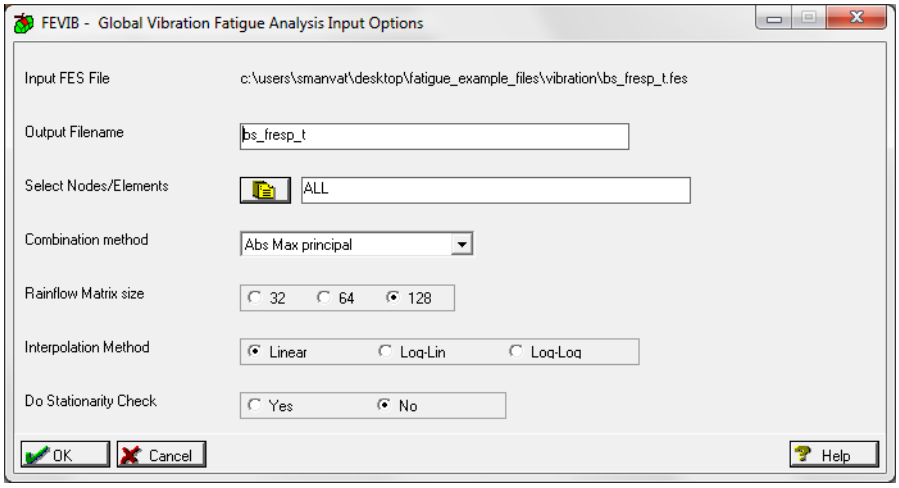


Figure 9-15 FEFAT Preprocessor Form

The following table explains each entry on the previous form:

Field	Description
Input FES File	This is the fatigue input file (`jobname.fes`) to be used in the fatigue pre-processing. The job must have already run at least through the PAT3FAT or FATTRANS translator to produce a `jobname.fes` file. This is achieved by carrying out a full or translate only submission from the job submit options in the MSC Fatigue menus, or by running PAT3FAT or FATTRANS in stand alone mode (see Job Setup (dtfat), 630). A fatigue input file can also be created using FEVIB's Utilities, 693. FEVIB accepts both frequency response and power spectrum types of jobs.
Output file name	The default is the jobname. After completion, a file called `jobname.fef` will exist. You will be requested to overwrite any existing output file of the same name if one exists. This output file can be read back into the pre- or postprocessor for result contouring (see Job Setup (dtfat), 630) or can be listed in tabular form (see Results Postprocessing, 692).
Select Nodes/Elements	This field accepts a list of nodes or elements or the word *ALL*. The job automatically detects whether a nodal or elemental-based analysis is to be performed from the `jobname.fes` file. The processing can be time consuming for many nodes or elements in an interactive mode but is generally quick relative to a typical time-based fatigue analysis. Nodes or element numbers may be specified in the following ways: 1,2,3,4 - or the equivalent - 1:4 - or `@pfatigue.ents` - which is an ASCII file containing a list of node or element id numbers. A complete list of all nodes/elements is available by pressing the List button.
Combination Method	For response PSD input you must select one of the six components. It is possible that one of the six actually represents something other than one of the components such as a maximum absolute principal. The `jobname.fes` file containing a response PSD from a FE analysis stores the six component values for each location for each frequency. The program makes no actual distinction as to what they actually are. If you select X-component it will simply use the first column of data of the 6 columns representing the 6 components. For transfer function input, your choices are: Abs. Max Principal, von Mises, Tresca/Shear. These values will be derived from the six components. How this is done is described in Theory section.

Field	Description
Rainflow Matrix Size	This sets the number of bins for cycle counting; more bins means a higher matrix resolution which will increase accuracy but also increase processing time and file sizes. The matrix size is remembered and any further jobs that are run in the same directory will use the size set here.
Interpolation Method	For the transfer function (frequency response) type input, it may be necessary to interpolate between frequency steps and/or actual power values. This options specifies how that interpolation should be performed: linearly on both axes, Log x and linear y, or log on both axes.
Do Stationarity Check	Selecting Yes provides a number of outputs which can be plotted. Two basic plots are generated: - a file defining the stationarity angle vs loadcase ID relationship (.LBP) - and for loadcases, a plot of frequency vs stationarity parameter (.EBP). This is presented as an error bar plot. All existing files will be overwritten and there is no .LBP file for single loadcase jobs. It is recommended that No is selected for a global analysis.

For an initial global analysis, you will be presented with a tabular listing of the top most damaged locations. For more result and postprocessing options, you can now proceed to Design Optimization, 674 or Results Postprocessing, 692.

Design Optimization

Having completed a global multi-node or multi-element analysis, the user will have identified an area of the structure that is either liable to fail at a life less than the design life or has such a long life that he wishes to explore manufacturing options using more cost effective methods and materials which will still achieve the target life. Alternatively, the user may already have some options for material selection or geometry detail which he wants to assess in terms of their effect on fatigue life.

The MSC Fatigue design optimization FEVIB provides a set of semi-automatic tools to assess fatigue design options. It is almost identical in its usage to the standard time-based fatigue module FEFAT. Unlike FEFAT, it does not begin with a rainflow histogram but rather, uses the same input file as the global analysis, the difference being that you are investigating a single location as opposed to the entire model (or a portion thereof). It supports a number of options including back calculation of parameter values which meet a target life, sensitivity studies on critical parameters, and an automatic material selection option based on fatigue criteria.

Having selected the node or element of interest, FEVIB will carry out a single fatigue calculation based on the default parameters from the global multi-node/ element analysis and present the results in more comprehensive form than that available in the global analysis. The design optimization analysis options are then presented on a main analysis page from which the user can set up the optimization calculations.

FEVIB presents its results in the form of analysis summary reports, 3-dimensional cycle histograms, and fatigue life sensitivity tables.

Stage 1: Module Operation

The operation of FEVIB from within the Patran environment is almost identical to its operation in stand alone mode. The only difference is that in stand-alone mode, the user must supply the jobname on entry to the program. (In direct mode from MSC Fatigue or Patran, this is passed to FEVIB automatically.)

The first screen to be presented when the program starts is shown in Figure 9-16. This screen will be skipped if FEVIB is entered from Patran and the screen in Figure 9-17 will be presented instead. This is because FEVIB already knows the jobname as specified from the MSC Fatigue menus. The node/element number will also be passed to FEVIB.

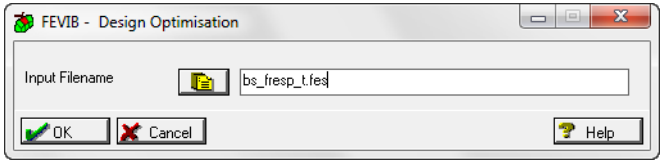


Figure 9-16 Design Optimization Job Entry Screen

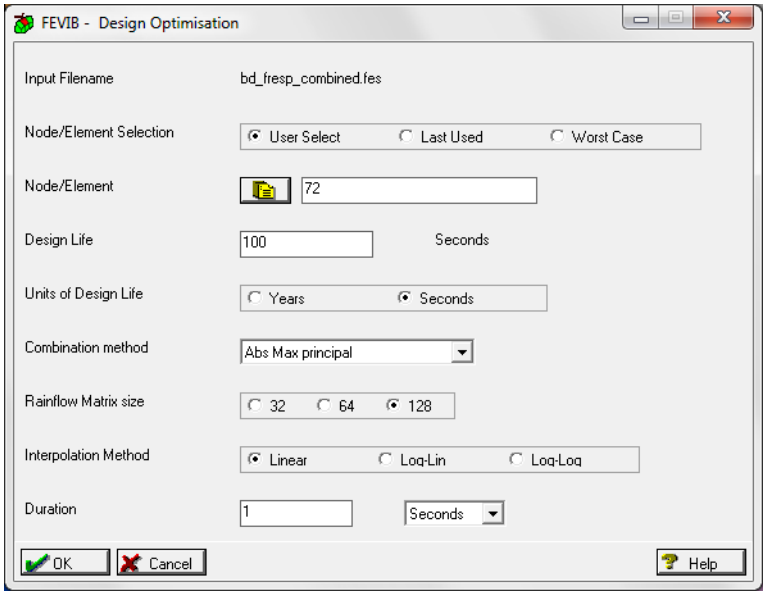


Figure 9-17 Design Optimization Job Entry Screen

The fields on these screens are described below.

Field	Description
Input File Name	This is the name of the job which is to be used in the fatigue design optimization analysis. The job must have already run at least through the translator stage using PAT3FAT or FATTRANS, to produce a `jobname.fes` file. This is achieved by carrying out a full analysis or translate only from the job submit options in the MSC Fatigue menus, or by running FEVIB utilities to create a simple input file. To select a jobname from a list of available jobs, use the List button. FEVIB accepts two types of FE input: response PSDs, and frequency response analysis (transfer functions). Once the jobname has been supplied, a new form will be displayed with the rest of the initial input options on it. These are described below.
Node/Element Selection	There are three options offered on this field: *Last Used* recalls the number of the node or element used in the last job. This number is shown in the Node/Element field. If the last job used a different geometry model, this option is unlikely to offer a meaningful node or element number. *User Select* allows for typing in a number in the Node/Element field shown below the Node/Element selection menu. A list of possible node or element numbers is available using the List button. The *Worst Case* node or element option is only available if a valid `jobname.fef` file exists. When this option is selected, the `jobname.fef` file is searched to find the node or element with the most damage as calculated by the global fatigue analysis. Once the critical node or element is found, its number is presented in the Node/Element field.
Node/Element	The number displayed in this field depends upon the choice made in the Node/Element selection described above. Use the List button to display a list of valid node or element numbers.
Design Life	The design life is a target life which is associated with the component or structure being analyzed. The life should be specified in the user units. These units are always either years or seconds. A design life MUST be entered here.
Units of Design Life	The units must be in years or seconds. Actually the units are always in seconds from a vibration fatigue analysis, but you may specify the design units in years for convenience.

Field	Description
Combination Method	For response PSD input you must select one of the six components. It is possible that one of the six actually represents something other than one of the components such as a maximum absolute principal. The `jobname.fes` file containing a response PSD from a FE analysis stores the six component values for each location for each frequency. The program makes no actual distinction as to what they actually are. If you select X-component it will simply use the first column of data of the 6 columns representing the 6 components. For transfer function input, your choices are: Abs. Max Principal, von Mises*, Tresca/Shear or one of the components. These values will be derived from the six components. How this is done is described in the Theory section (see Multiple Degree of Freedom Systems, 758).
Rainflow Matrix Size	This sets the number of bins for cycle counting; more bins means a higher matrix resolution which will increase accuracy but also increase processing time and file sizes. The matrix size is remembered and any further jobs that are run in the same directory will use the size set here.
Interpolation Method	For the transfer function (frequency response) type input, it may be necessary to interpolate between frequency steps and/or actual power values. This options specifies how that interpolation should be performed: linearly on both axes, Log x and linear y, or log on both axes.
Duration	The default data in the rainflow histogram is normalized to 1 second but may be rescaled to any duration of hours, minutes, or seconds. This field is used to define the total duration of the rainflow data. The value should be given as a real number greater than zero.

*Note that there are two methods that may be used. An environment variable called *PSDMTHD* can be set to *MSC Random* (actually it can be set to anything - it just needs to be set) to trigger the MSC Random method. This will affect the six components and the von Mises stress combination parameter (Abs. Max Principal and Tresca are not affected by this). The variable is set or removed by using the MSC Fatigue MENM environment variable utility. The variable can be set in the user's working directory, home directory, or globally in the MSC Fatigue installation. This modifies a file called envi.usr, envi.hom, or envi.sys respectively. See the Modifying the MSC Fatigue Environment (MENM), 1463.

When all fields are filled in appropriately, click the **OK** button. At this stage, FEVIB carries out an initial analysis using the original fatigue analysis parameters defined when the fatigue job was set up. The life computed from this "stage 1" analysis is used as a benchmark against which all subsequent optimization calculations can be judged.

The results from this analysis are presented in a summary table on the screen and also written to the `pfatigue.prt` file. See Figure 9-18.

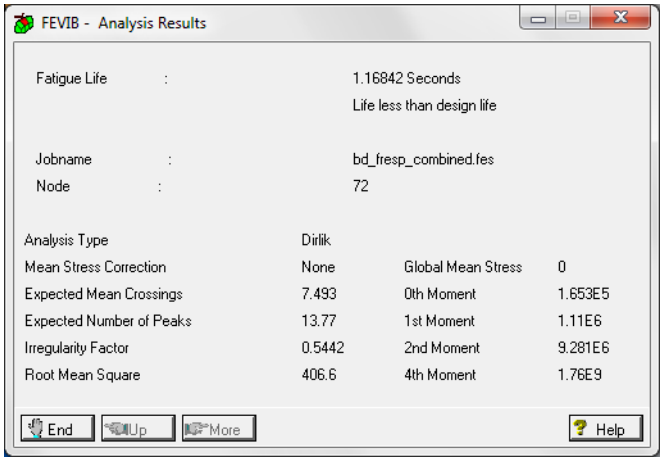


Figure 9-18 Results of the “stage 1" Fatigue Analysis for a Vibration Fatigue Job

If a design life has been defined, a message will be written under the life result indicating whether the design life has been met or not. The three possible messages are:

- Design life exceeded
- Life within a factor of 3 of the design life
- Life less than the design life

The other details presented summarize the analysis parameters. These details are specific to the type of job being carried out and not all parameters will always be present.

Some of the parameters are not defined in the global analysis such as other analysis methods, global offset stress, the Miner’s constant, and others. While these parameters are not available for editing at this stage, they are provided as analysis options in the design optimization input screens described later.

For the Dirlik + mean analysis option a cycle histogram file is created. The generic histogram name is made up from the jobname and the node or element number. This name is used for the cycles file which has the following naming conventions:

jobnamenn.cyh	Rainflow cycle histogram for node or element nn

Stage 2: Module Operation

After going through the initial reanalysis of a particular node or element, the main analysis screen is shown in Figure 9-19. From this menu, all the analysis options are available. The current jobname and node or element identity is shown at the top of the screen together with the design life. When a "back" or sensitivity analysis is defined, the type of analysis is reported in this area of the screen also.

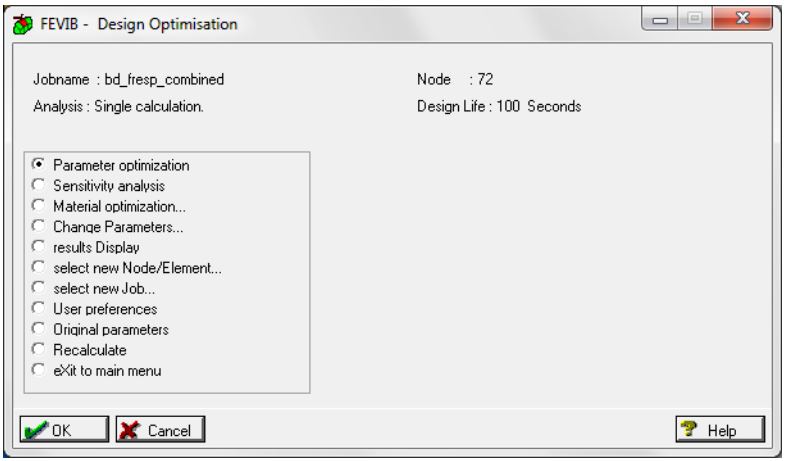


Figure 9-19 Fatigue Design Optimization Main Menu

To use this menu, choose the required option, set up the analysis parameters, and finally, when ready, select the Recalculate option to submit the analysis. A percentage complete message will inform the user of the progress of the calculations. A description of each menu pick follows.

Parameter Optimization

This option is the back calculation facility where a design life is supplied and FEVIB's automatic routines calculate the value of the chosen parameter that will achieve the target life; see Figure 9-20. There are three fatigue analysis parameters which may be used in this type of calculation. The parameters on which back calculation may be carried out are:

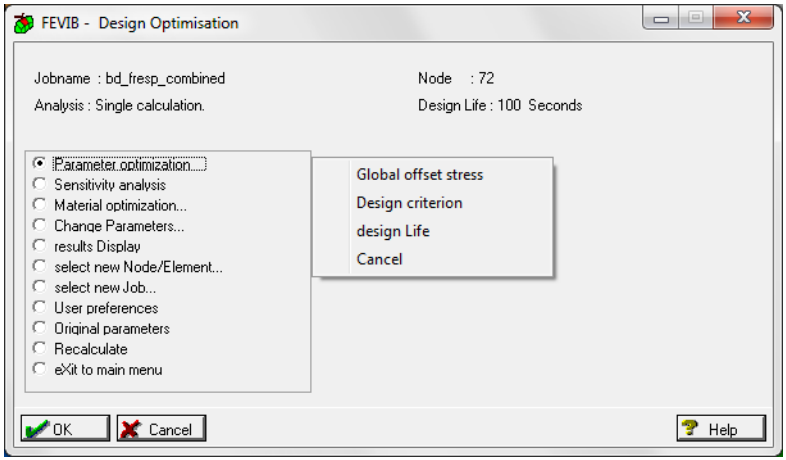


Figure 9-20 Parameter Optimization Submenu

Option	Description
Global Offset Stress	This back calculation parameter calculates the constant global stress value offset required to reach the specified design life. Once this parameter is calculated you are given the option to use it in subsequent sensitivity studies.
Design Criterion	This is the confidence of survival parameter which is based on the standard error of the S-N curves. Using this parameter will tell how much confidence the user can have in the product reaching the target life. However, the user should also consider the error in other parameters such as the stress computed in the FE analysis which may cause the life to be different from the estimate.
Design Life	This is not an optimization parameter but is used as a target for the optimization process. The design life may be changed or defined using this option. If a design life has not been indicated initially, the user will be prompted for one before being able to take advantage of any of the above back calculation options.

Having set up one of the optimization calculations, it is necessary to unset it in order to carry out any other kind of analysis. The easiest way to do this is to select the *Original parameters* option from the main menu. The other way is to select Change parameters followed by the parameter that was last set to back. The original default value will be offered and if accepted, the back calculation facility will be turned off. See also the section under User Preferences, 689 on Back calculation accuracy.

Sensitivity analysis

A sensitivity analysis allows the effect of variation in any of the input parameters on fatigue life to be explored, see Figure 9-21. For some parameters, all possible values are used in the sensitivity analysis. Parameters which fall into this category are:

- surface finish
- surface treatment
- mean stress correction method
- analysis method

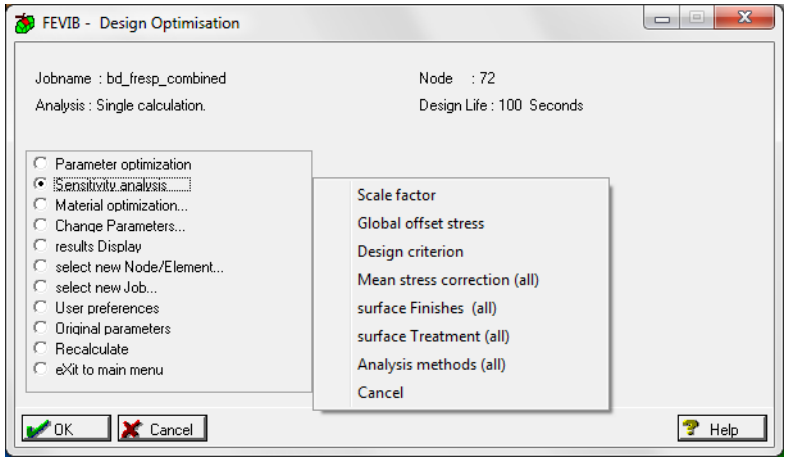


Figure 9-21 Sensitivity Analysis Submenu

To use one of these types of analysis, simply select the option followed by Recalculate on the main menu. For other parameters which are specified in a numerical form, the user is requested to enter a range of values for the chosen parameter. Parameters which fall into this category are:

- scale factor
- global offset stress
- design criterion

To select one of these types of analysis, simply select the option which will then present the user with a data input form at the bottom of the screen. In the box on this form, the user will be asked to provide a range of numbers for the parameter. Having done this, it is necessary to select the Recalculate option on the main menu.

Option	Description
Scale Factor	This factor is applied to the output stress PSD. The user may enter a range of values separated by spaces or commas. The user may also specify a range of values by inputting a start value, end value, and increment, i.e., (1, 10, 2). Note: This value needs to be a positive real number.
Global Offset Stress	This factor can be thought of as a stress offset. The user may enter a range of value separated by spaces or commas. The user may also specify a range of values by inputting a start value, end value, and increment, i.e., (1,10,2). A tabular display of the results will be displayed when the Recalculate button is pressed.
Design Criterion	This is the confidence of survival parameter which is based on the standard error of the S-N curve. Using this parameter will tell the user how much confidence he can have in the product reaching the target life. However, the user should also consider the error in other parameters such as scaling factor which may cause the life to be different from your estimate. The user may enter a single value in the input bar that appears at the bottom of the screen or a range of values separated by spaces or commas. The user may also specify a range of values by inputting a start value, end value, and increment, i.e., (1,10,2). A sensitivity plot can be created from this calculation.
Mean Stress Correction (all)	All mean stress correction methods are automatically calculated and a table of corresponding lives given when the *Recalculate* option is invoked.
Analysis Methods (all)	All analysis types are automatically calculated and a table of corresponding lives given when the *Recalculate* option is invoked.
Surface Finishes (all)	All surface finishes are automatically calculated and a table of corresponding lives given when the *Recalculate* option is invoked.
Surface Treatment (all)	All surface treatments are automatically calculated and a table of corresponding lives given when the *Recalculate* option is invoked. Changes in material properties are not modeled here but are available from the Material optimization submenu.

Material Optimization

The material optimization allows for changing to a different material, editing the parameters associated with the current material dataset, and searching for a better or worse material. These tools facilitate the optimization of the materials selection in terms of fatigue performance. For example, a new material may be found that offers the same fatigue performance but has a lower raw material cost and is easier to work with in the manufacturing process. Alternatively, a possible fatigue failure could be designed out of the product by switching material and these tools would give a selection of alternative materials based on a fatigue selection criterion. Figure 9-22 shows the Material optimization form.

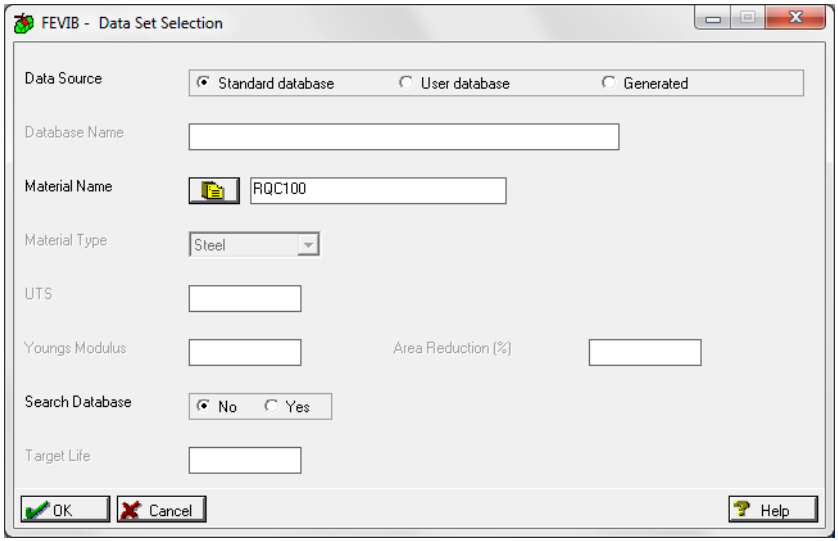


Figure 9-22 Material Optimization Form

Option	Description
Data Source	There are three sources of materials data in all MSC Fatigue analyzers. They are: The Standard Database, which can be the central database or a user-specific local database (which is usually a modified copy of the standard database). A user database which contains data in the format of the standard database but which is specific to the user, i.e. a custom database. Generated - which are generated from the UTS - the results of this type of calculation are an approximation, they should NOT be used in a final sign off.
Database Name	This field becomes live if User database is selected. The user database is created using the tools in PFMAT documented in Material Management (Ch. 4).
Material Name	This field becomes live if Standard or User database is selected. All materials currently available can be viewed using the List button.
Material Type	This field becomes live if 'Generated' is selected. Options available are Steel, Aluminum, Titanium, and Other. If Other is selected, then a Young's modulus and Area Reduction must be supplied in addition to the UTS.
UTS	See Material Type above.
Young's Modulus	See Material Type above.
Area Reduction (%)	See Material Type above.
Search Database	If search database is set to Yes then a range of materials will be evaluated and the10 best will be listed in a pick list. One of the ten should be selected for further consideration.
Target life	A target life is required so that the search database option can use it as a benchmark against which it can compare the relative performances of all the materials in the chosen database.

When the material choice has been optimized you the user will be returned to the main Design Optimization menu.

Change Parameters

This design optimization option allows for changing individual parameters or to reset individual parameters back to their original values. This form is shown in Figure 9-23.

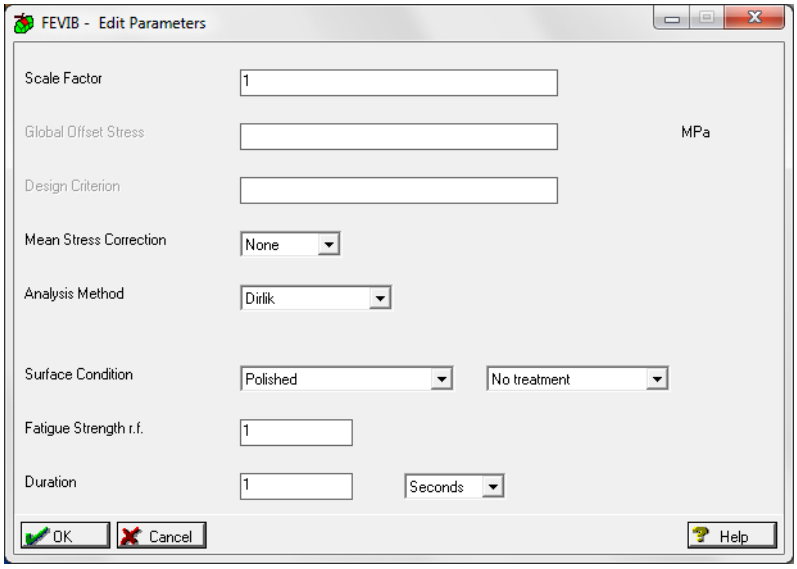


Figure 9-23 Change Parameters Form

Option	Description
Scale Factor	This value is applied to the output stress PSD. Since the PSD has units 12/Hz the scale factor is squared by FEVIB before being multiplied by the values in the output PSD.
Global Offset Stress	This factor can be thought of as a stress offset. The user must enter a single value. FEVIB can also use a variable called $GLBMEAN to define the global mean outside of the GUI. See the Modifying the MSC Fatigue Environment (MENM), 1463. Within Patran you cannot set the global mean to be used for Goodman or Gerber. The user must use the $GLBMEAN to set this.
Design Criterion	The % certainty of survival is a statistical parameter between 0.1 and 99.9% which is based on the standard error of the S-N curve. Using this parameter will tell you how much confidence you can have in the product reaching the target life. A low confidence is associated with long lives whereas the probability of reaching a short life is high. However, you should also consider the error in other parameters such as scaling factor which may cause the life to be different from your estimate. You can accept the default to reset to the original value or you can supply a single design criterion.
Mean Stress Correction	Goodman and Gerber mean stress correction methods are supported as well as no mean stress correction.

Option	Description
Analysis Method	The user may chose to change the analysis type or reset it back to its original value by accepting the default. The user will be presented with a submenu with the list of choices. These are Dirlik, Narrow Band, or All. All stipulates about seven different types of analyses. The difference between these is discussed in the Theory section.
Surface Condition (finish and treatment)	You may choose to change this surface finish or reset it back to its original value by accepting the default. The user will be presented with a submenu with the list of choices. These are: *Polished*, *Ground*, *Good Machined*, *Average Machined*, *Poor Machined*, *Hot Rolled*, *Forged*, *Cast*, *Water corroded*, *Seawater corroded, user defined, unchanged.* You may choose to change the surface treatment or reset it back to its original value by accepting the default. The user will be presented with a submenu with the list of choices. These are: *No Treatment*, *Nitrided*, *Cold rolled*, *Shot peened*.
Fatigue Strength r.f.	Since not all stress raisers may be modeled correctly in the FE analysis the effect of additional stress concentration factors on fatigue life may be modeled. Normally this is modified to a fatigue strength reduction factor Kf. Alternatively, an elastic stress concentration factor Kt can be entered and are available from standard reference texts such as Peterson's book. Kf factors must be in the range 1 to 100 inclusive.
Duration	The default data in the rainflow histogram is normalized to 1 second but may be rescaled to any duration of hours, minutes, or seconds. This field is used to define the total duration of the rainflow data. The value should be given as a real number greater than zero.

Results Display

The presentation of the results in both tabular and graphical form is handled from this menu. The options available are shown in Figure 9-24 and discussed below:

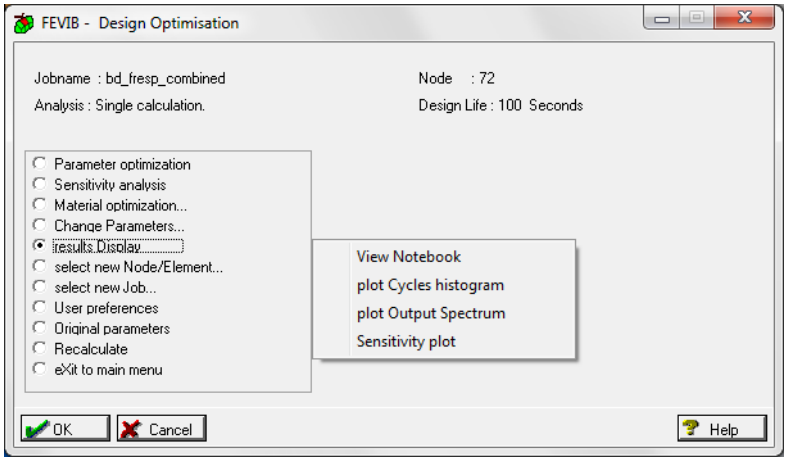


Figure 9-24 Results Display Submenu

Option	Description
View Notebook	Allows the review of the results of all analyses written to the Notebook (including the latest analysis if Notebook is set to On). To view the Notebook FEVIB uses whichever text processor has been nominated, e.g. vi on a Linux platform.
Plot Cycles Histogram	Plots the 3-dimensional rainflow cycle counted histogram using the MP3D module after scaling to the local stresses at the node or element being analyzed. A description of the graphical histogram display is given in Matrix Options, 356.
Plot Output Spectrum	This plots the output response PSD using the MTPD module as explained in Graphically Display a PSD, 692.

Select New Node/Element

Normally, design optimization will be carried out on the node or element which has the shortest life based on the assumption that the lives at all other nodes and elements will show at least the same change in life as the critical node/element. However, the lives at other nodes or elements will need checking, especially where the surface parameters or additional local effects such as mean stress or stress concentration are different from those at the critical node or element. When this option is selected, a new node or element entry screen is presented with the same select options used on the main input screen such as already shown in Figure 9-17. Having selected a new node or element, the user will be returned to the Design Optimization Analysis menu.

Select New Job

This option returns the user to the first input screen where the jobname is requested (see Figure 9-16). The current jobname is presented as a default.

User Preferences

The preferences that may be set here are generally items which are not normally changed for every analysis (i.e., they are not job specific). Each parameter is described in Figure 9-25.

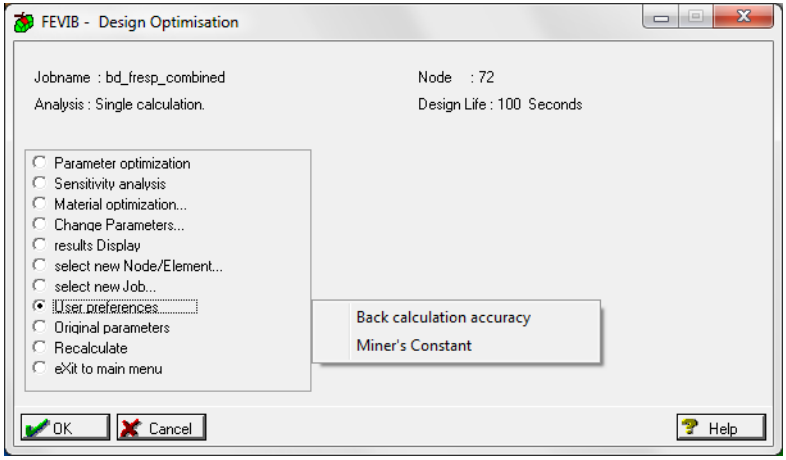


Figure 9-25 User Preferences Submenu

Option	Description
Back Calculation Accuracy	The normal convergence accuracy for the back calculation is 5% (i.e., the iteration will stop once the life is within 5% of the target or design life). Note: Higher accuracy will take longer for the calculation to converge.
Miner's Constant	This constant is normally set to a value of 1.0. Some situations may call for it to be set to a different value, usually less than 1.0 for more conservative life predictions. Caution: Once this value has been set, it will be used in all fatigue calculations carried out in the current directory including global multi-location jobs.

Original Parameters

If at any stage in the design optimization, the user wants to recall the original analysis parameters as defined in the global analysis, then this option will do this. This facility is particularly useful for turning off a previously defined back or sensitivity analysis setup. If the user only wants to reset certain parameters, then he should use the Change Parameters main menu pick.

Recalculate

Once the new analysis parameters have been defined, it is necessary to pick this option to start the re-analysis. Once this option has been selected, a message will appear to tell the user that the analysis parameters are being written to the `pfatigue.prt` file. A "fatigue analysis xx complete" message is used to report the stage of the analysis, where "xx" is a number between 0 and 100.

Exit to Main Menu

Picking this option causes FEVIB to return to the main menu, saving the analysis results summaries in the `pfatigue.prt` file.

Output Power Spectrum

Output power spectrum provides an important link between test and analysis in the integrated durability management of which MSC Fatigue is a core tool.

It allows the output response power spectrum to be output at a node or element to be exported and written to a standard time history `.dac` format (with a `.psd` extension). One or more nodes can be processed in a single run of this option resulting in a number of `.psd` files.

The output power spectrum `.psd` file is extremely useful to the understanding of local stress or strain response when working with strain measurements taken from a fatigue test component. These responses can be used in subsequent single location analyses such as those provide by Frequency Fatigue Life Estimation (MFLF), 695 which may provide even more insight into the results of the analysis.

The option automatically also allows for graphical display of the output response spectrum. You can also use the Graphically Display a PSD, 692 option from the main menu.

Output Power Spectrum creation consists of two forms. First you specify the input `jobname.fes` file. Then a form displaying the output `.psd` file names and a number of other inputs is shown. See Figure 9-26.

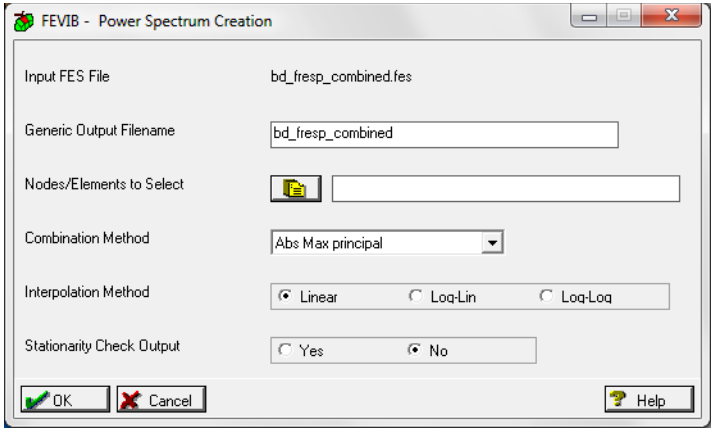


Figure 9-26 Power Spectrum Creation Form

The fields are as follows:

Field	Description
Input FES File	The name of a binary MSC Fatigue input file (`.fes`). Such a file is normally produced from PAT3FAT or FATTRANS or by converting an ASCII file to `jobname.fes` format using Utilities/Binary create from the FEVIB main menu.
Generic Output Filename	The power spectrum that will be created for each node or element will have a file name which is comprised of a generic root, taken from the input file name, and a node or element id number. A `.psd` extension will be automatically appended to each output.
Nodes/Elements to Select	Power spectra will be created at a user-defined number and designation of nodes or elements. Up to 100 locations can be processed. Ranges can be entered in the normal way but the word ALL will not be accepted because ALL could exceed the limit of 100 nodes/elements.

Field	Description
Combination Method	For PSD input you must select one of the components. It is possible that one of the components actually represents something other than one of the components such as a maximum absolute principal. The program makes no actual distinction. If you select X-component it will use the first column of data of the 6 columns representing the 6 components. For transfer function input, your choices are: Abs. Max. Principal, von Mises, Tresca/Shear. These values will be derived from the components. How this is done is described in the Theory section.
Interpolation Method	For the transfer function (frequency response) type input, it may be necessary to interpolate between frequency steps and/or actual power values. This options specifies how that interpolation should be performed: linearly on both axes, Log x and linear y, or log on both axes.
Stationarity Check Output	Turn this on if you wish to do a stationarity check to ensure that the principals are remaining stationary. This is equivalent to a multiaxial assessment in the time domain. The mechanics of this operation is described in the Theory section.

Graphically Display a PSD

Once a power spectrum has been created via the Output Power Spectrum option or created by any other means within PTIME, the time history or FEVIB, it can be graphically displayed. The module MTPD is used to display these plots. The explanation of power spectrum graphical displays is identical to that already discussed in Plot an Entry Option, 245.

Results Postprocessing

This option reviews the fatigue job output file `jobname.fef` and displays user selected items to a tabular listing using the module PFPOST.

Detailed descriptions of the operation of PFPOST are given in Reviewing Results (PFPOST), 363.

For vibration fatigue, the content of the `jobname.fef` file is different than that of the time-based fatigue result files. It consists of seven columns of data aside from the location (node/element) number itself. The columns of data in this file are:

Column 1	Irregularity Factor
Column 2	Root Mean Square
Column 3	Damage (per second)
Column 4	Life (seconds)
Column 5	Log of Damage

Column 6	Log of Life
Column 7	Angle Range

The actual format of this file is either a Patran nodal or elemental results file described in The Results Files (jobname.fef/fos), 414.

Utilities

This option consists of a number utilities for converting .dac files between various formats. Once the program has started, the user will be presented with a menu of options as described in Figure 9-27.

The options menu looks like this:

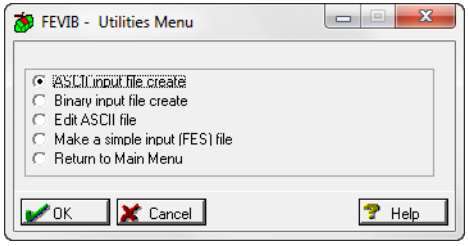


Figure 9-27 Utilities Menu Options

The utilities are identical to those described in the FEFAT module. See Utilities, 360.

FEVIB Batch Operation

FEVIB analysis can be operated in batch mode with the following keywords:

/OPT=	To specify main option. The capitalized letter of any option on the interactive form is generally the letter to use when specifying a batch operation (G, D, O, P, R, U, X) e.g.,/OPT=G
/UOPT=	Utilities options. A, B, L, C, M, R
/OPT=	Design optimization options. First issue the /OPT=D option and then specify the command again with the design optimization option (P, S, M, C, D, N, J, U, O, R) e.g., /OPT=D /OPT=P
/SOPT=	Design optimization suboptions. Parameter Optimization. G, D, L, C
/SOPT=	Design optimization suboptions. Sensitivities. G, D, F, T, A, C
/SOPT=	Design optimization suboptions. Results Display. V, C, O
/SOPT=	Design optimization suboptions. Preferences. B, M
/INP=	Input file name.

/OUT=	Generic output name.
/LOC=	Node or element list. Use 'ALL' for all locations.
/SCOMB=	Combination
/IMET=	Interpolation method.
/SCO=	Stationarity check. Y,N
/PSDP=	Power spectrum.
/NBINS=	Matrix size. 3, 6, 1
/LIFE=	Design life
/TUN=	Time units.
/GLOBOF=	Global offset stress.
/DC=	Design criterion.
/MSC=	Mean stress correction. None, Goodman, Gerber
/SUR=	Surface finish.
/TRE=	Surface treatment.
/KF=	Kf
/VIBM=	Vibration analysis method. D, N, A
/SNSRCE=	S-N data source. S, U, G
/DBASE=	Material database name.
/MTYPE=	Material type. S, A, T, O
/UTS=	Ultimate tensile strength.
/MAT=	Material name.
/YM=	Young's modulus.
/RAREA=	Reduction in area.
/SEARCH=	Search database. Y, N
/LIFE=	Search life.
/ACCY=	Accuracy.
/MINER=	Miner's sum.
/USEVAL=	Use optimized value. Y, N
/MODE=	Run mode.
/NRMS=	Number of RMS.
/ENT=	Node/element selection option for design opt., User, Last, Worst
/JOB=	Job filename
/PASK=	Plot option

/PLOAD=	Load case number
/*=TT	If the user wishes to see output sent to the screen, he must include this parameter exactly as shown. Any other value other than TT after the equal sign will send the output to a file by that name.

Design optimization mode is generally not used in batch mode.

Example:

```
fevib /opt=g/inp=mymodel/loc=all/ov=yes
```

This batch line would open `mymodel.fes` and perform the vibration fatigue analysis with defaults used for all other parameters. Any existing output files with the same prefix name as the input file would be overwritten.

FEVIB Batch Operation for Utilities

/OPT=U	To specify utility option.
/UOPTion=	The utility options are A, B, E, M, and R corresponding to the submenu picks under the utilities menu.
/OUTput=	Output file name. Only the jobname need be input. A file suffix of ASC or FES is assumed depending on the main menu option. The default is the input jobname.
/INPut=	The input file name. This is usually the jobname. A file suffix of ASC or FES is assumed depending on the main menu option.
/OVerwrite=	This parameter sets automatic overwrite of existing files on or off. /OV=Y sets overwrite on. /OV=N sets overwrite off. Off is the default.
/*=TT	If it is desirable to see output sent to the screen this parameter can be included exactly as shown. Any other value other than TT after the equal sign will send output to a file by that name.

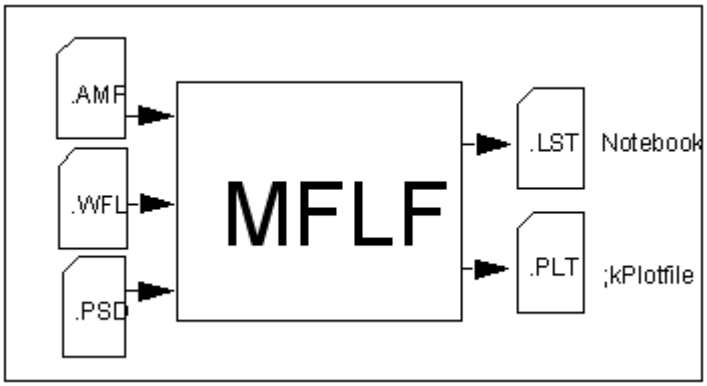
Example:

```
fevib /opt=u/uopt=a/inp=mymodel/out=mymodel/ov=y/*=tt
```

This example selects the utility option in FEVIB and executes the submenu option of creating an ASCII file from a binary file (A), with a jobname of mymodel. It will search for a file called `jobname.fes` and create an output file called `jobname.asc`.

Frequency Fatigue Life Estimation (MFLF)

The MFLF program offers a range of analysis methods by which an estimated fatigue life may be calculated from a power spectral density or from an amplitude spectrum. MFLF uses the S-N approach to fatigue life estimation but alternative fatigue methodologies may be used in conjunction with the rainflow histogram produced by MFLF.



The input file must contain a power spectrum or an amplitude spectrum expressed in SI units, e.g., MPa^2/Hz or MPa respectively. Unlike FEVIB, this module is a single location analyzer that accepts only a single response PSD as input. This PSD could come from just about any source. It can be a response PSD from a single location of a FE model, a time history that has been converted to a PSD via the MASD program, or a measured response PSD from test. The format of the input file must be of type DAC, but with a default extension of `.psd`.

See Vibration Fatigue Theory, 710 for technical background to Frequency Life Fatigue Estimation.

MFLF Module Operation

The MFLF module can be run in one of the following three modes:

- From the MSC Fatigue menu driven system.
- In stand alone mode by typing **mflf** at the system prompt.
- By incorporating the MFLF commands in macro.

The first two modes are interactive. Once running in interactive mode the MFLF module will display the following screen:

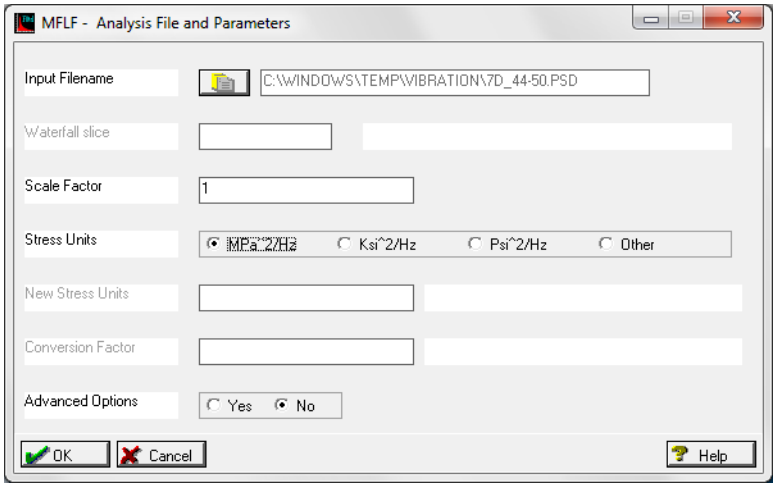


Figure 9-28 The First MFLF Screen

The fields are as follows:

Option	Description
Input Filename	The name of the file containing the data to be analyzed should be entered into this field. The data may comprise of either a Power Spectral Density or an Amplitude Spectrum (RMS, not Peak), optionally stored as a slice of a waterfall file. The default extensions are: ■ `.psd` for Power Spectral Density data ■ `.amp` for Amplitude Spectra ■ `.wfl` for waterfall data The frequency units at which data is measured must be in Hertz (Hz) (samples per second).
Waterfall slice [1 : 4] No units	Enter the "speed" (y-axis) value from which data is to be extracted. Note that speed is the usual parameter, but a waterfall file containing amplitude spectra may have time or buffer as the spectral value. The speed used will be closest match to the value entered. However, if the value entered is not within 25% of any of the speeds in the waterfall file then a warning is issued. Clicking the left mouse button over option also selects the corresponding data type.
Scale Factor	A scale factor can be specified if necessary to scale up or down the response. The default is unity.

Option	Description
Stress Units	This question identifies the units of stress contained in the input file. The units available depend upon the type of input file: `.amp` - Amplitude spectra measured in MPa, ksi and psi, are supported directly. The 'Other' option is available for spectra which contain stress data measured in other than these units. `.psd` - PSDs measured in MPa^2/Hz, ksi^2/Hz and psi^2/Hz are supported directly. The 'Other' option may be used for PSDs measured in any other units.
New Stress Units	For 'Other' units 2 additional questions are asked. The first requests the stress units (e.g. mmH_20) and the second resolves the relationship between these units of stress and the standard SI unit of stress, MPa.
Conversion Factor	The conversion factor of the new stress units to SI units of MPa
Advanced Options	Selecting Yes for this question gives access to the Advanced Options form. The form contains three questions: 1. Analysis Type 2. Cutoff Frequency 3. Ask for RMS Cutoff*

*The MSC Fatigue environment variable *$FLFNRMS* defined through the utiltiy MENM can also be used to set a user defined number of RMS Cutoff level to be used for stress clipping in the damage calculation. See the Modifying the MSC Fatigue Environment (MENM), 1463.

By default, the programs automatically determines the optimal number of RMS levels and then uses the user specified RMS Cutoff for the stress clipping only. Stress clipping has the effect of keeping the stress constant at stress bins above the RMS Cutoff level when summing fatigue damage. Less conservative fatigue lives are calculated by specifying RMS Cutoff levels below the automatically determined optimal number.

Figure 9-29 below summarizes the structure of the MFLF program and illustrates the routes through the program.

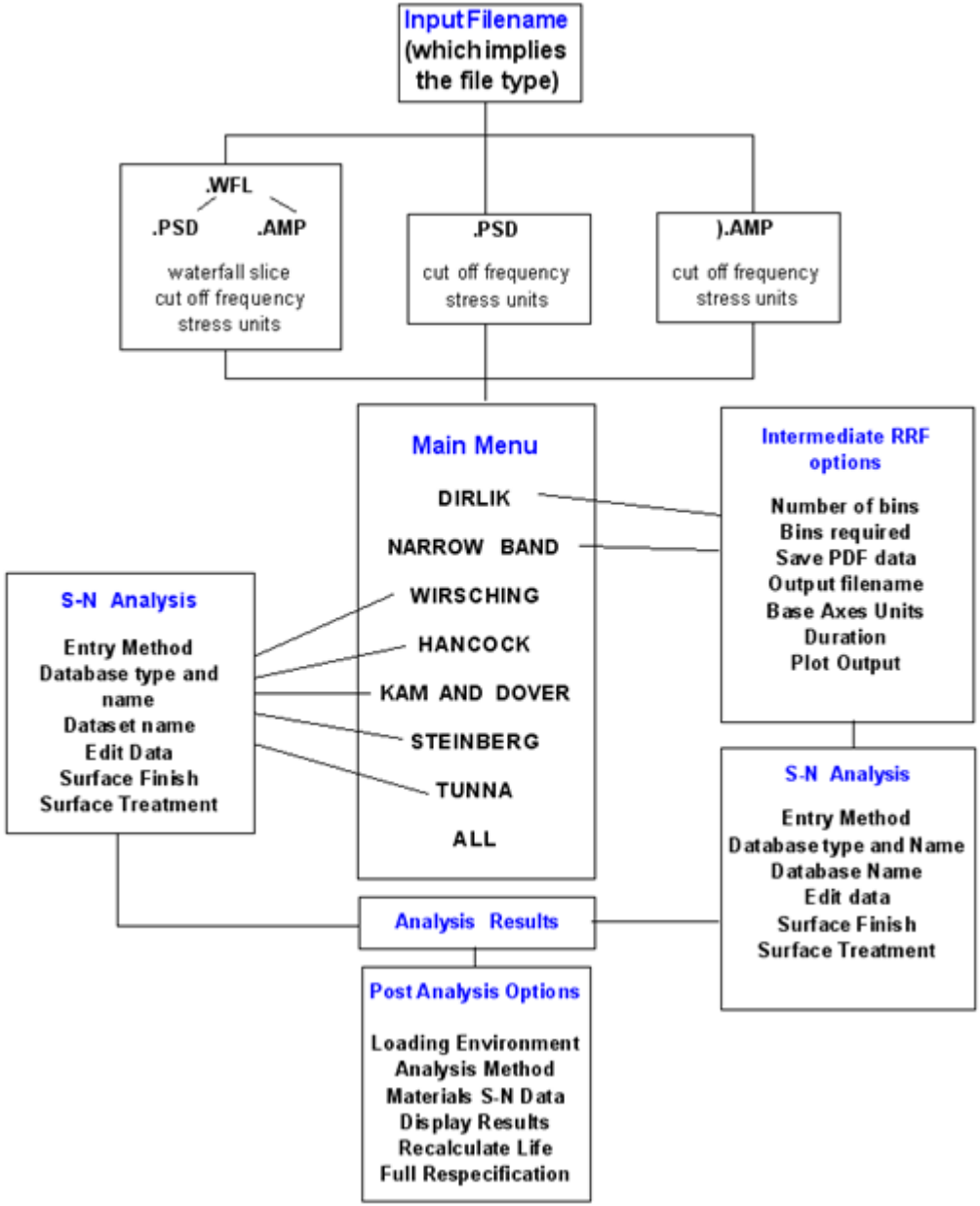


Figure 9-29 The General Module Structure

Intermediate Rainflow Range File Options

The next screen that is presented allows for specification of certain solution or analysis parameters.

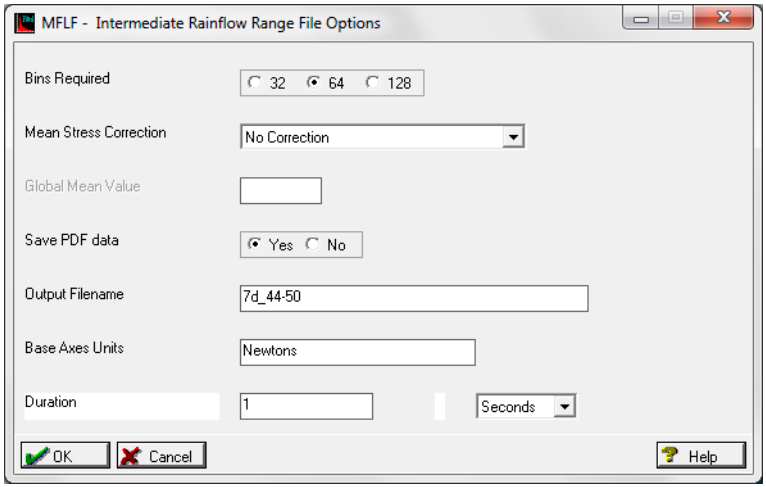


Figure 9-30 An Intermediate RRF Screen.

Figure 9-30 only allows for a Dirlik solution. To use other solutions such as Tunna, or Narrowband, you must do this in the postprocessing stage. Only Dirlik, Tunna and Narrow Band analyses generate rainflow stress range histograms. This screen allows you to save the rainflow range histogram that is generated.

Option	Description
Bins Required	This question is asking for the number of bins to be used along each dimension, Range (X) and Mean (Y), of the histogram. Three options are offered 32, 64 and 128. The higher the number, the higher the resolution of the output file, although once the data has been fully resolved it is pointless increasing the resolution further. Also, if the matrix file is to be used in another application, e.g. MSLF, then this feature can be used to ensure bin number compatibility with other matrices being used in that application.
Mean Stress Correction	You can apply the standard Dirlik method or the Dirlik method with a mean or the Dirlik method with a mean using either a Gerber or Goodman mean stress correction.
Global Mean Value	This is the global mean value of stress offset to be used in the specified mean stress correction method. Also the variable *FLFGMEAN* can be set globally. See the Modifying the MSC Fatigue Environment (MENM), 1463.
Save PDF data (Yes No)	The chosen analysis method generates an intermediate probability density function (PDF) hence the histogram of the stress ranges. This question offers the option of saving the PDF to a file or discarding it when the analysis is complete.

Option	Description
Output Filename	The question is asking for the name of a file in which to save the histogram.
Base Axes Units	The X and Y-axis represent the RANGE and MEAN values respectively and hence the units for both axes are identical. This question is asking for the X and Y-axis units. The default units are those from the input file.
Duration	The default data in the rainflow histogram is normalized to 1 second but may be rescaled to any duration of hours, minutes or seconds. This question asks for the total duration of the rainflow data. The answer given should be a real number greater than zero. If you want to correlate a life calculation between frequency life (**MFLF**) and stress life (**MSLF**) using the range-mean matrix option, you should enter the duration as the length of the time series used to create the range-mean matrix input to MSLF.

The Material S-N Analysis Screen

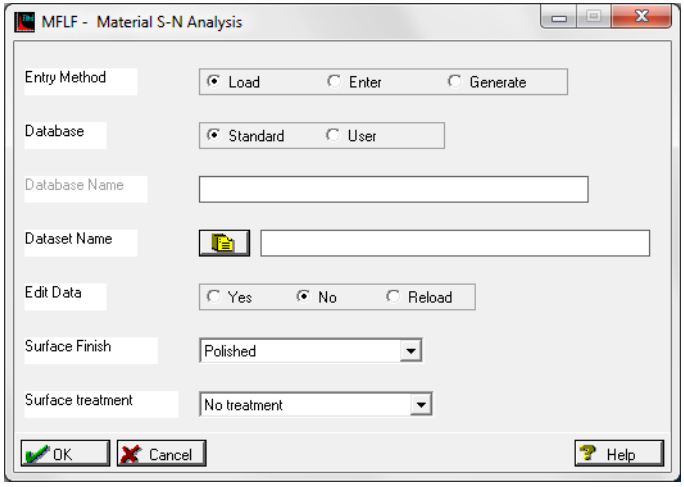


Figure 9-31 The Material S-N Analysis Screen

S-N damage curves can be selected for use in a life estimation in any one of the following ways:

- Loading an S-N data-set directly from the material database.
- Entering details of an S-N curve directly from the keyboard. The parameters entered will NOT be loaded into the database.

- Generating an S-N curve from an input value of the ultimate tensile strength, UTS. The generated parameters will *not* be loaded into the database.

Select the S-N type required.

Option	Description
Entry Method = Load	Selecting this option allows datasets to be loaded directly from the material database.
Database	It is possible to specify the database from which materials parameters will be extracted. The standard method is to look for a local database called `nmats.mdb` then, if that database does not exist, look in the central directory for a database named `nmatsmas.mdb` (which is distributed with the software). It is also possible to create named custom databases using the materials data manager. By selecting the User option, it is possible to select data from any database.
Database Name	This field is activated when the database field is set to "user". It is here that the user can enter the name and location of the material database to use.
Dataset Name	Typing the name of a dataset in this field causes **MFLF** to search the Materials Data Manager (PFMAT) for that dataset. The List button will provide a list of all material S-N datasets.
Edit Data Yes/No	If a dataset was specified in dataset name (above) then it's parameters can be edited before a life estimation is carried out (select Yes to Edit). The edit will take place only after the user leaves this screen.
Surface Finish	Surface finish can have a significant effect upon fatigue life, especially with higher strength materials. The default entry is 'Polished' as with a laboratory specimen, but a whole range of predefined finishes is provided. The surface finish correction factors that will be applied to the calculation are held in the `\mats` subdirectory. Selecting User Defined enables the user to define a surface correction factor.
Surface Treatment	Treatments that introduce stresses, usually compressive, into the surface of a component have a significant effect upon fatigue life. Correction factors for a range of surface treatments are available within **MFLF**.
Ultimate Tensile Strength, UTS	This field and the other fields listed below are all on the main form that comes up when the Edit Data field is set to Yes and the **OK** button has been selected. Enter the new UTS value in the units specified.
Stress Range Intercept, SRI1	This is the value of the intercept of the curve with the b1 slope on the stress range axis. Enter the new SRI1 value in the units specified.
First Slope, b1	Initial slope of the S-N curve - b1.
Transition Life, NC1	This is the number of cycles at which the slope of the S-N curve changes.

Option	Description
Second Slope, b2	This is the final slope of the S-N curve. Entering zero will generate an implicit fatigue limit.
R-Ratio of Test, R	This is a measure of the mean stress or the mean of constant amplitude signal or the mean of stress cycle. R=-1 is a fully reversed signal and is used in most material coupons.
Entry Method = Enter	**MFLF** also allows S-N curves to be entered at run time without using the database. The parameters to be entered have been selected so that no regression analysis is required. The actual parameters, slopes and intercepts, required to define the damage curve are calculated automatically. Note that, a line is assumed between the co-ordinates (N1,S1) and (1,UTS) on the stress axis.
Surface Finish and Surface Treatment	See Entry Method = Load above.
Ultimate Tensile Strength, UTS	This field and the other fields below it are all on the new form that comes up when the OK button has been selected. The S-N curve is strongly affected by the materials Ultimate Tensile Strength (UTS). Enter the UTS here.
First Life Point, N1	The default value for N1 is 1000.
Stress Amplitude at N1, S1	The default solve for stress amplitude at stress life coordinate pair 1 (cycles N1, S1) is 1000 MPa.
Second Life Point, N2	The default value for N2 is 1E6.
Stress Amplitude at N2, S2	The default solve for stress amplitude at stress life coordinate pair N2, S2 is 200 MPa.
Slope after N2, b2	By default the second slope of the fatigue curve (b2) is 0.0. The second slope, b2, is offered as zero which means it effectively represents a fatigue limit. For situations where an absolute fatigue limit is inappropriate some other slope should be entered.

Option	Description
R-Ratio of Test, R	This is a measure of the mean stress or the mean of constant amplitude signal or the mean of a stress cycle. R = -1 is a fully reversed single and is used in most material coupons.
Entry Method = Generate	Approximate S-N damage curves can be generated purely on the basis of ultimate tensile strength, UTS. The curves are constructed by fixing the stress axis intercept (1 cycle) at the value of the UTS, fixing the stresses at 1000 cycles and the endurance limit in accordance with the fraction of the UTS detailed below: **Cycles** / **Stress** **Ferrous Alloys.** 1 — 1.000 x UTS 1,000 — 0.900 x UTS 1,000,000 — 0.357 x UTS **Titanium Alloys.** 1 — 1.000 x UTS 1,000 — 0.800 x UTS 1,000,000 — 0.307 x UTS **Aluminum Alloys.** 1 — 1.000 x UTS 1,000 — 0.700 x UTS 500,000,000 — 0.258 x UTS **Other Alloys.** 1 — 1.000 x UTS 1,000 — 0.800 x UTS 100,000,000 — 0.274 x UTS
All Other Parameters	The parameters below are essentially as per the Entry Method = Enter. Please refer to that method in this table.

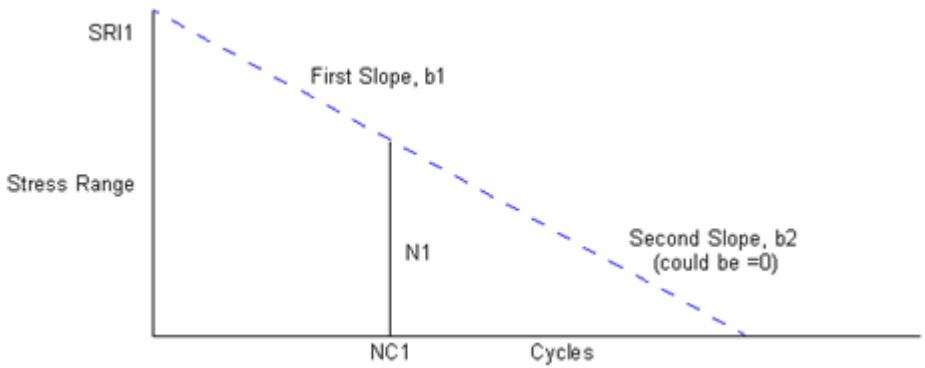


Figure 9-32 Diagram Illustrating the Editable Fields of an S-N Curve

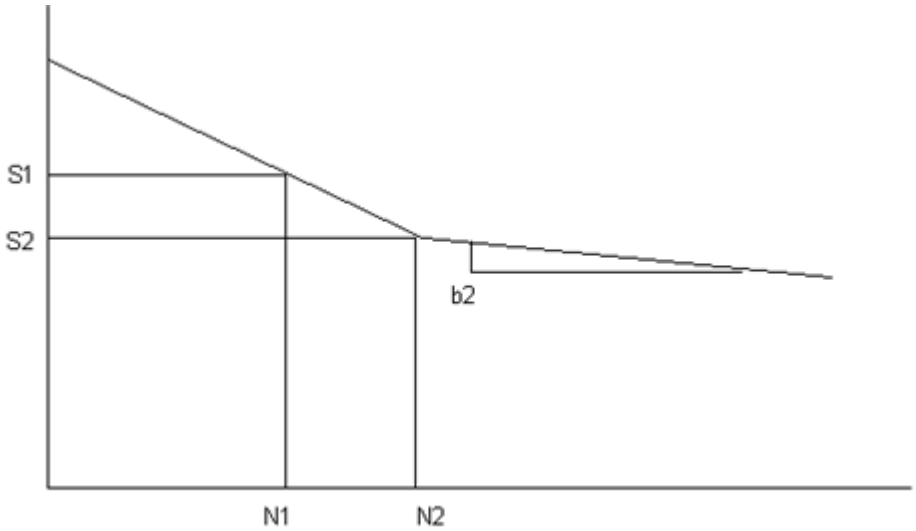


Figure 9-33 Diagram Illustrating the Slopes and Features of the S-N Curve

MFLF Geometry

The next page that is presented is the geometry correction screen.

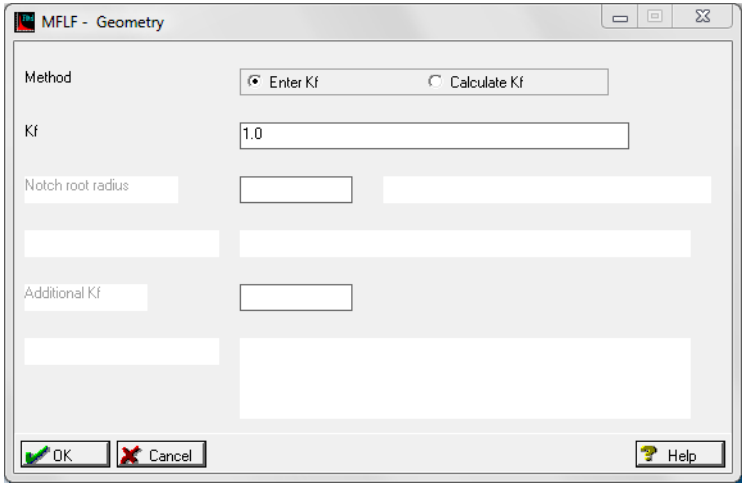


Figure 9-34 The MFLF Geometry Screen

The fields are as follows:

Option	Description
Method	You can either specify a K_f or you may calculate a K_f based on a stress concentration value K_t, a notch root radius and any additional Kf values you wish to add. See the next entry for a more detailed explanation.
K_t / K_f	The elastic stress concentration factor, K_t, is the ratio of the maximum stress at a stress raiser to the nominal stress computed by the ordinary strength- of-material formulae, using the dimensions of the net section. It can be used to account for the presence of a notch within a component or structure. The magnitude of the K_t required depends on the nature of the notch and its geometry. Values of stress concentration factors can be obtained from standard works such as: R.E Peterson's 'Stress Concentration Factors', John Wiley & Sons, Inc. 1974. Alternatively use can be made of the Time Correlated Damage - (MTCD), 1121 module. It is well known that small notches have less effect in fatigue than is indicated by K_t. This has led to the idea of a fatigue concentration factor, K_f, which is normally less than K_t, being introduced and being used to replace K_t within Neuber's rule. K_f is related to K_t according to: $K_f = 1 + (K_t - 1) / \{1 + \sqrt{(p' / r)}\}$ where: p' is a material constant dependent on grain size and strength and r is the notch root radius. If K_f is not known, then estimate the theoretical stress concentration factor, K_t, and select the calculate option, otherwise select the direct entry option.
Notch Root Radius	Enter the radius of the notch.
Notch Index q	This is just a label that will display the notch index q once you click the OK button.
Additional K_f	Enter any additional K_f value to be applied.
Calculated K_f	This is just a label that will display the calculated Kf once you click the OK button.

When all the fields are filled out from the various input screens the analysis will proceed and you will be presented with a summary page as follows:

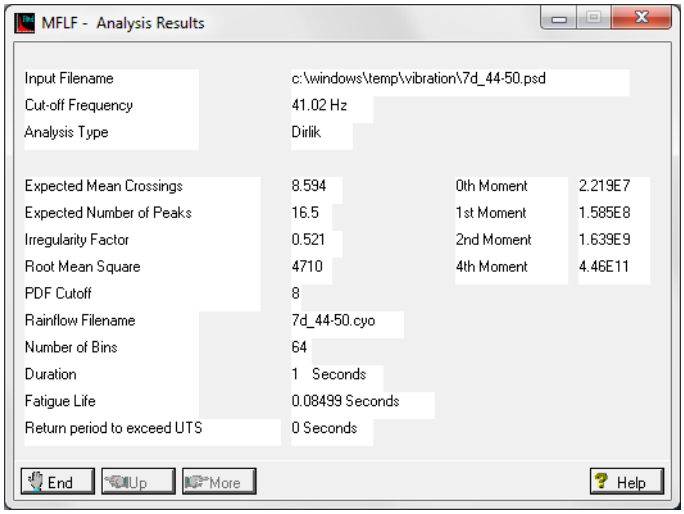


Figure 9-35 A Typical Page of Results

Once you accept the summary page you will then be place into the postprocessing menu where you can change parameters, materials, methods and a variety of other inputs to perform what-if studies and re-analysis.

MFLF Postprocessing Menu

Once an initial analysis is finished you are presented with the following menu for post-processing purposes.

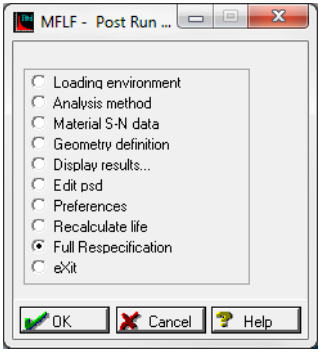


Figure 9-36 The Postprocessing Menu

The selections on this page offer the choice of rerunning the fatigue life estimation program and/or changing any of the initial input parameters without an intermediate exit.

Option	Description
Loading environment	This sends the user back to the loading screen (see Figure 9-28).
Analysis method	This sends the user back to the type of analysis selection screen (see Figure 9-37).
Material S-N data	This sends the user back to the material S-N screen (see Figure 9-31).
Geometry definition	This sends the user back to the geometry screen to modify Kf (see Figure 9-34).
Display results	This option loads the graphics module MP3D and displays any graphical results file. Please refer Matrix Options, 356 for it's usage instructions.
Edit psd	This option will allow you to graphically edit the PSD. This may be useful to investigate the contributions of various frequency content to damage. It uses the graphic editor MGED described in Graphical Edit, 236 in the PTIME module.
Preferences	This allows the user to change the cut-off frequency. The cut-off frequency is the position in the `.amp` file beyond which the data contribution from its equivalent PSD is considered negligible and is, therefore, not to be used during the analysis. The default offered is the frequency at which 99.9% of the area under the equivalent PSD curve is included - normally an acceptable value. The answer should be a positive real number.
Recalculate life	If any changes have been made this triggers a recalculation so that the effect of the changes can be assessed.
Full Respecification	This option restarts **MFLF** from its very beginning.
eXit	Quits **MFLF**, discards and edits to the materials database, and saves the results of the last run to the Notebook.

Analysis Method

The original analysis defaults to Dirlik, however you may change the analysis if desired after the initial calculation. These analysis methods are detailed in Frequency Domain Approaches of Life Estimation, 749. Dirlik is selected as the default for being the most generally applicable. It is sometimes useful to select "All" for a comparison of all the different analysis types. This does not significantly increase the computation time.

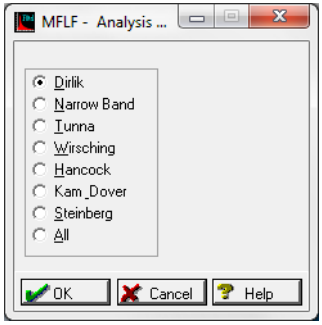


Figure 9-37 Selecting the Analysis Method

MFLF Batch Operation

It is recommended that, by default, the /OV=Y keyword be included in every batch command line, since if it is omitted and an output file with the specified name already exists, batch operation will cease.

A list of MFLF's batch keywords:

/INPut	Input file name
/TYPE	Input data type: P, A
/SLICE	Waterfall file slice
/COFREQ	Cut-off frequency
/SUNITS	Units of stress in input data: M, K, P, O
/YUNI	Entry field for less common stress units
/SCALE	Scaling factor with respect to MPa
/METHOD	Analysis method: D, N, W, H, K, S, A
/MATENTry	Mode of S-N curve entry: L, E, G
/SNDATa	S-N data-set name
/MATSRC	Source of materials data: S, U
/DBNAME	Material database name
/EDIT	Edit S-N data-set: Y, N
/SUFac	Surface finish
/TREAT	Surface treatment
/UTS	Ultimate tensile strength
/SRI1	First stress range intercept
/B1, /B2	First slope, Second slope
/NC1	First Knee

/N1, /N2	Cycles at first point, Cycles at second point
/S1, /S2	Stress at first point, Stress at second point
/SLOPE	Slope beyond second point
/GENTYP	Type of material to be generated: S, A, T, O
/BINs	Number of bins in histogram file: 32, 64, 128
/SAVPDF	Option to save in the intermediate PDF
/OUTput	Output histogram file name
/OVerwrite	Overwrite existing histogram option: Y, N
/XUNITs	Base units for histogram
/DURATion	Duration of data in histogram
/DURUNIts	Units of duration: N, M, H
/PLOt	Option to plot the histogram
/PLTNAMe	Histogram plot filename: Y, N

When batch processing with a series of different input files, it is necessary to use a new batch line and option definition for each new input. The new line must specify the option from the post-processing menu into which the new input will go.

Note: It is permissible to input more than one parameter on a batch line, but they must be different. Also, the order in which the batch keywords appear is not critical.

Vibration Fatigue Theory

Frequency based techniques for fatigue life prediction are now available to the analyst/designer through the Frequency Life Fatigue Estimation (MFLF and FEVIB) modules of the MSC Fatigue software package. This section is intended to provide the required technical background for the practical designer. Where more detailed information is required the reader is referred to the list of references provided in References (Ch. 17).

A Time or Frequency Approach?

Structural design can be conveniently categorized as either testing or analysis. By ***testing*** we usually mean the experimental measurement of stresses or strains on a laboratory or test track prototype. Or alternatively, testing might involve the measurement of noise or vibration rather than stresses. By ***analysis*** we usually mean the computation of structural response using a technique such as Finite Element Analysis (FEA).

In general there are two alternative options (or domains) which can be used for such testing or analysis. Conventionally the analysis might be carried out in the ***time domain***, where all inputs loadings and output responses are specified as time sequences or derivatives thereof.

For example, the stress history induced in a critical component of a car chassis while being driven over a road surface might be experimentally measured, or estimated at the design stage using appropriate analysis programs.

An alternative approach is to perform the analysis in the ***frequency domain***. In this case all loadings and output stress responses are represented as plots of energy at different frequencies.

A ***Power Spectral Density Function*** **(PSDF or PSD)** is the most common way of representing the loadings or responses in the frequency domain. The transformation between time domain, i.e., the time history of the loading, and the frequency domain, i.e., a PSD, should not trouble the reader. The PSD simply shows the frequency content of the time signal and is an alternative way of specifying the time signal. It is obtained by utilizing the ***Fast Fourier Transform*** **(FFT).** Figure 9-38 shows this equivalence for a typical structural response signal.

Transforming from the frequency domain to the time domain is also a relatively easy task which can be done using the ***Inverse Fourier Transform*** **(IFT).** However, when transforming in this direction the random phase angles attributable to each frequency component (which have not been kept when converting to the frequency domain) have to be generated or regenerated. This can be done such that a statistically equivalent signal can be reproduced.

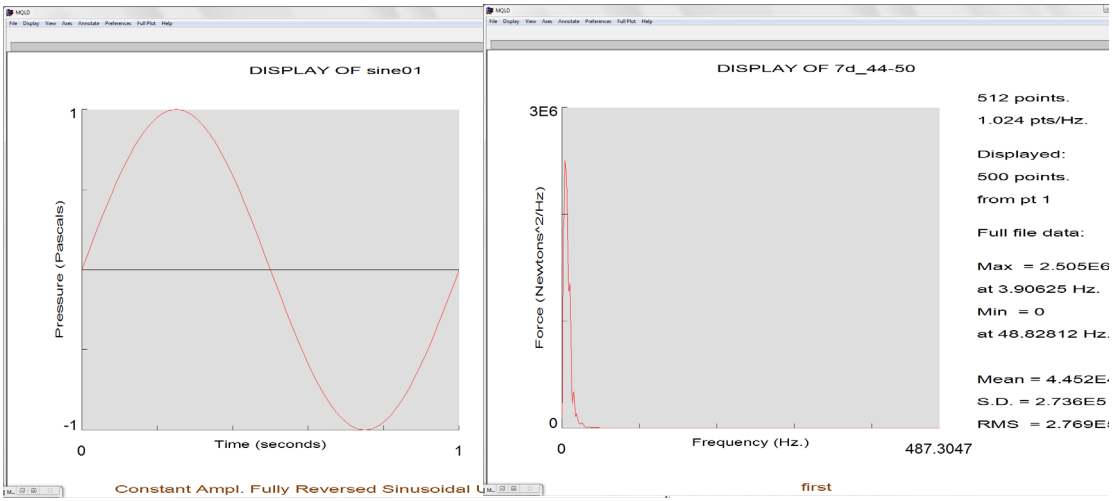


Figure 9-38 PSDs and the Transformation Between Time and Frequency Domains

Comments on Time versus Frequency Domain Approach

This explanatory text assumes that any prospective designer will be approaching either the task of laboratory test measurements or, alternatively, computer-based design. For each type of task a number of key questions may need to be addressed in order to decide if frequency domain techniques are the appropriate route. These questions are discussed below.

- Q. I am undertaking laboratory tests. Do I need to compare my results with, for instance, FEA results?

  An example from the automotive industry demonstrates that frequency based FEA can be a powerful qualitative as well as quantitative tool for the reliability assessment of certain components. If this is likely then frequency based tests and fatigue calculations may be appropriate.

- Q. Are data storage problems likely to be an issue, either in terms of hard disk space or speed of acquisition?

  An acquisition rate of one tenth of that for time domain measurements can usually capture the frequency domain data required for a fatigue analysis with the same level of accuracy. Furthermore, there can also be a significant reduction in storage space required for the frequency domain data. This is because PSDs are significantly easier to obtain, and store, than long time histories.

- Q. As part of the testing work being undertaken is any structural condition monitoring required, or beneficial?

Frequency based measurements are very useful for determining any changes in structural behavior or performance. Any change in the system transfer function (see later) is a useful indicator that some structural change has taken place such as the growth of a crack. If such structural monitoring is likely to be beneficial then frequency based tests and fatigue calculations may be appropriate. Alternatively, a vibration analysis may be the required objective in order to determine whether a structure, such as a car body, is acceptable. In this case, if fatigue is also an issue a frequency based analysis may well be appropriate.

- Q. What type of fatigue analysis is being undertaken?

  Is the fatigue calculation being undertaken with measured frequency domain data, or is the frequency domain data being generated using a computer analysis package? The answer to this question has a significant bearing on the number of assumptions which need to be considered before proceeding with a frequency based fatigue analysis. If, for instance, a PSD of structural response (stress or strain) has been measured, then only the characteristics of this measurement are important. If a structural computer analysis such as an FE analysis is being undertaken then the characteristics of the structural system also need to be considered.

- Q. What are my analysis options?

  It may be the case that, because of the nature of the structure you are analyzing, your analysis route is already determined. For instance, many deep water offshore oil platforms can only be satisfactorily designed in the frequency domain, thereby producing frequency domain results. In this case a frequency based fatigue calculation is the only option. Alternatively, the nature of the fatigue damage mechanism, or the structural system, may determine that only a time based approach is applicable. If either of these scenarios is true then the correct approach is already defined. More usually there is a choice, or perhaps both approaches in parallel are appropriate. In which case the additional questions below need to be addressed.

- Q. If I am directly measuring my response data (stress or strain) is it stationary, Gaussian and random?

  These are probably the issues which cause the most concern to design engineers.

  Firstly, on the question of how Gaussian (sometimes referred to as 'normality') particular data is. If we calculate the percentage of time that response data spends within a particular stress bin and plot this as a probability density function (PDF) we require that its PDF (probability density function) follows the Gaussian bell shape. Fortunately there is a theoretical explanation to explain why nearly all engineering components and structures exhibit Gaussian behavior. This is called the Central Limit Theorem. This states, in very general terms, that the response of any system will be Gaussian as long as the number of processes contributing to this system response is reasonably large and that no one process dominates. This is true even if the individual processes are not Gaussian. Practical fatigue calculations have shown that the fatigue predictor tool MFLF is quite robust to some variation from a strictly Gaussian signal. However, this is an issue which should be assessed before proceeding with any frequency based calculations.

If the signal is stationary, it means that the general characteristics, such as rms, don't change with time. For most engineering processes this is true. Furthermore, even when the signal characteristics are changing slowly with time, the complete response process can usually be broken up into a number of shorter stationary processes. An example of this, for offshore data, is given in the Case Studies Manual. This is again something which should be assessed before proceeding with any frequency based calculations.

Finally, and perhaps most importantly, is the signal random. If it isn't then a time based approach is probably appropriate. For instance, if a small number of transients dominate the fatigue damage, then it is almost impossible for a frequency based fatigue predictor, such as MFLF, to properly identify these. This is perhaps an example, such as that referred to in the Central Limit Theorem, where one process dominates the rest. If such transient or deterministic inputs are relatively small, in comparison to the rest of the response data, then a frequency based calculation may still be possible.

- Q. What determines if any data conforms to these assumptions?

  The only real test, in the context of a fatigue calculation, is how accurate the estimated fatigue life result is. Case studies have shown some results for wind turbine blade response data which helps us to make some practical judgments about this. These results tell us that unless a dominating deterministic component, such as a transient, is present in the signal then MFLF is surprisingly robust to small levels of non-stationarity and non-Gaussianality. However, real care has to be taken with transients and deterministic components.

- Q. Do advantages of working in the frequency domain outweigh possible errors?

  If a frequency based fatigue predictor is coupled up with a frequency based load predictor, such as an FEA program, then a designer has the ability to undertake rapid design optimization. He may then choose to use a time based calculation for the final 'proving' analysis calculation. In such a case, any loss of accuracy involved in working in the frequency domain is far outweighed by the increased design capability. This is therefore something which should be carefully considered.

- Q. If I am using data to perform a frequency based FEA, and subsequent fatigue calculation, is the data stationary, Gaussian and random?

  There are some circumstances when the response of a structural system will be Gaussian even if the input is not. However, in general, if you wish to perform a frequency based analysis you should place the same requirements on the input loading data that you would on any response data. Subject to system linearity (see below) this will ensure the response data is satisfactory.

- Q. Is the system linear?

  Before we can consider this we need to consider, in more detail, what is involved in computing a structural response estimate using, for instance, an FEA package. In order to obtain the output (PSD) response of any system we need its transfer function. This transfer function then obeys the following:

There are a number of experimental and analytical ways of obtaining the transfer function. The simplest and easiest method to visualize is the so-called sine sweep. Using this method the response of a system to a particular sine wave of a particular magnitude is noted, the frequency of the sine wave is then incremented by a small amount and the magnitude of response is noted again. By repeating the process a response vs frequency plot for all frequencies of interest can be produced which can then be used directly to evaluate the transfer function.

An alternative way of obtaining the transfer function is to apply an input of white noise which is an input that has a flat-topped PSD over all frequencies of interest. The response to this random input can be used directly to obtain the transfer function by dividing the output PSD by the (constant) height of the white noise input PSD.

These experimental techniques have their analytical equivalents. For instance, a series of unit load steady state (harmonic) analysis computations can be used to obtain the required transfer function ordinates. Alternatively a time domain analysis can be performed using white noise as the input and the response obtained can then be used to obtain the transfer function. It should be clear from the above that the transfer function is a characteristic of the system or structure. It is not until the transfer functions have been established that the actual loadings are applied. This highlights the usefulness of spectral techniques because the computational effort involved in computing the responses is trivial compared to that of computing the transfer functions themselves.

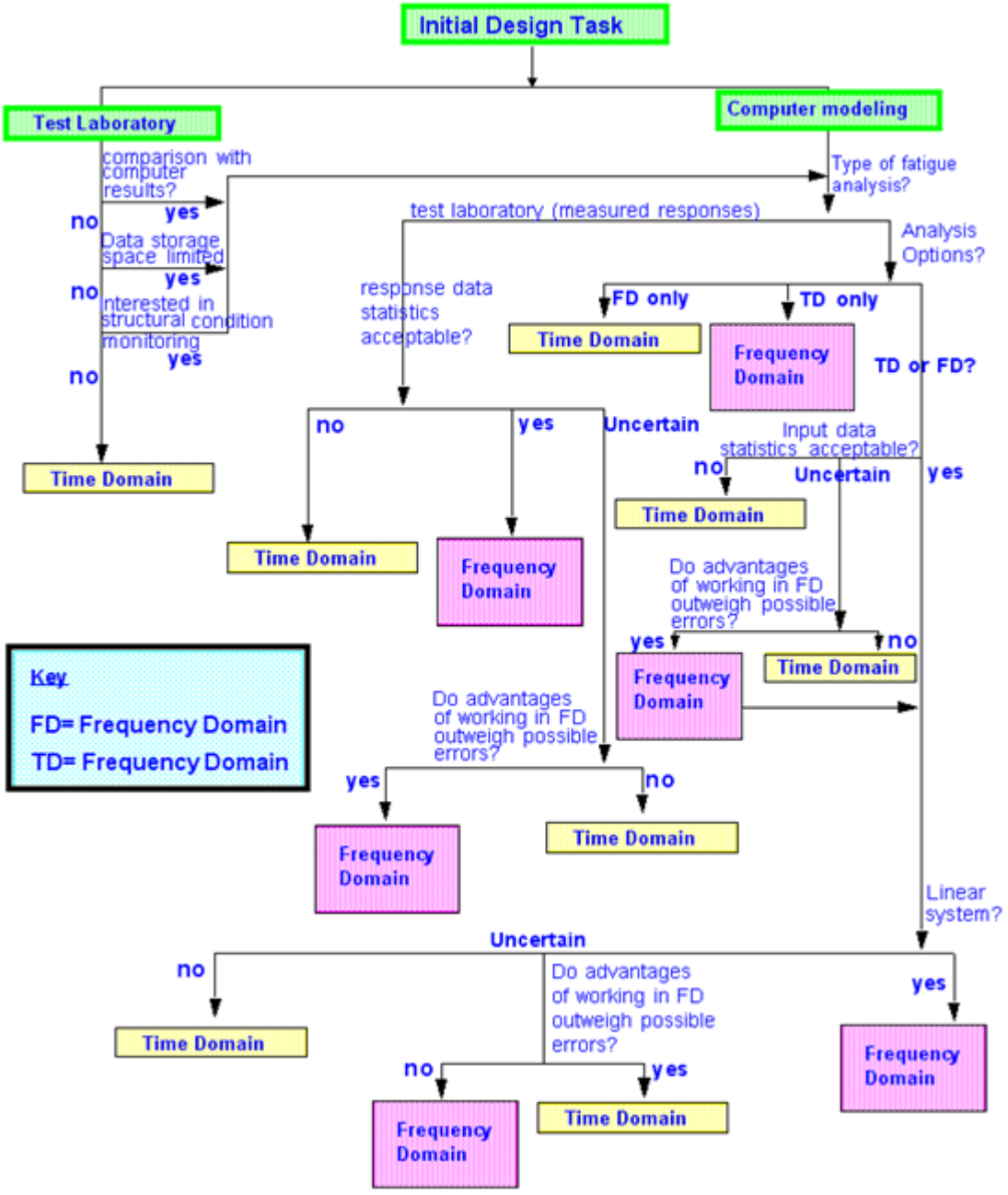


Figure 9-39 Frequency or Time Domain Decisions?

An Introduction to Random Process Theory

This section introduces random processes to facilitate an understanding in the theory behind vibration fatigue analysis. It is not intended that this serve as a definitive reference on the subject but rather a primer on the concepts involved. The description starts with an account on the theory of the Power Spectral Density Function (PSD) and progresses to cover single random processes and then multiple, partially correlated random processes.

Representing a Time Signal Using Fourier Series

Any periodic time history may be represented by the summation of a series of sinusoidal waves of various amplitude, frequency and phase. This is the basis of Fourier series expansion, a detailed explanation is given in (Ref. 100.). The Fourier series expansion of a time history y(t) is often expressed by (9-1).

$$y(t) = A_o + \sum_{n=1}^{\infty} \left\{ A_n cos\left(\frac{2\pi n}{T} \cdot t\right) + B_n sin\left(\frac{2\pi n}{T} \cdot t\right) \right\} \tag{9-1}$$

T		period

and

$$A_o = \frac{1}{T} \int_{-T/2}^{T/2} y(t)dt$$

$$A_n = \frac{2}{T} \int_{-T/2}^{T/2} y(t) cos\left(\frac{2\pi n}{T} \cdot t\right) dt$$

$$B_n = \frac{2}{T} \int_{-T/2}^{T/2} y(t) sin\left(\frac{2\pi n}{T} \cdot t\right) dt$$

The terms A_0, A_n and B_n are termed Fourier coefficients and provide information on the frequency content of the time history. A_0 represents the mean of the time history while A_n and B_n represent the amplitude of the various cosine and sine waves which when added together comprise the time history. Figure 9-40 shows the Fourier series expansion for a particular saw tooth time history. By the third summation the expansion is seen to give a reasonable representation of the original time history. Further summations improve the representation.

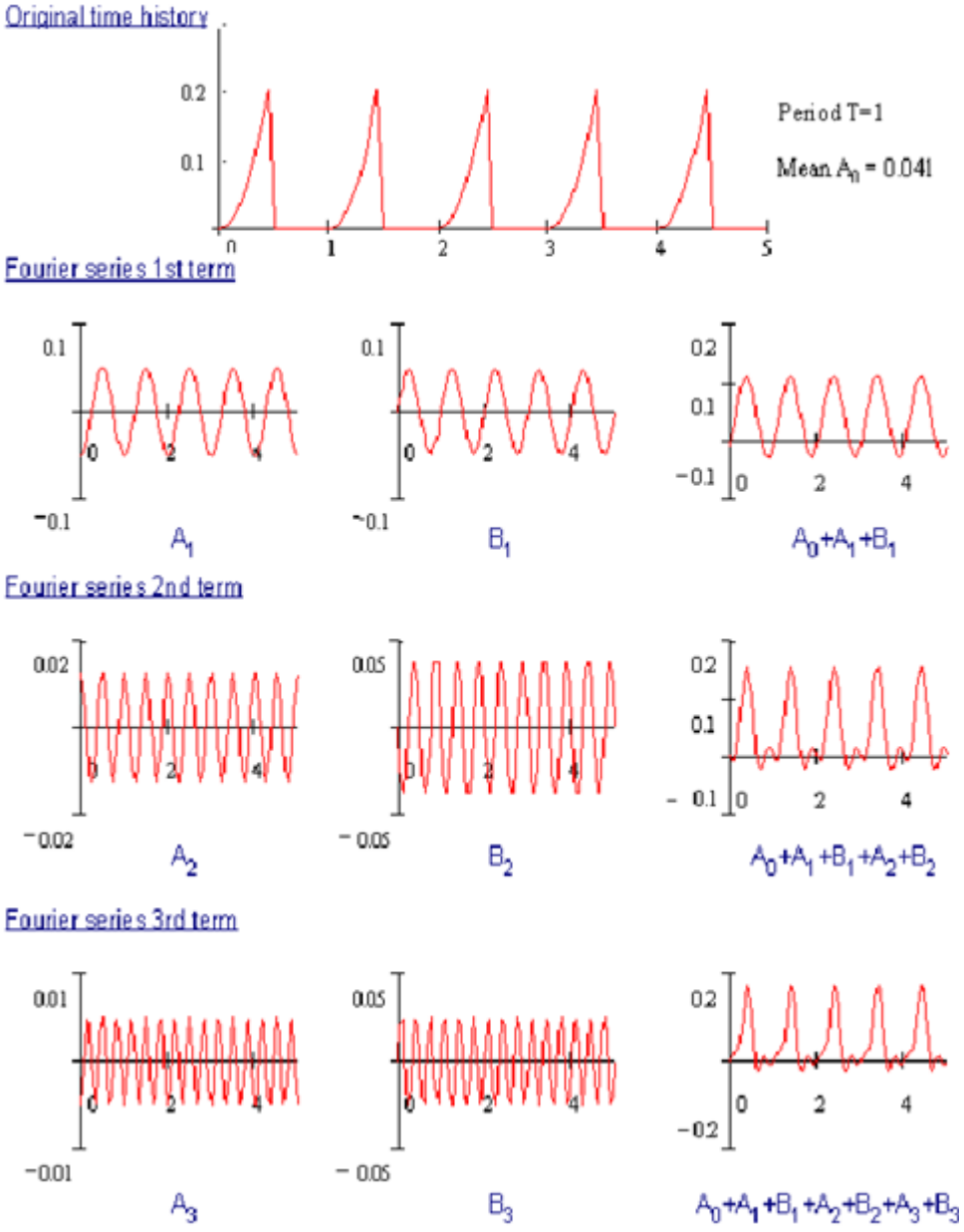


Figure 9-40 Fourier Series Expansion

The integrals within A_n and B_n effectively act as a filter and extract the amplitude of a cosine or sine wave of frequency n/T Hz. from the time history. This is accomplished using the relationships given in (9-2).

$$\int_{-T/2}^{T/2} \cos\left(\frac{2\pi m t}{T}\right) \cos\left(\frac{2\pi n t}{T}\right) = 0 \text{ for m} \neq n, \frac{T}{2} \text{ for m=n} \tag{9-2}$$

$$\int_{-T/2}^{T/2} \sin\left(\frac{2\pi m t}{T}\right) \sin\left(\frac{2\pi n t}{T}\right) = 0 \text{ for m} \neq n, \frac{T}{2} \text{ for m=n}$$

$$\int_{-T/2}^{T/2} \cos\left(\frac{2\pi m t}{T}\right) \sin\left(\frac{2\pi n t}{T}\right) = 0 \text{ for all m and n}$$

The Complex Fourier Series

In practice the Fourier coefficients A_0, A_n and B_n prove very cumbersome to manipulate algebraically. Complex number theory helps with this aspect as all three coefficients may be replaced by one complex coefficient C_n.

The first thing to realize is that the sum of a cosine and sine wave results in a sinusoidal wave of amplitude r and initial phase angle ϕ provided both waves have the same frequency. Figure 9-41 shows the summation of a cosine and sine wave with amplitudes A and B respectively and frequency ω.

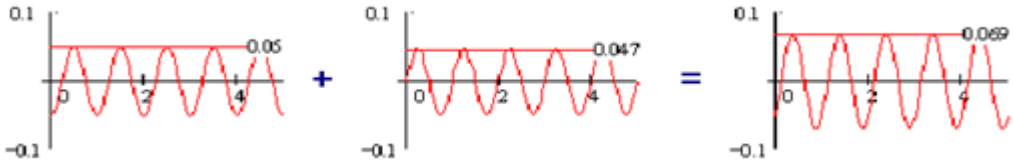


Figure 9-41 Summation of Cosine and Sine Waves

The amplitude r and initial phase angle ϕ of the resultant sinusoidal wave can be obtained using the relationship given in (9-3).

$$r_n = \sqrt{A_n^2 + B_n^2} \qquad \phi_n = atan\left(\frac{B_n}{A_n}\right) \tag{9-3}$$

It is convenient to express the Fourier coefficients A_n and B_n in terms of the complex coefficient Λ_n where $\Lambda_n = A_n - i.B_n$. The amplitude r and phase angle ϕ can then be obtained from the 'magnitude' and 'argument' of the complex coefficient as shown in (9-4).

$$r_n = |\Lambda_n| \quad \text{and} \quad \phi_n = \angle\Lambda_n \quad \text{where} \quad \Lambda_n = A_n - iB_n \tag{9-4}$$

Having expressed the amplitude and initial phase of the sinusoidal wave in complex terms it is now convenient to utilize the complex form of the exponential function to represent the frequency term. This technique is explained in (Ref. 100.), and follows the relationship given in (9-5).

$$e^{\pm i\left(\frac{2\pi n}{T} \cdot t\right)} \Leftrightarrow \cos\left(\frac{2\pi n}{T} \cdot t\right) \pm \sin\left(\frac{2\pi n}{T} \cdot t\right) \tag{9-5}$$

Using these relationships the original Fourier term $A_n \cos(\omega_n.t) + B_n \sin(\omega_n.t)$ can be replaced with the complex number representation given in (9-6).

$$A_n cos(\omega_n t) + B_n sin(\omega_n t) = \Re\left\{\Lambda_n e^{i\omega_n t}\right\} \tag{9-6}$$

(9-1) may therefore be written in complex form by (9-7).

$$y(t) = A_0 + \Re\left\{\sum_{n=1}^{\infty} \Lambda_n \cdot e^{i\left(\frac{2\pi n}{T}\right)\cdot t}\right\} \tag{9-7}$$

The expression may be simplified further by the introduction of negative frequencies. These have the effect of cancelling the imaginary components in (9-7) without using the $\Re()$ function. Using the concept of negative frequencies (9-7) can be expressed as (9-8).

$$y(t) = A_0 + \frac{1}{2}\sum_{n=1}^{\infty} \Lambda_n \cdot e^{i\left(\frac{2\pi n}{T}\right)\cdot t} + \frac{1}{2}\sum_{n=-1}^{-\infty} \Lambda_n \cdot e^{i\left(\frac{2\pi n}{T}\right)\cdot t} \tag{9-8}$$

Changing the limits of the summation the Fourier coefficients A_0 and Λ_n can be replaced with the single complex Fourier coefficient C_n as expressed in (9-9).

$$y(t) = \sum_{n=-\infty}^{\infty} C_n \cdot e^{i\left(\frac{2\pi n}{T}\right)\cdot t} \quad \text{Double Sided Fourier Series} \tag{9-9}$$

$$C_n = \frac{1}{2}(A_n - iB_n) = \frac{1}{T}\int_{-T/2}^{T/2} y(t) \cdot e^{-i\left(\frac{2\pi n}{T}\right)\cdot t}\, dt$$

(9-9) is said to be double sided because it uses both positive and negative frequencies to represent the Fourier coefficients. Figure 9-42 shows a plot of the Real and Imaginary parts of the complex Fourier coefficients C_n obtained from the expansion of the saw tooth wave plotted with respect to frequency n/T Hz.

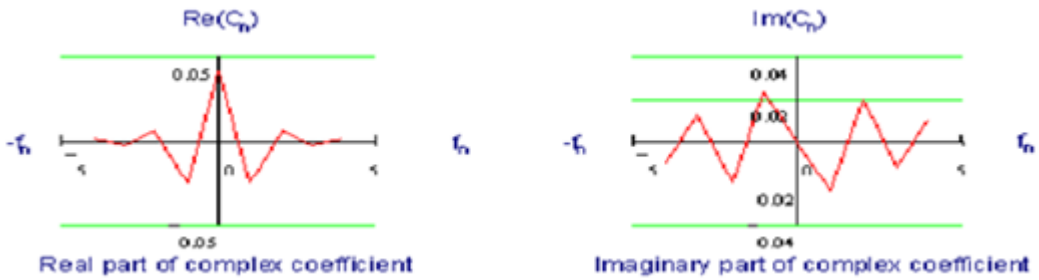


Figure 9-42 Plot of Real and Imaginary Parts of Complex Fourier Coefficient

The complex Fourier coefficient C_n is used to obtain the amplitude and phase of the nth sinusoidal wave using the formulation given in (9-10).

$$\text{amplitude } r_n = |C_n| + |C_{-n}| \qquad \text{phase } \phi_n = \angle C_n \tag{9-10}$$

Where $C_{-n} = A_n + i\,B_n$ and $C_n = A_n - i\,B_n$.

The Fourier Density Coefficient

The previous section introduced the complex Fourier series expansion, we are particularly interested in obtaining the complex Fourier coefficients C_n and these are discussed in this section. Each Fourier coefficient C_n is obtained for a frequency of n/T Hz, the frequency interval between each coefficient Δf is therefore 1/T Hz. This causes problems as the frequency at which the coefficients are calculated is dependent on the period T chosen. It is common practice to normalize the coefficients to eliminate the dependence on T. The normalized coefficients take the form of a 'density function' and the Fourier coefficient is obtained from the area under the density curve for the range Δf in question. This is shown in Figure 9-43.

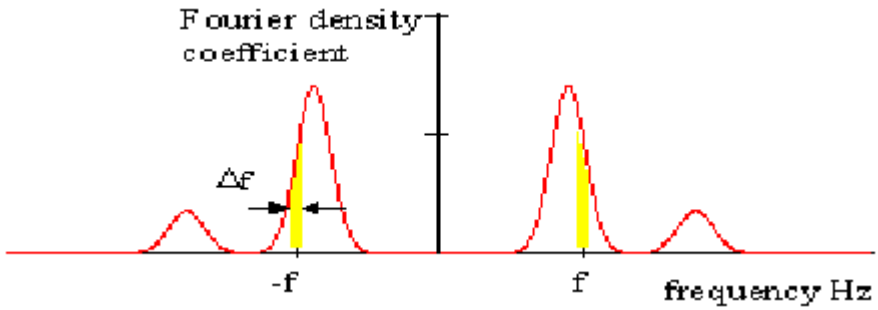


Figure 9-43 Fourier Density Coefficients

Using this normalization the complex Fourier series expansion given in (9-9) can be expressed by (9-11).

$$y(t) = \Delta f \cdot \sum_{n=-\infty}^{\infty} c_n \cdot e^{i\left(\frac{2\pi n}{T}\right)\cdot t} \tag{9-11}$$

$$c_n = \int_{-T/2}^{T/2} y(t) \cdot e^{-i\left(\frac{2\pi n}{T}\right) \cdot t} dt \quad \text{and} \quad \Delta f = \frac{1}{T}$$

The amplitude and phase of the sinusoidal wave with frequency f is now obtained by taking the magnitude and argument of the area under the density curve in the region Δ f. The function behaves much like a Probability Density Function.

If $\Delta(f \to 0)$ then the summation given in (9-11) can be expressed in integral form and c_n can be written as y(f). Changing the limits of integration between $-\infty$ and $+\infty$ the complex Fourier series is now expressed in integral form as (9-12). This is known as the Fourier Transform Pair. For future reference in this chapter y(f) shall be called the "Fourier Spectrum."

$$y(t) = \int_{-\infty}^{\infty} y(f) \cdot e^{i(2\pi f)t} df \quad \text{Inverse Fourier Transform} \tag{9-12}$$

$$y(f) = \int_{-\infty}^{\infty} y(t) \cdot e^{-i(2\pi f)t} dt \quad \text{Fourier Transform}$$

It is worth noting that many references use a different normalization to that given above, an alternative normalization is to use the square root of the period.

The Fourier Transform Pair

The 'Fourier Transform Pair' was derived earlier in integral form and is expressed in (9-12). It is now apparent that a time history y(t) can be completely expressed by the "Fourier Spectrum" y(f). The Fourier spectrum can therefore be thought of as another 'frequency domain' in which a time history may be expressed. In this sense, y(f) implies a description of the time history 'y' in the frequency domain 'f'. The 'Fourier Transform Pair' effectively transforms between the two domains.

In addition to the integral form, the 'Fourier Transform Pair' can also be described in a discrete form. This is particularly beneficial as most measured time histories are obtained in a discrete, digitized form with values taken over equally spaced intervals in time. In these circumstances the integral form is difficult to process and it is desirable to perform some numerical routine on the measured time history to transform it into the frequency domain. This transform is known as the "Discrete Fourier transform." The result obtained from a discrete transformation will tend to the result obtained from an integral transformation as the sample length and sampling frequency increase.

Cooley and Tukey (Ref. 101.) devised a very rapid discrete Fourier transform algorithm in 1965. It is known as the "Fast Fourier Transform (FFT)" and has a reverse process called the "Inverse Fourier Transform (IFFT)." There are a number of derivatives of these each using a different normalization. The discrete form of (9-12) is defined in (9-13).

$$y(t_k) = \frac{1}{T} \cdot \sum_n y(f_n) \cdot e^{i \cdot \left(\frac{2\pi k}{N}\right) \cdot n} \quad \text{Inverse Fourier Transform} \tag{9-13}$$

$$y(f_n) = \frac{T}{N} \cdot \sum_k y(t_k) \cdot e^{i \cdot \left(\frac{2\pi n}{N}\right) \cdot k}$$ Fourier Transform

Fourier Analysis of Random Time Histories

Under certain circumstances, random time histories may be expressed in the frequency domain. By definition, a random time history cannot be periodic. However, provided the time history is taken from a 'stationary random process' then it may be expressed in the frequency domain. A stationary process is defined as a process whose statistics are not affected by a shift in the time origin. (i.e. the statistics of a time history X(t) are the same as a time history X(t + τ) for all values of τ) For non-stationary processes the statistics obtained from a sampled time history would not be representative of those of the whole random process as these would be continuously changing. Priestley, (Ref. 102.), discusses stationarity at length. The analysis of non-stationary random processes is not considered in this text.

Figure 9-44 illustrates the application of the Discrete Fourier Transform pair to a random time history taken from a stationary random process.

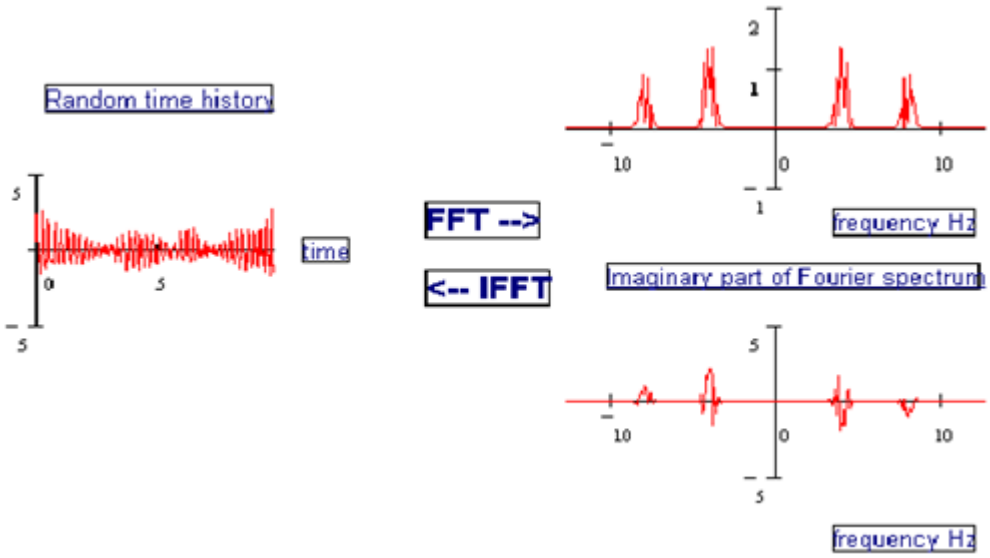


Figure 9-44 Frequency Domain Representation of a Random Time History

For real time histories the Fourier transform is symmetric about zero Hz. A Double Sided Spectrum is said to be symmetric if the relationships given by (9-14) are satisfied.

$$\Re\{f(\omega)\} = \Re\{f(-\omega)\} \quad \text{and} \quad I\{f(\omega)\} = I\{-f(-\omega)\} \tag{9-14}$$

It is usual to express a symmetric spectrum as a “Single Sided Spectrum.” The “Single Sided Spectrum” contains the same information as the “Double Sided Spectrum” but is often more convenient to use as negative frequencies are not considered. All measured time histories can be expressed as a Single Sided Spectrum. The relationship between single and double sided spectra is given by (9-15).

$$y_s(f) = 2 \cdot y_D(f) \tag{9-15}$$

Figure 9-45 illustrates the Single Sided Spectrum of the random time history given above.

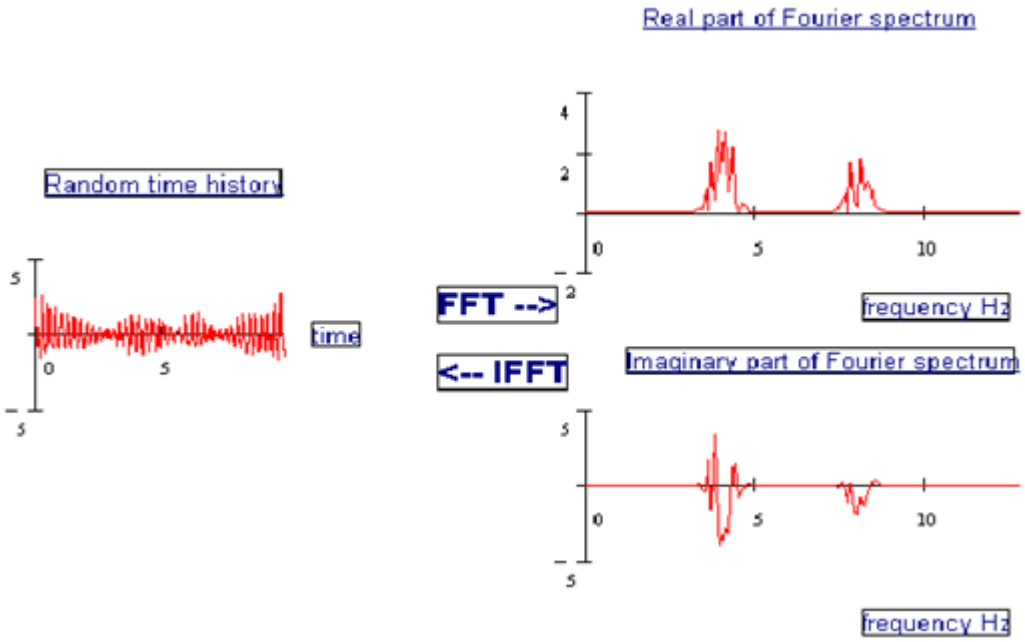


Figure 9-45 Single Sided Fourier Transform

Power Spectral Density Function (PSD)

We have considered the representation of a stationary random time history in the frequency domain. Often it is more beneficial to have a statistical description of the random process rather than the deterministic one. For this purpose, the "Power Spectral Density (PSD)" function is useful. A good historical and qualitative description of what the PSD actually is is given by Spence (Ref. 103.). The PSD gives a statistical representation of a stationary random process in the frequency domain. It is defined such that the area beneath the PSD represents the mean square amplitude of the random process. Many design standards, such as the ESDU papers (Ref. 104.) on wind turbulence and the design standards for offshore sea conditions, express the statistical properties of random processes in terms of PSDs. Using these in conjunction with a linear structural model allows PSDs of structural stresses and deflections to be calculated and fatigue life estimates to be obtained. These subjects are covered later in the thesis. The PSD is used in the same way as the Fourier Density spectra. The mean square amplitude of a sinusoidal wave of frequency f can be obtained by taking the sum of the area under the PSD at +f and -f over the interval ∂f as $\partial f \to 0$. This is illustrated in Figure 9-46.

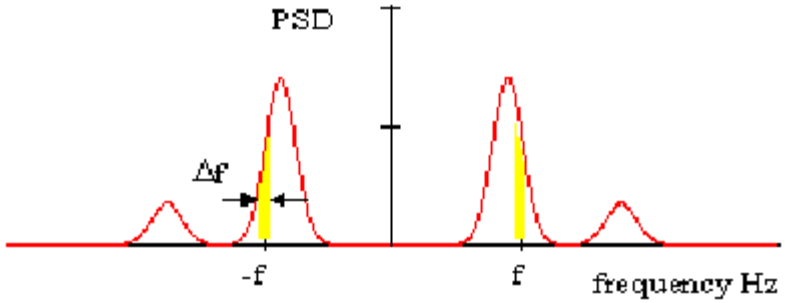


Figure 9-46 Double Sided Power Spectral Density (PSD)

PSDs may be complex and expressed as both double or single sided spectra in the same way as the Fourier density spectra. The relationships given in (9-14) and (9-15) hold for single and double-sided PSDs. PSDs calculated from measured time histories will always be real. The real part of a complex PSD S(f) is known as the co-spectral density function and the imaginary part as the quad-spectra density.

We will now consider the transformation from a single sided Fourier Density Spectra to a single sided PSD. The mean amplitude of the component sinusoidal waves over a frequency range Δ f is obtained from the Fourier spectra by taking the modulus of the area under the curve as expressed in (9-16).

$$Amplitude(f) = \Delta f \cdot y(f) \tag{9-16}$$

The area under a PSD represents the mean squared amplitude of the component sinusoidal waves where $MeanSquare(f) = 1/2 \cdot Amplidue^2$. We can equate the mean square amplitudes calculated from the PSD and the Fourier spectra and hence determine the transformation between the two, this is derived in (9-17).

$$\Delta f \cdot G(f) = \frac{1}{2} \cdot \Delta f^2 \cdot |y(f)|^2 \tag{9-17}$$

$$\therefore G(f) = \frac{1}{2 \cdot T} \cdot |y(f)|^2$$

The double sided PSD is determined in a similar fashion, the results are summarized in (9-18).

$$S(f) = \frac{1}{T} \cdot |y_D(f)|^2 \text{ or } S(f) = \frac{1}{T} \cdot (y_D(f) \cdot \overline{y_D(f)}) \text{ for double sided PSDSs or} \quad (9\text{-}18)$$

$$G(f) = \frac{1}{2T} \cdot |y_S(f)|^2 \text{ or } G(f) = \frac{1}{2T} \cdot (y_S(f) \cdot y_S(f)^*) \text{ for single sided PSDSs}$$

Where $y_D(f)^*$ is the complex conjugate of $y_D(f)$.

Both the modulus and complex conjugate forms of the transformations are correct however it is mathematically beneficial to use the complex conjugate form. It is also worth noting that many books adopt a different notation for the double and single sided PSDs. In this text the notation S(f) and G(f) is adopted for the double and single sided spectra respectively.

Earlier we commented on the use of different normalizations applied to the Fourier transform. It is quite common to see the $\sqrt{T}$ normalization being used and this changes the formulation expressed in (9-18) to the form $S(f) = |y_D(f)|^2$ or $S(f) = (y_D(f) \cdot y_D(f)^*)$. The normalization chosen for the Fourier transform is not important provided that the resultant PSDs are the same.

The PSD offers considerable information about the statistics of the random process. By definition the area under the PSD represents the mean square amplitude of the time history. For zero mean time histories the standard deviation may be calculated from the square root of this. Other statistical properties may be obtained using the spectral moments of the PSD. The nth spectral moment m_n of a PSD is defined by (9-19).

$$m_n(S) = \int_{-\infty}^{\infty} S(f) \cdot f^n \, df \quad (9\text{-}19)$$

The key statistical properties listed in (9-20) were derived by S.O. Rice (Ref. 99.) in 1954. These properties become important when considering the fatigue life of a structure. This is explained in detail in Characterization of Structural Response in the Frequency Domain, 747.

Standard deviation	$\sigma = \sqrt{m_0}$
Expected number of zero crossings	$E_0 = \sqrt{\frac{m_2}{m_0}}$
Expected number of peaks	$E_P = \sqrt{\frac{m_4}{m_2}}$
Irregularity factor	$\gamma = \frac{E_0}{E_P}$ (9-20)

The appearance of the PSD plot also conveys much information, Figure 9-47 shows four types of process. The distinction between narrow and broad banded processes becomes much more apparent from the PSD plots than the time histories.

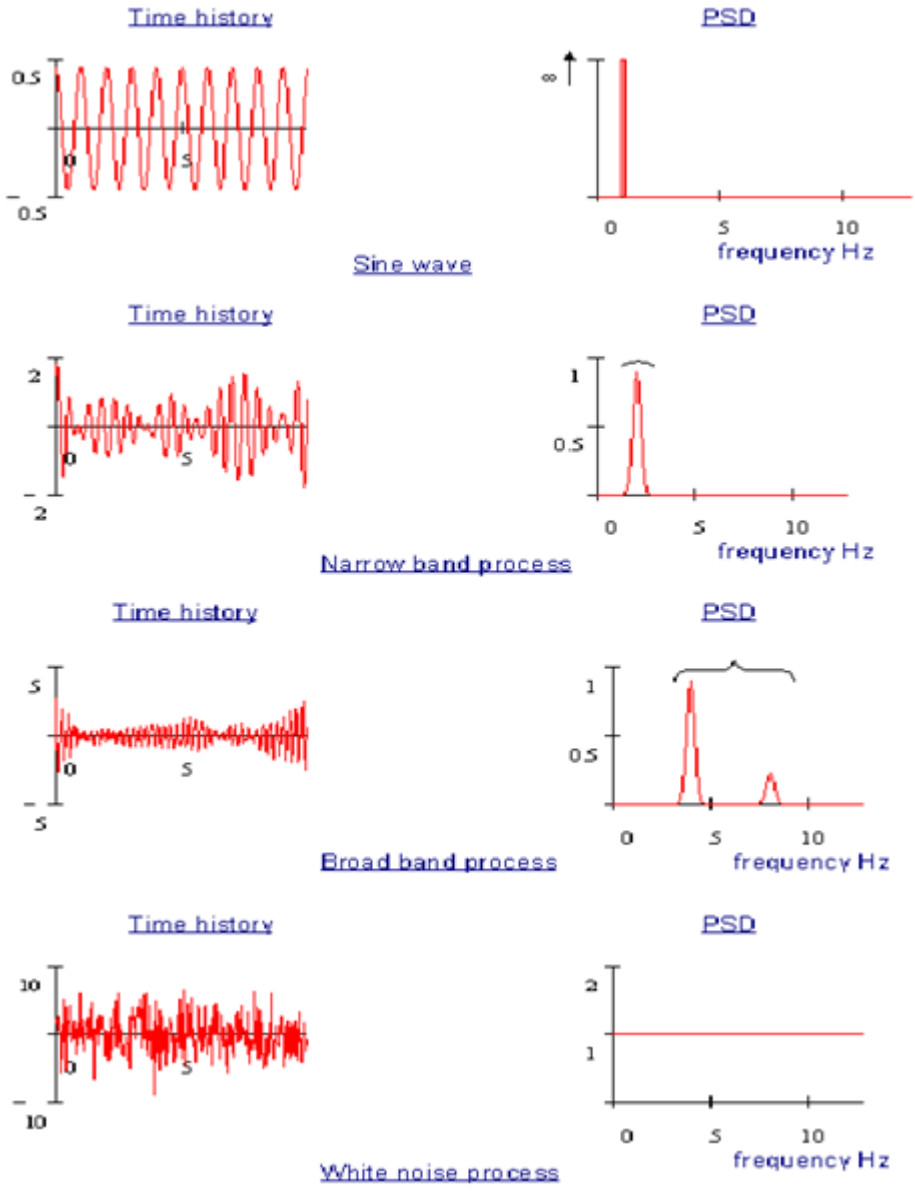


Figure 9-47 Time Histories and Corresponding PSDs

Time Signal Regeneration from PSDs

Earlier it was shown how a time history may be expressed in either the time or frequency domains. Transformation from one domain to the other is carried out using the Fourier transform pair. PSDs express the statistics of a random process in the frequency domain and in many instances it is desirable to obtain a

time history from a PSD. PSDs are given in many of the design standards where random loading is involved. If the designer wishes to determine how a non-linear structure will react to a typical random load history it is necessary to regenerate a characteristic time history from the design PSD, this can then be analyzed using a dynamic analysis program in the time domain. For linear structures this step is not necessary as the dynamic analysis may be carried out entirely in the frequency domain. This text deals with linear analysis however it is still beneficial to describe the process of time signal regeneration from PSDs.

Unlike the Fourier density spectra the PSD does not contain information about the phase relationships between the sinusoidal waves that make up the time history. It does however, contain information on the Mean square amplitude of each wave. In order to regenerate a time history it is necessary to reintroduce the phase relationships between each of the waves. For many signals it has been found that the phase relationships follow a uniform random distribution between 0 and π and with this knowledge it is therefore possible to regenerate a time history. Time histories obeying this trend are said to be 'Gaussian Random' processes. The regenerated time history will not be the same as the original measured time history since the phase angles are now different. It is still statistically characteristic though. There are a number of methods used for time history regeneration, this text looks at the most intuitive method of evaluating the random phase angles and taking the inverse Fourier transformation.

The time history is formed by the summation of a number of sinusoidal waves of differing frequency, amplitude and phase. The nth wave may be expressed in the form given in (9-21) where ϕ_{rnd} is the random phase angle between 0 and π radians.

$$y(t)_n = r_n \cdot cos(\omega \cdot t + \phi_{rnd}) \tag{9-21}$$

Using the complex relationships given in (9-3) and (9-4), the double sided PSD and random phase angle can be expressed in complex form as (9-22).

$$S(f)_n = T \cdot (A_n^2 + B_n^2) \quad \text{and} \quad \phi_{rnd_n} = atan\left(\frac{B_n}{A_n}\right) \tag{9-22}$$

Solving these two simultaneous equations yields values for the Fourier coefficients A_n and B_n defined in (9-4). Using the normalization described earlier the complex Fourier density function can be determined. Having obtained the Fourier density function the time history regeneration can be accomplished by taking the inverse Fourier transform. The formulation is given in (9-23).

$$.y(t)_n = \text{Inverse Fourier Transform of } y(f_n)$$

$$y(f)_n = T \cdot (A_n - i \cdot B_n)$$

and

$$A_n = \sqrt{\frac{S(f_n)}{T \cdot (1 + \varphi_n^2)}} \qquad B_n = A_n \varphi_n^2 \qquad \varphi_n = tan(\phi_{rnd_n}) \tag{9-23}$$

The method proves difficult to implement computationally however because large numeric precision errors occur when taking the tangent of a random number between 0 and π. This is not a serious problem in the context of this thesis but should be considered for other applications where time history regeneration is required. (9-24) gives the formulation for regenerating a time history from a single sided PSD.

$y(t)_n$ = Inverse Fourier Transform of $y(f_n)$

$y(f)_n = 2 \cdot T \cdot (A_n - i \cdot B_n)$

and

$$A_n = \sqrt{\frac{G(f_n)}{2 \cdot T \cdot (1 + \varphi_n^2)}} \qquad B_n = A_n \varphi_n^2 \qquad \varphi_n = tan(\phi_{rnd_n}) \tag{9-24}$$

Summary of Fourier Transformation and PSDs

Figure 9-48 illustrates the transformation between the time and frequency domains. The frequency domain is another way of representing a time history. Transformation between the time and frequency domains is accomplished using the Fourier Transform Pair. There are two forms of transform, the integral and the discrete transform. The discrete transform is particularly useful for transforming digitally measured time histories such as those recorded using data acquisition. The "Fast Fourier Transform" is now universally used for this purpose.

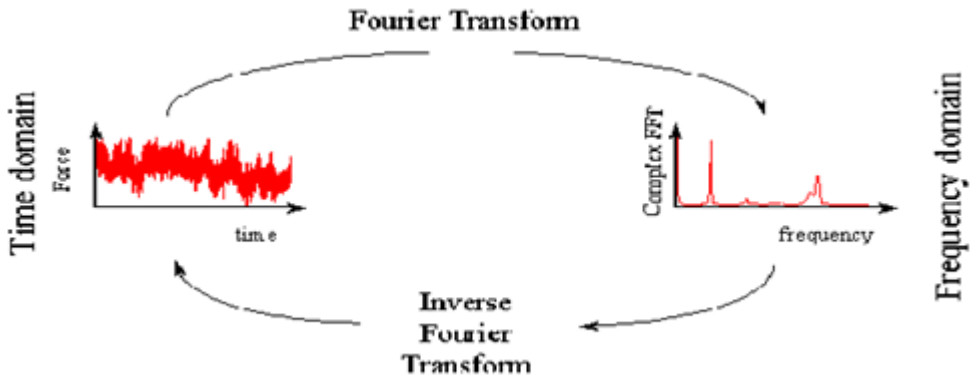


Figure 9-48 The Fourier Transform Between Time and Frequency Domains

The Fourier transform gives a Fourier Density Spectrum. This is complex and may be single or double sided. Single sided spectra are a special case of the double sided spectra where the negative frequencies are the complex conjugate of the positive frequencies. This condition arises when the time history is Real. All measured time histories are real and are usually expressed as single sided spectra.

It is often more convenient to deal with the statistics of a random process than either a time or frequency domain representation of the actual signal. For this reason the Power Spectral Density (PSD) is used. The PSD represents the Mean Square amplitude of the sinusoidal waves that comprise the time history. It is often necessary to regenerate a statistically representative time history from a PSD, the PSD does not contain information on the phase relationships between the sinusoidal waves and this therefore needs adding. Provided the process is 'Gaussian Random' then the phase relationships will follow a uniform random distribution. Figure 9-49 illustrates the transformation between the time domain and the PSD.

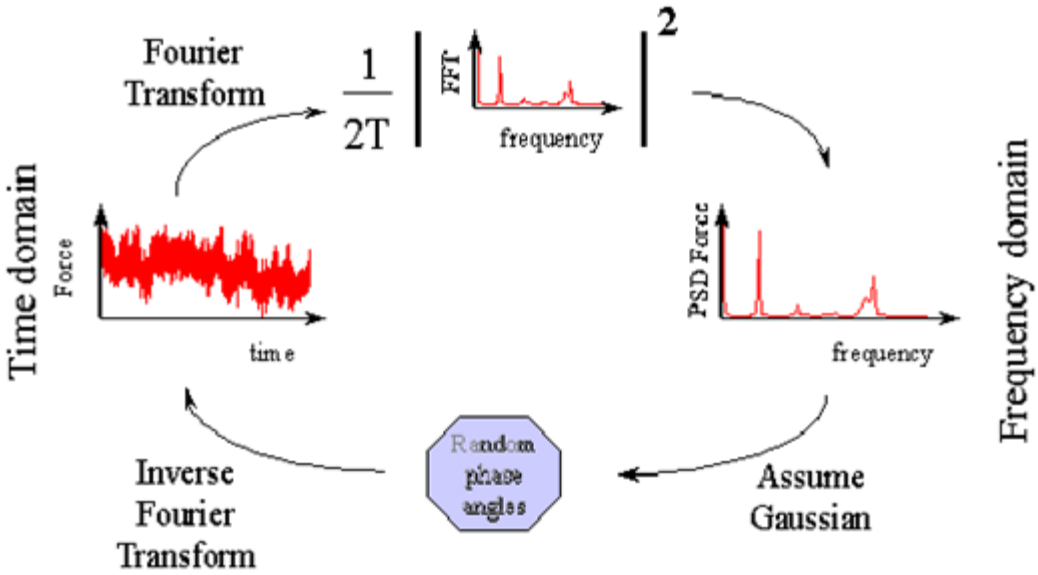


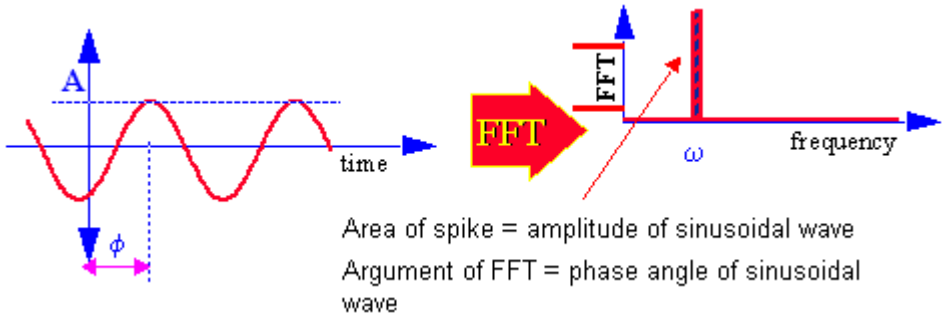
Figure 9-49 Transformation Between Time History and PSD

What Does the FFT Tell Us?

The figure below shows a physical representation of what the frequency domain graphs are showing.

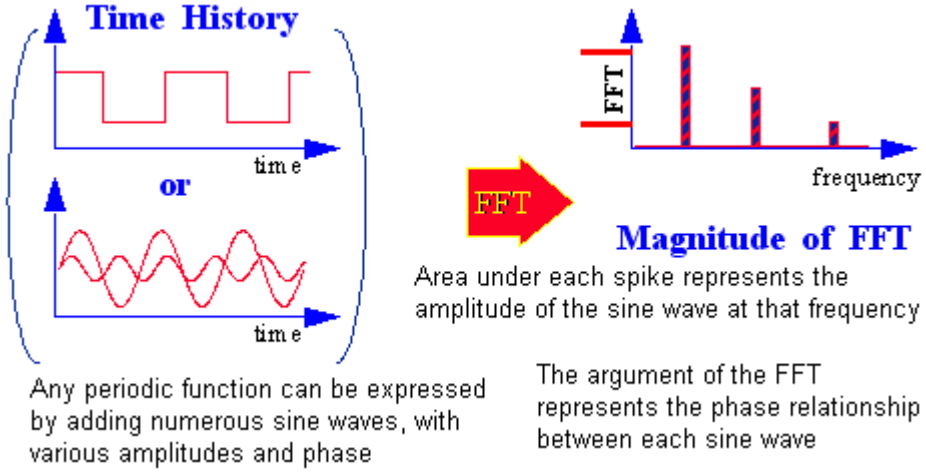
The FFT is a complex number given with respect to frequency. A sine wave of frequency ω, amplitude A and initial phase angle ϕ is represented in the frequency domain by a spike occurring at ω along the frequency axis. If the magnitude of the complex FFT is plotted, then the area under the spike is found to be the amplitude A of the sine wave. When the argument of the complex FFT is plotted then the area is found to be initial phase angle ϕ of the sine wave.

The frequency, amplitude and phase of the sine wave is therefore retained in both the time and frequency domains. As it is rather cumbersome to plot complex numbers, we generally plot two separate graphs; one for the amplitude and the other for the phase angle.



How Do We Use FFTs?

The French Mathematician J. Fourier (1768-1830) postulated that any periodic function can be expressed as the summation of a number of sinusoidal waves of varying frequency, amplitude and phase. Each individual sinusoidal wave can be expressed as a spike in the frequency domain and as the number of sine waves increase, the difference in frequencies between them tend to zero and so the spikes tend to merge into a continuous function. Therefore, if we want to find the amplitude and phase of the sinusoidal waves in a particular frequency range, say between 2 and 2.5 Hz, we can measure the area under the curve in that frequency range. As this area is given as a complex number, the amplitude content can be obtained by taking the modulus of this and the phase content from the argument.



For many engineering cases we are only interested in the amplitude of the various sine waves and are not concerned with the initial phases. In fact, in many cases we find that the initial phase angle is totally random, and so it proves unnecessary to show it. For this reason the Amplitude Spectral Density (ASD) function alone is usually shown. This is simply a plot of the modulus of the complex FFT given with respect to frequency.

Response of a Linear System to a Single Random Process

The work carried out so far has enabled us to represent a measured time history in the frequency domain in the form of a PSD. This measured time history may be that of wind speed measured using anemometers or stresses in a structural component measured using strain gauges. In the case of a wind turbine, for example, the PSD is likely to be given for the turbulent wind speed witnessed near the ground. It is important to understand how a structure will react to the dynamic wind loading. We could take the PSD and regenerate a statistically representative time history and then analyze this using a dynamic analysis program in the time domain but this is very time consuming as the analysis process is complicated. For structures which behave in a linear way the analysis may be done more efficiently in the frequency domain.

A linear system is one where the output is related to the input by a linear transfer function. In the case of a wind turbine blade the load on the turbine blade could therefore be expressed as (9-25).

e.g., the aerodynamic force on a turbine blade is given as the product of the wind speed v(t) and the linear aerodynamic transfer function g.

$$Force(t) = g \cdot v(t) \tag{9-25}$$

If the wind speed is expressed in terms of the Fourier density function then the aerodynamic force will then be given in the frequency domain using (9-26).

$$Force(t) = g \cdot v(f) \tag{9-26}$$

It is convenient to obtain the response as a PSD $S^F(f)$. From (9-18), the double sided PSD of force can be expressed as $S^F(f) = (1/T)(g \cdot v(f) \cdot g^* \cdot v(f)^*)$. If the wind speed is also expressed as a double sided PSD $S^v(f)$ then the PSD of force is given by a linear transfer function in the frequency domain as in (9-27).

$$S^F(f) = g \cdot g^* \cdot S^v(f) \quad \text{or} \quad S^F(f) = |g|^2 \cdot S^v(f) \tag{9-27}$$

This relationship also holds between single sided PSDs as expressed in (9-28).

$$G^F(f) = g \cdot g^* \cdot G^v(f) \quad \text{or} \quad G^F(f) = |g|^2 \cdot G^v(f) \tag{9-28}$$

Dealing With Multiple, Partially Correlated Random processes

The previous section describes the transformation of random time histories into the frequency domain using the Fourier transform pair. A simple relationship can be used to generate the Power Spectral Density function (PSD) which contains statistical properties of the random time history. We shall now look at an alternative method for generating PSDs. The method uses a statistical property of the time history, namely the Autocovariance function, as a method of obtaining the PSD. Although the Fourier transform is now universally used for obtaining PSDs, the method discussed here is important in understanding the concept of random processes. We shall expand on this theory to also include multiple random events, and will define the cross-power spectral density function to represent the statistical relationships between two random events.

Definition of the Autocovariance Function

The Autocovariance function is a statistical property that describes the frequency or periodic properties of a time history. The autocovariance function is defined in (9-29) where E{} is the expectation operator defined by

$$E\{y(t)\} = \frac{1}{T} \cdot \int_{-T/2}^{T/2} y(t)dt$$

$$C_{yy}(\tau) = E\{y(t) \cdot y(t+\tau)\} - E\{y(t)\}^2 \quad (9\text{-}29)$$

The notation C_{yy} indicates the autocovariance function for a single process y. The notation is introduced at this stage to distinguish between the 'cross-covariance function (C_{xy})' for two random processes x and y, this is discussed in this text. For all analysis carried out in this thesis random processes are assumed stationary and have a zero mean. (9-29) can therefore be rewritten as the infinite time average given by (9-30), the Autocovariance function.

$$C_{yy}(\tau) = \lim_{T \to \infty} \left[\frac{1}{T} \cdot \int_{-T/2}^{T/2} y(t) \cdot y(t+\tau)dt \right] \quad (9\text{-}30)$$

The autocovariance function is plotted for a number of time histories in Figure 9-50. For periodic processes with period T, the autocovariance function is also periodic with the same period. For stationary processes the autocovariance function is even, i.e. $C_{yy}(\tau)=C_{yy}(-\tau)$, and may be expresses as a single sided function. As stationarity is assumed for all random processes throughout this text the single sided autocovariance function is used.

Note: For t = 0, the autocovariance tends to the variance of the time history.

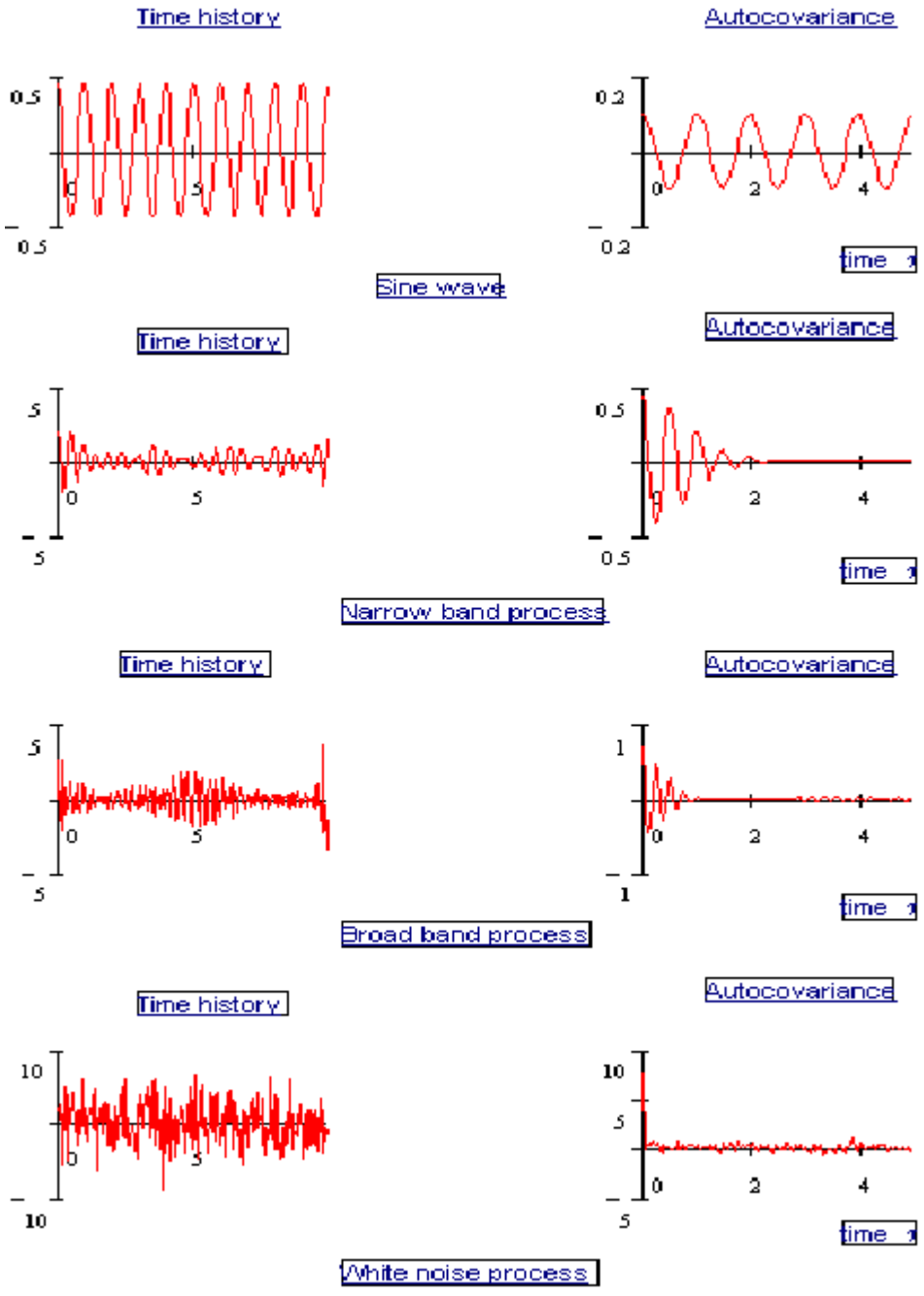
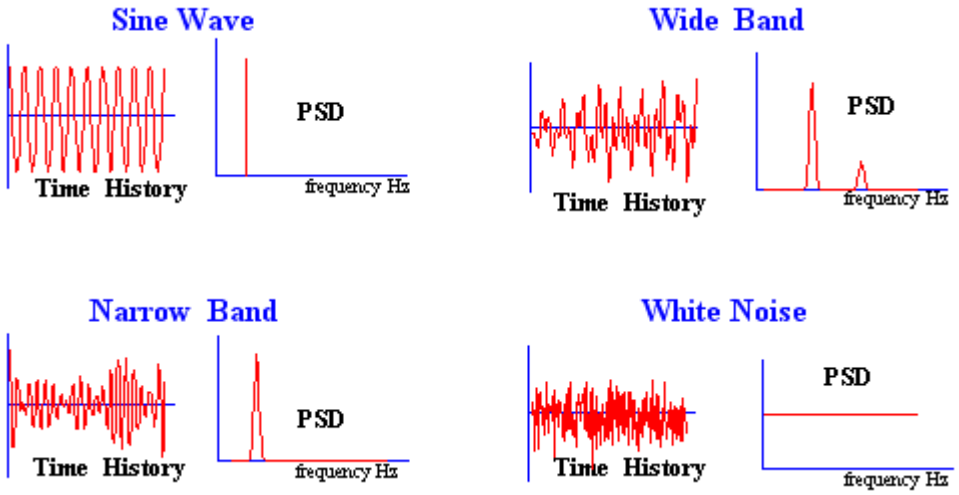


Figure 9-50 Time Histories and Corresponding Autocovariance's

A sinusoidal time history appears as a single spike on the PSD plot. The spike is centered at the frequency of the sine wave and the area of the spike represents the mean square amplitude of the wave. In theory this spike should be infinitely tall and infinity narrow for a pure sine wave, however because of the numerical analysis the spike will have a finite width and will therefore have a finite height. Remember, with PSD plots we are interested in the area under the graph and not the height of the graph.

A **narrow band** process is one which is built up of sine waves covering only a narrow range of frequencies. A narrow band process is typically recognized in the time history by the amplitude modulation, often referred to as a beat envelope.



Wide band processes are built up of sine waves over a broad range of frequencies. These are shown in the PSD plot as either a number of separate spikes or one wide peak covering many frequencies. This type of process is usually more difficult to identify from the time history but is typically characterized by its positive valleys and negative peaks.

White noise is a a special time history which is built up of sine waves over the whole frequency range.

Transformation Between Autocovariance and PSD

In this section it is shown that the Fourier transform pair can be used to transform between the autocovariance function and the PSD. In this sense the autocovariance function is similar to a time history representation of the PSD. In effect it is the time history of the mean square amplitude of the sinusoidal waves with a zero phase angle.

The autocovariance function for a zero mean, stationary random process is defined in (9-30). From (9-12) the term $y(t+\tau)$ may be expressed in terms of the Fourier transform pair as (9-31).

$$y(t+\tau) = \int_{-\infty}^{\infty} y(f) \cdot e^{i2\pi f(t+\tau)} df \tag{9-31}$$

Substituting this into (9-30) and simplifying yields (9-32).

$$C_{yy}(\tau) = \lim_{T\to\infty}\left[\frac{1}{T}\cdot\int_{-\infty}^{\infty}\int_{-T/2}^{T/2} y(f)\cdot e^{i2\pi f\tau}dt\cdot y(t)\cdot e^{i2\pi ft}df\right] \tag{9-32}$$

Now from (9-12) we know that

$$\lim_{T\to\infty}\left[\int_{-T/2}^{T/2} y(t)\cdot e^{i2\pi ft}dt\right] = y(-f) = y(f)^*$$

therefore, the autocovariance function may be expressed as (9-33).

$$C_{yy}(\tau) = \frac{1}{T}\cdot\int_{-\infty}^{\infty} y(f)\cdot y(f)^*\cdot e^{i2\pi f\tau}df \tag{9-33}$$

It is now possible to recognize the similarity with (9-18) used to transfer from the Fourier density function to the PSD. Using this relationship the autocovariance function may be expressed in terms of the PSD as (9-34), i.e. the inverse Fourier transform of the PSD $S_{yy}(f)$

$$C_{yy}(\tau) = \int_{-\infty}^{\infty} S_{yy}(f)\cdot e^{i2\pi f\tau}df \tag{9-34}$$

The transformation between the autocovariance function and the PSD can therefore be accomplished using the Fourier transform pair as shown in (9-35).

(9-35)

$C_{yy}(\tau) = \int_{-\infty}^{\infty} S_{yy}(f)\cdot e^{i2\pi f\tau}df$	The inverse Fourier transform
$S_{yy}(f) = \int_{-\infty}^{\infty} C_{yy}(\tau)\cdot e^{-i2\pi f\tau}d\tau$	The Fourier transform

The notation S_{yy} is adopted here to describe the Auto-power spectral density function as opposed to the cross-power spectral density function (S_{xy}) between two random processes x and y, we will look at this next.

Multiple Random Processes: The Principle of Cross-covariance

To this point we have considered the analysis of one random process, this section introduces the analysis of multiple random processes. Figure 9-51 shows two anemometers measuring wind speed placed side by side with separation d. From each anemometer a time history of the wind speed is produced. PSDs can be calculated from these and provided that the wind field is homogeneous, both PSDs will be the same. From this it is concluded that the mean square amplitudes of the sinusoidal waves which comprise the time histories are the same. The PSDs however give no information about the phase relationships between the two measured time histories.

In this section we are interested in the sequential relationship between the two time histories. If the two anemometers are far enough apart then the wind speed witnessed by one will be completely independent of

the other, they are said to be uncorrelated. As they are moved closer together then a correlation between the two time histories will be noted. Correlation occurs because the random turbulent wind incident on anemometer x is sufficiently large to also be influencing anemometer y.

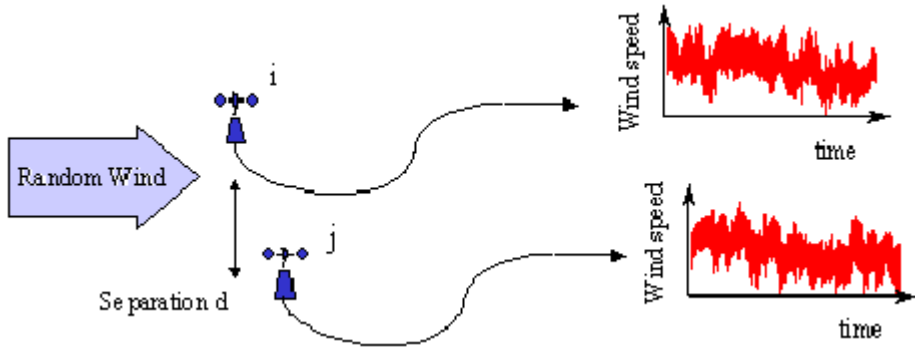


Figure 9-51 Multiple Random Processes

The covariance function may be used to find the sequential relationships between the two time histories in the same way as before. In this case (9-29) is written as (9-36). This is known as the Cross-covariance function. In this case we now compare time histories x and y.

$$C_{yx}(\tau) = E\{y(t) \cdot x(t+\tau)\} - E\{y(t) \cdot x(t)\} \tag{9-36}$$

The notation C_{yx} is used to describe the Cross-covariance function between the two random processes x and y. It effectively tells us what the effect on y would be should a random process hit x. Assuming that the random processes are stationary and have a zero mean then the cross-covariance function may be transformed to a PSD using the Fourier transform. The resulting PSD S_{yx} is now termed the 'Cross-power Spectral Density Function'. This relationship is expressed in (9-37).

(9-37)

$C_{yx}(\tau) = \int_{-\infty}^{\infty} S_{yx}(f) \cdot e^{i2\pi f\tau} df$	The cross-covariance function between processes x and y.
$S_{yx}(f) = \int_{-\infty} C_{yx}(\tau) \cdot e^{-i2\pi f\tau} d\tau$	The cross-power spectral density function between processes x and y.

Normalized Covariance and Time Scales

It is often convenient to normalize the covariance function in order to compare different time histories with different scales of measurement. This is achieved by dividing the covariance by the variance. The normalized covariance function is defined by (9-38), it has the same shape as the covariance function except that the ordinate is scaled to have a value of 1 for $\tau = 0$.

$$\rho_{yy}(\tau) = \frac{C_{yy}(\tau)}{C_{yy}(0)} = \frac{C_{yy}(\tau)}{\sigma_y^2} \tag{9-38}$$

The time scale is defined as the area under the normalized covariance function as (9-39). For a random time history this roughly equates to the mean zero crossing period of the time history. For a periodic time history this is not the case however as $T_y \to 0$.

$$T_y = \int_0^\infty \rho_{xy}(\tau) \cdot d\tau \tag{9-39}$$

The Coherence Function

Another way of representing the correlation between two random processes is through the coherence function. The coherence represents the degree of correlation between two random processes. Two uncorrelated events show zero coherence where as two fully correlated events show unit coherence. The coherence function is defined in (9-40) as a ratio of the cross-power spectral density function to the geometric mean of the two auto-power spectral density functions.

$$\gamma_{xy}(f) = \frac{|S_{xy}(f)|}{\sqrt{S_{xx}(f) \cdot S_{yy}(f)}} \tag{9-40}$$

Many design standards quote the auto-power spectral density function which the designer should use to verify the design. Where there are correlated multiple events it is often easier to express the correlation in terms of the coherence function than to specify many cross-power spectral density functions separately. The designer may then derive his own cross-spectral density functions by rearranging (9-40). For certain analyses, such as wind turbulence, the coherence function is expressed as a function of frequency and separation distance.

Response of a Linear System to Multiple, Partially Correlated Random Loads

Figure 9-52 illustrates the loading on a wind turbine blade due to two random processes. In order to calculate the reaction to this loading, it is insufficient to simply sum the reactions from the two PSDs. Instead, we must sum the reactions from the PSDs and cross-PSDs. The cross power spectra contain information on the joint statistics of the two processes. If the two processes are correlated then the sequencing effect of the two processes may act to either increase or decrease the overall loading on the blade. A very good mathematical explanation of this can be found in Newland (Ref. 105.).

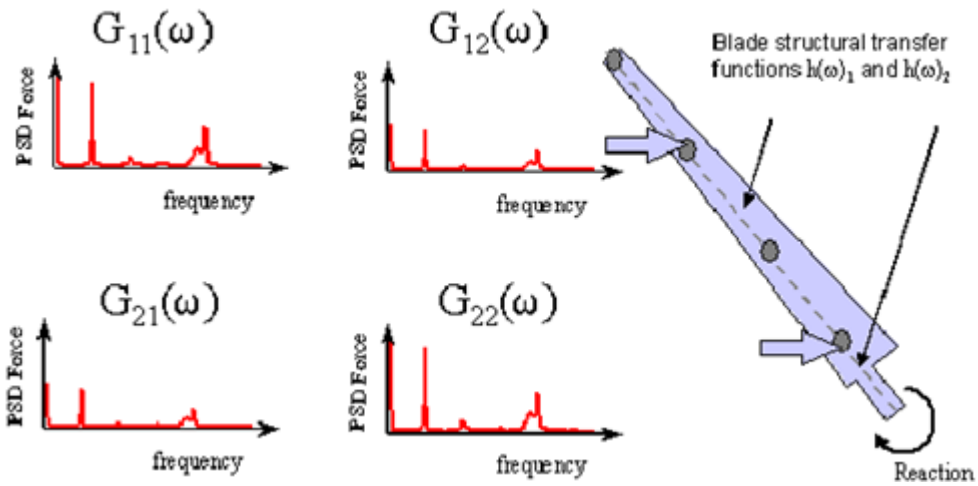


Figure 9-52 Reaction to Multiple Random Loading

The total reaction force on the blade can be found in the time domain from (9-41).

$$Force(t) = \sum_i g_i \cdot v_i(t) \tag{9-41}$$

If the wind speed is expressed in terms of the Fourier density function then the aerodynamic force will then be given in the frequency domain using (9-42).

$$Force(f) = \sum_i g_i \cdot v_i(f) \tag{9-42}$$

It is convenient to obtain the response as a PSD $S^F(f)$. From (9-18) the double sided PSD of force can be expressed as $S^F(f) = (1/T) \cdot (\sum g_i \cdot v_i(f) \cdot \sum g_j^* \cdot v_j(f)^*)$. If the wind speed is also expressed as a double sided PSD, $S^v(f)$, then the PSD of force is given by (9-43).

$$S^F(f) = \sum_i \sum_j g_i \cdot \bar{g}_i \cdot S_{ij}^v(f) \tag{9-43}$$

This relationship also holds between single sided PSDs as expressed in (9-44).

$$G^F(f) = \sum_i \sum_j g_i \cdot g_i^* \cdot G_{ij}^v(f) \tag{9-44}$$

Matrix Form of the Cross-power Spectral Density Function

The Auto- and cross-power spectral density functions may be expressed in matrix form, this permits the rapid calculation of responses using matrix algebra. The auto-power spectral values are located along the leading diagonal while the cross-power terms are located in the remaining cells as illustrated in Figure 9-53.

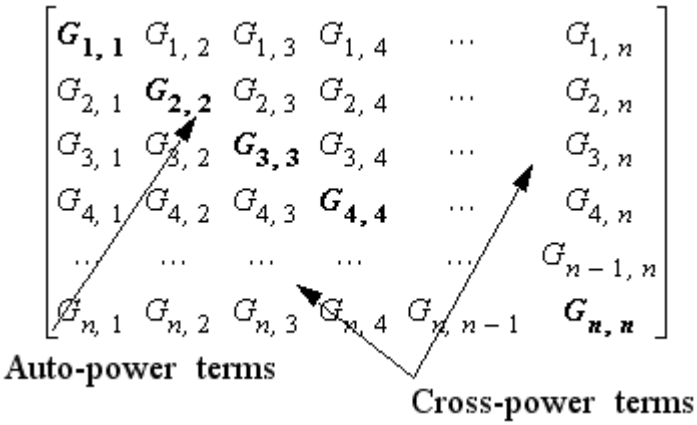


Figure 9-53 Cross-power Spectral Density Matrix

Using matrix algebra, (9-44) may be expressed as (9-45).

$$G^F(f) = [g]^T \cdot [G(f)] \cdot [\bar{g}] \tag{9-45}$$

Where [G(f)] is the cross-power matrix and [g] is a vector of gain factors.

Time Domain Characterization of Fatigue Life Estimation

With MFLF a traditional S-N curve as shown in Figure 9-54 is used to model the material properties of the components being analyses.

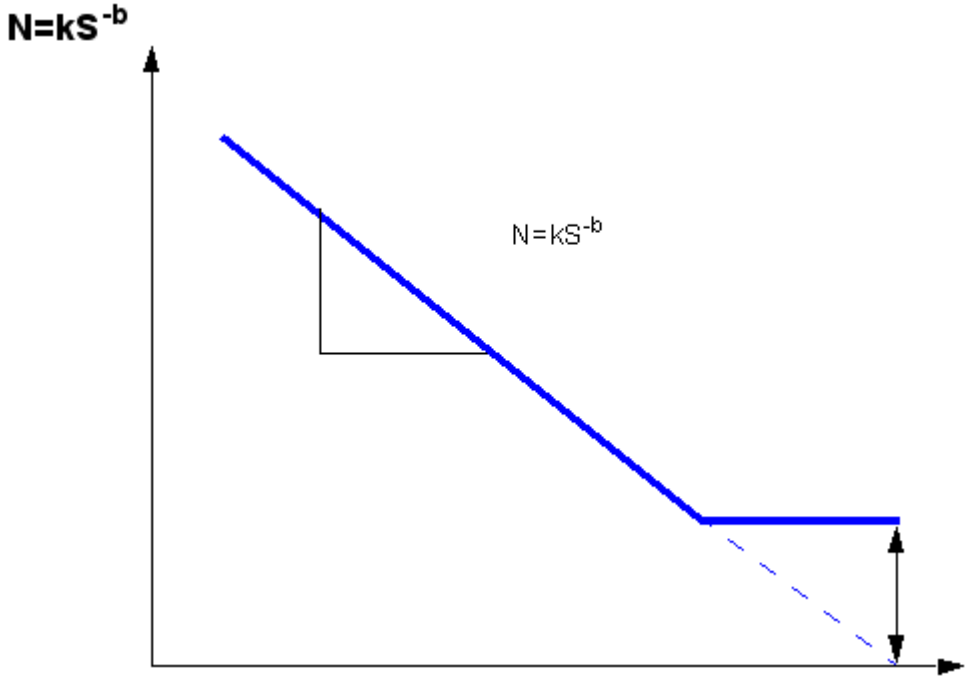


Figure 9-54 S-N Relationship - an Experimental Law for Constant Amplitude Loading

This figure simply shows that, under constant amplitude cyclic loading, a linear relationship exists between cycles to failure N and applied stress range S when plotted on log-log paper.

This is an alternative form of the S-N curve to that used for the rest of the MSC Fatigue tools (see appendix 2) where,

$$b = -\frac{1}{b1} \tag{9-46}$$

$$k = (SRI1)^{-\frac{1}{b1}} \tag{9-47}$$

Because real signals rarely conform to this ideal constant amplitude situation, an empirical approach is used for calculating the damage caused by stress signals of variable amplitude. Despite its limitations, Miner's rule is generally used for this purpose as shown in Figure 9-55.

The relationship between the form of S-N curve used for MFLF (material constants b and k) and the convention used for the rest of MSC Fatigue tools (material constants b1 and SRI1):

$$MFLFN.S^{b} = K \tag{9-48}$$

$$FEFATN^{-b1}.S = SRI1 \tag{9-49}$$

i.e. $\log N + b.\log S = \log K$

The above two equations are valid for all N and S, therefore

$$b = -\frac{1}{b1} \tag{9-50}$$

$$k = SRI1^{-\frac{1}{b1}} \tag{9-51}$$

and

$$b1 = -\frac{1}{b} \tag{9-52}$$

$$SRI1=K^{-b} \tag{9-53}$$

An example...

b1 = -0.102

SRI1 = 1868 MPa

b=9.8

K= 1.147×10^{32} $(MPa^{9.8})$

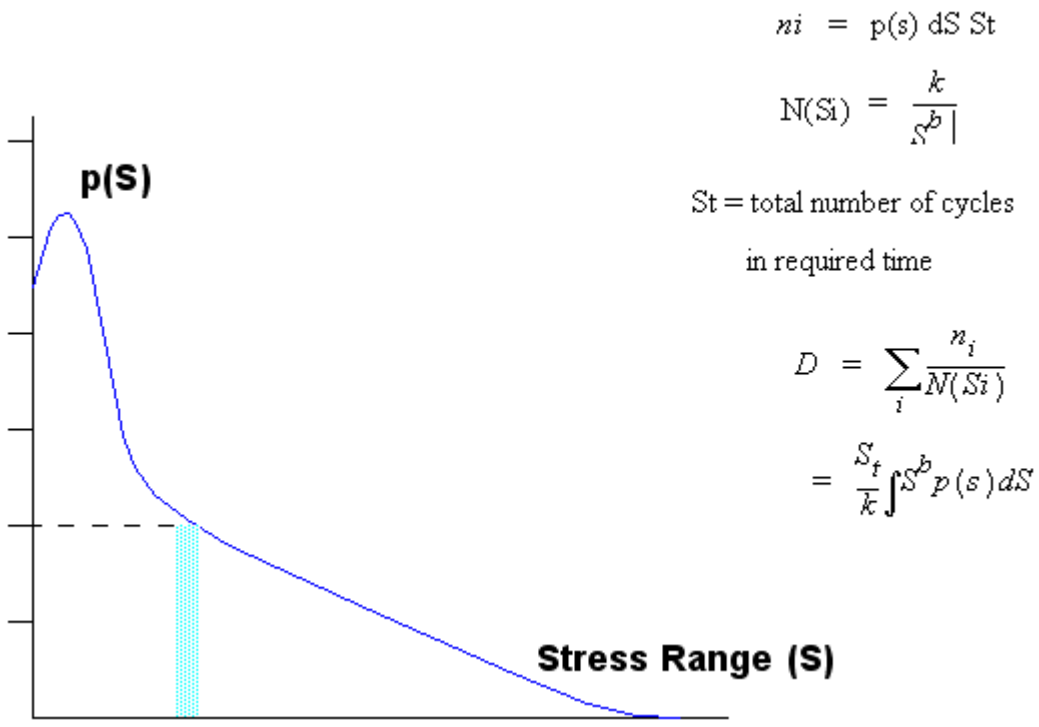


Figure 9-55 Palmgren-Miner Cumulative Damage Hypothesis - Linear Damage Rule and Probability Density Functions (PDFs)

Percentage of damage (Damage Ratio)

$$D = \sum_i \frac{n_i}{N(S_i)} \tag{9-54}$$

n_i= actual counted number of cycles

$N(S_i)$=allowable (from S-N plot) number of cycles

The linear relationship in Figure 9-55 assumes that the damage caused by parts of a stress signal with a particular range can be calculated and accumulated to the total damage separately from that caused by other amplitudes. A ratio is calculated for each stress range, equal to the number of actual cycles at a particular stress range n divided by the allowable number of cycles to failure at that stress N (obtained from the S-N curve). Failure is assumed to occur when the sum of these ratios, for all stress ranges, equals 1.0.

If n is obtained from a time history of stress, then the most convenient way of storing the information is in the form of a probability density function (PDF) of stress ranges. A typical representation of this function is the diagram in Figure 9-55.

Since **rainflow ranges** of stress are generally regarded to give the best indication of the fatigue damaging potential of a random signal, it is the PDF of rainflow ranges which is the desired result. Rainflow ranges have

been widely used for estimating fatigue damage from random signals since Matsuishi and Endo first introduced the concept to the scientific community over twenty years ago (Ref. 77.). When the loading sequence is specified as a time history, the procedure for calculating rainflow ranges is relatively simple. An example of the way rainflow ranges are extracted from a time signal is given in Figure 9-56. For the potential of a random signal, it is the PDF of rainflow ranges which is the desired result. Rainflow ranges have been widely used for estimating fatigue damage from random signals since Matsuishi and Endo first introduced the concept to the scientific community over twenty years ago. When the loading sequence is specified as a time history, the procedure for calculating rainflow ranges is relatively simple.

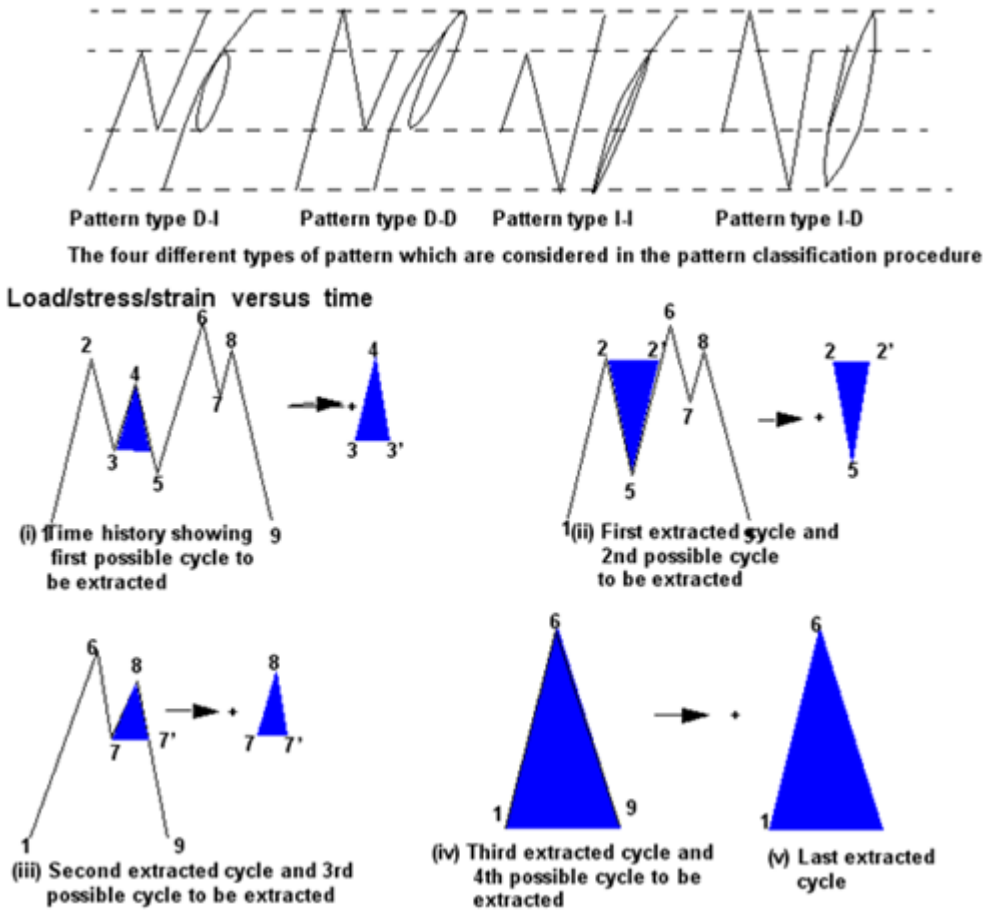


Figure 9-56 Rainflow Cycle Counting

An important characteristic of any time signal is related to its frequency content. Figure 9-57 shows two different types of structural response. In each case both the time series and its corresponding PSD are shown. The right hand response contains predominantly one frequency as indicated by its PSD. The corresponding time signal shows a classical **narrow band** response, i.e., constant frequency, slowly varying envelope. The structural response on the left contains a spread of frequencies, as indicated by its PSD. This can also be

identified in its corresponding time series. This type of response is termed **wide band**. If the PSD has a flat top extending (theoretically) to infinity (as is the case with so-called white noise or shot noise) then the process is often referred to as **broad band**.

An important distinction between these two types of responses, in terms of their fatigue damaging content, can be determined by considering the form of peak pdf's shown in Figure 9-58. Most importantly, for the narrow band signal virtually all peaks occur above the mean level. Because of this, and the fact that the envelope is slowly varying, the PDF of peak position is identical to that of (half) cycle ranges. Also, because of the slowly varying envelope these cycle ranges are also rainflow ranges.

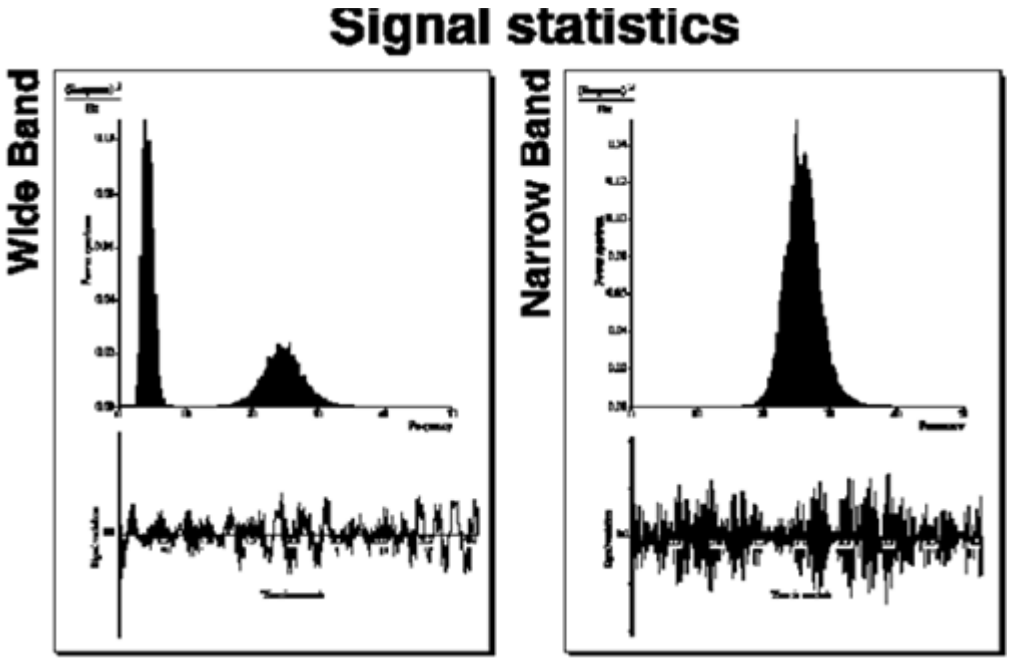


Figure 9-57 Signal Statistics

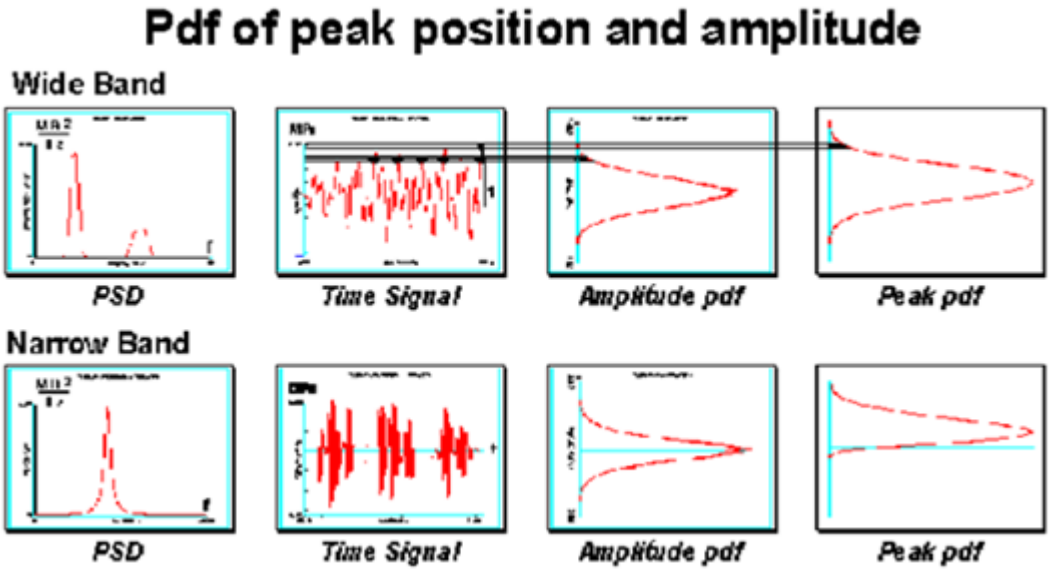


Figure 9-58 Pdf of Peak Position and Amplitude

A numerical indication of this characteristic can be determined from the number of so-called ***zero crossings*** and ***peaks*** in the signal. Figure 9-59 shows a section cut out from a typical wide band signal. E[0] represents the number of (upward) zero crossings, or mean level crossings for a signal with a non-zero mean. E[P] represents the number of peaks in the same sample. These are both normally specified for a typical one second sample. The ***irregularity factor*** (γ) represents the number of zeros divided by the number of peaks. This is discussed further in the next section.

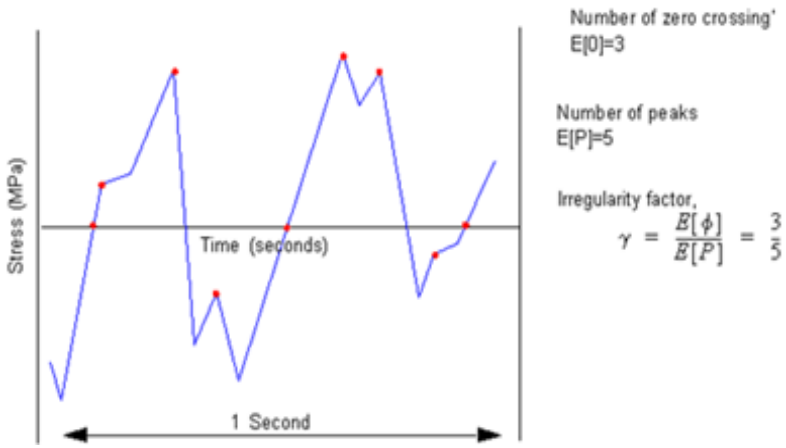


Figure 9-59 Expected Zeros, Peaks, and Irregularity Factor

Characterization of Structural Response in the Frequency Domain

Since we are concerned with structural systems analyzed in the frequency domain, we require a method for extracting the PDF of rainflow ranges directly from the PSD of stress. The characteristics of the PSD which are used to obtain this information are the nth moments of the PSD function: as shown in Figure 9-60 below.

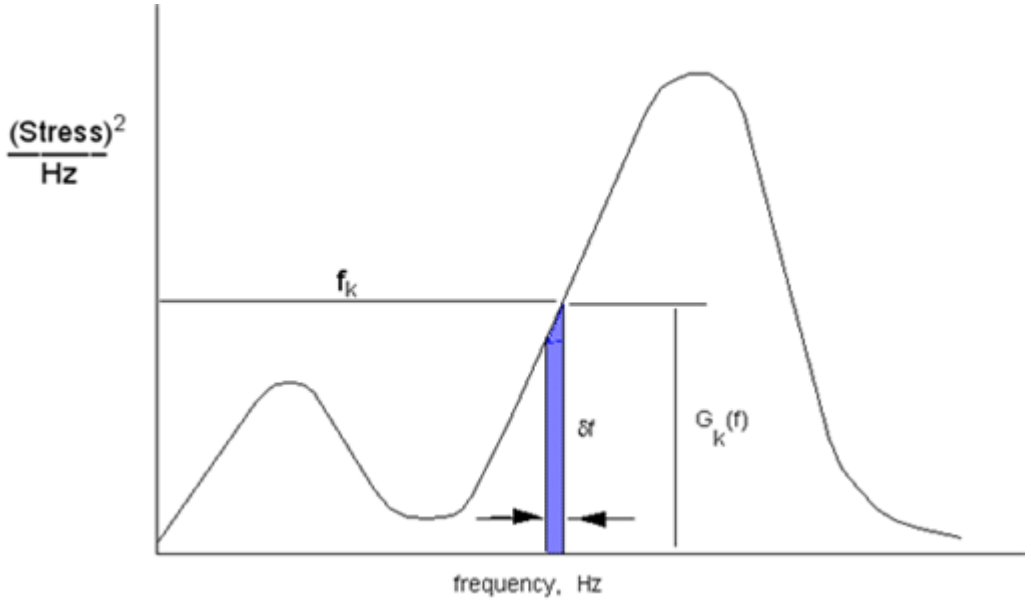


Figure 9-60 Moments from a PSD

Some very important statistical parameters can be computed from these moments such as the expected number of zeros and peaks per second. From these the irregularity factor, γ, can be computed which, as described earlier, gives an indication of the spread of frequencies present in the signal: This is illustrated in the equations below.

$$Mn = \int_0^{\infty} G(f)df = \sum_{k=1}^{m} f_k^n Gk(fk)\delta f \tag{9-55}$$

$$E[0] = \left[\frac{m_2}{m_2}\right]^{\frac{1}{2}} \quad \text{Zero Crossing} \tag{9-56}$$

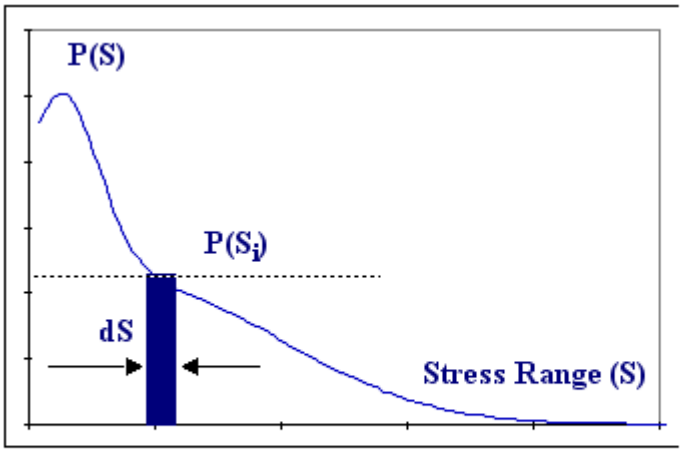
$$E[P] = \left[\frac{m_4}{m_2}\right]^{\frac{1}{2}} \quad \text{Peaks} \tag{9-57}$$

$$\gamma = \frac{E[0]}{E[P]} = \left[\frac{m2^2}{m0m^4}\right]^{\frac{1}{2}} \quad \text{Irregularity Factor} \tag{9-58}$$

The value of γ varies between 1.0 and 0.0. A value of 1.0 corresponds to a narrow band signal which, as the term implies, means that the signal contains only one predominant frequency. A value of 0.0 implies that the signal contains an equal amount of energy at all frequencies.

Probability Density Functions (PDFs)

The most convenient way, mathematically, of storing stress range histogram information is in the form of a probability density function (PDF) of stress ranges. A typical representation of this function is shown below. It is very easy to transform from a stress range histogram to a PDF, or back. The bin widths used, and the total number of cycles recorded in the histogram are the only additional pieces of information required.



To get PDF from rainflow histogram divide each bin height by

$$S_t \times dS$$

S_t = total number of cycles

dS = bin width

The probability of the stress range occurring between $S_i - \frac{dS}{2}$ and $S_i + \frac{dS}{2} = P(S_i) \cdot dS$

Fatigue Damage for Random Response Histories

Once the stress range histogram has been converted into a stress range PDF then there is an elegant and efficient equation to describe the expected fatigue damage caused by this loading history. The probability density function P(S) simply represents the characteristics of the loading. In order to compute fatigue damage over the lifetime of the structure (T) the form of material (S-N) data must also be defined using the parameters k and m. In addition, the total number of cycles in time T must be determined from the number of peaks per second E[P]. If the damage caused in the time T is greater than 1.0 then the structure is assumed to have failed. Or alternatively, the fatigue life can be obtained by setting T = 1.0 and then finding the fatigue life in seconds from the equation.

Fatigue Damage from Random Response Histories

Damage ratio then becomes

$$n_i = p(S) \cdot dS \cdot S_t$$

$$N(S_i) = \frac{K}{S^m}$$

S_t = total number of cycles in required time = E[P]T

$$D = \sum \frac{n_i}{N(S_i)} = \frac{S_t}{K} \int S^m P(S) dS$$

$$D = \frac{E[P]T}{K} \int S^m P(S) dS$$

P(S) is therefore the all important output

Frequency Domain Approaches of Life Estimation

Narrow Band

The first frequency domain method for predicting fatigue damage from PSD's made use of the so-called narrow band approach (Ref. 77.) mentioned above, which assumes that the PDF of peaks is equal to the PDF of stress amplitudes. The narrow band solution was then obtained by substituting a function of the Rayleigh PDF of peaks for the PDF of stress ranges. The problem with this solution is that by using the Rayleigh PDF, positive troughs and negative peaks are ignored and all positive peaks are matched with corresponding troughs of similar magnitude regardless of whether they actually form stress cycles. For wide band response data the method therefore overestimates the probability of large stress ranges and so any damage calculated will tend to be conservative. The narrow band formula is illustrated here.

The total number of cycles is given by E[P].T in this formula. Expected damage:

$$E[D] = E[P] \frac{T}{K} \int S^b p(S) ds \tag{9-59}$$

where s=stress range.

Expected damage:

$$E[D] = E[P] \frac{T}{K} \int S^b \left[\frac{S}{4M_0} e^{\frac{-S^2}{8M_0}} \right] ds \tag{9-60}$$

Note: E[P].T = total number of cycles in time T = S_t as used earlier

Many expressions have been proposed to correct this conservatism. Most were developed with reference to offshore platform design where interest in the techniques has existed for many years. In general, they were produced by generating sample time histories from PSD's using Inverse Fourier Transform techniques, from

which a conventional rainflow cycle count was then obtained. The solutions of Wirsching (Ref. 78.), Chaudhury and Dover (Ref. 79.), and Hancock (Ref. 80.) were all derived using this approach.

Wirsching's Equation:

$$E[D] = E[D]_{NB} \bullet [a(b) + [1 - a(b)](1 - \varepsilon)^{c(b)}] \tag{9-61}$$

$$a(b)=0.926-0.033b; \qquad c(b)=1.587b-2.323; \qquad \varepsilon = \sqrt{1-\gamma^2}$$

is called the bandwidth parameter which is an alternative version of the irregularity factor.

Hancock's Equation

$$S_h^b = (2\sqrt{2Mo}^b)^b \left[\gamma\Gamma\left(\frac{b}{2}+1\right)\right] \tag{9-62}$$

This solution is given in the form of an equivalent stress range parameter S_h, where;

$$S_h^b = \int_0^\infty S^b p(S) ds \tag{9-63}$$

Chaudhury and Dover Equation

Their equation was derived using a similar approach and is given in (Ref. 79.):

The fatigue damage can then easily be obtained by substituting this into the general damage equation used when deriving the narrow band solution;

$$E[D]= E[P]\frac{T}{k}S_h^b \tag{9-64}$$

Tunna Equation

He (Ref. 96.) (Ref. 97.) proposed a different formulation using a revised form of the Rayleigh PDF for stress ranges as follows:

$$p(s) = \frac{s}{4\gamma mo} e^{\frac{-\delta^2}{8\gamma mo}} \tag{9-65}$$

For $\gamma = 1.0$ this formula becomes the narrow band formula given earlier.

Steinberg's Three-band Technique:

As a rough estimation for the fatigue damage, Steinberg proposed a three-band technique (Ref. 98.). The basis for this method is the Gaussian distribution. The instantaneous stresses (or accelerations) between $+1\sigma$ (σ is the root mean square) and -1σ are assumed to act at the 1σ level 68.3% of the time. The instantaneous stresses (or accelerations) between $+2\sigma$ and -2σ are assumed to act at the 2σ level of 95.4-68.3, or 27.1% of the time. The instantaneous stresses (or accelerations) between $+3\sigma$ and -3σ are assumed to act at the 3σ level 99.-95.4, or 4.33% of the time. These values are shown below for the 3 bands:

- 1σ values occur 68.3% of the time
- 2σ values occur 27.1% of the time
- 3σ values occur 4.33% of the time

The fatigue damage is thus estimated based on these three stress levels.

The approach of Steinberg leads to a very simple solution based on the assumption that no stress cycles occur with ranges greater than 6 rms values. The distribution of stress ranges is then arbitrarily specified to follow a Gaussian distribution. This defines the stress range cycles to occur with the following probability.

Steinberg Solution

$$S_{eq,\text{stein}} = f(m_0)$$

$$S_{eq,\text{stein}} = [0.683 \cdot (2 \cdot \sqrt{m_0})^m + 0.271 \cdot (4 \cdot \sqrt{m_0})^m + 0.043(6 \cdot \sqrt{m_0})^m]^{\frac{1}{m}}$$

It is possible to define the maximum stress range used in the subsequent fatigue analysis. If this is set at 6 rms it is possible to see that large stress ranges are omitted using this approach. MSC Fatigue will set this value automatically, if not over ridden, to be somewhere around 9 rms for stress range! Anything less than this is likely to result in an under prediction of fatigue damage. Of course this is counter balanced by the fact that medium range stress cycles of levels between 4 and 6 rms are over predicted. Nevertheless it must be stated that this approach is very questionable. It is included only as a means of allowing designers in the electronics industry, who are used to the Steinberg approach, a means of comparison.

Dirlik's Empirical Formulation

Since these solutions assume that the PDF of rainflow ranges is the factor which controls fatigue life, a better approach is to estimate this directly from the PSD without using the narrow band approach as a starting point. Both empirical and theoretical expressions have been produced in this way. They are generally only applicable to the offshore loading conditions for which they were developed.

Dirlik (Ref. 81.) has produced an empirical closed form expression for the PDF of rainflow ranges, which was obtained using extensive computer simulations to model the signals using the Monte Carlo technique. Dirlik's solution is illustrated in the equations below:

$$p(s) = \frac{\frac{D1}{Q}e^{\frac{-z}{Q}} + \frac{D_2}{R^2}e^{\frac{-Z^2}{2R^2}} + D_3 Z e^{\frac{-Z^2}{2}}}{2(mo)^{1/2}} \tag{9-66}$$

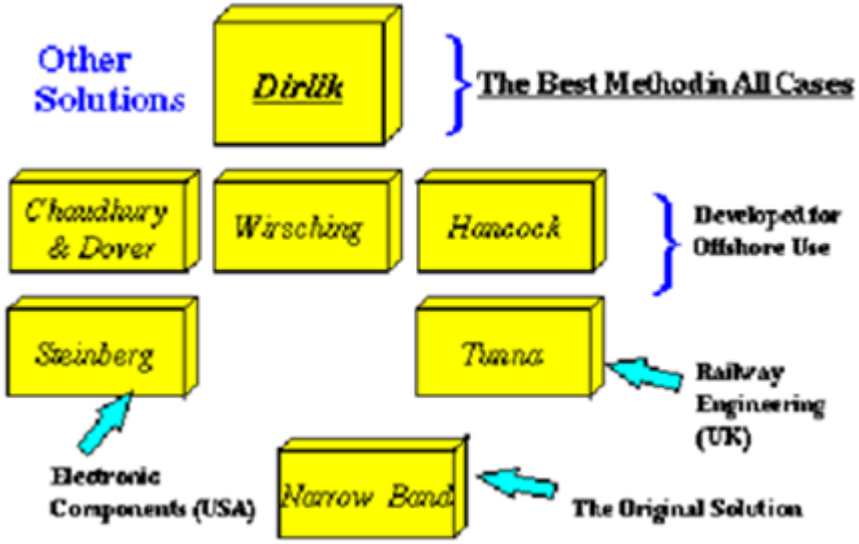
$$z = \frac{s}{2(mo)^{1/2}}x_m = \frac{m_1}{m_0}\left[\frac{m_2}{m^4}\right]^{1/2} D1 = \frac{2(X_m - \gamma^2)}{1+\gamma^2}D_2 = \frac{\left(1-\gamma-D_1+D_1^2\right)}{1-R} \tag{9-67}$$

$$D_3 = 1 - D_1 - D_2 Q = \frac{1.25(\gamma - D_3(D_2 R))}{D_1} R = \frac{\gamma - X_m - D_1^2}{1 - \gamma - D + D_1^2} \tag{9-68}$$

Dirlik's empirical formula for the PDF of rainflow ranges has been shown to be far superior, in terms of accuracy, than the previously available correction factors. However, the need for certification of the technique before its use meant that theoretical verification was required. This was achieved in 1988 (Ref. 82.) when a theoretical solution for predicting rainflow ranges from the moments of the PSD was produced. A detailed description of this method is given in (Ref. 84.).

An essential requirement of any theoretical solution was a statistical rather than spatial description of rainflow ranges. This was initiated in (Ref. 83.) and further extended later (Ref. 84.) to the form shown in Figure 9-61. The horizontal axis represents time and the vertical axis, stress range. -ve and +ve time is indicative of past and future events.

All the above methods are included in MSC Fatigue; this is because certain industries have developed design standards which assume that a particular approach will be used. Unless you are restricted by the requirements of a standard, we would always recommend that you use the Dirlik analysis as this generally proves the best approach for all applications.



Rainflow Range Definition

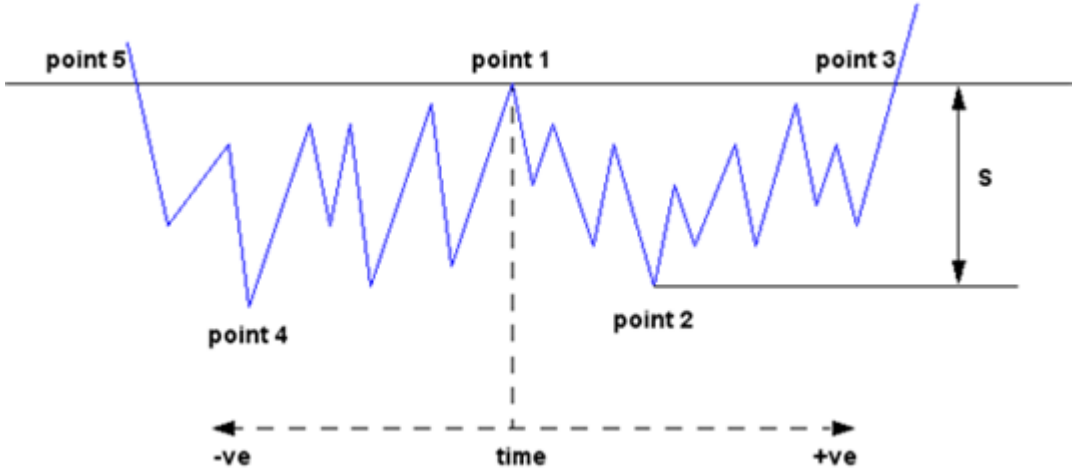


Figure 9-61 A New Rainflow Range Definition

In practical terms this definition asserts that the probability of three events U1, U2, and U3 can be used to define the probability of a rainflow range S occurring with a peak at the level corresponding to point 1 and a trough at a level corresponding to point 2.

Obviously there are a number of configurations that can occur with the same distance or range between point 1 and point 2 as shown in Figure 9-61. The total probability of a particular rainflow range is therefore the summation of these individual probabilities. The definition is expressed in terms of peak and trough values

and the relationships between these. This is not surprising since fatigue damage is determined by ranges of stress, which are themselves determined by peak and trough values.

The problem then reduces to one of finding the individual probabilities (for each value of ip) of U1, U2, and U3 and can be solved using the Markov chain model.

In order to reduce the computational effort required to obtain a solution, the assumptions of normality and stationarity were used. By assuming that the signal is stationary, one can assume that the signal is statistically equivalent after reflection about a vertical axis. U1 and U2 can then be treated in the same way. By assuming the signal is Gaussian, one can assume that the signal is statistically equivalent after reflection about a horizontal axis. Event U3 can then be treated in the same way as events U1 and U2.

The task of obtaining these three events then reduces into that of obtaining the long run absorption probabilities of transitions into states 1 and 2. These are:

- Given the assumption that a signal starts from some peak at level ip, what is the probability that it will move (via any number of peaks and troughs) to some other level kp directly, without going back to (or above) the level of ip (the first peak) or to a level below kp. Level kp is termed state 1.
- Given the assumption that a signal starts from some peak at level ip, what is the probability that it will move (via any number of peaks and troughs) to some other level below kp, without going back to (or above) the level of ip or into the level of kp. This level below kp is termed state 2.

Of course this information must be evaluated for every configuration of peak and trough in the signal.

If the Markov assumption is valid, that the probability of going to any adjacent peak (or trough) in the signal is dependent only on the position of the current trough (or peak), then the probabilities of the above multi-transition events can be obtained from the one step peak-to-trough and trough-to-peak probabilities using Markov Chain theory. This assumption is generally true for typical engineering situations.

The one step transition probabilities are obtained using the (approximate) solution of Kowalewski (Ref. 85.) for the PDF of adjacent peaks and troughs in a Gaussian signal. The reader is referred to (Ref. 85.) for a detailed description of the method of evaluating the Markov Chain model.

This solution is more complicated to evaluate computationally than Dirlik's expression which was given earlier. Furthermore, Dirlik's expression has been shown (see (Ref. 88.)) to be at least as accurate. For these reasons it is not included in the MFLF and FEVIB programs. It is presented here only to provide the theoretical background required for certification bodies to approve the overall approach.

All of the frequency domain techniques described in this note require the process being analyzed to be stationery, Gaussian and random. In theory this can be a very onerous requirement but in practice a considerable divergence from rigorous compliance can be tolerated.

Vibration Analysis using Finite Elements

In the previous section we discussed how fatigue life is determined from a PSD of stress. What is required now is a finite element analysis capable of evaluating such PSDs for each node in a structural component. The analysis described in this section requires the following:

1. A finite element model of the component.
2. A list of nodes to which input loads are applied.

3. Autopower and cross-power spectral density functions (PSDs & CPSDs) of the actual input loads.
4. A list of output nodes over which to calculate the fatigue damage.

An overview of the analysis procedure is given in Figure 9-62.

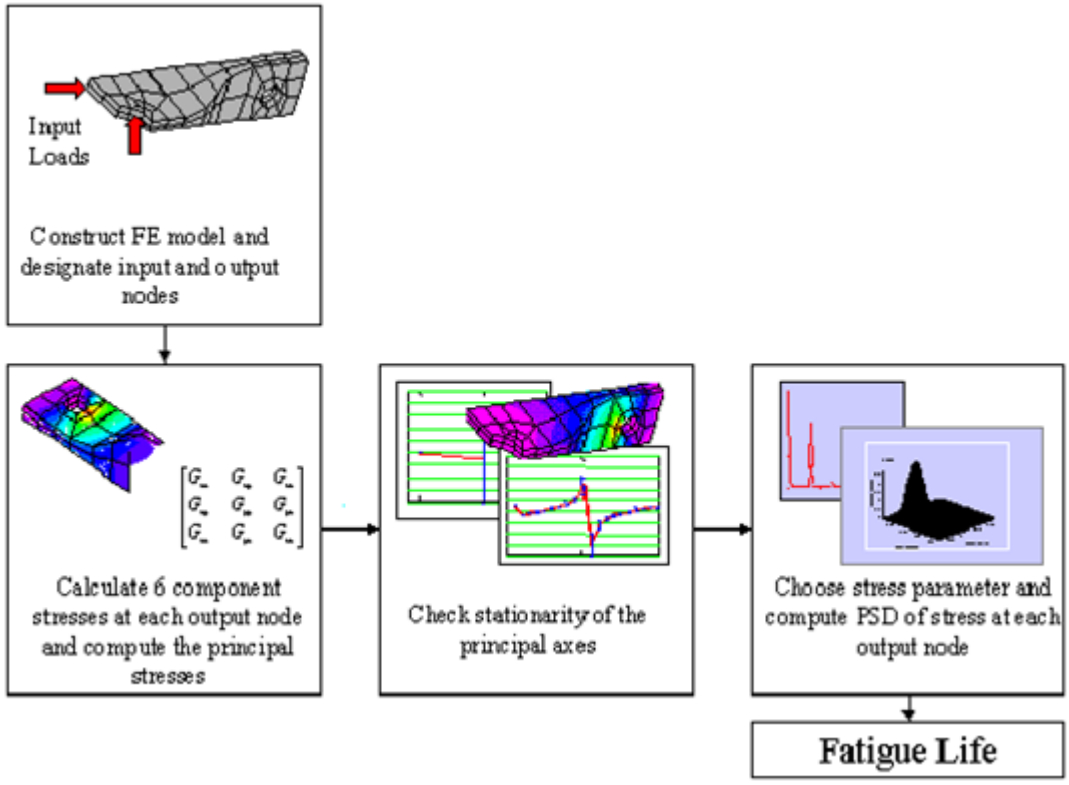


Figure 9-62 Overview of the Analysis Process

The theories employed are best described by example. We will start with a simple one-degree of freedom system subjected to a single random input load, the concept of a linear transfer function is introduced in this example. For the second example we consider the case of a two degree of freedom system subjected to two partially correlated random loads, the matrix form of the transfer equation and cross-power spectra is introduced here. Finally, we consider the case of a multiple degree of freedom system subjected to multiple partially correlated random loads giving six component stresses at the node under investigation. In this example we look at resolving the component stresses into a single effective stress PSD (e.g. maximum principal, von Mises, etc.), and also consider the problem of multiaxiality and discuss how to detect this.

One Degree of Freedom System with One Random Load

If a sinusoidally varying force is applied to a linear structure then the structure will respond with a sinusoidally varying stress of the same frequency as the applied load, this is illustrated in Figure 9-63.

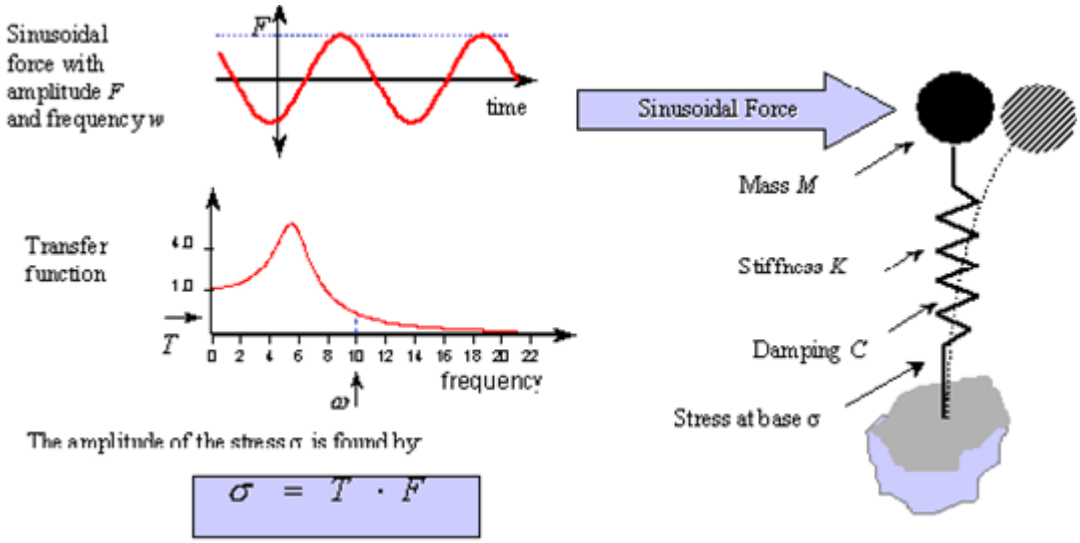


Figure 9-63 The Concept of a Linear Transfer Function

For a linear structure we expect that an increase in the amplitude of the forcing function will cause a proportional increase in the stress. This leads to the concept of a response parameter relating the amplitude of stress to the amplitude of the forcing function.

The amplitude of the stress varies with respect to the frequency of the applied load and so the response parameters are frequency dependent. The Transfer Function is defined as a plot of the response parameters with respect to frequency as shown in Figure 9-63. We can therefore use the transfer function to predict the amplitude stress of the structure by multiplying the amplitude of the load F by the Transfer Function T for a particular frequency of applied load.

The transfer function can also be used in the frequency domain to relate input load PSDs to output stress PSDs, this is discussed in An Introduction to Random Process Theory, 717 and is given in (9-69).

$$G^{\sigma}(\omega) = h(\omega) \cdot h(\omega)^{*} \cdot G_{ii}(\omega) \quad \text{or} \quad G^{\sigma}(\omega) = |h(\omega)|^{2} \cdot G_{ii}(\omega) \tag{9-69}$$

Where $G^{\sigma}(\omega)$ is the PSD of stress, $G_{ii}(\omega)$ is the PSD of input stress, $h(\omega)$ is the linear transfer function and $h(\omega)^{*}$ is the complex conjugate of $h(\omega)$. ω is the frequency expressed in circular measure (rad/sec).

For a single degree of freedom system the linear transfer function is easily obtained using (9-70).

$$h(\omega) = \frac{k}{-\omega^{2} \cdot m + i \cdot \omega \cdot c + k} \tag{9-70}$$

Where, m is the nodal mass, c is the element damping and k is the element stiffness.

Two Degree of Freedom System with 2 Partially Correlated Random Loads

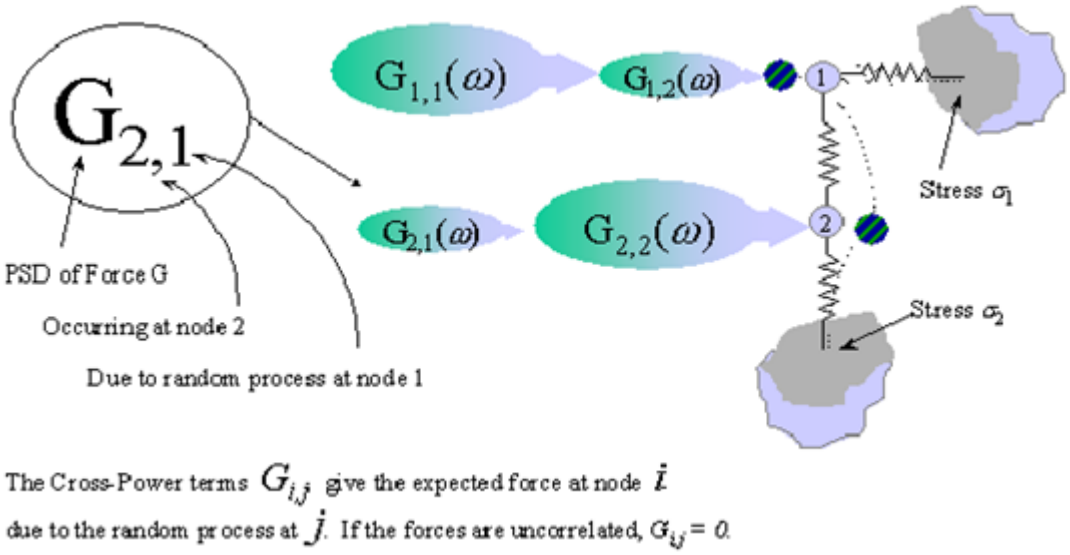


Figure 9-64 Two Degree of Freedom System, Partially Correlated Random Loads

Each random load is described by its Auto-power spectral density function (PSD). If both random loads are partially correlated we must also include cross-power spectral density functions (CPSD) to adequately model the sequential effects between the two processes. The PSD of stress witnessed at the base of the structure was derived earlier and is given by (9-71).

$$G^{\sigma}(\omega) = \sum_{i}\sum_{j} h_i(\omega) \cdot h_i(\omega)^* \cdot G_{ij}(\omega) \tag{9-71}$$

Where $G^{\sigma}(\omega)$ is the PSD of stress, $G_{11}(\omega)$ and $G_{22}(\omega)$ are the autopower spectra of inputs 1 and 2 respectively, and $G_{12}(\omega)$ and $G_{21}(\omega)$ are the cross power spectral density function between inputs 1 and 2. $h_1(\omega)$ and $h_2(\omega)$ are the transfer functions between stress and load input at nodes 1 and 2 respectively.

To improve computational efficiency and simplify the arithmetic, it is desirable to express (9-71) in matrix form as (9-72).

$$G^{\sigma}(\omega) = \left[H(\omega)\right]^{T} \cdot \left[G(\omega)\right] \cdot \left[H(\omega)\right]^{*} \tag{9-72}$$

Where $G^{\sigma}(\omega)$ is the PSD of stress, $[H(\omega)]$ is a vector of transfer functions defined as

$$\left[H(\omega)\right] = \begin{bmatrix} h_1(\omega) \\ h_2(\omega) \end{bmatrix}$$

$[H(w)]^*$ is the complex conjugate of $[H(\omega)]$ and $[H(\omega)]^T$ is the transpose. $[G(\omega)]$ is a symmetric, square matrix of auto- and cross-power spectral densities of loading, the auto-power terms lying along the leading diagonal.

$$\left[G(\omega)\right] = \begin{bmatrix} G_{11}(\omega) & G_{12}(\omega) \\ G_{21}(\omega) & G_{22}(\omega) \end{bmatrix}$$

Multiple Degree of Freedom Systems

We deal with multiple degree of freedom structures in exactly the same way as single degree of freedom structures, however, the FE analysis now yields a transfer function in terms of six component stresses at the output node. This is illustrated in Figure 9-65.

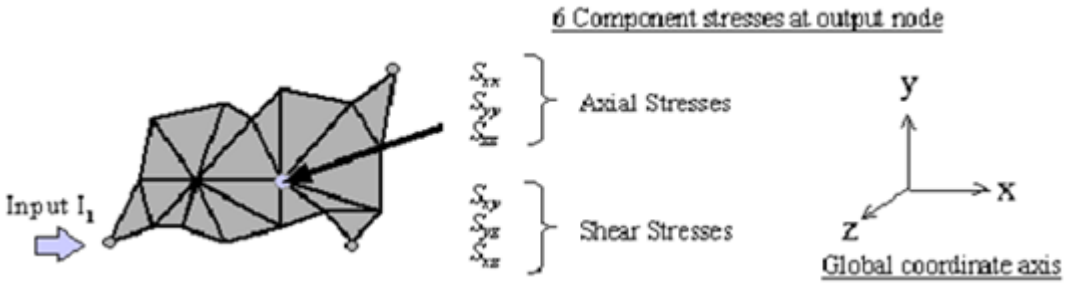


Figure 9-65 Transfer Function Calculated as Six Component Stresses

The finite element solver returns six component stresses at each output node in response to a single input load, these being three axial stresses and three shear stresses along the global coordinate or the element coordinate axis. We require a single stress result for use in the transfer function, this can either be the maximum principal or an effective stress parameter based on some yield criterion such as Tresca or von Mises.

The three principal stresses can be determined from the six components using matrix routines for determining eigenvalues and eigenvectors. The components are expressed in matrix form, as shown in (9-73). The eigenvalues yield the principal stresses while the eigenvectors yield the directional cosines of the principal axes.

$$\sigma_{principal}(\omega) = eigenvalues(S(\omega)) \quad Direction(\omega) = eigenvectors(S(\omega)) \tag{9-73}$$

Where $\sigma_{\text{principal}}(\omega)$ is a vector of the three principal stresses, Direction(ω) is a matrix of the direction cosines for the three principal axes normalized to yield three unit column vectors, the nth column referring to the nth eigenvalue. $S(\omega)$ is a 3 x 3 matrix of the component stresses defined as

$$S(\omega) = \begin{bmatrix} S_{xx}(\omega) & S_{xy}(\omega) & S_{xz}(\omega) \\ S_{xy}(\omega) & S_{yy}(\omega) & S_{yz}(\omega) \\ S_{xz}(\omega) & S_{yz}(\omega) & S_{zz}(\omega) \end{bmatrix}$$

The eigenvalues and eigenvectors are complex retaining both the amplitude and phase information of the transfer function. It is desirable to combine these two matrices to form a single tensor matrix with three columns each representing the vector of a particular principal stress. The magnitude of each vector will now

represent the magnitude of the principal stress. As the eigenvector matrix is a unit matrix, we can evaluate the new stress tensor matrix s as (9-74).

$$\sigma(\omega) = \{diag((\sigma_{principal}(\omega)) \cdot Direction(\omega)^T)\}^T \quad (9\text{-}74)$$

Where diag(υ) is a function to return a diagonal matrix of the vector υ.

The single stress parameter can now be determined depending on the user's choice.

Maximum principal stress	$\sigma_{max} = max(\sigma_1, \sigma_2, \sigma_3)$
von Mises* stress	$\sigma_{vonMises} = \sqrt{\frac{1}{2} \cdot (\lvert\sigma_1 - \sigma_2\rvert^2 + \lvert\sigma_2 - \sigma_3\rvert^2 + \lvert\sigma_3 - \sigma_1\rvert^2)}$
Tresca effective stress	$\sigma_{Tresca} = max(\lvert\sigma_1 - \sigma_2\rvert, \lvert\sigma_2 - \sigma_3\rvert, \lvert\sigma_3 - \sigma_1\rvert)$

*when using the MSC Random method, the following equation is used:
$\sigma^2_{vonMises}(f)/Hz = S^2_x+S^2_y+S^2_z+3(S^2_{xy}+S^2_{xy}+S^2_{xy})$-Cross($S_x$,$S_y$)-Cross($S_y$,$S_z$)-Cross($S_z$,$S_x$)
where $\sigma^2_{vonMises}(f)/Hz$ is the PSD of von Mises stress calculated at every frequencye and S^2_i and S^2_{ij} are the PSDs of the components of the stress tensor transfer function. The Cross PSD terms are complex and the sign is kept when adding.

Note: The three stress parameters above are real, it is no longer necessary to retain the complex notation as phase is not required.

The Case of Multiaxiality

In other parts of this guide we talk about multiaxiality (or biaxiality) and categorize structures as uniaxial, proportional biaxial, and non-proportional biaxial. In terms of the principal stress tensor these three cases directly relate to the following descriptions:

1. Uniaxial - One principal stress is much larger than the other two, which tend to zero. The stress tensor is stationary and doesn't rotate.
2. Proportional, multiaxial - Two or three principals are greater than zero but increase and decrease proportionally. Again the stress tensor doesn't rotate.
3. Non-proportional, multiaxial - Two or three principals are greater than zero and increase and decrease disproportionately to each other. The stress tensor is seen to rotate.

The first two cases can be dealt with using the maximum principal or a von Mises stress invariant, in these cases we notice that the direction of the stress tensor is stationary and the fatigue crack will propagate in a relatively constant direction.

The last case, non-proportional multiaxial, results in a non-stationary principal stress tensor, its direction is continually changing thereby influencing the crack growth and direction of propagation. In this instance the engineer must be aware of the extent of this directional variation to ascertain the suitability of the analysis. Small angular variations of the tensor result in small errors, however as the angular variations increase the effect on fatigue analysis become more apparent. It is important for the engineer to know what has caused

the rotating principal axis in order to ascertain how far to trust the results. In the next section we look at what causes multiaxial stress states and how the angular variation of principal stress tensor can be determined.

There are three causes of rotating principal stresses:

Class I: Rotating Principal Stress Axes Caused by Multiaxial Loading

In some structures multiple input loading will give rise to rotating principal stresses, this is not always the case and so we must have a method of detecting it. Figure 9-66 illustrates two examples, one where multiple input loading does affect the stress state, and the other where it does not.

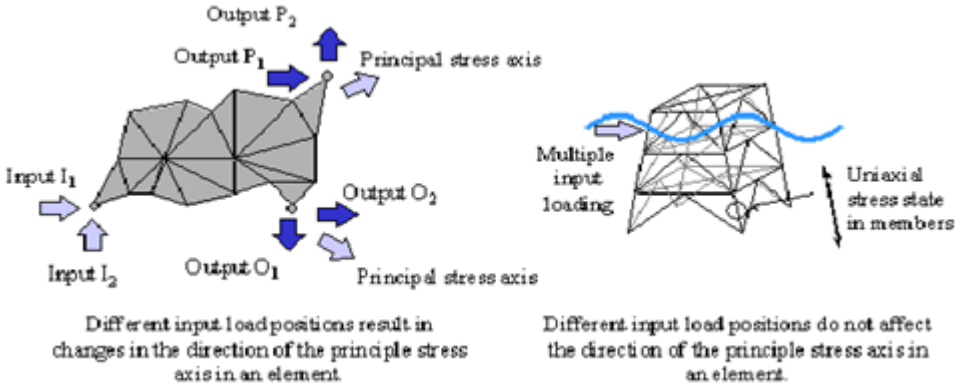


Figure 9-66 Rotating Principal Stress Axes Caused by Multiaxial Loading

Class I rotating principle stresses can be identified by comparing the direction of the principal stress axes for different load positions. It is worth noting that a non-proportional, multiaxial load state need not result in a rotating principal stress state, we are interested in the 'stress state' not the load state.

Class II: Rotating Principal Stress Axes Caused by Differential Damping

Differential damping can give rise to a multiaxial stress state where the principal axis is seen to periodically oscillate. For this to be significant then the structure has to exhibit fairly high levels of differential damping, as this is not common for most structures then this potential cause of multiaxiality should not present many problems with most engineering applications. It is still worth noting however, that Class II rotation is commonly accompanied by Class III and in these cases the Class III is usually the cause of the problem.

Figure 9-67 illustrates the effect. Damping causes a phase lag between input and output frequencies and differential damping between both output components will cause phase lags between O_1 and O_2, hence causing variations in the direction of the principal stress axis.

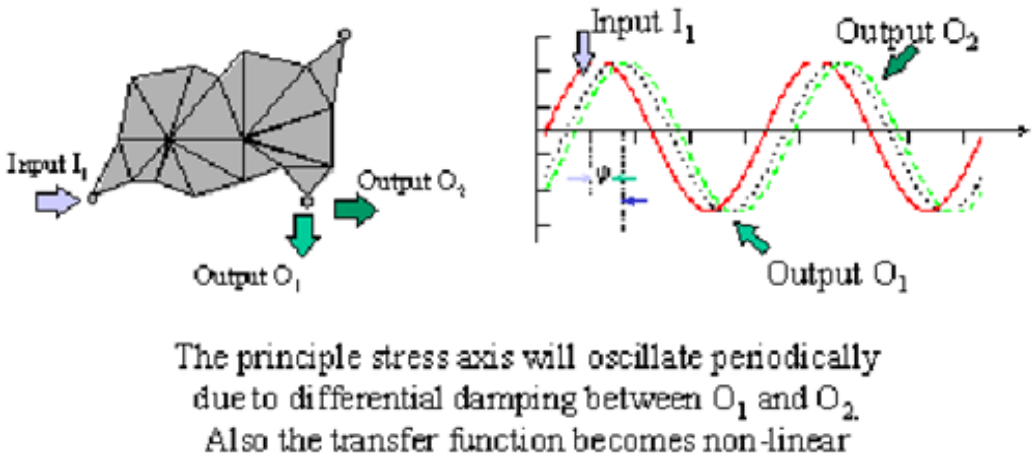


Figure 9-67 Rotating Principal Stress Axes Caused by Differential Damping

To see how this effect works consider the complex stress vector σ shown in Figure 9-68. We will deal first of all with a simple biaxial stress state on a surface element where the stress perpendicular to the surface is zero (i.e. k = 0). Later in the text we will deal more generally with the full triaxial case.

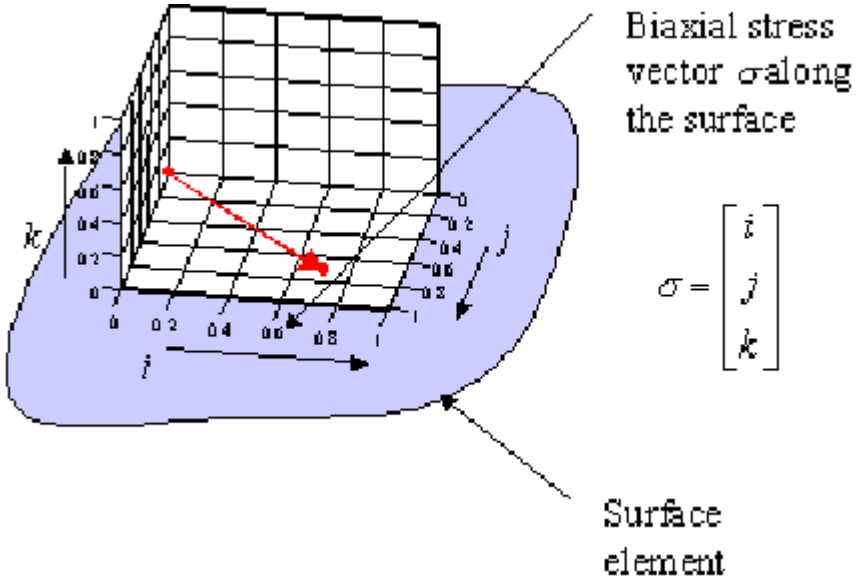


Figure 9-68 Visualizing a Principal Stress Vector On the Surface of a Component

Vector σ is complex and each element of the vector i, j, and k contains information on the amplitude and phase of the transfer function stress in that particular direction. In an isotropic material with no differential

damping, the argument of i, j and k are identical indicating that the components are in phase. Consider now how the stress vector responds to a unit sinusoidal input load, this is illustrated in Figure 9-69.

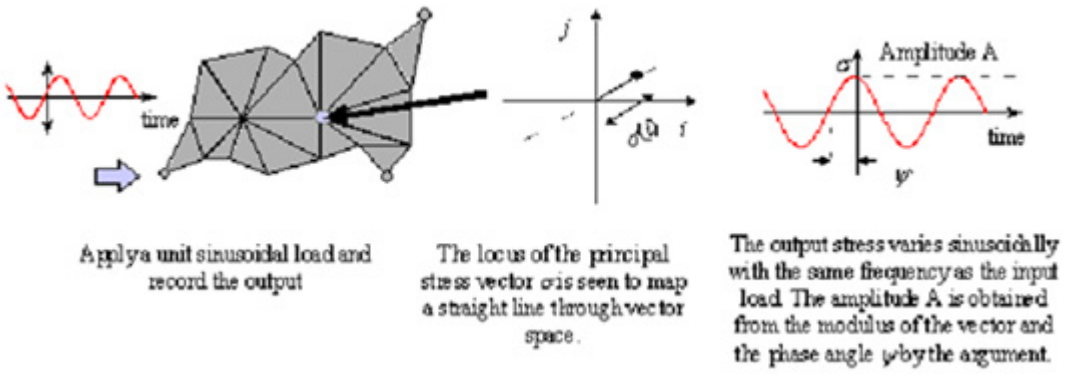


Figure 9-69 Stress Response Vector For a Component with NO Differential Damping

Consider now the stress response to a unit sinusoidal load in a non-isotropic material where the damping properties are different in the i and j directions, this is illustrated in Figure 9-70.

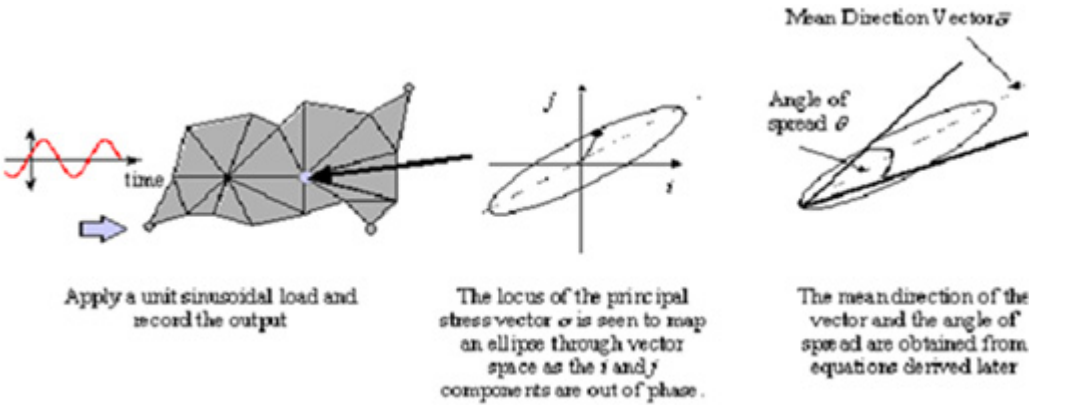


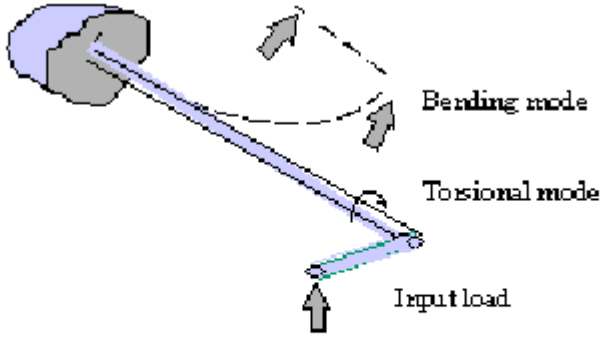
Figure 9-70 Stress Response Vector For a component With Differential Damping.

Comparing the spread of the vector for each frequency identifies class II nonstationarity. If this test were positive it would suggest a problem with the actual FE model as most engineering structures exhibit only small damping values.

Class III: Rotating Principal Stress Axes Caused by Exciting Vibration Modes

In some structures the direction of the principal axis may be dependent on the frequency of the applied load. This phenomenon can occur when the applied load is exciting more than one mode. As each mode approaches its natural frequency, the stresses associated with that mode increase, thus causing a variation of

the principal stress axis. Figure 9-71 shows a classic example of Class III nonstationarity, with a simple cantilever beam in combined bending and torsion.



Vertical input results in combined torsion and bending in the shaft. The direction of the principle stress axis will vary with frequency as the load excites each mode separately

Figure 9-71 Rotating Principal Stress Axes Caused by Exciting Vibration Modes

Class III nonstationarity is tested by comparing the direction of the principal stress axes for varying frequencies. Differences in the phase angles would indicate frequency dependent rotating principal stresses.

Note:	It might be possible to analyze structures exhibiting this type of behavior if the actual loads lie outside the critical frequency areas.

In summary, by far the most significant of these three Classes are Classes I and III. Class I is often more apparent in the immediate vicinity of applied loads or a sudden change in structural geometry. Class III will arise where torsion and axial modes are witnessed, but the engineer may find the frequencies affected to be sufficiently distanced from those at which the loads are applied.

Class II seldom arises on its own, this would likely indicate a problem with the model. However, it is common to find Classes II and III occurring together due to the differential damping between two modes being exited near their natural frequency.

Displaying the Angular Variation of the Principal Stresses.

The design engineer needs to determine the extent to which the principal stress tensor rotates at a particular node in order to decide whether to proceed with the analysis at that point or whether to change to a time based multiaxial fatigue analyzer. It is worth noting that although rotating principal stresses may occur in a component in some localized areas these will have no affect on other nodes within the model and the engineer is free to consider this analysis technique in those stable areas. Also, in many instances the designer will have no choice in the analysis technique (maybe time implications, etc.). In this case knowing the extent of the principal stress rotation and the Class causing this will prove crucial to the confidence of the results.

The results of the axis stationarity test are presented in two stages. Firstly, the maximum angular variation is calculated for all three classes and displayed on a contour plot of the finite element model. Areas of high and low rotations can easily be detected and the engineer quickly ascertains whether a problem exists or not. See Figure 9-72.

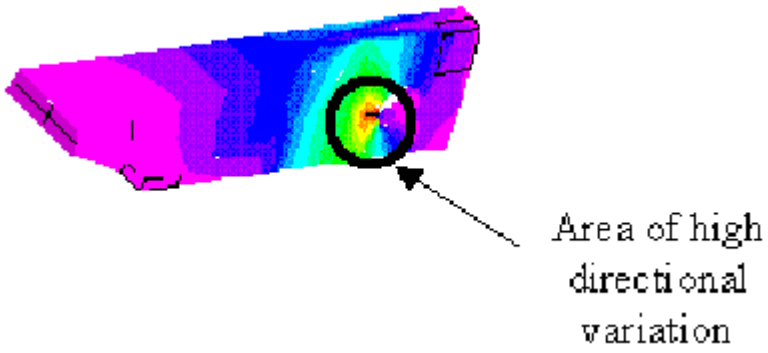


Figure 9-72 Contour Plot of Principal Axis Stationarity

Should a problem be detected, the engineer will need to ascertain the cause of this, either Class I, II, or III. The cause is easily deduced using two error bar plots, the first discriminates between Class I and the other two classes. The second discriminates between Classes II and III. These are shown in Figure 9-73.

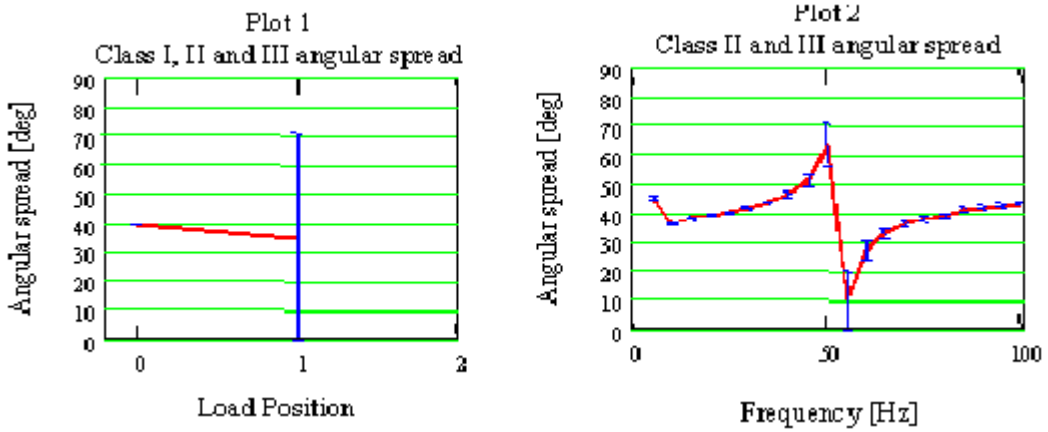


Figure 9-73 Error Bar Plot to Show Extent of Class I, II and III Principal Stress Rotation

Calculating the Angular Variation of Principal Stress Axes

The angle between two real vectors a and b is determined from the scalar product expressed in (9-75).

$$\psi(a, b) = acos\left(\frac{a \cdot b}{|a| \cdot |b|}\right) \tag{9-75}$$

Where ψ is the angle between the two vectors a and b.

In our implementation, we are dealing with complex vectors where the modulus yields the magnitude of the principal stress and the argument yields its phase relative to the phase of the input load. In this case, (9-75) is unsatisfactory and a better definition is obtained from (9-76). (The benefit of this definition becomes apparent when calculating Class II situations).

$$\psi(a, b) = acos\left(\frac{|a \cdot b|}{|a| \cdot |b|}\right) \tag{9-76}$$

Where ψ is the angle between the two vectors a and b.

The two error bar plots in Figure 9-73 help to distinguish the classes of axis rotation. In the following sections we look at how the values in the plots are determined.

Calculating the Values for Plot 1

For this analysis, we must calculate the maximum angular range of the stress vector for each input load and also show how the median direction vector varies between each input load position. Figure 9-74 shows a plot of the loci of the stress vectors over all frequencies for one particular load position. The plot represents a biaxial stress state in the i/j plane, the elliptical orbits show Class II non-stationarity. The total angular spread for Classes II and III is derived from the envelope of the vectors as shown in Figure 9-74.

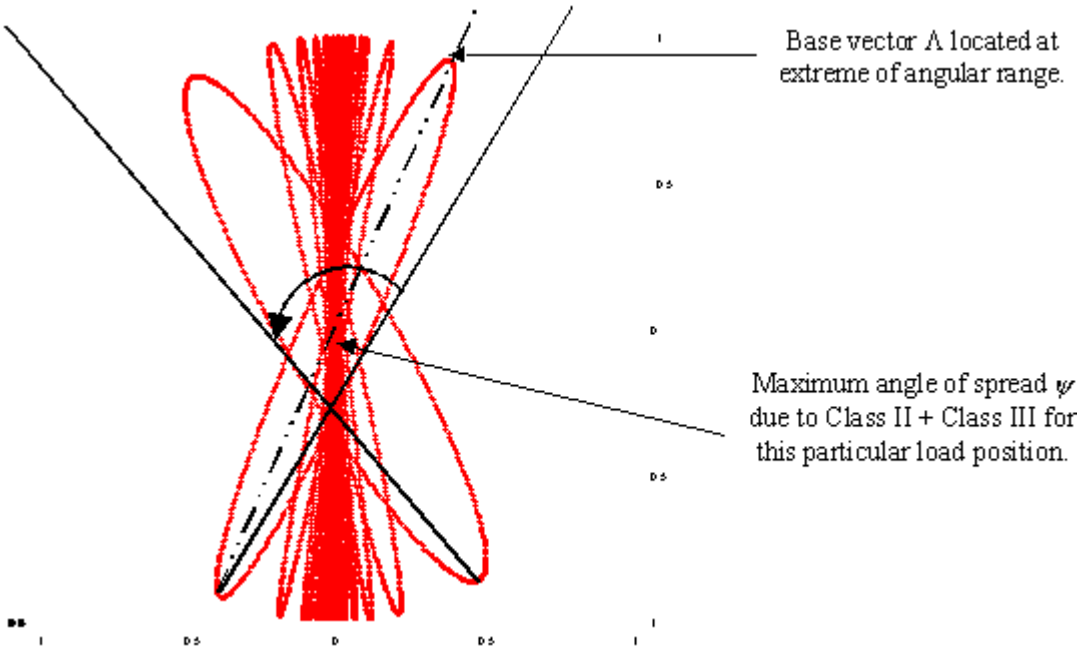


Figure 9-74 Plot Showing Loci of Unit Stress Vectors and Maximum Angular Spread

To calculate the total angular spread of the principal stress axis, we first need to determine an extreme vector lying at the edge of the range, the angle of every other principal stress vector can then be determined relative to this. This step becomes necessary as we use (9-76) to determine the angle y and this returns a scalar value with no indication as to whether the angle is positive or negative. To find an extreme vector we choose any arbitrary vector in the range and proceed to locate the vector with the largest angular deviation from it. Provided that the stress state is biaxial or hydrostatic triaxial then this vector will always be an extreme. (We will look at full triaxial stress states later.) The angle between an arbitrary vector and an extreme Λ is given in (9-77).

$$\Lambda_{angle} = max\left\{\psi(\bar{a}, \overline{\sigma_n}) + \frac{1}{2}\cdot\psi(\sigma_n, \sigma_n)\right\} \text{ for all n values} \qquad (9\text{-}77)$$

Where Λ_{angle} is the angle between an arbitrary vector a and an extreme vector Λ. $\bar{\sigma}$ denotes the mean direction vector of $\bar{\sigma}$ defined as

$$\bar{\sigma} = \begin{bmatrix} |i| \\ sign(i) \cdot sign(j) \cdot |j| \\ sign(i) \cdot sign(k) \cdot |k| \end{bmatrix} \text{ where } sign(n) = \begin{cases} 1 \text{ if } \Re_n \geq 0 \\ -1 \text{ if } (\Re_n < 0) \end{cases}$$

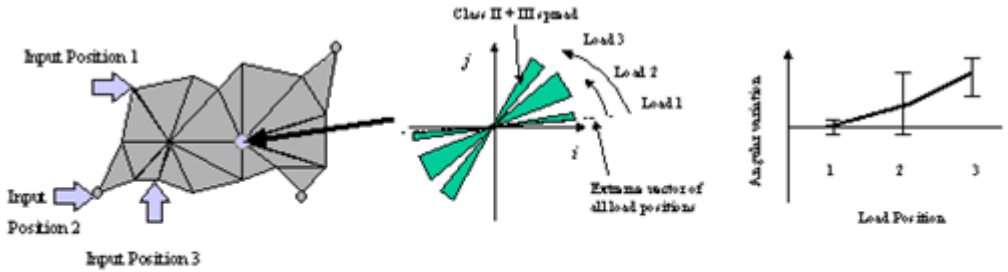
In other words, the angle between vectors a and $\bar{\sigma}_n$ is the angle between their mean directions plus half the angle of spread of vector $\bar{\sigma}_n$ due to the elliptical orbit of the locus.

Having determined an extreme base vector Λ we can proceed to find the next vector with the maximum angle from this. This now gives the total angular spread of the principal stress axis. In this calculation we also need to include the angle given by the elliptical orbits associated with both vectors. The equation is given in (9-78).

$$\psi_{max} = max\left\{ \psi(\Lambda, \overline{\sigma_n}) + \psi(\sigma_n, \sigma_n) \right\} + \psi(\Lambda, \Lambda) \quad \text{for all n values} \tag{9-78}$$

Where ψ_{max} is the total angular spread.

The Class I axis variation monitors how a principal stress axis rotates with respect to the different input positions. Associated with each load input is an angular spread due to Classes II and III, for Class I analysis we calculate these with respect to a new base vector at the extreme of all the vectors from all frequencies and load positions. Figure 9-75 illustrates the point.



Error bars denote extent of Class II + Class III spread,
solid line denotes median vector direction for each load case

Figure 9-75 Illustrating The Calculation of Class I Principal Axis Rotation

Calculating the Values for Plot 2.

For each load position it is necessary to distinguish between a Class II or a Class III problem. This analysis proceeds along the same lines as that described earlier:

1. Locate a base vector at the extreme of the load position under consideration.
2. Calculate the angle between the base vector and all other mean direction vectors for each frequency giving the Class III drift. (This is represented by the solid line on error bar plot 2.
3. Calculate the angular spread of the elliptical locus for each frequency.

The procedure is illustrated in Figure 9-76.

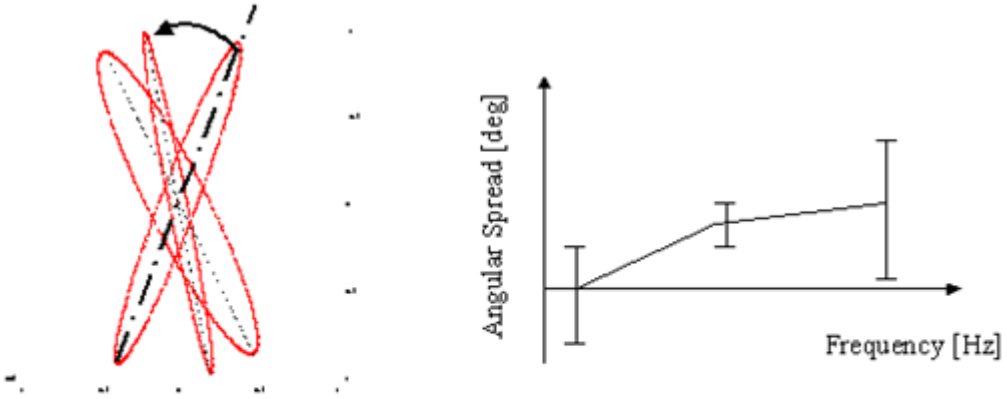


Figure 9-76 Illustrating The Calculation of Class II & III Principal Axis Rotation

Dealing with Triaxial Stress States

Most fatigue analysis is concerned with surface stresses and these are unlikely to have a triaxial stress state unless acted upon by hydrostatic forces, such as with submarines and pressure vessels. The analysis technique for tracking the principal stress vectors is still valid under these conditions, though it does become unreliable when dealing with a varying triaxial stress state where all three axes rotate. The unreliability is due to the method adopted in determining the extreme base vector. With the full triaxial case, we can no longer assume this is the vector with the greatest deviation from any arbitrary vector in the range. Figure 9-77 shows the loci of the principal stresses in a triaxial stress state.

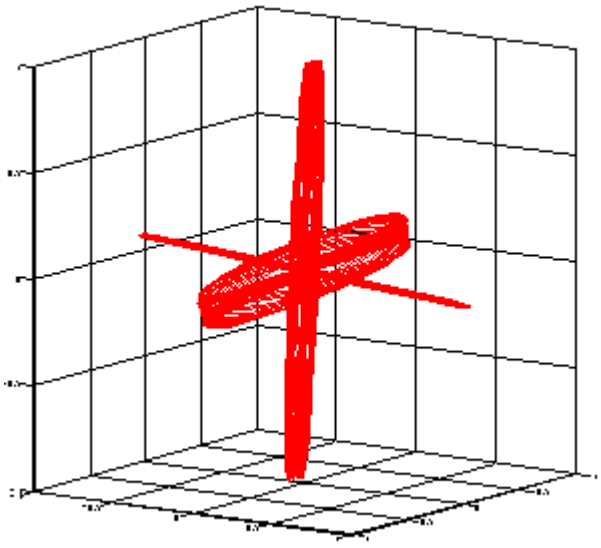


Figure 9-77 Loci Plot of a Triaxial Stress State

10 Weld Analysis

- Introduction 772
- Spot Weld Job Setup (dtfat) 774
- dtfat Batch Operation 777
- Job Setup (Classic Modules) 777
- Spot Weld Analyzer (SPOTW) 792
- Polar Display (MPOD) 820
- Spot Weld Analysis Theory 828
- Seam Weld Analysis 835
- Seam Weld Job Setup (dtfat) 851
- dtfat Batch Operation 857
- Seam Weld Job Setup (MFATFE) 857

Introduction

MSC Fatigue supports a number of approaches for the fatigue analysis of welded structures. This chapter discusses methods available for analysis of spot welded and seam welded structures.

The multi-disciplinary fatigue solver dtfat and the Classic module SPOTW are MSC Fatigue module that analyze the fatigue life of spot welds using the S-N method. Theoretical background can be found in Spot Weld Analysis Theory, 828 in this chapter.

In setting up a proper spot weld analysis it is important to first have a sound understanding of the operation of MSC Fatigue within the Patran environment. It is suggested that you first read and understand Using MSC Fatigue - The Classic Modules (Ch. 3). Job setup for a spot weld analysis is very similar to a job setup for other fatigue analysis types with only a few differences, as pointed out in this chapter.

Certain limitations or restraints exist in setting up a spot weld analysis. These are mainly the fact that the analysis MUST be set up within a preprocessor, and that only results from MSC.Nastran bar elements representing the spot welds can be used.

The method requires spot welds to be modeled as stiff MSC.Nastran beam elements. The forces transmitted through these beam elements are used to calculate the structural (nominal) stresses in the weld nugget and the adjoining sheet metal at intervals around the perimeter of the nugget. These stresses can then be used to make fatigue life predictions on the spot weld using a S-N (total life) method.

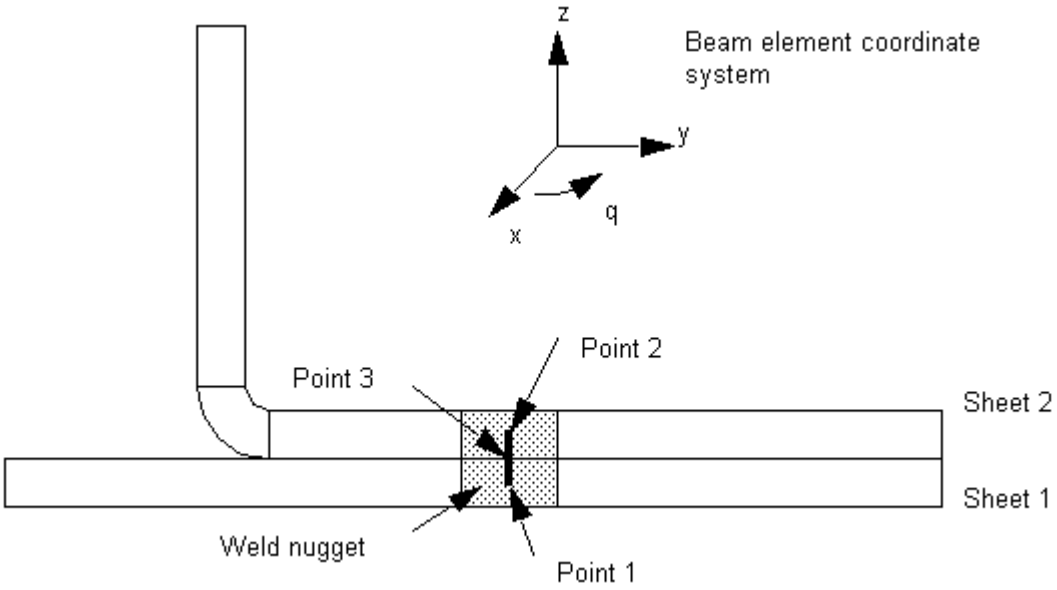


Figure 10-1 Schematic of Typical Spot Weld

In a finite element analysis, the weld is modeled in MSC.Nastran as a stiff beam element joining the mid-plane of two sheets. The length of the beam element should be $0.5(s_1+s_2)$ where s_1 and s_2 are the thicknesses of sheets 1 and 2 respectively. Point 3 is on the axis of the weld nugget and at the interface of the 2 sheets, i.e. $0.5s_1$ from Point 1. All forces and moments are taken to be in the MSC Fatigue beam element coordinate system. This is taken to be a Cartesian system with the Z axis going from Point 1 to Point 2. The forces and moments at points 1 and 2 will be those applied by the spot welds on the sheets, and the forces and moments at point 3 will be those applied by the upper section (between point 3 and point 2) on the lower section (between point 1 and point 3).

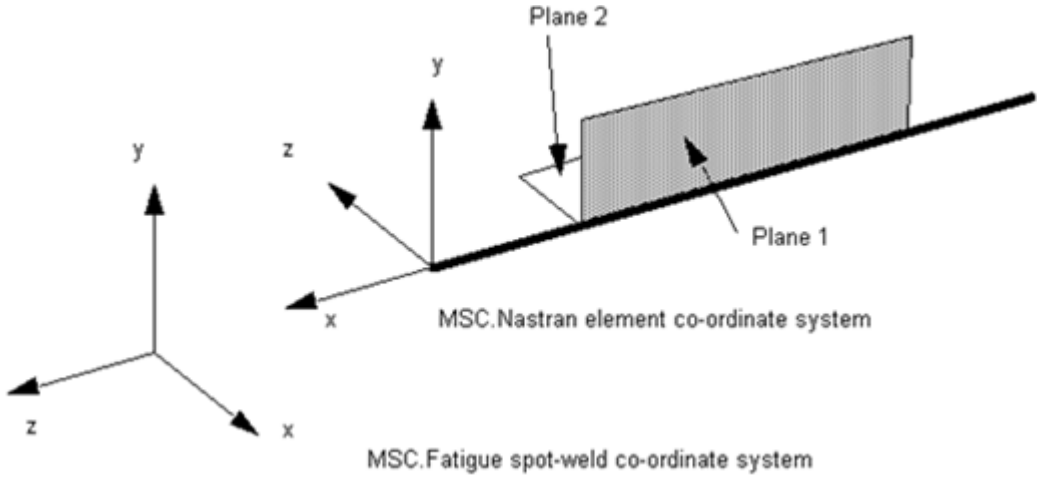


Figure 10-2 Relationship of MSC Fatigue Spot Weld Coordinate System to MSC.Nastran

The important thing here is to know how the MSC.Nastran beam element coordinate system relates to that used in the new spot weld calculation software, and how the results in MSC.Nastran translate to those required by the fatigue analysis tool. The relationship between the coordinate systems is shown above. The MSC.Nastran beam element has positive X direction from Point A to Point B. In MSC Fatigue Point A is Point 1 and Point B is Point 2. The positive Z direction in MSC Fatigue is the positive X direction in MSC.Nastran. The Y axes are the same in both coordinate systems. The required transformations are built into the PAT3FAT translator.

In addition, if the two plates are dissimilar in material or thickness, the spot welds in each group should all have the same directionality - with the axis of the beams going from Sheet 1 to Sheet 2. The directions of beam elements can be checked and reversed using one of the Patran utilities.

Some items of note with this spot weld analyzer are:

1. You must use a preprocessor such as Patran to create the spot welds on your FE mesh. MSC Fatigue does not provide any modeling tools to help in the creation of these spot welds.
2. Very stiff MSC.Nastran beam or bar elements (CBAR) must be used to model the spot welds. The analysis results must be from a MSC.Nastran analysis, specifically bar forces and moments.
3. The spot welds (bars/beams) must be perpendicular to the two sheets. If not, errors may occur because the bending moments will not be correct.

4. You do not need to define the radius of the spot weld nugget in the beam itself. The actual cross section of the beam is unimportant, but it should be stiff in relation to the surrounding shells. The compliance of the joint comes from the shell elements. Torsional stiffness is normally set to zero.

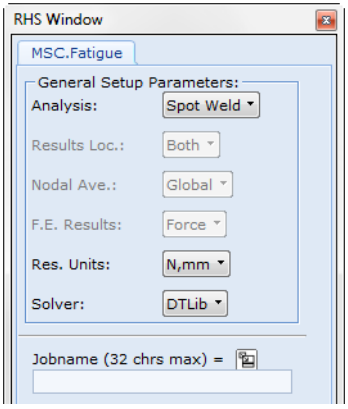
CWELD Modeling

Refer to the NASTRAN Users Guide for modeling CWELDS.

MSC Fatigue supports results from all CWELD modeling optioins - ELPAT, ELEMID, GRIDID, ALIGN and PALTPAT.

Spot Weld Job Setup (dtfat)

The setup starts in the MSC Fatigue forms within Patran. You must set the analysis type to Spot Weld. There are three basic inputs just as for any other analysis types, those being Solution Parameters, Material and Loading Information. Results post-processing and Job Control are also described in this section.



Solution Parameters (Spot Weld)

The following table lists DTLib solution parameters that are associated with Spot Weld analysis as shown on Solution Parameters form below.

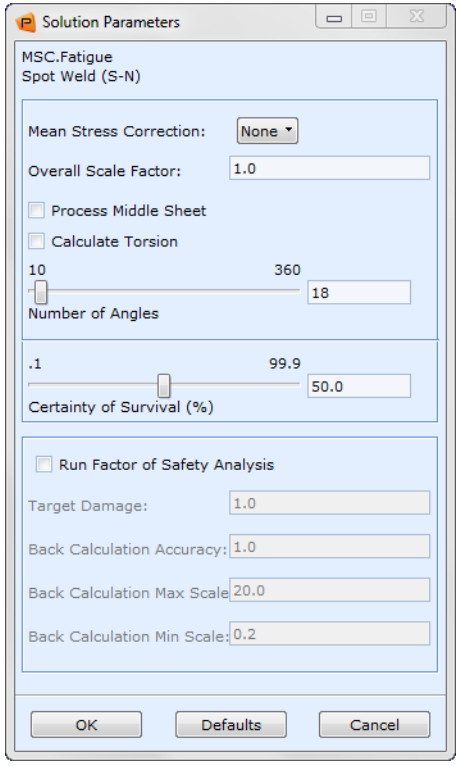


Table 10-1 Solution Parameters

Parameter	Options	Description	Default
Mean Stress Correction			**None**
	None	No mean stress correction applied.	
	Simple	This method uses a simplification of the FKM mean stress correction, defined by a single mean stress sensitivity factor MSS. It allows an equivalent stress amplitude S_e at the material R-ratio (RR) to be calculated based on the stress amplitude S_a and mean S_m of each cycle.	
	FKM	The FKM method as implemented is based on the method described in the FKM Guideline “Analytical Strength Assessment of Components in Mechanical Engineering”, Tr. E. Haibach. 2003.	

Table 10-1 Solution Parameters

Parameter	Options	Description	Default
Overall Scale Factor	0 < **Value** <= 1e10	The scale factor to apply to the combined data prior to performing damage calculation.	**1.0**
Process Middle Sheet	**True, False**	The Rupp method for spot weld life prediction has not been validated for the prediction of fatigue damage occurring at the middle sheets for spot welds joining more than two sheets. It should also be noted that such failures are relatively rare, difficult to reproduce in the laboratory, and difficult to detect in practice. For this reason, the software provides an option (by setting this property to **False**) to omit any fatigue calculation on the middle sheets, which might otherwise give a false positive failure prediction.	**False**
Calculate Torsion	**True, False**	By setting this option to **True,** it enables fatigue calculations to be made based on the shear stress around the periphery of the spot weld in each sheet, due to the torsional load on the spot weld.	**False**
Number of Angles	10 < **Value** <= 360	In spot weld calculations, predictions of life are made at intervals around the periphery of the weld in the two sheets and in the weld nugget. This property controls the number of values of the angle at which the fatigue calculations are made. For example, if **Number Of Angles** = 18, calculations will be made at 20 degree intervals.	**18**
Certainty of Survival	0 <= **Value** <= 100	Percentage certainty of survival	**50**
Factor of Safety Analysis	**True/False**	This option performs a back-calculation on stress to compute a scale factor, which when applied, results in the damage value specified below. For cases where the scale factor is greater than 1.0, the damage target is exceeded and the local stress is "safe".Note that the full sets of fatigue results from this analysis are based on the application of the computed scaling factor. A separate analysis should always be performed without the "Factor of Safety Analysis" option turned on.Note that the full set of fatigue results from this analysis are based on the application of the computed scaling factor. A separate analysis should always be performed without the "Factor of Safety Analysis" option turned on.	**False**
Target Damage	0 < **Value** <= 1E30	The target damage for back calculation.	**1**

Table 10-1 Solution Parameters

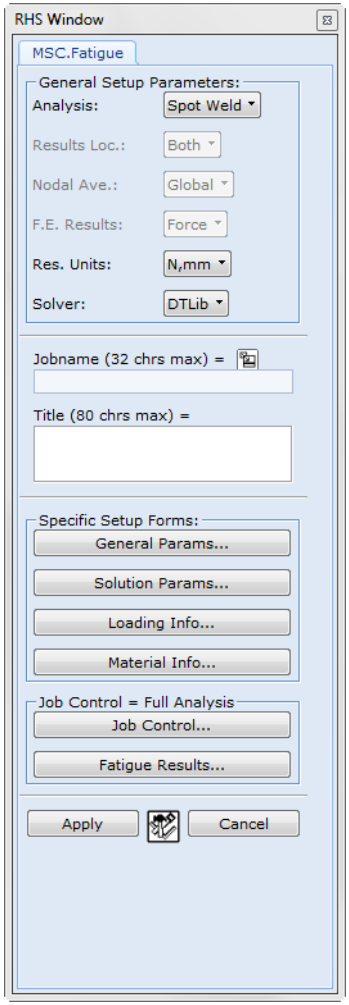
Parameter	Options	Description	Default
BackCalcAccuracy	0.01 <= **Value** < 100	This defines a percentage error on the target damage. The back calculation will iterate until the calculated damage matches the target damage within this tolerance.	**1**
BackCalcMaxScale	20 <= **Value** <= 5.e6	This defines the maximum value of the scale factor that will be reported in the analysis results.	**5**
BackCalcMinScale	0 <= **Value** <= 0.5	This defines the minimum value of the scale factor that will be reported in the analysis results.	**0.2**

dtfat Batch Operation

The setup starts in the MSC Fatigue forms within Patran or MSC Fatigue. You must set the analysis type to Spot Weld. There are three basic inputs just as for any other analysis types, those being Solution Parameters, Material and Loading Information. Results post-processing and Job Control are also described in this section.

Job Setup (Classic Modules)

The setup starts in the MSC Fatigue forms within Patran. You must set the analysis type to Spot Weld. There are three basic inputs just as for any other analysis types, those being Solution Parameters, Material and Loading Information. Results post-processing and Job Control are also described in this section.


RHS Window
MSC.Fatigue
General Setup Parameters:
Analysis:
Spot Weld
Results Loc.:
Both
Nodal Ave.:
Global
F.E. Results:
Force
Res. Units:
N,mm
Solver:
DTLib
Jobname (32 chrs max) =
Title (80 chrs max) =
Specific Setup Forms:
General Params...
Solution Params...
Loading Info...
Material Info...
Job Control = Full Analysis
Job Control...
Fatigue Results...
Apply
Cancel

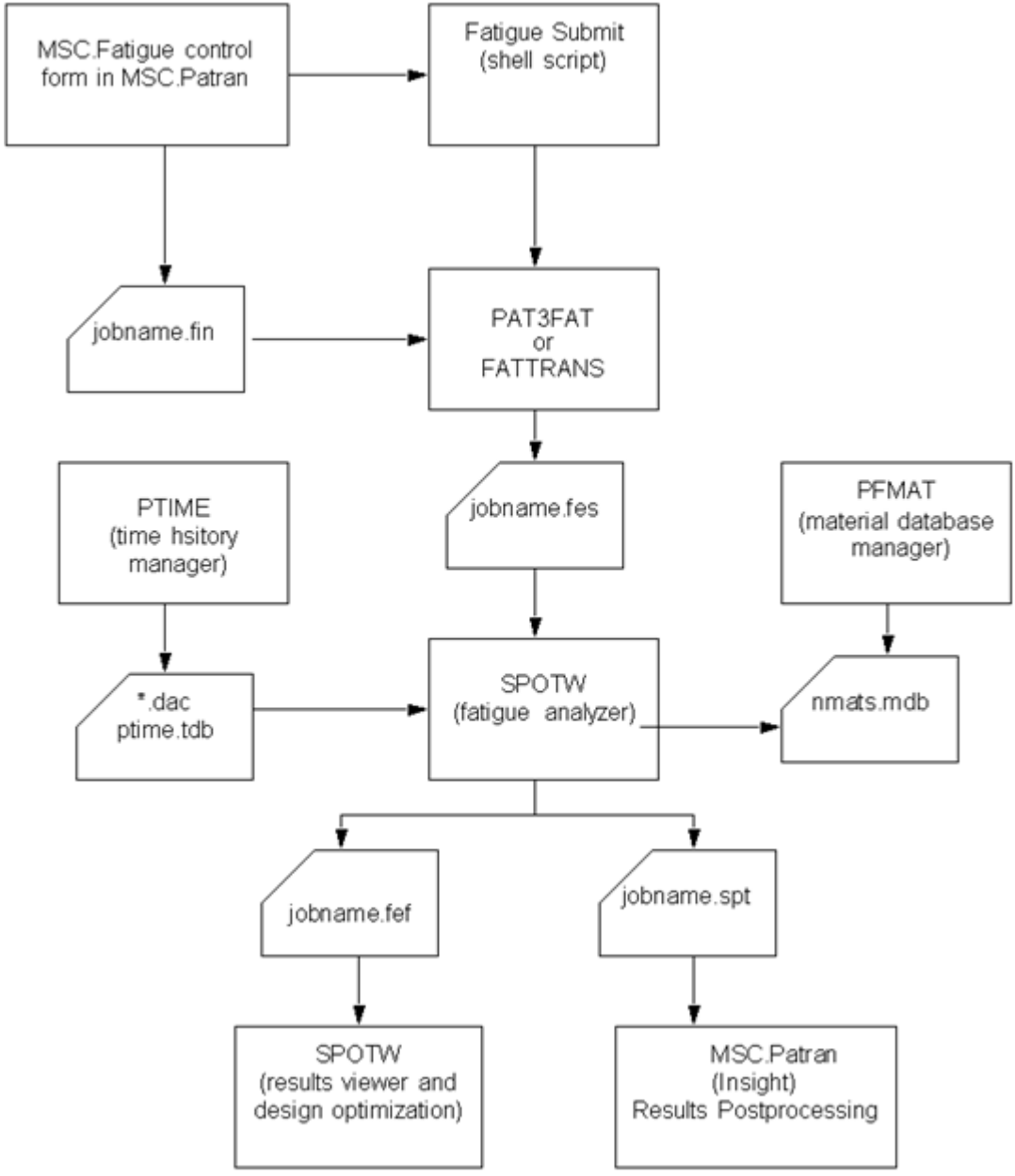


Figure 10-3 Spot Weld Analysis Submittal Schematic

Basic Information

All programs in the MSC Fatigue system may be executed by typing the names of the program at the system prompt. These programs may ask questions which are not normally presented to you since they are executed as batch jobs when called from MSC Fatigue Pre & Post or Patran. The programs normally used in a Spot Weld Analysis are listed below.

1. Data Preparation

PFMAT	Materials Database Manager
PTIME	Time History Database Manager and ASCII Time History File Convertor
PVXMUL	A Peak-Valley Extraction Program for Reducing Lengthy Time Histories
MFD	A Multi-file Display Program

2. Global Multi-Spotweld Analysis

PAT3FAT	Model Database (Patran) to Fatigue Input Translator
FATTRANS	New Model Database (Patran) to Fatigue Input Translator
SPOTW	Fatigue Preprocessor (rainflow cycle counting)

3. Results Postprocessing and Design Optimization

SPOTW	Results Listings, Polar Plots
SPOTW	Design Optimization Tools

4. General Utilities

FEFAT	FES File ASCII/Binary Convertor
PFTRM	Terminal Driver
CONFIL	Binary to Binary File Convertor

Analysis Route

The actual programs needed to complete a global multi-node or element total life or crack initiation analysis are:

FatigueSubmit	Shell script (necessary for submittal from MSC Fatigue Pre & Post or Patran)
PAT3FAT	Translator (creates the fatigue input file `filename.fes`)
FATTRANS	New translator (creates the fatigue input file `filename.fes`)
SPOTW	Fatigue Analyzer (life prediction)

The programs and options must be used in this order. The results may be reviewed using the results listing options in SPOTW or by inspecting the ASCII results files (`jobname.fef` and `jobname.spt`) using a text editor or by inspecting marker plots in Patran/Insight.

Necessary files

When a global multi-element fatigue analysis is set up using MSC Fatigue Pre & Post menus or Patran, these are the files necessary to run the analysis and the files that are created.

jobname.fin	This file contains all the analysis parameters that were defined in the main and subordinate MSC Fatigue forms, (e.g., loading time history data file names). In addition, the analysis type and job titles are also defined in this file. A full description of this file is contained in The Job Information File (jobname.fin), 369.
Database	The groups of nodes and elements for which the fatigue life is to be calculated are contained in the database. For a spot weld analysis, it is necessary to have carried out a finite element analysis in MSC.Nastran with each spot weld defined as a CBAR element joining two sheets of shells. The beam element cross-sectional forces and moments are scaled, superimposed and used to calculate the structural stresses from which life is calculated. The results must be in the database including group information comprising the spot weld elements themselves to be used in the analysis.
Additional Files	Other files that are necessary to complete a successful fatigue analysis are the time history files (`ptime.adb`, `ptime.tdb` and `*.dac` or *.pvx), and the materials database (`nmats.mbd`) which is generally held in a central location and not necessary to be located in the user's local directory.

The first of these files is ASCII and may be edited using a standard text file editor. Although this method of defining the MSC Fatigue job parameters is not as automated as the MSC Fatigue menus in Patran, it does offer a simple and rapid method of changing a few parameters without the encumbrance of a menu structure.

File that may be created during an analysis run are summarized below:

File Name	Description
***jobname*.fin**	Job parameter file (ASCII).
***jobname*.fes**	MSC Fatigue Input file (Binary).
***jobname*.asc**	ASCII version of the ***jobname*.fes** file.
***jobname*.fpp**	MSC Fatigue intermediate results file (Binary).
***jobname*.msg/log**	MSC Fatigue message and log files (ASCII).
***jobname*.sta**	Job status file (ASCII).
***jobname*.fef**	Global multi-node/element results file (ASCII).
***jobname*.spt**	Flat result file containing detailed spot weld results (ASCII)
***jobname*nn.nd /.ndm**	Spot diameter-life XY data (ASCII/Binary)
***jobname*.abo**	Job abort file (ASCII).

File Name	Description
***jobname*.fpr**	Job currently active alert file (ASCII).
***jobname*nn.pod**	Polar damage data (damage as a function of angle around the spot weld (Binary)
***jobname*nn.pol**	Polar life data for spot weld(Binary).
***jobname*nn.por**	Polar stress range data for spot weld (Binary)
***jobname*nn.rst/.rsm**	Spot diameter-life XYdata (ASCII/Binary)
***jobname*nn.plt/.ptm**	Sheet thickness-life XY data (ASCII/Binary)
***.dac, *.cyh**	Loading time history/rainflow matrix files (Binary).
***.xyd**	K solution XY data (ASCII).
***.tem**	Plotting format data (ASCII PCL file).
***_tmpl**	Results template files (ASCII).
***.adb/.tdb/.mdb**	Time history and Materials database description files (ASCII/Binary/Binary).

After the translator has been run (described in The Translator (PAT3FAT or FATTRANS), 294) and the fatigue input file (`jobname.fes`) has been created, the Spot Weld Fatigue Analyzer, SPOTW, is run. There is no pre-processing stage in Spot Weld Analysis. When run in interactive mode, this program asks for a number of input parameters which are passed in through the `jobname.fes` file when run from the MSC Fatigue menus within Patran. A full description of file content is provided in Description of Files, 369.

Solution Parameters

The only parameter that can be set on this form is the design criterion (% Certainty of Survival). The nature of the calculation, including mean stress correction, is predetermined. Some other features of the calculation, such as the number of calculation points, can be controlled by running the spot weld analyzer interactively or in batch mode.

Only a design criterion is displayed in this form as a certainty of survival. The default is 50% and is based on the scatter of the S-N curve. For example, to be 96% certain that the life will be achieved, set the slider bar at 96. This value is used to modify the S-N curve according to the standard error scatter parameter (SE). The design criterion parameter will be meaningless if the value of SE is 0. A Design Criterion value of 50 leaves the S-N curve unmodified.

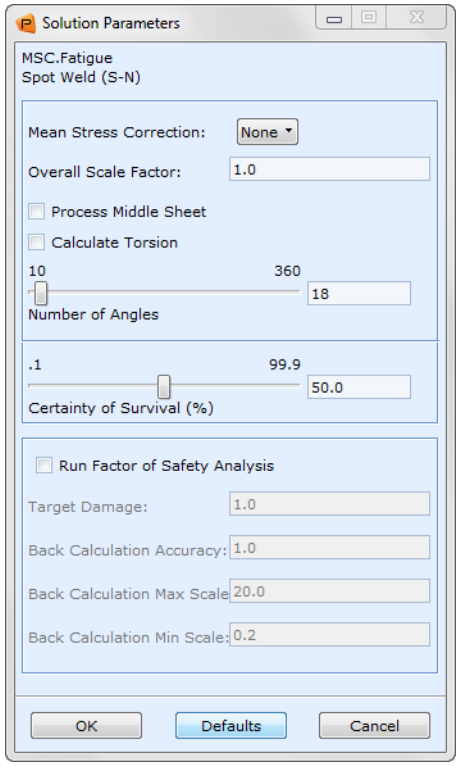


Figure 10-4 Spot Weld Solution Parameters Form

Materials Information

The basic S-N curves for the nugget and sheet metal are standard, meaning they have two slopes, a transition point, and statistics that are described by a log-normal distribution on life. In addition to the normal S-N data, given for R=0, the software also uses a single mean stress sensitivity factor. The mean stress sensitivity factor M is used to calculate an equivalent stress amplitude S0 at load ratio R=0 for each cycle with amplitude S and mean Sm.

There are a handful of S-N curves delivered in the materials database for spot weld analysis. There are also two generic spot weld S-N curves delivered, one for the spot weld nugget and one for the sheets. These are generic and may not be appropriate to your spot weld application, in which case you will have to provide your own data.

The Material Information form is similar to that for other analysis types and operates in the same fashion. The major difference is that you must supply three S-N curve definitions for each grouping of spot welds that you wish to analyze. The spot weld module allows analysis of spot welds modeled three different ways: as bar elements, as CWELD elements, and as hex elements.

For modeling spot welds as bars or CWELDs, the user must set up groups with only the bar elements that represent the spot welds. If the spot welds have been modeled as hex elements, the spot weld module assumes that all hex elements in the database represent spot welds. The user must select the type of spot weld element and it's associated group. This group will appear in the first column of the spreadsheet on the form. The other columns are for specifying the S-N curves for the nugget and the two sheets as well as the nugget radius and sheet thicknesses. To add more spot weld groups (up to 165) simply add more rows to the spreadsheet.

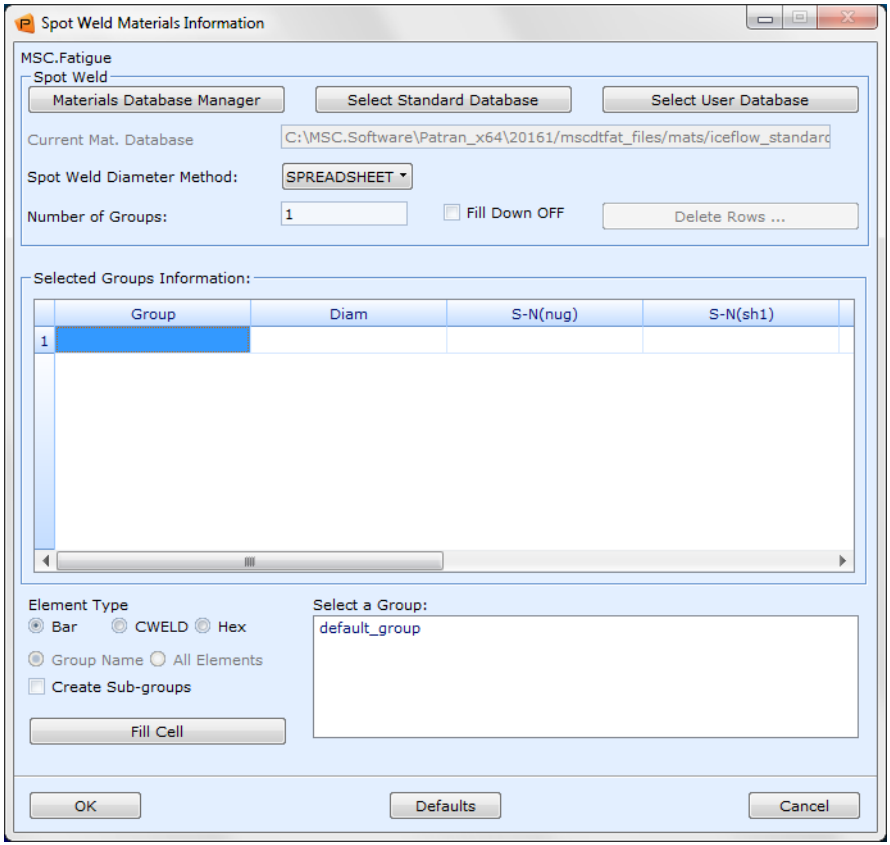


Figure 10-5 Spot Weld Materials Information Form

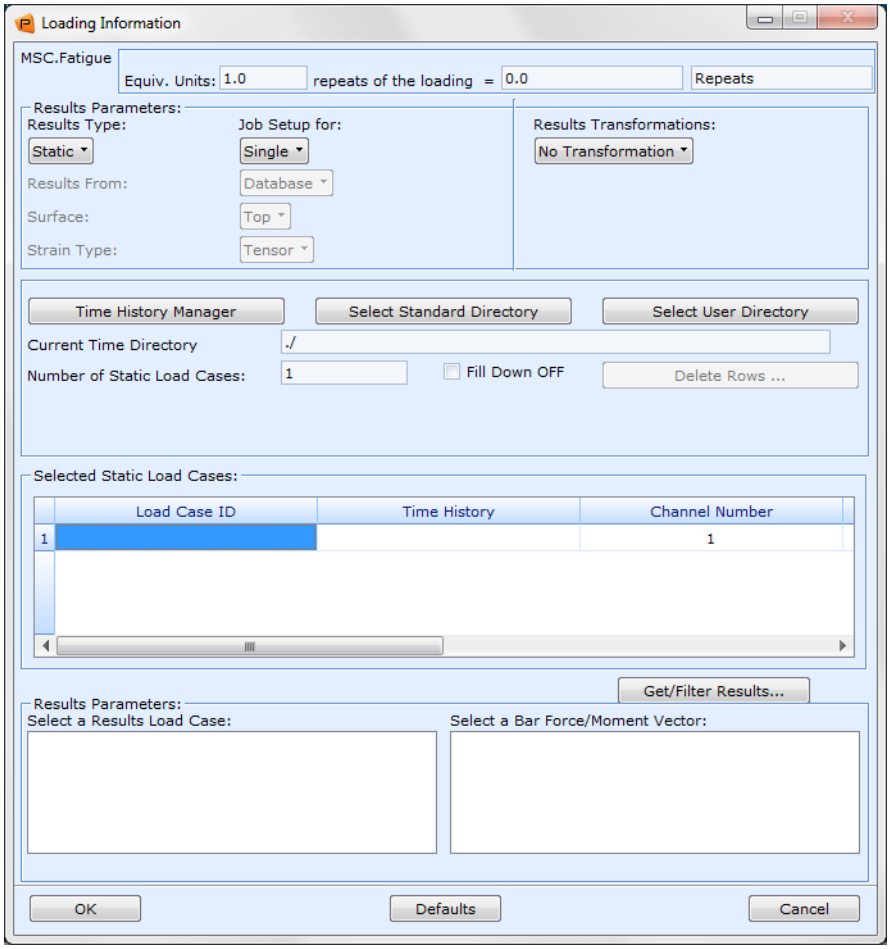
The cells of the spreadsheet are filled are filled in automatically from data derived from the information in the first spreadsheet cell. Default values are supplied for missing data. To change the system supplied values, select any cell (making it active) with the cursor. A listbox, databox or a pulldown menu will appear below the spreadsheet from which you make your selection. The data selected will then occupy the selected cell and the adjacent cell will become active for data input. A list of the items for the spreadsheet are explained here.

Parameter	Description
Group	When this cell is active, a list of spot weld modeling options and sub-options become visible. Select the spot weld modeling option appropriate. The **Bar** or **CWELD** options assume the bars have all been placed in one exclusive group. The available groups will appear in a listbox for selection. A sub-option Create Sub-groups will split a selected group into a series of groups based on the thickness pairs found at the end of the bar elements. Selecting the Hex modeling option will look at all the hex elements in the database and group the hex elements according to common calculated diameters. For any spot weld type selected, the Fill Cell button must be executed for the Group cell to be filled. **Note:** Even though groups allow name spaces, for use with fatigue regions they cannot have any spaces in the name, either leading, trailing or anywhere in between. For the Bar or CWELD options, if the Create Sub-groups sub-option is not selected, it is best for the group selected to have elements with the following in common: they must represent welds with the same diameter and having the same S-N dataset, they must all start from sheet metal (Sheet 1) with the same thickness and S-N dataset and they must all end on sheet metal (Sheet 2) with the same thickness and S-N dataset. The spot weld axis must be positive from Sheet 1 to Sheet 2. If Sheet 1 and Sheet 2 are identical, the direction is immaterial.
Diam	For the **Bar** option, the diameter is calculated and loaded automatically from the minimum thickness found in the first bar element. The user is able to use an external file to control the thickness/diameter mapping (**spotweld.sys**). A default diameter of 1 is used if the minimum thickness is zero. The user may enter a diameter in the range 1< d < 10 (mm) to change the automatically calculated value. For the **CWELD** option, the diameter is extracted from the stored CWELD data. A default diameter of 1 is used if the extracted diameter is zero. For the **CHEX** option, the diameter is extracted from the stored CHEX data. A default diameter of 1 is used if the extracted diameter is zero.
S-N(nug)	This cell is automatically loaded with the Spot_nugget_generic material dataset. An alternative dataset may be selected by selecting this cell and picking one of the available spotweld datasets from the data box. Selecting one of these from the listbox will fill the active cell with the material name and make the adjacent cell active.

Parameter	Description
S-N(sh1)	This cell is automatically loaded with the Spot_sheet_generic material dataset. An alternative dataset may be selected by selecting this cell and picking one of the available spotweld datasets from the data box. Selecting one of these from the listbox will fill the active cell with the material name and make the adjacent cell active.
T(sh1)	The thickness for Sheet 1 is loaded up automatically based on the information for the group(s) in the database. The user may change this thickness by entering a suitable thickness. This parameter is used in the structural stress calculation.
S-N(sh2)	This cell is automatically loaded with the Spot_sheet_generic material dataset. An alternative dataset may be selected by selecting this cell and picking one of the available spotweld datasets from the data box. Selecting one of these from the listbox will fill the active cell with the material name and make the adjacent cell active.
T(sh2)	The thickness for Sheet 2 is loaded up automatically based on the information for the group(s) in the database. The user may change this thickness by entering a suitable thickness. This parameter is used in the structural stress calculation
SF	A multiplier can be specified for each group of spot welds. The default is 1.0 (no modification). See discussion below.

Loading Information

The loading information is almost identical to that of the other analysis types. The only difference is that you specify bar forces instead of stresses from the database. A list of vector results will be displayed in the listbox of results. You may pick either the translational forces or rotational moments. The software will extract both even though only one is selected. Results can only be extracted from the database and not from external files. This, of course, assumes that the bar force results have been requested properly from a MSC.Nastran analysis and imported into the database.

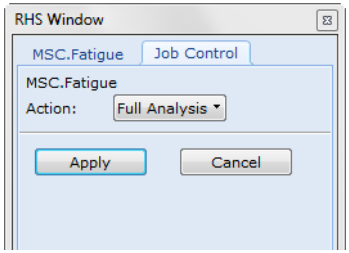


You are referred to Loading Information Form, 70 for more detailed information about this input. Remember for spot weld analysis:

1. Only results imported into the Database are supported.
2. You select Translational Forces or Rotational Moments and not stress or strain tensors.
3. Everything else is identical to setting up any other analysis type.

Job Control

The Job Control form allows you to submit a Full Analysis. A Partial Analysis is not possible. Most other activities available from the Job Control form are as with the other analysis types. When running Interactive, the SPOTW program will be invoked. When running a Full Analysis, SPOTW will run in the background.



Since control of the spot weld analysis job is so similar to any other analysis jobs, you are referred to Job Control, 864. Remember though for spot weld analysis:

1. Partial Analysis behaves the same as Full Analysis (there is no separate preprocessing stage)
2. Interactive spawns SPOTW
3. The Calculate Normals option is inappropriate for a spot weld analysis

The following files are created after a spot weld analysis is complete.

File name	Description
jobname.fin	Fatigue job parameter data.
jobname.fes	FE stress and fatigue input file.
jobname.fef	Fatigue results file.
jobname.msg	Message file.
jobname.sta	Job status file.
*jobname*nn.spt	Detailed fatigue results file

Job Execution Status Messages

When a job is submitted, it will pass through three to five phases. The user will be informed through the status option of the progress of the job. Both success and error messages are displayed. The following list summarizes some of the typical, normal operation messages which the user may experience. Not all the messages will be displayed since the status file is updated very quickly in some cases. In certain cases, the status file may not be available, in which case a "Try again" message will appear. When execution is through MSC's Analysis Manager, these messages appear in the Analysis Manager message window.

Phase 1

```
JOB jobname HAS BEEN SUBMITTED BUT HAS NOT STARTED EXECUTION
JOB HAS BEGUN EXECUTION
WRITING THE JOB (.FIN) FILE
```

Phase 2

```
PAT3FAT" reading the neutral file...
PAT3FAT" reading the.FIN file...
PAT3FAT" reading the FE results...
PAT3FAT" writing the.FES file...
PAT3FAT" terminated normally
or
FATTRANS" reading the neutral file...
FATTRANS" reading the .FIN file...
FATTRANS" reading the FE results...
FATTRANS" reading the .FES file...
FATTRANS" terminated normally
```

Phase 3

```
Fatigue analysis module loaded and running
Processing spot weld n/m
Fatigue analysis completed successfully
```

In addition there may be other messages giving status of other aspects of the job. Error messages are also displayed via these status messages.

What To Do When a Job Stops

If the status message does not appear to be updating, it is possible that the job has halted due to an error. In many cases, that error message will be reported through the status facility. However, if it is not reported, you can investigate the problem by opening another window and examining the following file:

jobname.msg: This file will contain all the status messages for the job including any error messages.

Some hints on determining why a job has failed:

1. If the `jobname.fin` file and the `database` exist in your directory, try running the job interactively by typing:

   `pat3fat` `jobname`

   or by typing:

   `fattrans` `jobname`

   then checking the message file.
2. If the `jobname.fes` file exists, run the SPOTW program interactively and watch for error messages. Type **spotw** at the system prompt.
3. If the `jobname.fes` file exist, but SPOTW still gives errors, try running FEFAT. Type **fefat** at the system prompt. Use the Utilities menu to examine the file contents.

Also check the file called `batlog.lst` if it exists for any additional clues if none of the above helps.

Important:	If a job inadvertently quits, sometimes a `jobname.fpr` file is left in the directory. This file is created during submission to detect a running job so that inadvertent submissions while a job is in progress of the same jobname are detected. In some cases, it may be necessary to remove this file before resubmitting the job.

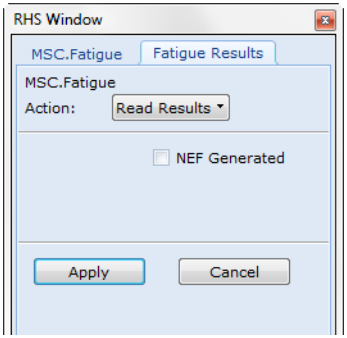
Error Messages

See Error Messages (App. 19) for a description of error messages and possible solutions.

Results

A Patran element style `.els` file called `jobname.fef` which contains the usual MSC Fatigue information, plus a few additional pieces of information, is created from the analysis and is read in to the database. The results file columns are: damage, log-damage, life (repeats), log-life (repeats), life (equivalent units), log-life (equivalent units), failure location, angle, node, maximum force and are apparent when accessing them from the postprocessor Results application.

The Log of Life in Repeats and the Log of Life (equivalent Units) are log (base10) of the life in repeats or user units respectively.



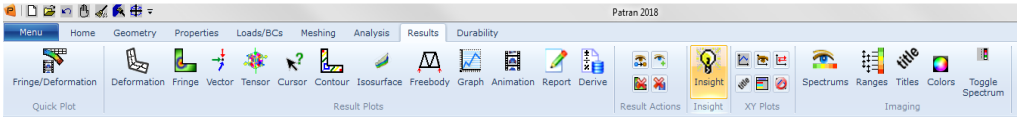
Also, a flat file called `jobname.spt` which contains damage and maximum and minimum stress is created which can be processed by the SPOTW module itself. The best option is to use Insight as opposed to the Results application.

As with other analysis types, these options remain basically the same for spot weld analysis, therefore you are referred to Postprocessing Results, 100. The differences are:

1. List Results, Re-Analyze, and Optimize will spawn SPOTW to the appropriate functionality requested.
2. Factor of Safety is not available.
3. When Results are read into the database they exist in the form of element centroidal values for the various bar elements that represent the spot welds. This makes the standard postprocessing tools (fringe, contour plots) ineffective. The best is to use Insight marker plots.

Marker Plots in Insight

The best way of visualizing spot weld fatigue analysis results on the FE model is using Insight. This is done by selecting the Insight toggle from the main menu bar or MSC Fatigue Pre & Post or within Patran.



Once the Insight tool has opened, set the action to Create and the tool to Marker, choose a name for the marker plot. Use the results selection button to select the results to be plotted. The results available are:

Damage	Damage
Log of Damage	Damage reported in log units which is interpreted to be 10 raised to the reported value. This is log(base10) of damage.
Life in Repeats	Life in repeats of the defined load history service.
Log of Life in Repeats	Life in log units which is interpreted to be 10 raised to the reported value. This is log(base10) of life.

Location of Failure	This refers to spot weld analyses, where the failure can occur in Sheet 1, Sheet 2 or the Nugget.
Angle of Failure	This is the angle at which the damage is largest in a spot weld analysis. The angle is relative to the spot weld x-axis.
Node ID of Failure	A spot weld may be predicted in either of the 2 sheets, or in the weld nugget itself, corresponding to the ends and the middle of the CBAR element. If failure occurs in one of the sheets, the result here is the corresponding Node ID. Otherwise this result is 0.
Maximum Force Encountered	This is the maximum translational force that is transmitted through the spot weld during the loading history
Life (equivalent units)	Life in user-defined units.
Log of Life (equivalent units)	This is life in user-defined log units which is interpreted to be 10 raised to the reported value. This is log(base10) of life.

Select the result to be plotted; one of the most useful to plot is Log of Life (equivalent units).

Use the Marker Attributes button to choose the kind of marker to be applied to the model. The recommended choices are:

■ Color:	Mapped
■ Type:	Sphere
■ Scale:	Screen
■ Scale Factor	0.03

Set the target for posting the results to Elements and target only the groups of beam elements for which the spot weld results were calculated.

Spot Weld Analyzer (SPOTW)

The MSC Fatigue analysis module SPOTW performs a number of tasks including Global Analysis, Results Listing and Design optimization. Module operation of each of these tasks is described in detail in this section.

The operation of SPOTW can be in three modes: by spawning from the MSC Fatigue Pre & Post or Patran environment, in stand alone mode by typing **spotw** at the system prompt or in batch mode. The only difference is that in stand alone mode, the user must supply the jobname. (In direct mode from Patran, these are passed to SPOTW automatically). Once SPOTW has been initiated in either of these modes, two windows will be presented with the Motif driver. The top, small form is a generic form and allows for general program control. This is discussed in detail in Module Operations (Ch. 18) for the Motif driver.

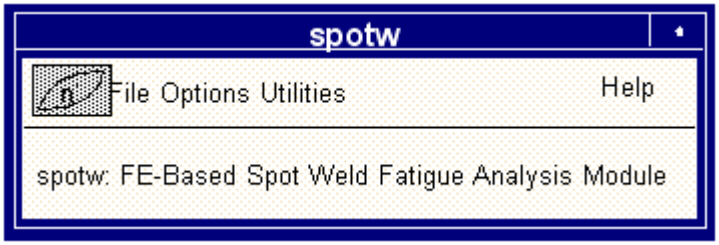


Figure 10-6 SPOTW Utility Form

The main menu appears as follows. Each item is discussed in this section.

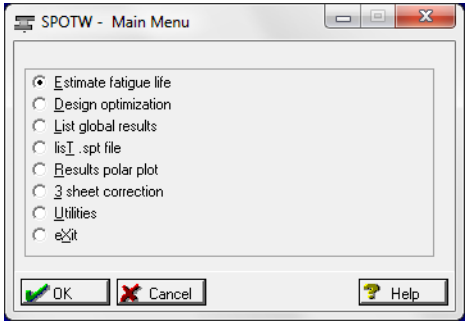


Figure 10-7 SPOTW Main Menu Form

1. **Estimate Fatigue Life** - This option will rerun the analysis interactively for you. If this it is a computationally intensive operation, you may find it more convenient to run in batch mode. See Estimate Fatigue Life, 794.
2. **Design Optimization** - This option will put you in another menu to allow for sensitivity studies and design optimization tasks. See Design Optimization, 797.
3. **List Global Results** - This option will read the `jobname.fef` file and tabularly display the results. See List Global Results, 814.
4. **List .spt File** - This option will read the `jobname.spt` file and tabularly display the results. See List .spt File, 814.
5. **Results Polar Plot** - This option will read the `jobname.spt` file and graphically display the results for a given element in a polar plot. The plot is most useful when using log scale. Two of three curves will appear on each plot representative of the location, at either sheet and at the weld nugget (if it is calculated). Each curve point represents the damage at each angle calculated. See Results Polar Plot, 815.
6. Three Sheet Correction. See Three Sheet Correction, 816.
7. The utilities are identical to those used in other modules. See Utilities, 360.
8. **eXit** - This options will exit you from the program.

Estimate Fatigue Life

Note that the fatigue analysis may take some time. It may be worth considering operating SPOTW in batch. Batch operation is discussed in SPOTW Batch Operation, 819.

When the "Estimate Fatigue Life" option has been selected, the user will be presented with a number of questions. The first question asks for the input file name. Click the **OK** button once a file name (`jobname.fes`) has been selected. Use the List button to list all available input files. These files have been created by the PAT3FAT or FATTRANS translator. The default will be the last `jobname.fes` created. Once a valid file name has been entered, the user will be presented with a summary of the `jobname.fes` that has been opened. Each of these parameters may be changed or edited.

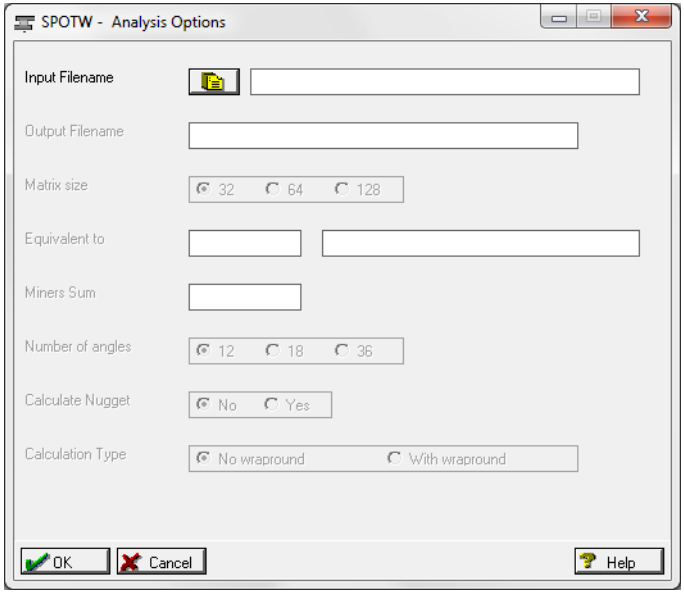


Figure 10-8 SPOTW Global Analysis Form

The following table explains each entry on the previous form.

Field	Description
Input Filename	This is the fatigue input file name (`jobname.fes`) to be used in the fatigue pre-processing. The job must have already run at least through the PAT3FAT or FATTRANS translator to produce a `jobname.fes` file. This is achieved by carrying out a full or translate only submission from the job submit options in the MSC Fatigue menus of Patran, or by running PAT3FAT in stand-alone mode (see The Translator (PAT3FAT or FATTRANS), 294). To select a jobname from a list of available jobs, use the List button. Once the file name has been supplied and the screen inputs accepted, the rest of the input options will be displayed.
Output filename	The default is the jobname. After processing, files called `jobname.fef` and `jobname.spt` will exist. You will be requested to overwrite any existing output file of the same name if one exists.
Matrix Size	The software creates a rainflow matrix at each calculation point. The matrix can have 32x32, 64x64 or 128x128 bins. Larger matrices will produce more accuracy, especially for simple or constant amplitude loadings, at the expense of computation time. Matrices output by the program in design optimization mode will have the same number of bins as specified here. Select the desired number of bins here.
Equivalent to	In these two fields, the number of equivalent units is set. If for example, the time history is equivalent to 5 hours of operation, then “hours” would be a suitable name and 5 would be a suitable number (although you may enter any name or any number).
Miners Sum	The Miners sum is set to 1 by default. Some calculation methods require the Miners sum to be modified. Reducing the Miners sum will cause the predicted life to be reduced by the same factor. A value of 1 is recommended for constant amplitude loadings, and 0.5 for variable amplitude loadings (Linear Damage Summation, 1380).

Field	Description
Number of Angles	By default, SPOTW will make fatigue calculations at 10 degree angles around the spot weld, making 36 angles in all. You may reduce calculation times, at the expense of some accuracy, by changing the number of angles to 18 (20 degree intervals) or 12 (30 degree intervals).
Calculate Nugget	By default, SPOTW will carry out calculations at intervals as described above, in both Sheets 1 and 2 and in the weld nugget, making a total of up to 108 fatigue calculations per spot weld. In practice, welds rarely fail by cracking through the weld nugget (unless the weld is too small compared to the sheet thickness, or improperly made). For this reason, it may be desirable to reduce calculation time further by omitting calculation of the weld nugget.
Calculation Type	Select the method used to cycle count the spot weld time histories. No wrap around ignores the last point as a potential turning point whereas wrap around mode puts the first point again to allow the last point to be a turning point.

When this form is complete, select **OK** and the analysis will run. While the analysis is running, a form appears which indicates (spotwelds calculated/total number of weld spots), current element, current node (0 indicates the weld nugget is being calculated) and the current angle.

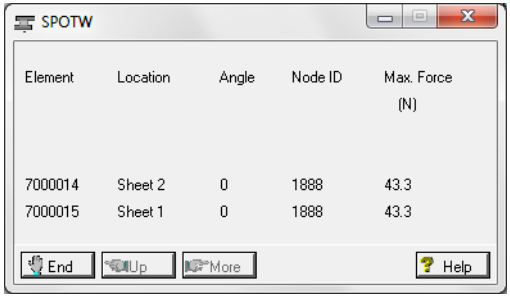


Figure 10-9 Job Status Form

If you wish to stop the analysis, select Pause. You will be prompted to quit or continue the analysis.

When the analysis is complete, a results page appears (see Figure 10-10). This provides a summary of the results for the ten worst spot welds. The information provided is Damage, Life in Repeats, Life in Equivalent Units. If More is selected, more information is provided about the same list of elements. This form is shown in Figure 10-11.

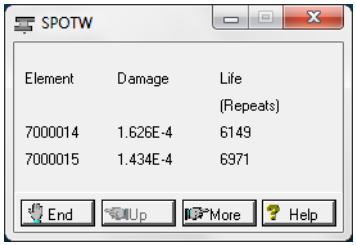


Figure 10-10 Summary Results Page 1

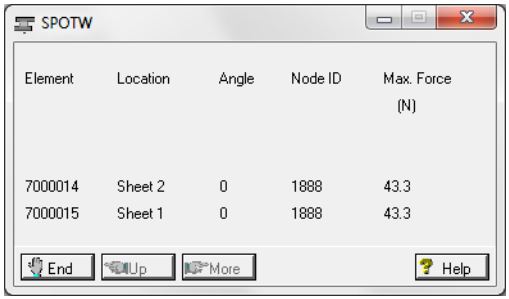


Figure 10-11 Summary Results Page 2

The information on this form for each element is, the results location (Sheet1, Sheet2 or the Nugget), the angle at which most damage is predicted, the node number (0 if failure is at the nugget) and the maximum Force through the spot weld.

Design Optimization

Having completed a global multi-element analysis, the user will have identified an area of the structure which is of particular interest, for instance, it may be likely to fail before or close to the design life, or it may comfortably exceed the design life, offering opportunity to reduce cost by reducing the strength.

The design optimization option within SPOTW provides a set of semi-automatic tools to assess fatigue design options. It makes calculations for single elements based on the information in the `jobname.fes` and `jobname.fef` files. It supports a number of options including back calculation of parameter values which meet a target life, sensitivity studies on critical parameters and a material selection option.

Having selected the node or element of interest, SPOTW will carry out a single fatigue calculation based on the parameters from the global multi-element analysis and present the results in more comprehensive form than that available in the global analysis. The design optimization analysis options are then presented on a main analysis page from which the user can set up the optimization calculations.

SPOTW presents its results in the form of analysis summary reports, 3 dimensional cycle or damage histograms, fatigue life sensitivity tables, life versus parameter plots, polar damage versus angle plots, and stress history plots.

Stage 1: Module Operation

The operation of SPOTW from within the Patran environment is identical to its operation in stand-alone mode.

The first screen to be presented when the program starts is shown in Figure 10-12. Certain information about the analysis that has been carried out is stored in the results (`.fef`) file. For this reason, it is necessary to specify the results file as well as the input file name.

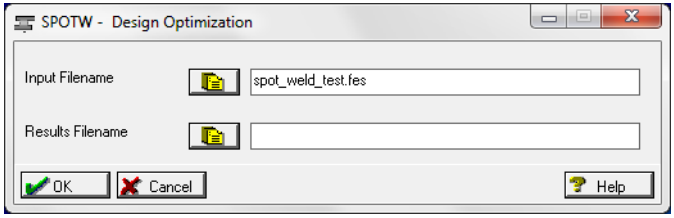


Figure 10-12 Design Optimization Job Entry Screen 1

When this form is accepted the form illustrated in Figure 10-13 is seen.

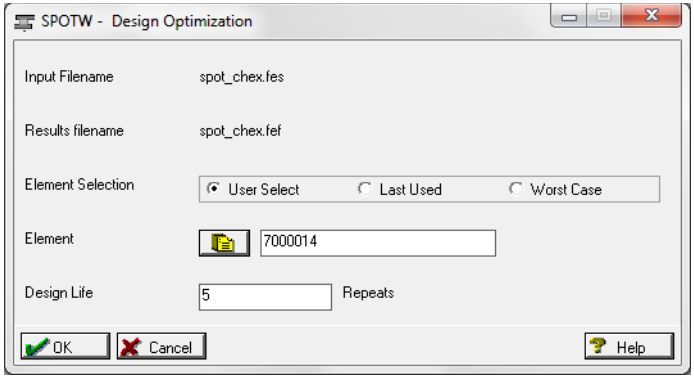


Figure 10-13 Design Optimization Job Entry Screen 2

The fields on these screens are described below.

Field	Description
Input Filename **and** **Output Filename**	These are the names of the job which is to be used in the design optimization analysis. The job must have already run to produce a `jobname.fes` file and a `jobname.fef` file. The `.fef` file need not have the same name as the `.fes` file, but should be based on it. To select a jobname from a list of available jobs, use the List button. Once the jobnames have been supplied and the screen inputs accepted, the rest of the initial input options will be displayed.
Element Selection	There are three options offered on this field: Last node/element used recalls the number of the node or element used in the last job. This number is shown in the Node Number field. If the last job used a different Patran geometry model, this option is unlikely to offer a meaningful element number. User entry allows for typing in a number in the Element Number field shown below the Element Entry menu. A list of possible element numbers is available using the list button. The Worst case element option searches the jobname.fef file to find the element with the most damage as calculated by the global fatigue analysis. Once the critical element is found, its number is presented in the Element Number field.
Element Number	The number displayed in this field depends on the choice made in the Element Entry described above. Use the list button to display a list of valid element numbers.
Design Life	The Design Life is a target life which is associated with the component or structure being analyzed. The life should be specified in the user units. These units and the number of these units equivalent to 1 repeat of the time history are displayed under the Design Life field. A design life must be entered here. When all fields are filled in appropriately, click the OK button. At this stage, SPOTW carries out an initial analysis using the original fatigue analysis parameters defined when the fatigue analysis was carried out. The life computed from this stage 1 analysis is used as a benchmark against which all subsequent optimization calculations can be judged. The results from this analysis are presented in a summary table on the screen and also written to the `pfatigue.prt` file. See Figure 10-14.

Note:	Results of this calculation may be slightly different from that given by the global analysis for the same weld spot. This is due to a slightly more accurate matrix discretization method used for design optimization.

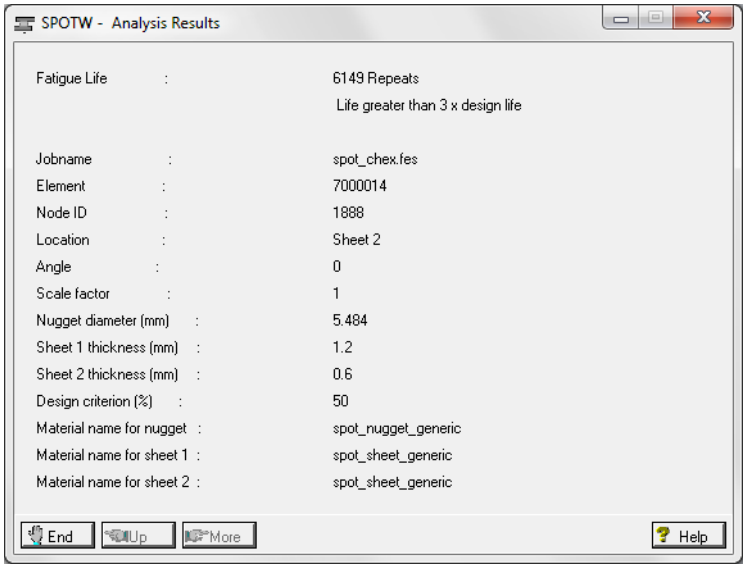


Figure 10-14 Results of the Stage 1 Fatigue Analysis for a Spot Weld job

The calculated fatigue life is reported, and underneath this a message is written which indicates whether the design life has been met or not. The three possible messages are:

- Design life exceeded
- Life within a factor of 3 of the design life
- Life less than the design life.

The other details summarize the analysis parameters and certain other results information. The jobname is given, together with the selected Element ID. The node ID helps to identify which end of the spot weld is more likely to fail. This will be set to 0 if the weld spot is predicted to fail through the nugget. Further information is provided about the location of the failure by telling whether the weld is more likely to fail in Sheet 1, Sheet 2 or the Nugget, and indicating at which angle (relative to the gauge coordinate system) the gauge is most likely to crack. The rest of the information gives the dimensions of the spot weld and the sheet metal, the material datasets used in the analysis, and the design criterion.

Stage 2: Module Operation

After going through the initial reanalysis of a particular node or element, the main analysis screen appears as shown in Figure 10-12. From this menu, all the analysis options are available. The current jobname and element identity is shown at the top of the screen together with the design life. When a back or sensitivity

calculation is defined, the type of analysis is reported in this area of the screen also. Menu options which are followed by 3 dots indicate a submenu hangs from that option.

To use this menu, choose the required option, set up the analysis parameters and finally, when ready, select the Recalculate option to submit the analysis. A percentage complete message will inform the user of the progress of the calculations. A description of each menu pick follows.

Parameter optimization

This option is the back calculation facility where a design life is supplied and SPOTW's automatic routines calculate the value of the chosen parameter that will achieve the target life; see Figure 10-12.

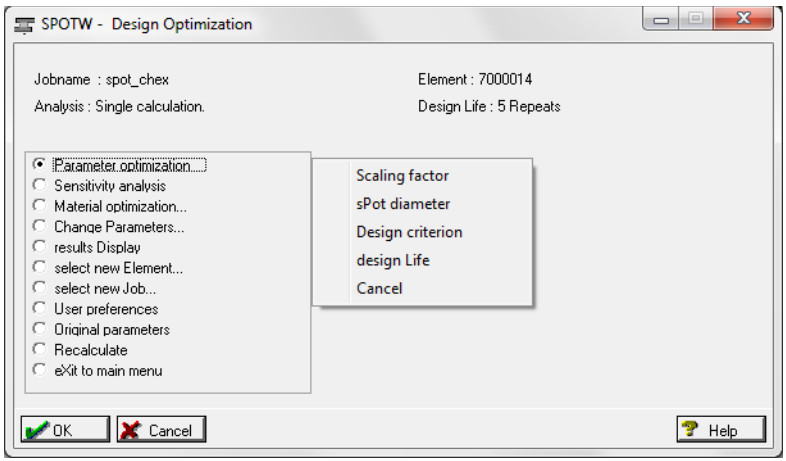


Figure 10-15 Optimization Main Menu with Parameter Optimization Submenu

There are four fatigue analysis parameters which may be used in Parameter Optimization. The parameters on which back calculation may be carried out are:

Option	Description
Scaling Factor	This factor can be thought of as a multiplier of the combined superimposed load input, or of the local forces and moments, and hence the stresses. Errors in the FE mesh could be evaluated with this feature.

Option	Description
Spot Diameter	The spot diameter affects the structural stress calculations (see Spot Weld Analysis Theory, 828 for details) and hence the life calculation. If there is scope for varying the diameter of a weld spot, this option may allow the optimum size to be determined.
Design Criterion	This is the confidence of survival parameter which is based on the standard error of the S-N curves. Using this parameter will tell how much confidence the user can have in the product reaching the target life. However, the user should also consider the error in other parameters computed in the FE analysis which may cause the life to be different from the estimate.
Design Life	This is not an optimization parameter but is used as a target for the optimization process. The design life may be changed or defined using this option.

Sensitivity Analysis

A sensitivity analysis allows the effect of variation in any of the input parameters on fatigue life to be explored, see Figure 10-16.

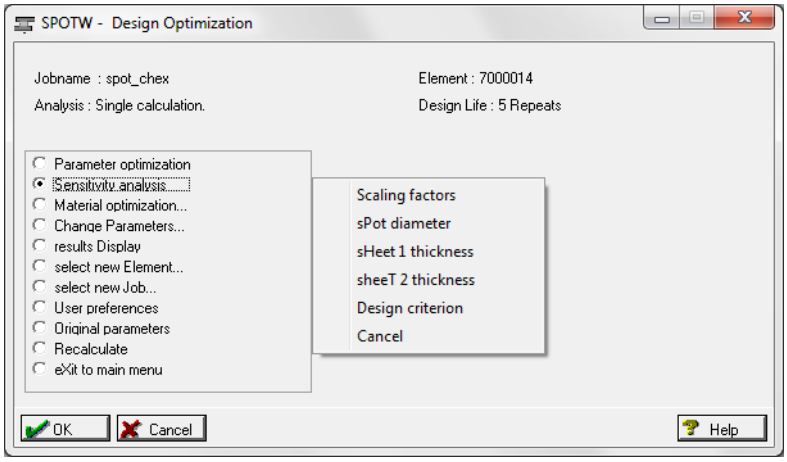


Figure 10-16 Sensitivity Analysis Submenu

To select one of these types of analysis, simply select the option which will then present the user with a data input form. In the box on this form, the user will be asked to provide a range of numbers for the parameter. Having done this, it is necessary to select the Recalculate option on the main menu.

Option	Description
Scaling Factor	This factor can be thought of as a multiplier of the combined superimposed load input, or of the local forces and moments, and hence the stresses. The effect of possible errors in the FE mesh could be evaluated with this feature. The user is required to enter a range of values in the data input form that appears when one of the options is selected. The values may be separated by spaces or by commas. The user may also specify a range of values by inputting a start value, end value and increment e.g. (1 10 2). A sensitivity plot can be created from this calculation.
Spot Diameter	The spot diameter affects the structural stress calculations (see Spot Weld Analysis Theory, 828 for details) and hence, the life calculation. This option allows the sensitivity of the design to spot weld diameter to be determined. The user is required to enter a range of values in the data input form that appears when one of the options is selected. The values may be separated by spaces or commas. The user may also specify a range of values by inputting a start value, end value, and increment e.g. (3 9 1). A sensitivity plot can be created from this calculation. Values of between 2 and 10 mm are allowed.
Sheet 1 Thicness **Sheet 2 Thickness**	The thickness of the sheet metal affects the structural stress calculations, and hence the predicted life. See Spot Weld Analysis Theory, 828 for details. This option allows the user to estimate the sensitivity of the design to sheet thickness. The user is required to enter a range of values in the data input form that appears when one of the options is selected. The values may be separated by spaces or by commas. The user may also specify a range of values by inputting a start value, end value, and increment e.g. (1 3 0.5). A sensitivity plot can be created from this calculation. Allowable values are between 0.5 and 5 mm. This option should be used with great caution. In practice, changing the sheet thickness may change the distribution of loads through the structure and will certainly affect the moments transmitted through the spot welds. Results obtained by changing the sheet thickness in Design Optimization should be checked by altering the FE model and rerunning the stress analysis and fatigue analysis.
Design Criterion	This is the confidence of survival parameter which is based on the standard error of the S-N curves. Using this parameter will provide the user with a prediction of the distribution of lives with the selected range of probabilities. However, the user should also consider the distribution in other parameters which may affect the fatigue life, such as load and geometric variations. The user is required to enter a range of values in the data input form that appears when one of the options is selected. The values may be separated by spaces or by commas. The user may also specify a range of values by inputting a start value, end value and increment e.g. (5 95 10). A sensitivity plot can be created from this calculation. Allowable values are in the range 0 - 100 %.

Material Optimization

The material optimization form allows for changing to a different material dataset for either of the sheets, or for the weld nugget. This facility may help the optimization of the design through selection of a better (or worse and less expensive) material. Note that the S-N curves required for this analysis are Spot Weld S-N curves, and may have no obvious relationship to the parent plate from which the spot welds are formed. Figure 10-17 shows the Material Optimization Form.

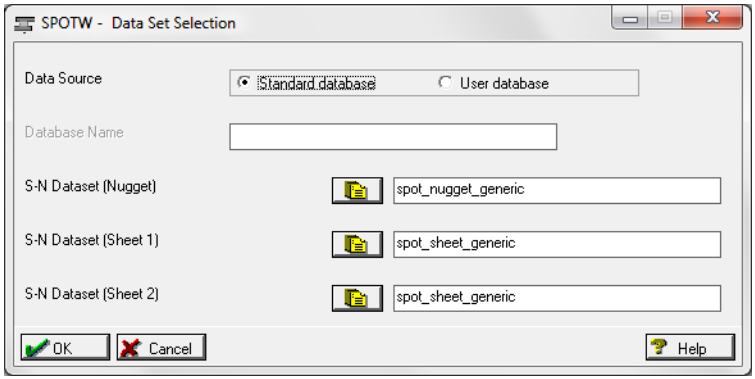


Figure 10-17 Material Optimization Form

Option	Description
Data Source	There are two sources of materials data used in the Spot Weld Analyzer. They are: The Standard Database, which can be the central database or a user specific local database (which is usually a modified copy of the standard database). A user database which contains data in the format of the standard database but which is specific to the user, i.e. a custom database.
Database Name	The field becomes live if User database is selected. The user database is created using the tools in Material Management (Ch. 4).
S-N Dataset (Nugget)	Select a Nugget S-N curve. All suitable materials currently available can be viewed with the list button.
S-N Dataset (Sheet 1)	Select a suitable S-N curve for Sheet 1. All suitable materials currently available can be viewed with the list button.
S-N Dataset (Sheet 2)	Select a suitable S-N curve for Sheet 2. All suitable materials currently available can be viewed with the list button.

Change Parameters

This design optimization option allows for changing individual parameters or to reset individual parameters back to their original values. This submenu us shown in Figure 10-18.

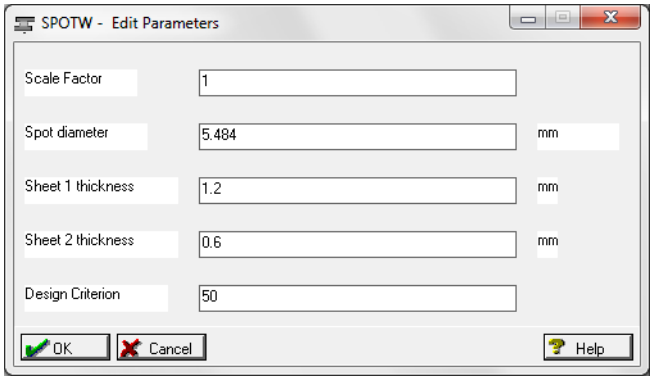


Figure 10-18 Change Parameters Submenu

Option	Description
Scale Factor	This factor can be thought of as a multiplier of the combined superimposed load input, or of the local forces and moments, and hence the stresses. You can accept the default to retain the original value, or you can supply a single scale factor.
Spot Diameter	The spot diameter affects the structural stress calculations (see Spot Weld Analysis Theory, 828 for details) and hence, the life calculation. You can accept the default to retain the original value, or you can supply a single spot diameter. Allowable values are in the range 2 - 10 mm.
Sheet 1 and 2 Thickness	The thickness of the sheet metal affects the structural stress calculations, and hence the predicted life. See Spot Weld Analysis Theory, 828 for details. Allowable values are between 0.5 and 5 mm. This option should be used with great caution. In practice, changing the sheet thickness may change the distribution of loads through the structure, and will certainly affect the moments transmitted through the spot welds. Results obtained by changing the sheet thickness in MSC Fatigue should be checked by altering the FE model and rerunning the stress analysis and the fatigue analysis.
Design Criterion	This is the confidence of survival parameter which is based on the standard error of the S-N curves. Allowable values are in the range of 0 - 100%.

Results Display

The presentation of the results in both tabular and graphical form is handled from this menu. The options available are shown in Figure 10-19 and discussed below:

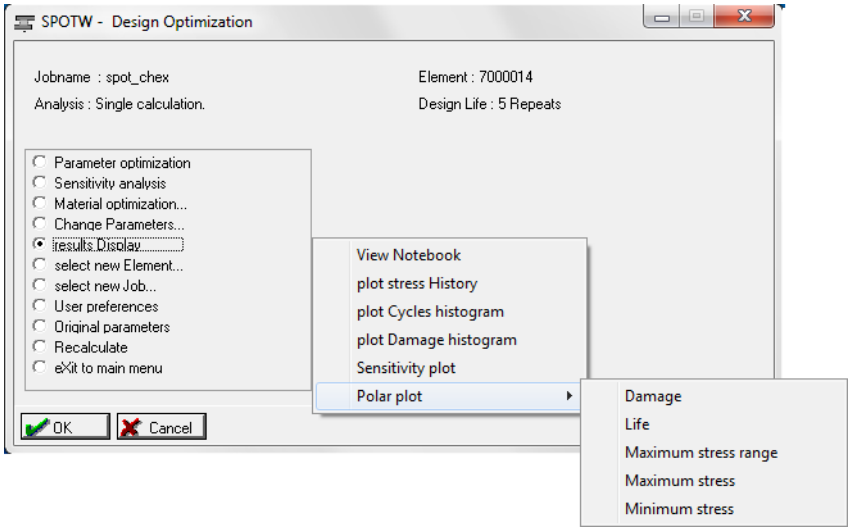


Figure 10-19 Results Display Submenu

Option	Description
View Notebook	Allows the review of the results of all analyses written to the Notebook (including the latest analysis if the Notebook is set to On). To view the Notebook, SPOTW uses whichever text processor that has been nominated, e.g. vi on an Linux Platform.
Plot Stress History	This option plots the effective stress history from the worst (shortest life) calculation point (location/angle) on the current spot weld. A description of the time history display is given in Plot an Entry Option, 245.
Plot Cycles Histogram	Plots the 3-dimensional rainflow cycle counted histogram of the effective stress at the worst calculation point on the current spot weld. A description of the graphical histogram display is given in Matrix Options, 356.
Plot Damage Histogram	Plots the 3-dimensional damage histogram which is related to the rainflow cycle counted histogram at the worst calculation point of the element being analyzed. A description of the graphical histogram display is give in Matrix Options, 356.
Sensitivity Plot	Displays an X-Y sensitivity plot when one of the options under Sensitivity Analysis from the main selection screen is chosen. This plotting option is only accessible immediately after running a sensitivity analysis. Various files are created which allow this plot also to be created under the Results button of the main MSC Fatigue form. An example of this type of display is shown in Figure 10-20. See Results Display, 344 for more details and also Sensitivity Plots, 107.
Poar Plot	Displays a polar plot of Damage, Life, Maximum Effective Stress Range, Maximum Effective Stress or Minimum effective stress as a function of angle around the spot weld. The angle is defined as the angle around the axis of the spot weld with respect to the MSC Fatigue spot weld x-axis. An example of this type of display is shown in Figure 10-21. Typically three datasets are plotted - for Sheet 1, Sheet 2, and the Nugget (unless of course the Nugget has not been calculated). See Polar Display (MPOD), 820 and Results Polar Plot, 815 for more details.

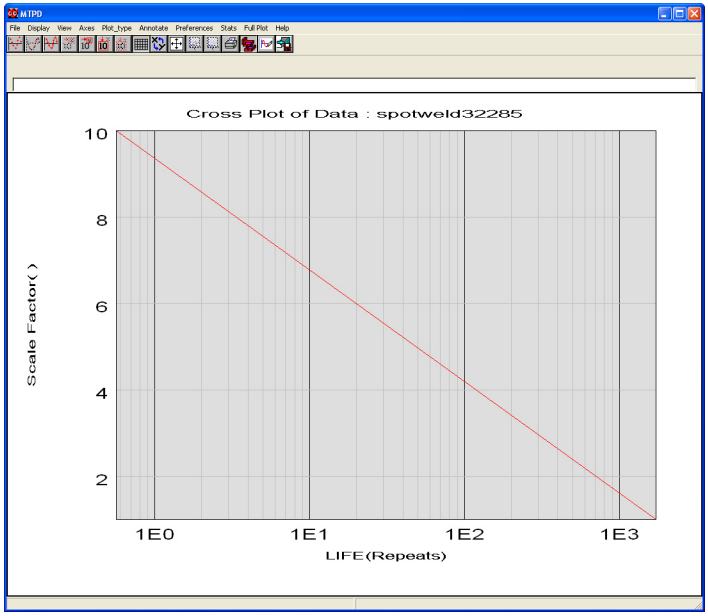


Figure 10-20 Sensitivity Plot

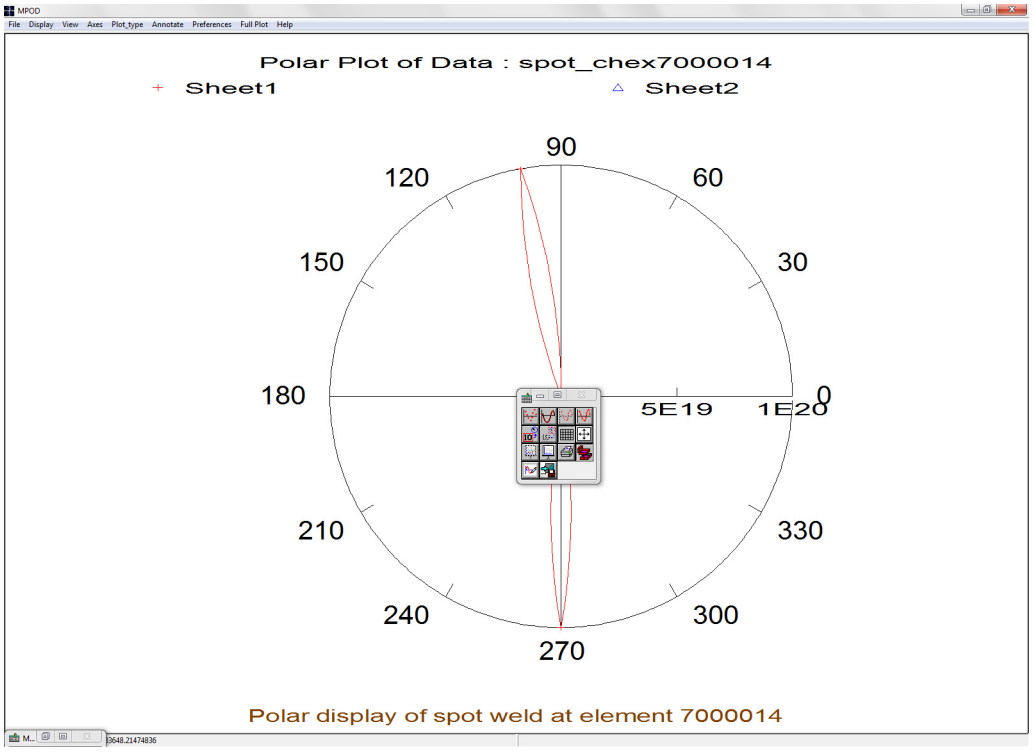


Figure 10-21 Polar Display of Maximum Effective Stress Range

Many of the options available from the top pull-down menus are generic to the MSC Fatigue modules and are described fully in Module Operations (Ch. 18) along with the commands that are applicable in the Command databox. Those specific to this display are described here.

Option	Description
DISPLAY	
Join/Points	Displays the plot as either a continuous line by joining the points together or displays only the data points.
Join Points	Displays both the lines joining the data points and the data points themselves
Lines	Displays the data points as radial lines joining the origin to the data point in question. The length of the line represents the damage, stress etc.
VIEW	
Full Plot or Full R	Displays all the data within the visible window.
Window R	Requests a minimum and maximum R value in the Command databox for display.

Option	Description
Zoom In / Out	Zooms in or out (away from the plot).
AXES	
Log R / Linear R	Converts the Radial Axis to Log Scale / Linear Scale.
Grid / Grid Off	Creates a plot grid of radial and circumferential lines, and turns it off.
Box On / Off	Turns the circle around the plot on or off.
Del RA lines	Hitting the R or A key produces either a circumferential line through the current radial position of the cursor, or a radial line through the current angular position. Selecting Del RA lines deletes these lines.
PLOT TYPE	
Point Skip	Allows the display to be plotted for every nth data point. For example if every other data point is to be plotted use 2.
Hide Set / Show Set	Hides a currently displayed data set / Shows a currently hidden data set.
Shape Off / On	Shows the data points as crosses rather than shapes.
ANNOTATE	
Set Title / Delete Title	Allows for setting or deleting a title from the plot. The title must be input through the Command databox.
Add Text / Delete Text	Allows for adding or removing additional text or titles on the plot. The text is input through the databox and automatically placed at a predefined location on the plot. To delete the text, click on it with the mouse after selecting the Delete Text option. Confirmation of the text will be requested.
Move Text	To place added text or titles use this option. First select the text with the cursor and then use the cursor to place the text in the new location.
MISCELLANEOUS	
P	If the P key is pressed at any time, the current option is terminated and the whole screen is re-drawn.
A	Hitting the A key produces a radial line through the current angular position.
R	Hitting the R key produces a circumferential line through the current radial position of the cursor.

Select New Element

Often, design optimization will be carried out on the element that has the shortest life. However, the lives at other elements may also be of concern. When this option is selected, a new element entry screen is presented with the same select options used on the main input screen such as already shown in Figure 10-15. Having selected a new element, the user will be returned to the Design Optimization Analysis menu.

Select New Job

This option returns the user to the first input screen where the job names are requested. See Figure 10-15. The current job names are presented as defaults.

User Preferences

The preferences that can be set here are the back calculation accuracy and the Miners Damage Sum. See Figure 10-22.

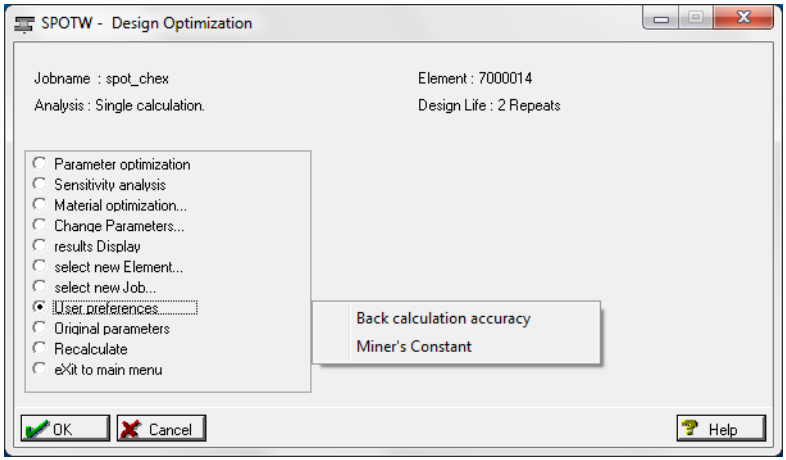


Figure 10-22 User Preferences Submenu

Option	Description
Back Calculation Accuracy	The normal convergence accuracy for the back calculation is 5% (i.e. the iteration will stop once the life is within 5% of the target or design life). Higher accuracy will take longer for the calculation to converge.
Miners Constant	This constant is normally set to a value of 1.0. Some situations, notably variable amplitude spot weld calculations, may call for it to be set to a different value.

Original Parameters

If at any stage in the design optimization, the user wants to recall the original analysis parameters as defined in the global analysis, then this option will do this. This facility is particularly useful for turning off a previously defined back or sensitivity analysis setup. If the user only wants to reset certain parameters, then he should use the Change Parameters main menu pick.

Recalculate

Once the new analysis parameters have been defined, it is necessary to pick this option to start the reanalysis.

Exit to Main Menu

Picking this option causes SPOTW to return to the main menu.

List Global Results

List Global results provides a simple way of viewing a summary of a results file (`.fef` file). When this option is selected, the form illustrated in Figure 10-23 appears. Use the list button to select the `jobname.fef` file required.

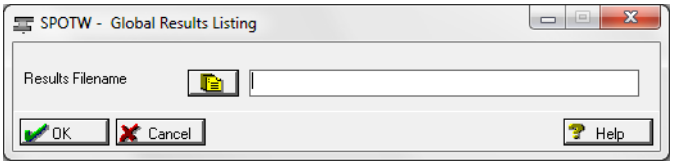


Figure 10-23 Results File Selection Form

When this form is accepted, a summary of fatigue results appears as illustrated in Figure 10-10 and Figure 10-11.

List .spt File

The List .spt File allows the user to list results in a more detailed way for individual spot welds. When this option is selected, the form illustrated in Figure 10-24 appears.

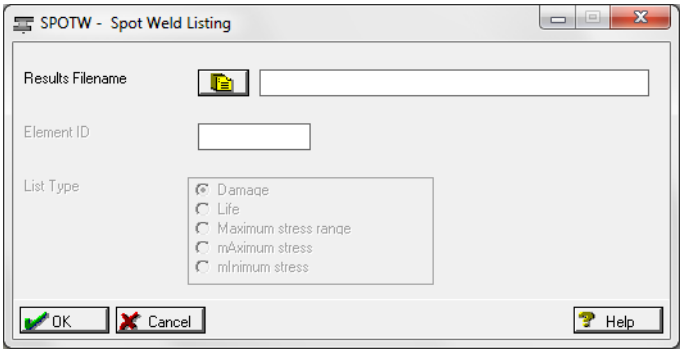


Figure 10-24 Spot Weld Listing Form

Option	Description
Input filename	Input here the name of the `.spt` file to be listed, or select using the List button. The `.spt` file is a flat text file which is described in more detail in Description of Files, 817. When you have done this, select OK and the rest of the form will then be available
Element ID	Enter here the element ID for which you wish to list information. Alternatively accept the default which is WORST. Accepting the default will list data for the element with the worst damage. The List button can be used to show a list of available elements.
List type	Select here the information to be listed.

When this form has been accepted, a results form similar to that shown in Figure 10-25 appears. This form has a listing of the parameter selected, for Sheet 1, Sheet 2, and the Nugget (if calculated), for all the angles calculated.

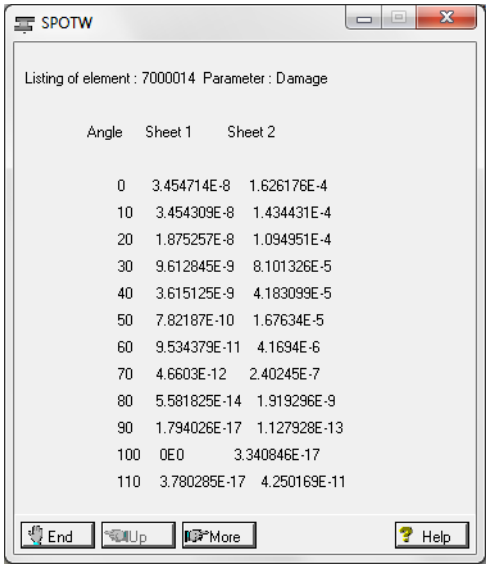


Figure 10-25 Typical .spt File Listing

Results Polar Plot

This option allows the user to create a polar plot of Damage, Life, Maximum Effective Stress Range, Maximum Effective Stress or Minimum effective stress as a function of angle around the spot weld. The angle is defined as the angle around the axis of the spot weld with respect to the MSC Fatigue spot weld x-axis.

When this option is selected a form appears as shown in Figure 10-26.

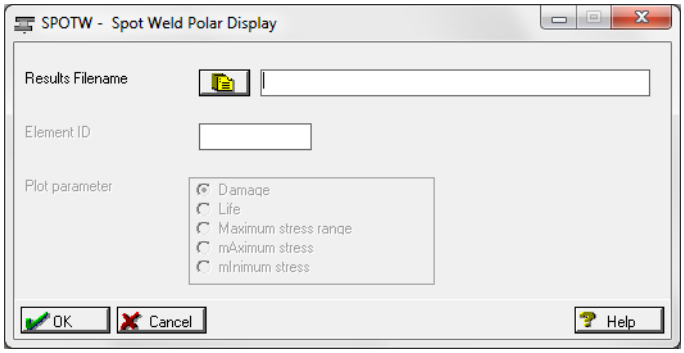


Figure 10-26 Results Polar Display Form

Option	Description
Results Filename	Enter here the name of the flat results file `jobname.spt`. Alternatively, use the List button to select a file from the list.
Element ID	Enter here the element ID for which you wish to create a polar display. Alternatively accept the default which is WORST. Accepting the default will plot data for the element with the worst damage. The List button can be used to show a list of available elements.
Plot parameter	Enter here the parameter to be plotted. Based on the parameter chosen, SPOTW will create a 2-parameter polar display file. The options are: Damage - jobnamenn.pod Life - jobnamenn.pol Maximum Stress Range - jobnamenn.por Maximum Stress - jobnamenn.pma Minimum Stress - jobnamenn.pmi

An example of this type of display is shown in Figure 10-21. Typically 3 datasets are plotted - for Sheet 1, Sheet 2, and the Nugget (unless of course the Nugget has not been calculated). This functionality is duplicated in Design Optimization, 797.

Three Sheet Correction

The current techniques incorporated into the spot weld analyzer have been developed to make calculations for cases where two sheets are spot welded together. In general, joints with three or more sheets spot welded together are more difficult to make efficiently and undesirable from a durability point. They should be

avoided as much as possible in design. Sometimes they may be unavoidable, or alternative designs may be uneconomic.

There is no reason why you should not analyze cases where three sheets are spotwelded together by treating them as two separate spot welds, but the analysis methods currently used are not validated for these cases. This problem is the subject of a current research project. Until a validated solution to this problem has been found, a temporary fix has been provided, called the "3 sheet correction."

If you model the three sheet spot welds as two spot welds, you very often get failures predicted on the middle sheet, which rarely occur in practice. For this reason, a simple fix has been implemented. First, run a spot weld analysis, treating three sheet spot welds as two normal spot welds joined together. Selecting the "3 sheet correction" option, and the `jobname.spt` result file just created will detect three sheet spot welds in the `.spt` file and will create a new `jobname.fef` results file in which failure at the middle sheet (the common node between the two spot welds) is ignored. The worst result for the remaining calculation points is written to both spot weld elements in the resulting `.fef` file.

This option makes post-processing the results easier by eliminating spurious predicted failures at the middle sheet. Note that if the middle sheet really would fail, this won't be predicted either! However this does not appear to happen much in practice.

Description of Files

For descriptions of the Job Information File (`jobname.fin`), the Submit Script (`FatigueSubmit`), and Fatigue Input File (`jobname.fes`), see Description of Files, 369 onwards.

The main differences in terms of file formats between Spot Weld Analysis and other global analyses are:

1. The `jobname.fes` file for a spot weld analysis contains the CBAR element forces and moments rather than stresses or strains.
2. SPOTW does not have a preprocessing stage and therefore does not produce a `jobname.fpp` file.
3. The data in the results file, `jobname.fef`, is specific to spot weld analysis.
4. A more detailed ASCII results file, `jobname.spt`, is also generated by SPOTW.

The Global Results File (jobname.fef)

The results from a Spot Weld analysis are written to an elemental results file, which is a standard Patran results text file. An example of a Spot Weld Results file is shown in Figure 10-27. The format of this file type is described in The Results Files (jobname.fef/fos), 414.

```
SPOT WELD ANALYSIS
  10
5       hours
64    36    1    0    0.5
  65596      0
123456789-1234
   .0000000E+00-.2000000E+02   .1000000E+21   .2000000E+02   .5000000E+21   .2069897E+02
.1000000E+01 .0000000E+00 .0000000E+00 .7077472E+02
  65597      0
   .0000000E+00-.2000000E+02   .1000000E+21   .2000000E+02   .5000000E+21   .2069897E+02
.1000000E+01 .0000000E+00 .0000000E+00 .5313958E+02
  65598      0
   .0000000E+00-.2000000E+02   .1000000E+21   .2000000E+02   .5000000E+21   .2069897E+02
.1000000E+01 .0000000E+00 .0000000E+00 .1923844E+02
  48962      0
   .3677947E-17-.1743439E+02   .1359454E+18   .1713337E+02   .6797271E+18   .1783233E+02
.2000000E+01 .3300000E+03 .7027400E+05 .3686838E+03
```

Figure 10-27 A Typical jobname.fef File From a Spot Weld Analysis

The header of the file contains the following information.

Analysis:	SPOTWELD ANALYSIS
Columns:	10
Equivalent Units:	5 Hours
Number of bins:	64
Number of angles:	36
Calculate Nugget:	1 (Yes)
Units:	0 (N - mm - Nmm - MPa)
Miners Sum:	0.5

Then for each element there are 10 columns of information

1	Damage	**6**	Log of Life (equivalent units)
2	Log of Damage	**7**	Location of failure
3	Life in Repeats	**8**	Angle of failure
4	Log of Life in Repeats	**9**	Node ID of failure
5	Life (equivalent units)	**10**	Maximum force encountered

The Flat Results File (jobname.spt)

The flat results file is an ASCII file which stores information about the damage, maximum stress and minimum stress for every calculation point. It also stores that maximum force through each element. A typical example of a `.spt` file is shown in Figure 10-28. The first line in Figure 10-28 consist of: the element number, the node number (0 indicates this block of data represents the weld nugget), the maximum force, the angle of worst damage (0 may mean no damage) and the worst damage.

The next 36 lines have 4 columns - angle, damage, maximum stress, minimum stress.

Following this block the next two blocks contain the same data but for Sheet 1 and Sheet 2.

```
65596    0          70.7747        0      0E0
         0          0E0       3.51079     2.1259
         10         0E0       4.29129     2.59852
         20         0E0       5.00047     3.02796
         30         0E0       5.66606     3.43099
         40         0E0       6.29919     3.81438
.
.
.
         330        0E0       1.62199     0.98217
         340        0E0       1.92957     1.16842
         350        0E0       2.6706      1.61714
65596    262426                70.7747     0      0E0
         0          0E0       11.9866     7.25832
         10         0E0       12.4859     7.56062
         20         0E0       12.6478     7.65867
.
```

Figure 10-28 Section of jobname.spt File

SPOTW Batch Operation

Batch operation has the following batch keywords:

/INPut	The name of the input job (/INP=myjob).
/OUTput	The name of the output file (/OUT=newname).
/OPTion	This is the option as shown on the main screen (e,l,s,d,x) (/OPT=e).
//NBINS	This is the matrix (bin) size (32, 64, 128) (/SIZ=128).
/NUMEQU	Number of Equivalent Units.
/MINers	This is the Miner's Sum. Valid from 0.5 to 2.0 generally. (/MIN=1.0).
/NANGD	Number of Angles for Calculation.
/NUGGET	Calculate Nugget? (Y or N).
/OVerwrite	Overwrite (yes or no) (/OV=yes).
/*	Output to.... (TT = screen, Filename = to file, None = None).

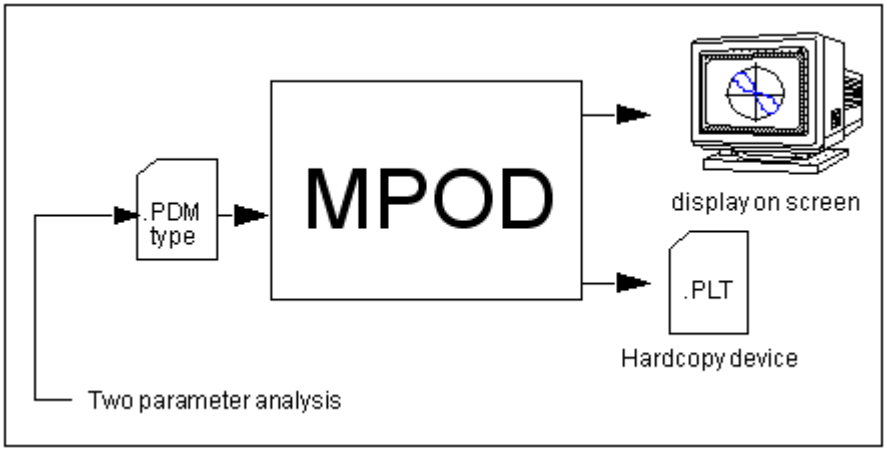
Example:

```
spotw/opt=e/inp=myjob/ov=y/*=tt
```

This batch line would run a spot weld analysis in batch where the first option is called out (Estimate Fatigue Life) and the jobname is myjob. All the defaults are used for the rest of the inputs such as matrix size and Miners sum. The results files will be called `jobname.fef` and `jobname.spt` by default.

Polar Display (MPOD)

The polar display module, MPOD, displays paired [X-Y] data files, normally'theta-R' files with a .pdm extension. Displays may be scaled in various ways. Functions for windowing specific fields and picking off coordinate pairs are also available.



MPOD cross-plots polar plots that are generated by both the multiaxial fatigue analysis modules, MMLF and FEMLF, and the spot weld analyzer, SPOTW.

MPOD Module Operation

The MPOD module can be run in one of the following three ways:

- From the MSC Fatigue menu systems as spawned from the MMLF, FEMLF and SPOTW modules
- In stand alone mode by typing **mpod** at the system prompt
- By incorporating the MPOD batch commands.

Modes one and two are interactive. Once running in interactive mode the MPOD module will prompt with the following screen:

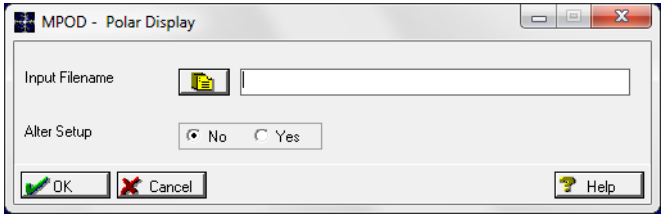


Figure 10-29 Naming the MPOD Input File

MPOD plots the contents of a paired (θ ,R) data file. By default it expects this file to be a standard paired file with an extension of .pdm and so if such a file is to be displayed, it is only necessary to enter its name, the extension may be omitted.

MPOD will expect to find the input data files to be resident in the user's directory. However, other directories can also be accessed if the complete file specification, (path name and file name) is entered. If the Alter setup question it set to Yes, then a second window appears. The purpose of each of its fields is as follows:

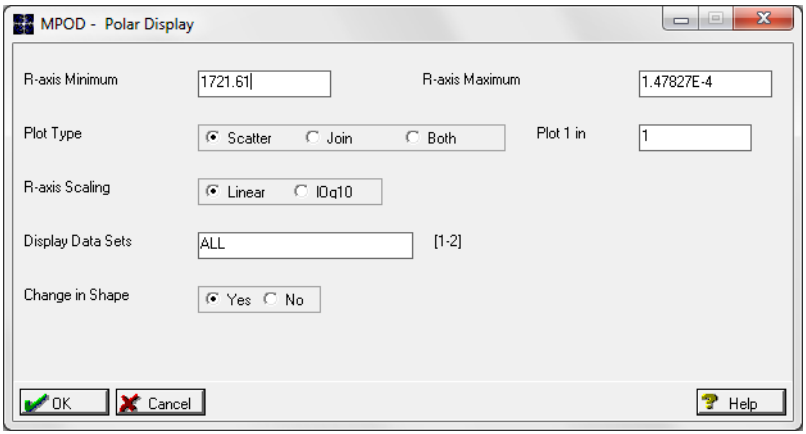


Figure 10-30 Altering the MPOD Setup File

The input fields are as follows:

Option	Description
R-Axis Minimum	For the purpose of scaling the display, the minimum value of R to be included in the plot may be specified in physical units. The minimum value of the R parameter present in the data file is given as the default limit. If a value greater than the minimum value is specified then some data will be excluded from the plot.
R-Axis Maximum	For the purpose of scaling the display, the maximum value of R to be included in the plot may be specified in physical units. The maximum value of the R parameter present in the data file is given as the default limit. If a value smaller than the maximum value is specified then some data will be excluded from the plot.
Plot Type	Having specified the R axis limits, it is now possible to define if the plot is to be generated by displaying the θ-R coordinate pairs discretely (a scatter plot) or if the data are to be plotted by joining up between adjacent values (a joined plot) or both.
Plot 1 in n	Not every point in the file has to be displayed. To display a plot more quickly, enter a number n and only every n'th plot will be displayed. It also allows hard copies to show adequately the pattern of data in a file with many points without taking too long to physically produce. The selection of either scattered or joined plots is arbitrary. However, scattered plots will, by and large, take longer to plot than joined plots. Joined plots are generated some 30% faster than scatter plots and so they are recommended for longer data sets. In either case, cross-plotting data can be time consuming and care must be taken in selecting the data to plot.

Option	Description
Scaling Options (Scale R)	Two scaling options for the R axis are possible. Linear θ - Linear R Linear θ - Log R Linear-Linear scaling is the default. Set the required option with the left mouse button.
Display Data Sets [1-n]	If the file selected has more than 1 data set, then they can be plotted selectively. Enter the number of the data sets you wish to have plotted. The numbers can be separated by commas. For example 1, 5, 18, will display the 1st, 5th, and 18th data sets.
Change in Shapes	Shapes on/off allows the user to have each data set plotted with a different shaped symbol. For example one set of data points can be plotted with small crosses, another with small triangles, another with small squares etc. *Change in Shape=no* plots all the data set with the same shaped symbol. When the above fields have been filled, selecting OK will cause MPOD to plot the data.

Clicking the **OK** button either after altering the setup or without altering the setup will result in a polar plot. When the polar plot is displayed, various manipulation options are available.

Polar Plot Manipulation

The screen is split into 3 regions:

- the prompt line dialog box
- the menu along the top of the form
- the graph area in the centre of the screen.

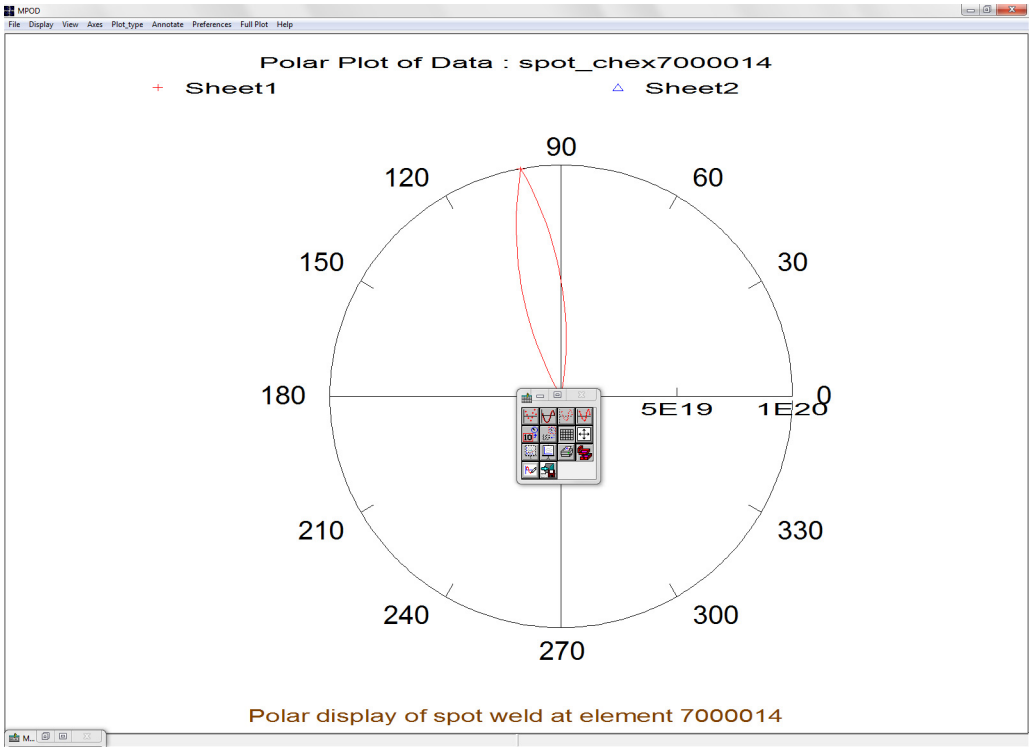


Figure 10-31 The MPOD Graphics Screen

The dialog area is used for the following:

- Supplemental questions following option picks
- Entering prompt mode codes

Use the mouse cursor to select the menu options.

The general operation of the graphics menu system is fully described in Module Operations (Ch. 18), but the options available specific to MPOD are all illustrated in the diagram below.

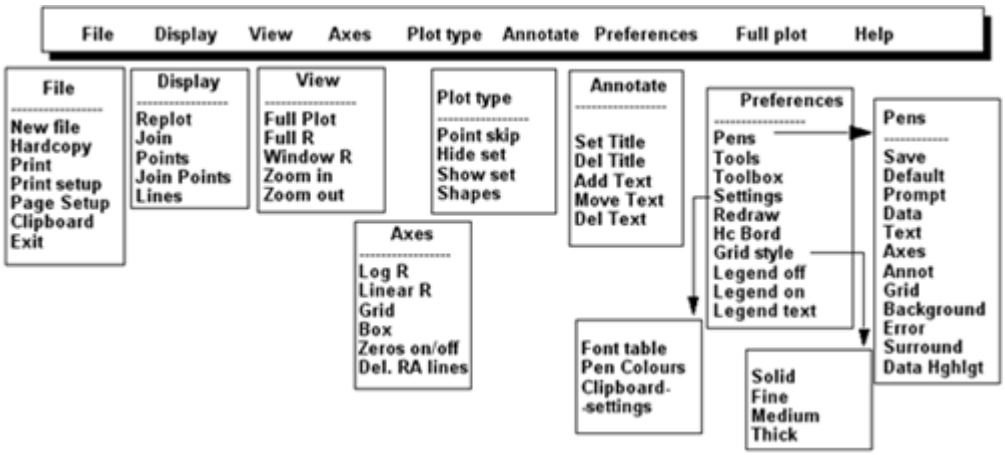


Figure 10-32 The MPOD Graphics Menu System

The menu options all have corresponding prompt mode codes. They can be typed in the dialogue box (click in it to make it active).

The following lists explain the menu options and also give their corresponding prompt codes. Note that the lists do not cover all menu options because many of the standard options are fully explained elsewhere.

Also, note that some options do not appear at all times - for example the Show Set/Hide Set options do not appear if there is only one set.

Option	Command	Meaning
Main Menu options		
Options	OP	Display the available prompt options.
New File	NE	Return to the file input screen
Replot	PL	Replot without change. PL is useful after many changes have been made to the plot with Redraw Off, or if the display has been corrupted in any way.

Option	Command	Meaning
Display View and Axes options		
Join	JO	Plot the data by joining the data values with a line
Points	PO	Plot the data point themselves, scatter plot
Join Points	JP	Plot points and join them
Lines	TO	Plot data by displaying radial lines to the values
XMIN = XX		Alter the scaling on the specified axis
XMAX = XX		The commands may be concatenated with commas
YMIN = YY		For example: YMAX = YY XMAX=500,XMIN=-300,YMIN=-500,YMAX=1000
Window R	WR	Pick two values on the Y-axis and plot all between them
Log R	LR	Replot with a logarithmic R-axis
Linear R	NR	Replot with a linear R-axis
Full X/Y	FX/FY	Redraw the plot to include the full X and Y axes
Zoom in	ZI	Zoom in 2 times on the centre of the plot
Zoom out	ZO	Zoom out 2 times on the centre of the plot
grid#	GR	Superimpose the plot on a grid
Box	TI	Graduate the axes with ticks
Zeros On/Off	ZON/ZOFF	Displays zero lines for the X and Y axes
Plot Type and Annotate Options **Note:** The title string is stored within the extra details area of the data file for which the string is defined against the keyword GRPTITLE. This means that when the data are re-plotted once MPOD has exited, the title will remain intact.		
Point Skip	SK	Plot every n'th point and skip the rest
Set Title	STI	Sets a title which will appear below the plot
Del Title	DTI	Deletes the title below the plot
Add Text	AT	Add text at the position of the mouse cursor
Move Text	MT	Move existing text
Del Text	DT	Delete a block of text
Preferences\ Legend On/Off **Note:** These are mostly standard options. However, if between 1 and 6 sets are being plotted then Legend On/Off and Legend Text appears on the menu. It enables each data set to be given an editable label which be toggled on and off the plot.		

Option	Command	Meaning
Keyboard Options **Note:** These are normally available when the cursor is selected by typing CU at the prompt when the cursor is over the plot, e.g. to draw a circle on the R-axis, press the R key, and a circle will be drawn through the cursor.		
Display Value	V	Pick off a value. Equivalent to pressing the LEFT HAND mouse button
Define Window	W	Define a window. The window is rectangular and its left corner will be the position of the cursor when W is pressed, and the right corner the position of the cursor when the right mouse button is pressed
Save Pair Data	S	Save coordinate pair in the extra details area
Replot	P	Replots entire screen
Hardcopy	H	Hardcopy plot of entire screen including menu and dialogue box. The plot title will take the form PLn.PLT. Use PLTCON to convert it to formats such as HPGL (.HPS)
Draw Circle	R	Draws circle at R
Draw Angle	A	Draws an angle at θ
Quit	Q	Quit from cursor mode
Exit or return options	EX and #EX	Exit **MPOD**

MPOD Extra Details Keywords

User-Defined Titles

MPOD allows for the inclusion of a user-defined main title and subtitle within a particular plot. The main title appears at the top of the plot and the subtitle at the bottom. Both titles correspond to text strings which are stored in the extra details area of the data file being plotted. The keywords corresponding to each title are detailed below.

Extra Data Keyword	Text String Value
$TITLE	<Main Title Text String - top of plot>
GRPTITLE	<Sub-Title Test String - bottom of plot >
<KEYWORD>	<User defined keyword that corresponds to a data point picked off by the user.>

MPOD Batch Operation

MPOD runs in all the standard batch modes.

When run in batch mode MPOD will, by default, produce a hardcopy with the plot file taking on the next available name of the form `plN.plt` from the plot index.

If the /OPT=PL command is received in batch, MPOD will return to interactive mode and plot the data to the screen.

A list of MPOD's batch keywords:

/INP	Input file name
/RMIN	Minimum on R-axis to display
/RMAX	Maximum on R-axis to display
/TYPE	Plot Type (Scatter/Join/Both)
/ONEIN	Plot 1 in value
/RSCAL	R-axis scaling (Linear/Log10)
/SETS	Data sets to display
/SHA	Whether to change shape between data sets
/OPT	Graphical plot options

Spot Weld Analysis Theory

A software system has been developed in the MSC Fatigue environment which permits fatigue life predictions to be made for automotive spot-welds joining two steel sheets. The method uses bar element forces to calculate the "structural stresses" in each spot-weld nugget and the adjacent sheets using the methodology described by Rupp, Störzel and Grubisic. The system described here supports the use of dynamic stresses derived from road load data, using either a quasi-static or transient approach to stress history determination. The method is geometry-independent and suitable for application to large models (because it does not require local mesh refinement). The system provides a convenient way for users of Patran, MSC.Nastran, and MSC Fatigue to predict the location and life of fatigue sensitive spot-welds. See (Ref. 32.) through (Ref. 38.) in References (Ch. 17) for more detail.

Resistance spot welds are very commonly used in the automotive industry in the fabrication of all manner of components and structures, and the durability of such structures is very often controlled by the strength of the spot welds. The cost of tooling up for a single weld spot as part of an automated manufacturing process may be around $30,000, and this can more than double if a weld spot has to be added during production to remedy a problem. These costs may be minimized if the life of spot welds can be predicted at an early stage in the design process, though the reduction in development time and improvement in quality is likely to be more significant.

Smith and Cooper addressed the problem of life prediction of shear spot welds using a fracture mechanics approach. They noted that a spot weld could be "....considered to be a circular solid surrounded by a deep circumferential crack, which when loaded in a combination of Mode I and Mode II, would grow a branch crack in the direction of maximum local Mode I". They showed that good predictions of life could be made on the basis of calculated crack growth rates, and used their calculations to generate some simple design curves. The method was based on detailed finite element modeling of simple spot-welded lap-joints loaded in shear. This method would need further development in order to cover all the possible weld configurations

used in automotive structures and to deal with the variable amplitude out-of-phase loadings to which they are subject. The results of this might be a simple design code for spot-welds along the lines of BS 7608 with families of load-life curves for different classes of spot-weld. In practical FE models of automotive structures there is no scope for such detailed modeling of individual spot welds.

In fact, load is a rather poor parameter for correlating the fatigue strength of spotwelds under different loading conditions. Radaj and Sheppard note that durability of spot welds of a variety of configurations and loadings can be better understood through numerical analysis of the local stresses at the weld spot edge on the inside of the plate - the structural stresses around the weld.

Rupp, Störzel, and Grubisic describe the calculation of these structural stresses, and also carry out fatigue life predictions based on maximum and minimum stresses and a load spectrum. The software described here is closely based on the work of Rupp et al, but combines their method for structural stress calculation with the methods of stress scaling and superposition and access to transient FE results normally used in MSC Fatigue.

The method requires spot welds to be modeled as stiff beam elements in MSC.Nastran. The forces transmitted through these beam elements are used to calculate the structural (nominal) stresses in the weld nugget and the adjoining sheet metal at intervals around the perimeter of the nugget. These stresses can then be used to make fatigue life predictions on the spot weld using a S-N (total life) method.

The software system consists of some modified MSC Fatigue modules and a spot-weld fatigue analyzer called SPOTW. The system currently only supports fatigue calculations on spot welds joining two sheets. In the FE model, the spot-welds should be represented by stiff beam elements joining the mid-planes of the two sheets of shell elements, and perpendicular to both. The length of the spot weld and the sheet separation should therefore be half the sum of the sheet thicknesses. There is no need for any refinement of the mesh around the spot-welds. The only requirement for the shell elements used to model the sheets is that they transmit the correct loads to the bar elements. In fact it seems that best results are achieved when the dimensions of the shell elements are quite large - more than twice the diameter of the weld nuggets.

A typical spot-weld is illustrated in Figure 10-33. The shaded part is the spot weld “nugget”. In a finite element analysis, the weld is modeled in MSC.Nastran as a stiff beam element joining the mid-plane of two sheets. The length of the beam element will be $0.5(s_1+s_2)$ where s_1 and s_2 are the thicknesses of sheets 1 and 2 respectively. Point 3 is on the axis of the weld nugget and at the interface of the 2 sheets, i.e. $0.5s_1$ from Point 1. All forces and moments are taken to be in the MSC Fatigue beam element co-ordinate system illustrated. This is taken to be a Cartesian system with the Z axis going from Point 1 to Point 2. This is different both from the arrangement used by Rupp et al. and that used in MSC.Nastran, but is a little more simple.

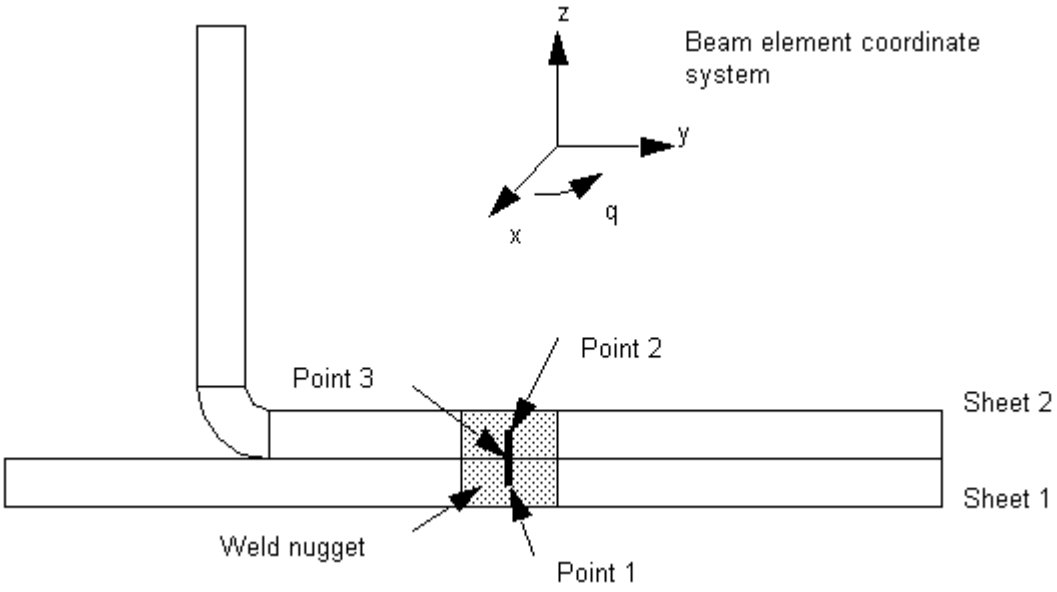


Figure 10-33 Schematic of Typical Spot Weld

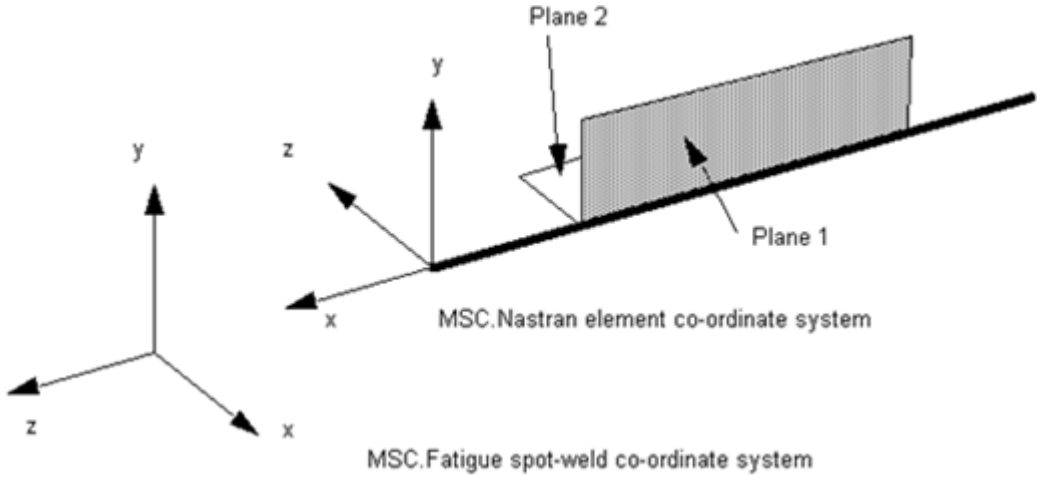


Figure 10-34 Relationship of MSC Fatigue Spot Weld Coordinate System to MSC.Nastran

The translator extracts forces and moments $F_{x,y,z}$ and $M_{x,y,z}$ in the MSC Fatigue coordinate system, and in the conventional right-handed sense, from the results in the database, for each of the three specified points. These forces and moments (except M_z) are used to calculate nominal stresses (structural stresses) on the inner surface of Sheet 1 and Sheet 2, and in the weld nugget at the interface of the two sheets, at intervals around the circumference of the spot weld (θ =0 degrees to 360 degrees by increments of 10 degrees). The forces and moments at points 1 and 2 are those applied by the spot welds on the sheets, and the forces and moments at point 3 will be those applied by the upper section (between point 3 and point 2) on the lower section (between point 1 and point 3).

Stress Calculations

The stresses are calculated as follows:

Point 1:

The equivalent stress on the inner surface of the sheet as a function of angle θ around the circumference of the spot weld is:

$$\sigma_{v1} = -\sigma_{max}(F_{x1})\cos\theta - \sigma_{max}(F_{y1})\sin\theta + \sigma(F_{z1}) + \sigma_{max}(M_{x1})\sin\theta - \sigma_{max}(M_{y1})\cos\theta \quad (10\text{-}1)$$

$$\sigma_{max}(F_{x1}) = \frac{F_{x1}}{\pi ds_1} \quad (10\text{-}2)$$

$$\sigma_{max}(F_{y1}) = \frac{F_{y1}}{\pi ds_1} \quad (10\text{-}3)$$

$$\sigma(F_{z1}) = K_1\left(1.744\frac{F_{z1}}{s_1^2}\right) \quad for \quad F_{z1} > 0 \tag{10-4}$$

$$\sigma(F_{z1}) = 0 \quad forF_{z1} \le 0 \tag{10-5}$$

so that only the tensile component of the axial force in the nugget contributes to damage, and:

$$\sigma_{max}(M_{x1}) = K_1\left(1.872\frac{M_{x1}}{ds_1^2}\right) \tag{10-6}$$

$$\sigma_{max}(M_{y1}) = K_1\left(1.872\frac{M_{y1}}{ds_1^2}\right) \tag{10-7}$$

Note that K_1 = 0.6 sqrt(s_1) and d is the diameter of the weld nugget, dimensions in mm. Forces are in N and moments in Nmm.

Point 2:

The equivalent stress on the inner surface of the sheet as a function of angle θ around the circumference of the spot weld is:

$$\sigma_{v2} = -\sigma_{max}(F_{x2})\cos\theta - \sigma_{max}(F_{y2})\sin\theta - \sigma(F_{z2}) - \sigma_{max}(M_{x2})\sin\theta + \sigma_{max}(M_{y2})\cos\theta \tag{10-8}$$

$$\sigma_{max}(F_{x2}) = \frac{F_{x2}}{\pi ds_2} \tag{10-9}$$

$$\sigma_{max}(F_{y2}) = \frac{F_{y2}}{\pi ds_2} \tag{10-10}$$

$$\sigma(F_{z2}) = K_2\left(1.744\frac{F_{z2}}{s_2^2}\right) \quad forF_{z2} < 0 \tag{10-11}$$

$$\sigma(F_{z2}) = 0 \quad forF_{z2} \ge 0 \tag{10-12}$$

so that only the tensile component of the axial force in the nugget contributes to damage, and:

$$\sigma_{max}(M_{x2}) = K_2\left(1.872\frac{M_{x2}}{ds_2^2}\right) \tag{10-13}$$

$$\sigma_{max}(M_{y2}) = K_2\left(1.872\frac{M_{y2}}{ds_2^2}\right) \tag{10-14}$$

Note that K_2 = 0.6 sqrt(s_2) and d is the diameter of the weld nugget, dimensions in mm. Forces are in N and moments in Nmm.

Point 3:

From the forces calculated for point 3, nominal stresses are calculated at intervals around the circumference of the weld nugget, say at 10 degree intervals. The method of Rupp then suggests that the direct stress be calculated on multiple planes at 10 degree intervals, i.e. use a stress-based critical plane method. This would mean 36 x 18 = 648 calculations for each weld nugget. This is very computationally intensive, especially in view of the fact that spot welds do not usually fail by cracking through the nugget. For this reason, two faster approaches are considered: to ignore the possibility of nugget failure altogether, and to use the absolute maximum principal stress as the damage parameter, as used in MSC Fatigue (only 36 calculations). This is calculated as follows:

$$\tau = \tau_{max}(F_{x3})\sin^2\theta - \tau_{max}(F_{y3})\cos^2\theta \tag{10-15}$$

$$\sigma = \sigma(F_{z3}) - \sigma_{max}(M_{x3})\sin\theta - \sigma_{max}(M_{y3})\cos\theta \tag{10-16}$$

$$\tau_{max}(F_{x3}) = \frac{16F_{x3}}{3\pi d^2} \tag{10-17}$$

$$\tau_{max}(F_{y3}) = \frac{16F_{y3}}{3\pi d^2} \tag{10-18}$$

$$\sigma(F_{z3}) = \frac{4F_{z3}}{\pi d^2} \quad when F_{z3} > 0 \tag{10-19}$$

$$\sigma(F_{z3}) = 0 \quad when \ F_{z3} \le 0 \tag{10-20}$$

$$\sigma_{max}(M_{x3}) = \frac{32M_{x3}}{\pi d^3} \tag{10-21}$$

From the shear and direct stresses on the nugget, the in-plane principal stresses can be calculated from:

$$\sigma_{1,3} = \frac{\sigma}{2} \pm \sqrt{\left(\frac{\sigma}{2}\right)^2 + \tau^2} \tag{10-22}$$

The principal stress with the greatest magnitude is taken as the damage parameter.

Material Properties

The system requires an S-N curve for each metal sheet and for the weld nugget at load ratio R=0, plus a mean stress sensitivity factor and a standard error parameter. The formulation of the S-N curve is as follows:

$$\Delta S = SRI1(N_f)^{b_1} \tag{10-23}$$

for $N_f<N_{c1}$ the transition life. For $N_f>N_{c1}$ a second slope b_2 is used. It is possible to correct each cycle with amplitude S and mean stress S_m to calculate an equivalent stress amplitude S_0 at R=0:

$$S_0 = \frac{S + MS_m}{M + 1} \tag{10-24}$$

Rupp described generic S-N curves for sheet steel and weld nuggets. There is quite a wide-scatter band, which is partly a reflection of the fact that this data represents spot-welds in a variety of steels, including mild and high strength. Better predictions may be possible if S-N data specific to the materials being used is available.

Damage Calculation

Actual damage calculations use typical MSC Fatigue techniques for the S-N methods as described fully in Fatigue Theory (Ch. 16).

Specific to spot weld analysis, damage calculations are carried out at 10 degree intervals around the spot weld in both sheets and in the weld nugget. There are therefore 108 fatigue calculations per spot-weld. At each calculation point, the effective stress history is calculated either directly from the force and moment results from a transient FE analysis, or by scaling and superimposing the results of a number of static load cases according to the quasi-static method. The stress history is then rainflow cycle counted to form a range-mean histogram. Rainflow cycles are converted to equivalent stress amplitude for R=0, then damage is calculated and summed using Miner's rule. The results are written to two files: a Patran `jobname.els` file containing summary results for postprocessing in Patran, and a more detailed file for postprocessing by SPOTW.

The method for life prediction of spot-welds described here is somewhat computationally intensive. Computation time is roughly proportional to the number of data points in the load histories. Substantial reductions in computation time can therefore be achieved by judicious filtering of the loading inputs.

Stress Factors

The spot weld analyzer works by using the cross-sectional forces and moments in the beam elements to calculate structural stresses in the spot weld nugget, and the two sheets being joined. These structural stresses are then used to make the fatigue calculation. More information on this is available in Spot Weld Analysis Theory, 828 in this chapter. The structural stresses are modified by empirical factors that take into account size and loading type effects. These factors are different for material types 1-99 (ferrous metals) and types 100-199 (aluminum alloys). The stress factors are applied to the stresses due to the x and y forces on the beam element, the z forces and the x and y moments. The factor takes the general form:

$$SF = F \times diam^{e1} \times thickness^{e2} \tag{10-25}$$

where F is a factor and e1 and e2 are the diameter and sheet thickness exponents. For steels these factors are by default assumed to be as follows:

Component	Factor F	Diameter Exponent e1	Thickness Exponent e2
Fx,y	SFFXY = 1.0	DEFXY = 0.0	TEFXY = 0.0
Mx,y	SFMXY = 0.6	DEMXY = 0.0	TEMXY = 0.5
Fz	SFFZ = 0.6	DEFZ = 0.0	TEFZ = 0.5

For aluminium alloys, the factors are as follows:

Component	Factor F	Diameter Exponent e1	Thickness Exponent e2
Fx,y	SFFXY = 0.4	DEFXY = 0.5	TEFXY = -0.25
Mx,y	SFMXY = 0.4	DEMXY = 0.5	TEMXY = -0.25
Fz	SFFZ = 1.0	DEFZ = 0.0	TEFZ = 1.0

Expert users may wish to specify their own values. This can be achieved by setting an environment keyword using the environment manipulation program MENM. See Modifying the MSC Fatigue Environment (MENM), 1463. The keyword to be set is "SPOTWPAR" and the value it must take is the 9 values detailed in the table above, in the correct order, separated by commas:

SPOTWPAR=SFFXY,DEFXY,TEFXY,SFMXY,DEMXY,TEMXY,SFFZ,DEFZ,TEFZ

Seam Weld Analysis

Introduction

There are basically two approaches to modeling seam welds in MSC Fatigue. The first approach is the use the multi-discipline solver dtfat. The dtfat solver allows a number of different weld types to be used in a single analysis, such as fillet welds and overlap welds. The MSC Fatigue GUI has been updated to automatically identify the weld regions associated with these weld types. The second approach is the use the Classic solver MFATFE, which is a second-generation multi-discipline fatigue solver. MFATFE only recognizes fillet welds.

Both approaches are available for predicting the fatigue life of seam welded thin sheet structures. It compliments BS7608 for thicker sections. The method, using the S-N approach, has been established and tested at Volvo Car and analytical predictions are in good agreement with test data. One advantage of this solution is speed. The solver only needs to consider the weld area which is extracted automatically and the whole model need not be analyzed.

In setting up a proper seam weld analysis it is important to first have a sound understanding of the operation of MSC Fatigue within the Patran environment. It is suggested that you first read and understand Using MSC Fatigue - The Classic Modules (Ch. 3). Job setup for a seam weld analysis is very similar to job setup for other fatigue analysis types with only a few differences as pointed out in this chapter.

General Procedure

The seam weld fatigue analysis method implemented in this engine is based upon that originally proposed by Volvo Car Corporation and Chalmers University of Technology, and developed in cooperation with nCode. The method was developed particularly for automotive components welded from thin steel sheets (typically 1-3 mm), and most of the industrial applications of this method have been to this type of structure. However, the structural stress-based approach it uses is of quite general applicability and there is no reason in principal why it should not be applied to structures with thicker sheets.

The method is basically quite similar to the standard S-N method, but with some special features to deal with welds.

Note the following:

1. The method was originally developed for thin sheet welded structures, modelled and analyzed in NASTRAN using predominantly CQUAD4 elements. (The original work is described in "Fatigue Life Prediction of MAG-Welded Thin-Sheet Structures", by Mikael Fermér, Magnus Andréasson and Björn Frodin. SAE Technical Paper 982311).
2. As originally described by Fermér et al, the method used grid point forces from the FE solution to calculate structural stresses at the weld toe, as this approach was considered to provide a good level of mesh insensitivity compared to standard stress recovery methods, which rely on extrapolation from gauss points. Subsequently, it was realized that a good level of mesh-insensitivity could also be achieved by using the CUBIC stress recovery option in NASTRAN, and implementation of the method in software relied on CUBIC stress recovery. (The CUBIC method derives stresses at each node in each element from the displacements and rotations of the nodes.) Although use of stresses from other FE codes is quite possible, significant differences in the results may be caused by the differences in stress recovery methods. If it is necessary to use stresses from other FE codes (e.g. ABAQUS) better results may be obtained through the use of quadratic elements.
3. From version 6.0, optionally grid point forces may be used to determine the stresses to be used for fatigue analysis. The details of the method have been refined from that described in SAE 982311 and are described in detail later. To use grid point forces, they must be present in the FE results file, and EntityDataType = ForceMoment must be set on the Analysis Group properties.
4. From version 6.0, optionally nodal displacements and rotations may be used to determine the stresses in the elements around the weld. To use this option, nodal forces and displacements must be available in the FE results file, and EntityDataType = Displacement must be set on the Analysis Group properties. Details of this method are described later.
5. There are some specific guidelines for meshing the welds, which will be discussed shortly, but essentially the structure must be meshed predominantly with CQUAD4 elements (or equivalent) representing the mid-planes of the metal sheets, and the weld bead represented by a single or double row of shells. The mesh around the weld should be regular, with elements of around 5 mm in size if possible (in practise the size of the elements tends to be dictated by the dimensions of the weld and the need for good-shaped elements). Triangular elements in the vicinity of the weld should be avoided where possible.
6. If Stresses (preferably CUBIC) or Displacements are being used, unaveraged node-at-element stresses are recovered for the weld toes and weld roots, based on the elements adjacent to the weld, and used to make the fatigue calculations. Where weld throat calculations are made, the centroid stress from the weld throat element is used.
7. From version 7.0, the option to calculate stresses at the mid-point of the element edges at the weld toes and roots is extended to FE stresses and stresses derived from displacements. These methods either recover or derive unaveraged (node-on-element) stresses.
8. The method determines the contribution of bending to the total stress, and from this determines whether the weld is essentially "stiff" or "flexible". A different SN curve may be required for these cases, and interpolation is made between the curves based on the degree of bending.

Grouping and Weld Types

Different types of welds have to be analyzed differently, depending upon their geometry, the process by which they are created and the failure modes to be considered. In MSC Fatigue, welds must therefore be collected into different groups for analysis according to which category they fall into. In practice, this means that welds have to be grouped into separate Material Regions, each of which has the property SeamWeldType set to an appropriate value. The supported weld types (SeamWeldType) are as follows.

Fillet

A fillet weld is a Metal Inert Gas (MIG) or Metal Active Gas (MAG) weld joining two sheets at an angle. The weld illustrated is a single sided weld. The most likely failure (fatigue crack) locations at the weld toe or weld root are marked. These are the locations where the fatigue damage will be evaluated. Note that weld throat failure is unlikely unless a weld is of poor quality or the weld bead under-dimensioned.

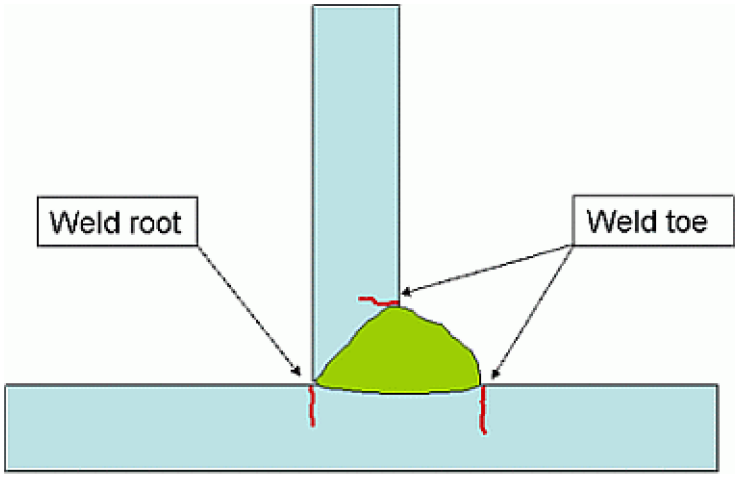


Figure 10-35 Fillet weld cross-section showing likely failure locations

Overlap

An overlap weld is similar to a fillet weld, except that the plates being joined are parallel. Note the marked possible failure locations – these are the locations where the analysis engine will predict the fatigue damage.

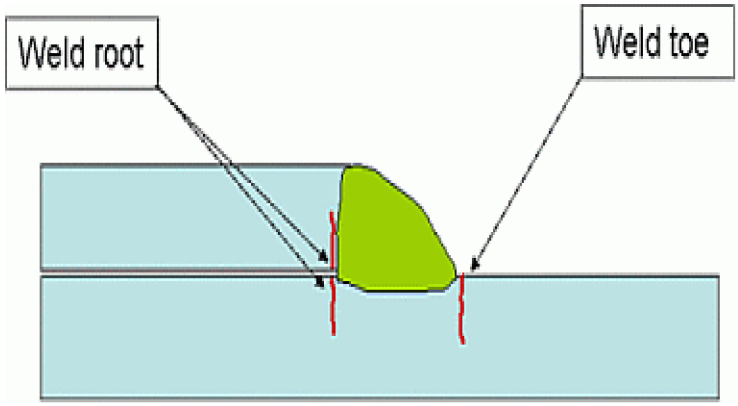


Figure 10-36 Overlap weld cross-section showing likely failure locations

Laser Overlap

In the case of the Laser Overlap weld, failures initiating and propagating through the sheets are classed as weld root failures. The other possible failure mode is cracking through the weld itself. This is classified as a weld throat failure. The possible failure locations are illustrated in Figure 10-37. Note that the sheet separation has been exaggerated for the sake of clarity.

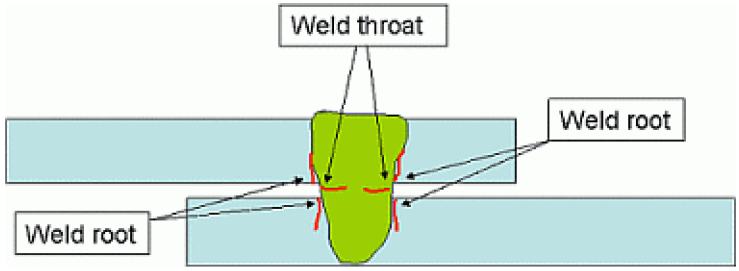


Figure 10-37 Laser overlap cross-section showing likely failure modes (sheet separation exaggerated)

Laser Edge Overlap

Laser edge overlap welds are handled in a very similar way to overlap welds, except that because of the small dimensions of the weld bead, failure through the weld throat becomes a possibility.

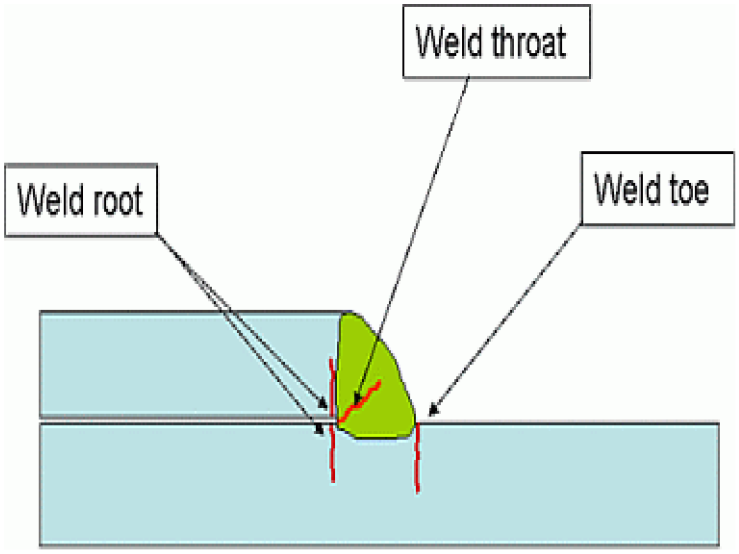


Figure 10-38 Laser edge overlap weld cross-section and possible failure locations

Generic

The "Generic" option is provided so that some users may reproduce a conservative option previously available in MFATFE. In this option, damage is calculated for the top and bottom surfaces of all shell elements attached to the weld elements. All these locations are treated as weld toes. For example:

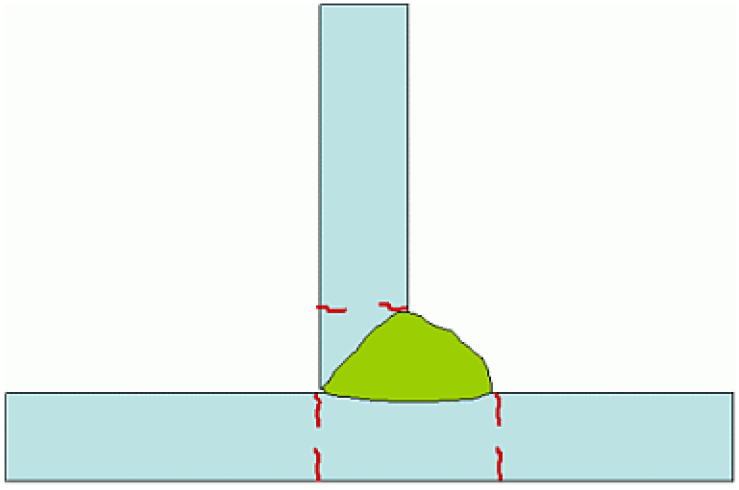


Figure 10-39 Generic option "failure" locations. All positions are treated as weld toes.

Modelling Guidelines and Calculation Points

In order to obtain reasonably accurate fatigue analysis results, it is necessary to adhere to certain FE modelling guidelines. There are some general guidelines, and some that are specific to the different weld types.

General Guidelines

The structures to be analyzed should be meshed predominantly with 4-noded quadrilateral elements (CQUAD4 or equivalent) representing the mid-planes of the metal sheets. The weld bead should be represented by a single or double row of shells. The mesh around the weld should be regular, with elements of around 5 mm in size, and triangular elements in the vicinity of the weld should be avoided.

The elements representing the weld bead are used to identify the weld, and these elements should be placed in a unique group for each weld or weld type. In addition, the weld element normals must be aligned so that the normals point outwards, that is towards the welder (except for Laser Overlap welds). The orientation of the sheet element normals is not important, except that within a single sheet they should be aligned in compliance with normal good practice.

Averaged nodal stresses are recovered for the weld toes and weld roots, based on the elements adjacent to the weld and used to make the fatigue calculations. Where weld throat calculations are made, the centroid stress from the weld bead element is used (averaged from the nodes or element edges).

Note:	All seam weld results for the root, toe and throat locations identified must have a non-zero thickness. All elements identified as part of results locations must be connected directly to the weld elements defined by the user. If you ensure that all weld elements and all shell elements connected directly to these have none zero thickness, you will not see a "zero thickness" error message.

In the past it has been recommended to “close” the ends of welds as illustrated in (a) of Figure 10-40. However, when using FE stresses, a simplified approach such as that illustrated in (b) appears to give quite good results whilst being significantly easier to build. When grid point forces are used, it may be beneficial to close the weld ends.

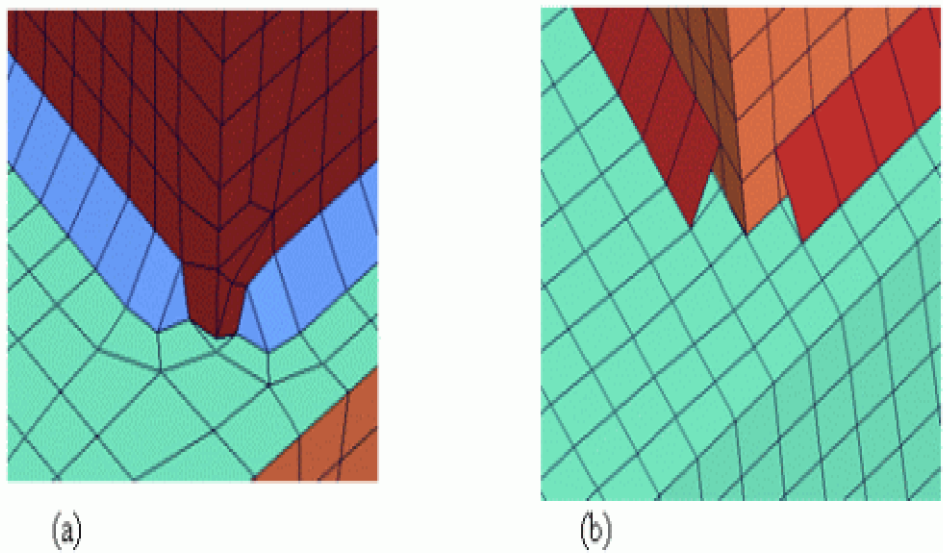


Figure 10-40 Simplified modelling of weld ends. Approach (b) is adequate.

Fillet

Fillet welds may be modelled either by a row of inclined shell elements, as illustrated in Cross-section of fillet weld modelled with single inclined element, or by two rows, as illustrated in Figure 10-42.

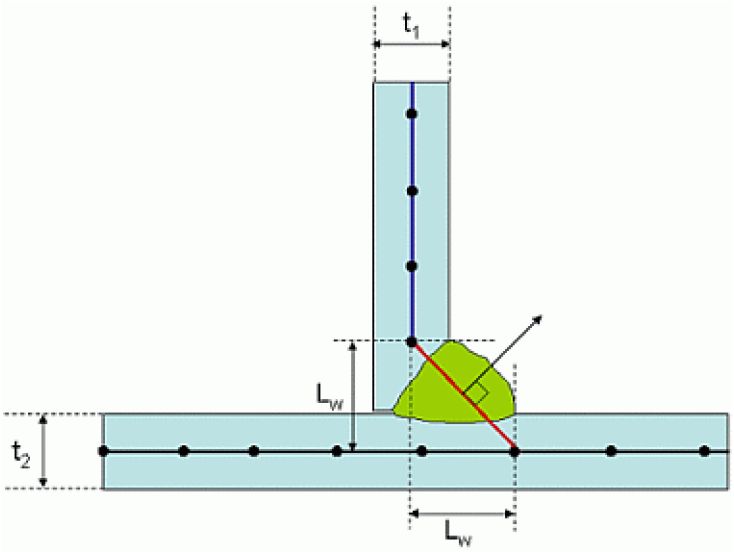


Figure 10-41 Cross-section of fillet weld modelled with single inclined element

1. The weld elements are in a different property set.
2. The weld element normal points "towards the welder".
3. The nodes of the weld elements are in line with the weld toe.
4. The length L_w should be determined by the actual dimensions of the weld toe. A typical recommendation might be $L_w = t_1 + t_2$
5. The thickness of the weld element should represent the weld throat thickness, typically $L_w/(\sqrt{2})$.

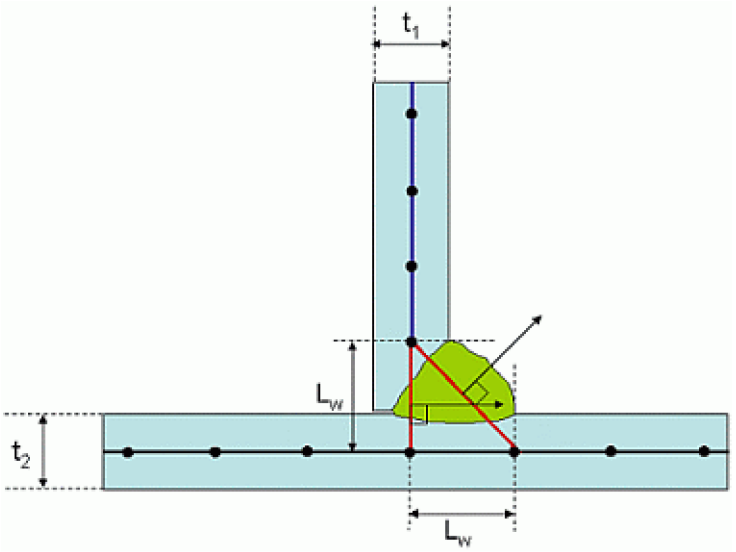


Figure 10-42 Cross-section of fillet weld modelled with two rows of elements

1. Make sure that the weld element normals are aligned as shown. This orientation is required for the correct identification of weld toe and weld root.
2. A weld element thickness of $L_w/(2\sqrt{2})$ or in other words about $0.35(t_1 + t_2)$ is recommended, although the stiffness of this joint formulation will not be very sensitive to the weld element thickness.

Overlap and Laser Edge Overlap Welds

Overlap and Laser Edge Overlap welds are modelled in the same way, except that the dimensions of the weld bead will be smaller for a Laser Edge Overlap weld. The principles are the same as for fillet weld modelling; a row of nodes should be created along the line of the weld toe and 1 or 2 rows of elements used to model the weld, with the thickness of these weld elements representing the weld throat.

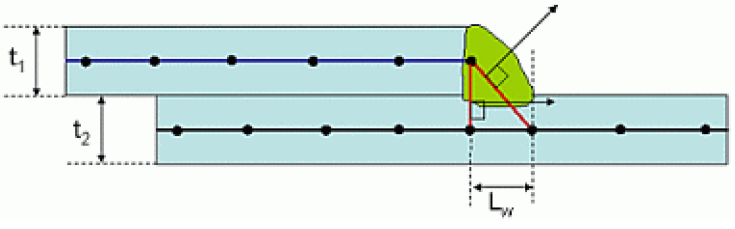


Figure 10-43 Overlap weld modelling – two element approach

For Overlap welds, typical leg length $L_w = t_1 + t_2$ and the weld element thickness should then be around $0.27(t_1+t_2)$.

The approach can be simplified, replacing the double row of elements by a single row, as illustrated in Figure 10-44. This method is more suitable for Laser Edge Overlap welds, because it allows weld throat fatigue calculations to be made; reasonable weld throat stresses require a single row of elements to be used.

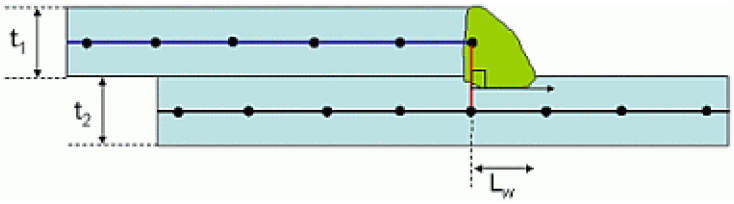


Figure 10-44 Simplified modelling approach for overlap or laser edge overlap welds

In this case, for Overlap welds, the weld element thickness should be set to 2x the thickness of the thinner of the two sheets, but in any case not less than 3 mm.

For Laser Edge Overlap welds, the thickness of the weld element should correspond to the weld throat, which will be about 0.7x the upper sheet thickness.

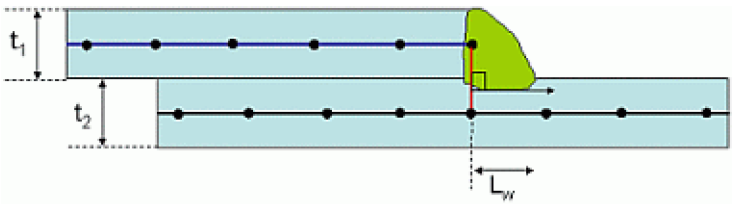


Figure 10-45 Alternative method for overlap and laser edge overlap welds

Laser Overlap Welds

For Laser Overlap welds, the thickness of the weld element should represent the weld width. The weld width should be 90% of the thickness of the thinner sheet, but in any case not less than 1mm.

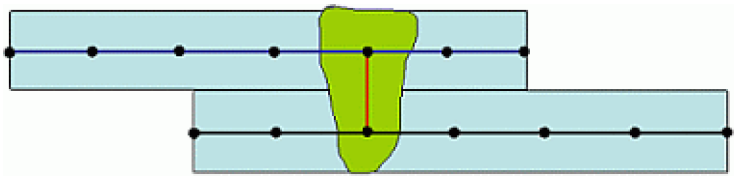


Figure 10-46 Laser overlap weld

Calculation Points and Stress Recovery

This section describes how the correct calculation points and stresses are identified for each weld type.

In general, the process is as follows, based upon the weld elements and weld element normals:

- Identify weld toe and root elements.
- Identify the nodes on those elements that are shared with the weld elements.

Currently, stresses are being used directly:

- Recover stresses in element coordinates, for both surfaces of each element, with the "weld top" surface first.
- For weld toe and root elements, if WeldResultLocation = Node, the stresses are averaged nodal stresses in local (element) coordinates.
- For weld throat elements, the stresses are averaged from at the centroid in local (element) coordinates.

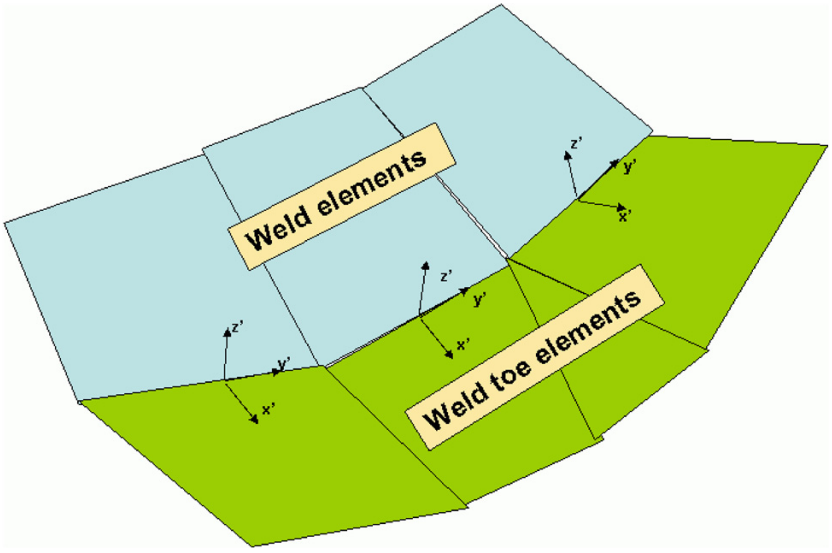


Figure 10-47 Local coordinate systems for weld toe elements

We will now look at the specific rules for each element type.

Fillet Welds

For fillet welds, the weld toe and root elements are outlined in Figure 10-48: The weld throat elements are not used for analysis.

The weld toe elements are the elements in front of the weld that share nodes with the weld throat elements. The elements at the end of the weld line are optional. Whether or not they are used is controlled by the property WeldEndElements (Exclude or Include). Currently, stresses are being recovered and WeldEndElements is set to Include, these elements will be included in the analysis, i.e. a fatigue calculation will be made for this element. We recommend excluding these elements from the calculation (set WeldEndElements = Exclude).

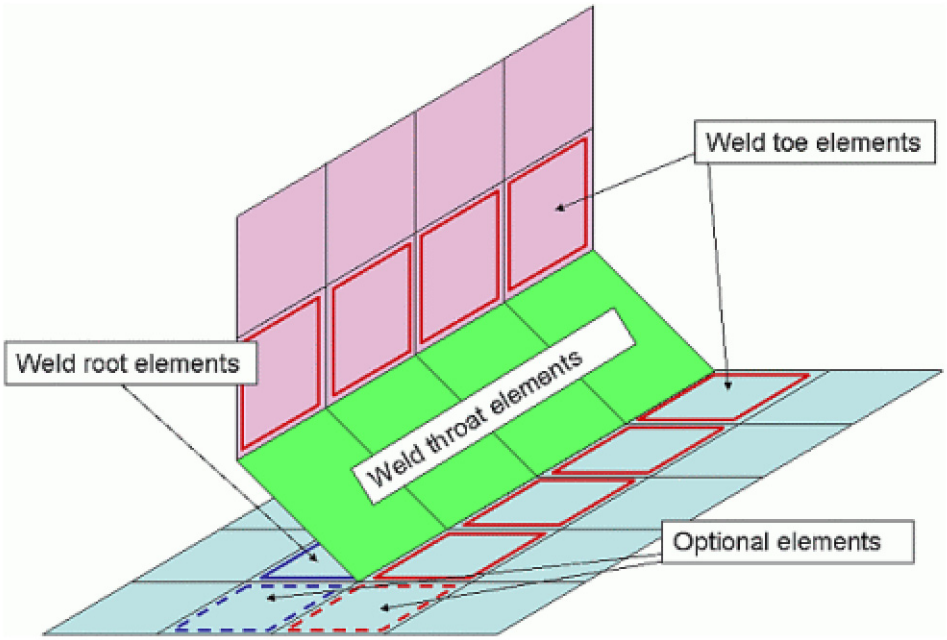


Figure 10-48 Weld toe and root elements for analysis for fillet welds (single element method)

The locations and "weld top" surfaces for calculation are indicated in Figure 10-49 and Figure 10-50 below.

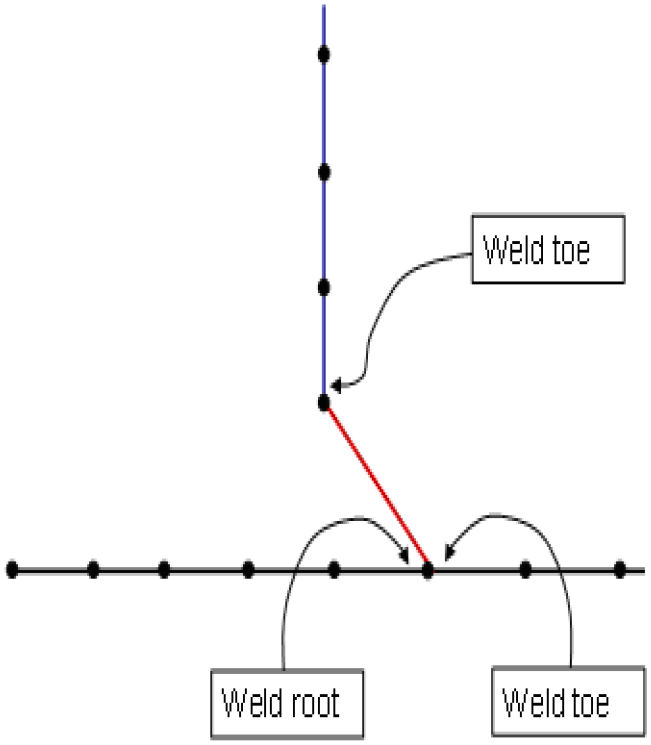


Figure 10-49 Weld toe and root calculation points and surfaces (weld top) for fillet weld – single element method

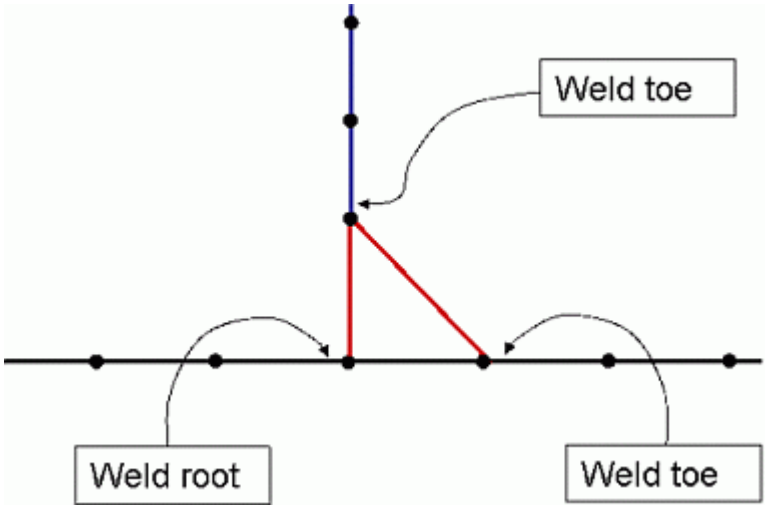


Figure 10-50 Weld toe and root calculation points and surfaces (weld top) for fillet weld – two element method

Overlap Welds

For overlap welds, the weld toe and root elements are outlined in Figure 10-51. The weld throat elements are not used for analysis.

Again, the elements at the end of the weld line are optional. Whether or not they are used is controlled by the property WeldEndElements (Exclude or Include). This is currently set to Include. We recommend excluding these elements.

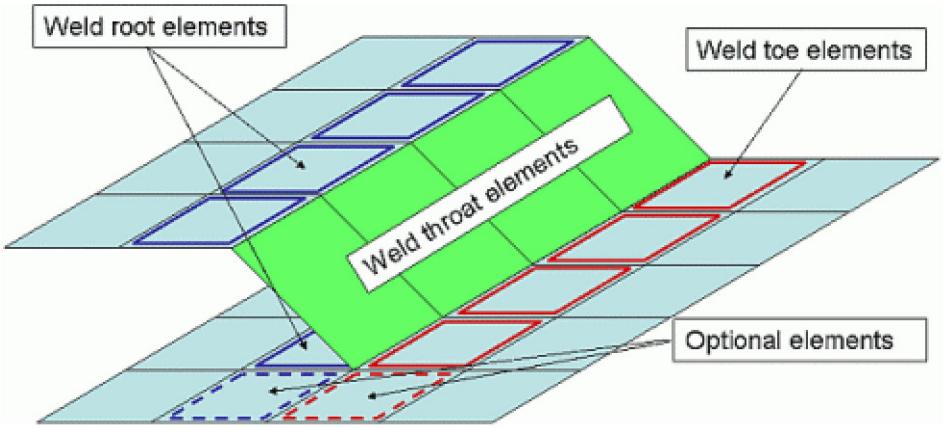


Figure 10-51 Weld toe and root elements for analysis for overlap welds (single element method)

The locations and "weld top" surfaces for calculation are indicated in Weld toe and root calculation points and surfaces (weld top) for overlap weld – two element methodWeld toe and root calculation points and surfaces (weld top) for overlap weld – single inclined element method and Weld toe and root calculation points and surfaces (weld top) for overlap weld – single vertical element method below:

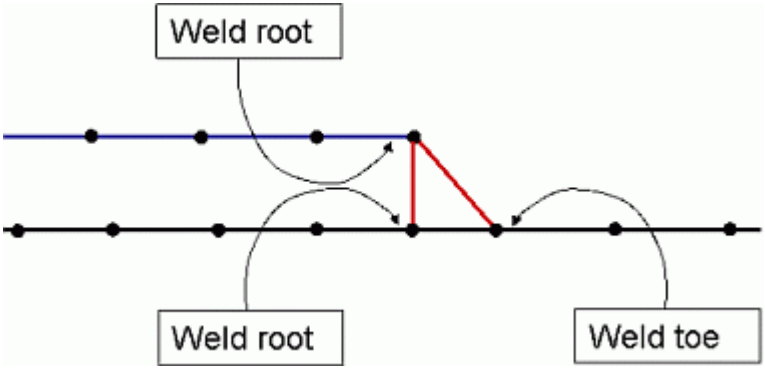


Figure 10-52 Weld toe and root calculation points and surfaces (weld top) for overlap weld – two element method

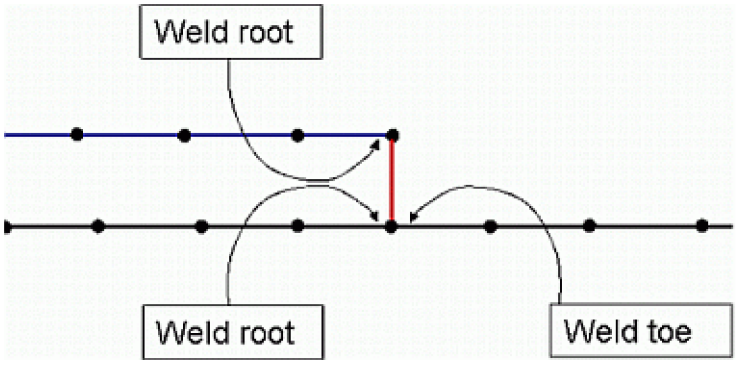


Figure 10-53 Weld toe and root calculation points and surfaces (weld top) for overlap weld – single inclined element method

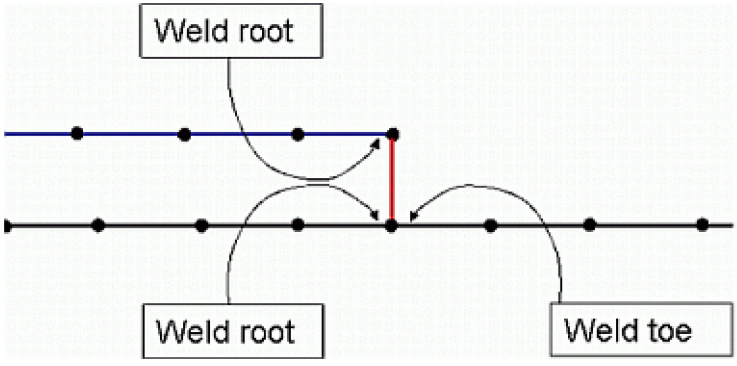


Figure 10-54 Weld toe and root calculation points and surfaces (weld top) for overlap weld – single vertical element method

Laser Overlap Welds

For Laser Overlap welds, the weld root and throat elements are outlined in Figure 10-55. Because a crack might initiate from either side of the weld throat, both sides of the weld throat element must be considered.

Again, the elements at the end of the weld line are optional. Whether or not they are used is controlled by the property WeldEndElements (Exclude or Include). This is currently set to Include. We recommend excluding these elements.

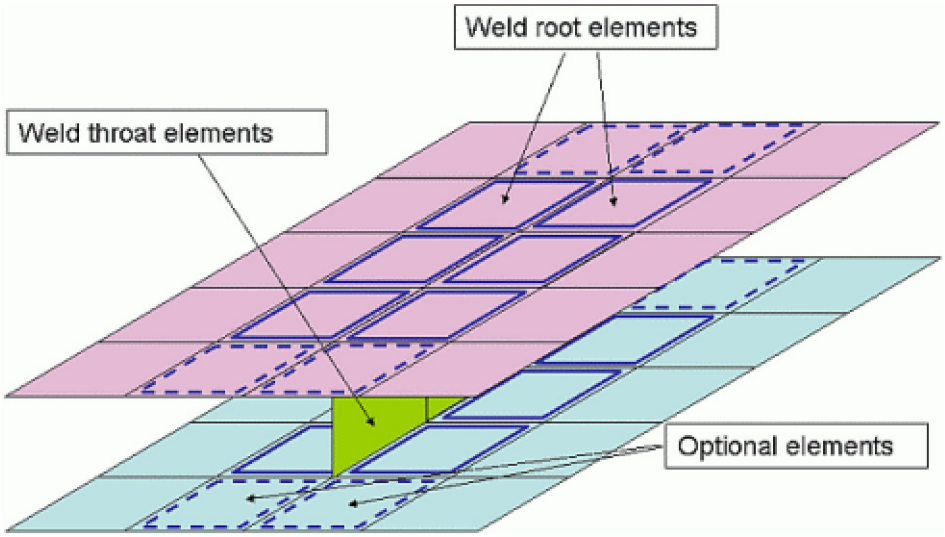


Figure 10-55 Weld root and throat elements for analysis for laser overlap welds

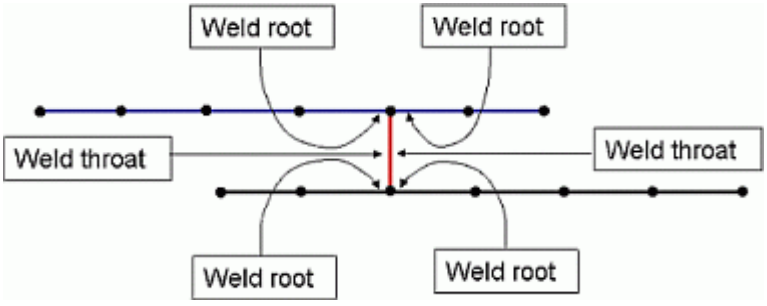


Figure 10-56 Weld root and throat calculation points and surfaces for laser overlap weld – single vertical element method

Laser Edge Overlap

Laser Edge Overlap welds are handled in the same way as Overlap welds, except that the weld throat may also be calculated. In this case, it is important to use a single row of elements to represent the weld. A double row of elements will not generate realistic weld throat stresses.

The calculation points and surfaces for a Laser Edge Overlap weld are illustrated in Figure 10-57.

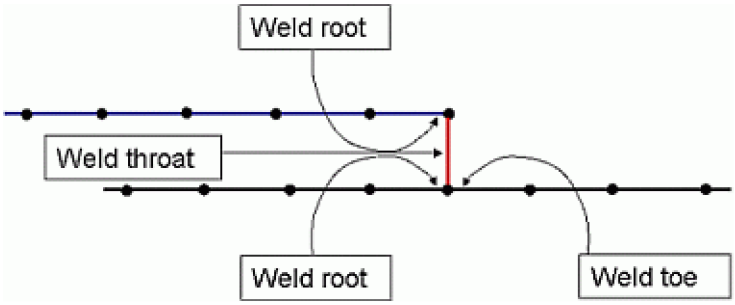


Figure 10-57 Weld toe, root and throat calculation points and surfaces for laser edge overlap weld – single vertical element method

Generic

If the WeldType is set to Generic, all surfaces of elements attached to the weld elements are used, and treated as weld toes. See the example in Example of calculation points and surfaces for generic method:

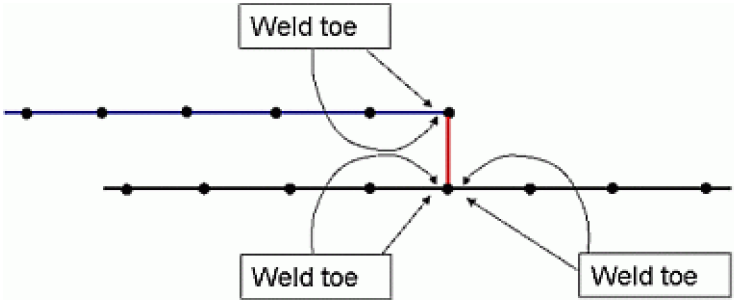
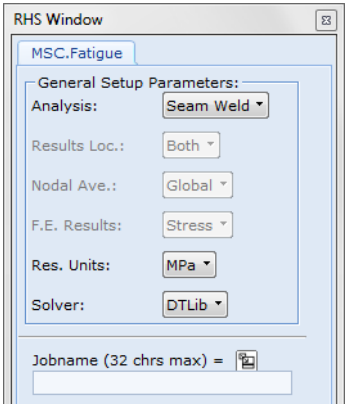


Figure 10-58 Example of calculation points and surfaces for generic method

For details on the method of solution, please see the Fatigue Theory Guide.

Seam Weld Job Setup (dtfat)

The setup starts in the MSC Fatigue forms within Patran. You must set the analysis type to Seam_Weld. There are three basic inputs just as for any other analysis types, those being Solution Parameters, Material and Loading Information. Results post-processing and Job Control are also described in this section.


RHS Window
MSC.Fatigue
General Setup Parameters:
Analysis:
Seam Weld
Results Loc.:
Both
Nodal Ave.:
Global
F.E. Results:
Stress
Res. Units:
MPa
Solver:
DTLib
Jobname (32 chrs max) =

Solution Parameters (Seam Weld)

The following table lists DTLib solution parameters that are associated with Seam Weld analysis as shown on Solution Parameters form below.

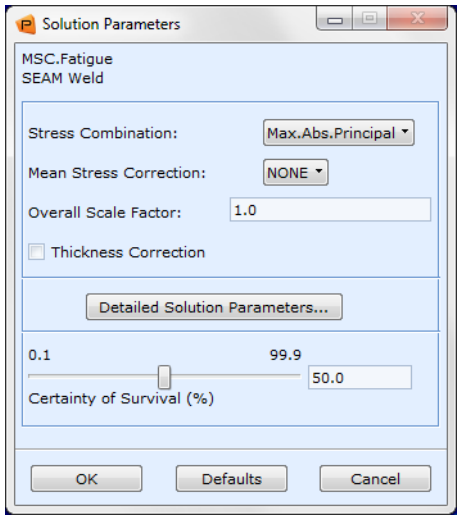


Table 10-2 Solution Parameters

Parameter	Options	Description	Default
Stress Combination Method			**Max Abs Principal**
	Max Abs Principal	The absolute Maximum Principal stress is defined as the principal stress with the largest magnitude	
	Critical Plane	The Critical Plane option is available only for stress flagged as 2-D. The normal stress is calculated and rainflow counted on multiple planes.The critical plane is the plane with the most predicted fatigue damage. The planes on which the normal stress is determined have normals that lie in the plane of the physical surface, i.e., in the x-y plane of the 2-D stress results coordinate system. The orientation of each plane is defined by the angle ϕ made with the local x-axis.	
Mean Stress Correction			**None**
	None	No mean stress correction applied.	
	FKM	The FKM method as implemented is based on the method described in the FKM Guideline “Analytical Strength Assessment of Components in Mechanical Engineering”, Tr. E. Haibach. 2003.	

Table 10-2 Solution Parameters

Parameter	Options	Description	Default
Overall Scale Factor	0 < **Value** <= 1e10	The scale factor to apply to the combined data prior to performing damage calculation.	**1.0**
Thickness Correction	**True, False**	A thickness (size effect correction) may be applied, based upon the thickness t of each element. This works as follows: If t <= Tref (the reference thickness or threshold), there is no effect. If t > Tref the fatigue strength is reduced at all lifetimes by a factor (Tref/t)^n , where n is the thickness exponent. (We use the factor to increase the stress.)	**False**
Certainty of Survival	0 <= **Value** <= 100	Percentage certainty of survival	**50**

Multiaxial assessment is carried out in the same way as for the Standard S-N analysis. The following options are available:

None - No multiaxial assessment is carried out.

Simple Biaxiality - A simple biaxiality assessment is carried out.

Standard - A more sophisticated biaxiality assessment is carried out.

Material Properties and Interpolation

The material dataset required for a seam weld calculation consists of a parent dataset of type "nCode SN

Seam Weld" containing basic material properties, and either one or two children containing S-N curves.

If two curves are present, one must represent "flexible" or bending loading conditions (r = 1) and the other "stiff" or membrane loading conditions (r = 0).

The generic (parent) data is as follows:

Parameter Name	Description
Material type	Numeric code defining the type of material
UTS	Ultimate tensile strength
E	Modulus of elasticity

Parameter Name	Description
M1	Mean stress sensitivity factors for FKM mean stress correction
M2	
M3	
M4	
TRef	Reference thickness/threshold for sheet thickness correction
n	Sheet thickness correction exponent
RR	R-ratio (ratio of minimum to maximum load) of the tests used to define the S-N curve
rth	Threshold value of the bending (r) ratio, used in interpolation between “stiff” and “flexible” curves. If left blank, a value of 0.5 will be assumed.
Comments	
References	

In addition, each child dataset (there must be either 1 or 2 children) has the following additional properties.

Parameter name	Description
BendingRatio	Bending or flex ratio of the curve. If two curves are present, BendingRatios must be 1 and 0 to identify “flex” and “stiff” curves.
SRI1	Stress range intercept
b1	First fatigue strength exponent
Nc1	Transition life
b2	Second fatigue strength exponent
SE	Standard error of $\log_{10}(N)$. This is used to calculate the life adjusted to a certain probability of failure/survival.
NFc	Numerical fatigue cutoff life. Beyond this life, damage will be assumed to be zero.

Weld Groups

MSC Fatigue provides methods to automatically identify Toe and Root elements, based on Weld Type. The user must first create a group which includes all the weld bead elements. This should be done for all seam

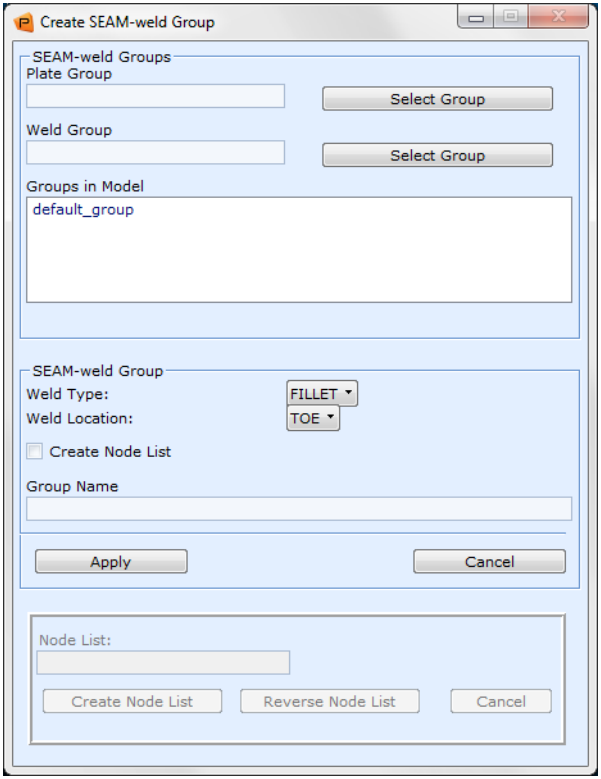
welds in the model. If desired, all weld beads for similar weld types can be loaded to a single group, for example for fillet welds. Note that only weld bead elements should be included in this group.

The group creation process is started by clicking in the **Group** cell on the **Material Information** form spreadsheet. This automatically brings up the **Create Seam Weld Group** form. On this form, specify the group containing elements attached to the weld bead(s). This is known as the **Plate Group**. Next the weld bead group is selected. This is known as the **Weld Group**.

Next, the Weld Type is selected from the pull-down menu on the **Create Seam Weld Group** form. A name should be entered in the **Group Name** text box on the **Group** form. This name will serve as the basis for TOE, ROOT, and other groups created during the automation process.

To launch the automation process, click the **Apply** button on the **Group** form, and wait for the process to complete. When the process is complete, the Material Information form with take focus and will be populated with **Group**, Material Name, weld **Type**, weld **Location**, and other information.

If another weld group is to be created, increase the **Number of Groups** on the **Material Information** form by 1, click in the first available row in the spreadsheet, and follow the process outlined above for each addition weld group.

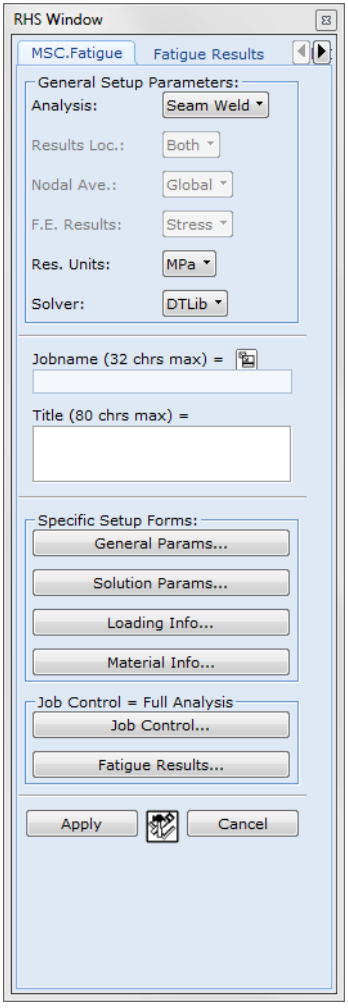


dtfat Batch Operation

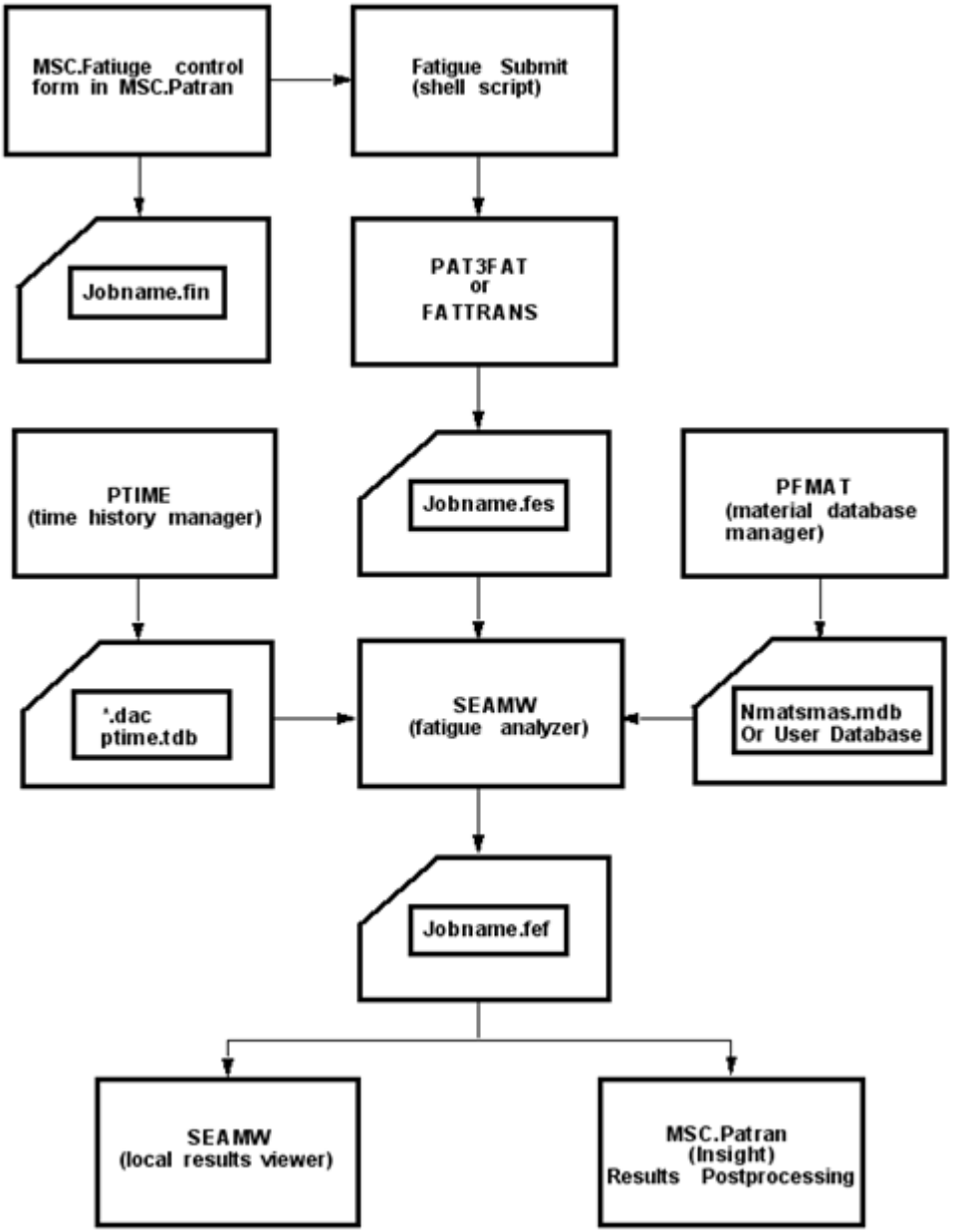
The setup starts in the MSC Fatigue forms within Patran. You must set the analysis type to Seam_Weld. There are three basic inputs just as for any other analysis types, those being Solution Parameters, Material and Loading Information. Results post-processing and Job Control are also described in this section.

Seam Weld Job Setup (MFATFE)

The setup starts in the MSC Fatigue forms within Patran. You must set the analysis type to Seam_Weld. There are three basic inputs just as for any other analysis types, those being Solution Parameters, Material and Loading Information. Results post-processing and Job Control are also described in this section.



Seam Weld Analysis Submittal Schematic



Basic Information

All programs in the MSC Fatigue system may be executed by typing the names of the program at the system prompt or by the Tools menu if used. These programs may ask questions which are not normally presented to you since they are executed as batch jobs when called from MSC Fatigue Pre & Post or Patran. The programs normally used in a Seam weld analysis are listed below.

1. Data Preparation

PFMAT	Materials Database Manager
PTIME	Time History Database Manager and ASCII Time History File Convertor
PVXMUL	A Peak-Valley Extraction Program for Reducing Lengthy Time Histories
MFD	A Multi-file Display Program

2. Global Multi-Seam weld Analysis

PAT3FAT	Model Database (Patran) to Fatigue Input Translator - file (jobname.fes)
FATTRANS	New Model Database (Patran) to Fatigue Input Translator - file (jobname.fes)
SEAMW	Fatigue Analyzer (and rainflow cycle counting)

3. Results Post-processing and Design Optimization

SEAMW	Results Listings
SEAMW	Design Optimization Tools

4. General Utilities

SEAMW	FES File ASCII/Binary Convertor - used for manual editing
MCONFIL	Binary to Binary File Convertor

Analysis Route

The actual programs needed to complete a global multi-node or element total life analysis are:

FatigueSubmit	Shell script (necessary for submittal from MSC Fatigue Pre & Post or Patran)
PAT3FAT	Translator (creates the fatigue input file `jobname.fes`)
FATTRANS	New translator (creates the fatigue input file `jobname.fes`)
SEAMW	Fatigue Analyzer (life prediction)

The programs and options must be used in this order. The results may be reviewed using the results listing options in SEAMW (for detailed understandings) or by inspecting the ASCII results file (`jobname.fef`) using a text editor or by inspecting marker plots in Patran/Insight.

Necessary files

When a global multi-element fatigue analysis is set up using MSC Fatigue Pre & Post menus or Patran, the following files are necessary to run the analysis.

`jobname.fin`	This file contains all the analysis parameters that are defined in the main and subordinate MSC Fatigue forms, (e.g., loading time history data file names). A full description of this file is contained in Job Control, 864.
Database	The groups of nodes and elements for which the fatigue life is to be calculated are contained in the database. For a seam weld analysis, it is necessary to have carried out a finite element analysis in MSC.Nastran with the seam weld defined by a group of CQUAD4 element joining two sheets of thin shells. The seam weld element CQUAD4 elements should be modeled relatively stiff but without causing matrix ill-conditioning problems. Guidelines for modeling the weld elements are provided in the previous discussion. The element stress results from which the nodal stresses are derived must be in the database including group information comprising the seam weld elements and the adjacent elements connecting to the peripheral nodes of the seam weld elements.
Additional Files	Other files that are necessary to complete a successful fatigue analysis are the time history files (`ptime.adb, ptime.tdb` and `*.dac or *.pvx`), and the materials database (`nmatsmas.mdb`) which is generally held in a central location and it is not necessary for this file to be located in the user's local directory. However, users may prefer to define a company database that can be stored at a convenient location (see User Database).

The fin file is ASCII and may be edited using a standard text file editor. Although this method of defining the MSC Fatigue job parameters is not as automated as the MSC Fatigue menus in Patran, it does offer a simple and rapid method of changing a few parameters without the encumbrance of a menu structure, for the experienced user.

File that may be created during an analysis run are summarized below:

jobname.fin	Job parameter file (ASCII).
jobname.fes	MSC Fatigue Input file (Binary).
jobname.asc	ASCII version of the jobname.fes file (via the utilities in SEAMW solver).
jobname.msg/log	MSC Fatigue message and log files (ASCII).
jobname.sta	Job status file (ASCII).
jobname.fef	Global multi-node/element results file (ASCII).
jobname.abo	Job abort file (ASCII).
***.dac, *.cyh**	Loading time history/rainflow matrix files (Binary).
***.ent**	Analysis Node list (used for specially selected entities).
***.tem**	Plotting format data (ASCII PCL file).

***_tmpl**	Results template files (ASCII).
***.adb/.tdb/.mdb**	Time history and Materials database description files (ASCII/Binary/Binary).

After the translator has been run (described in The Translator (PAT3FAT or FATTRANS), 294) and the fatigue input file (`jobname.fes`) has been created, the Seam Weld Fatigue Analyzer, SEAMW, is run. There is no separated pre-processing stage in Seam Weld Analysis. When run in interactive mode, this program asks for a number of input parameters which are passed in through the `jobname.fes` file when run from the MSC Fatigue menus within Patran. A full description of file content is provided in Description of Files, 369.

Solution Parameters

The only parameter that can be set on this form is the design criterion (% Certainty of Survival). The nature of the calculation, including mean stress correction, is predetermined. The default design criterionis 50% and is based on the scatter of the S-N curve. For example, to be 96% certain that the life will be achieved, set the slider bar at 96. This value is used to modify the S-N curve according to the standard error scatter parameter (SE), specified in the materials database. The design criterion parameter will be meaningless if the value of SE is 0. This is used to imply “not known” or undefined in which case only 50% should be used.A Design Criterion value of 50 leaves the S-N curve unmodified.

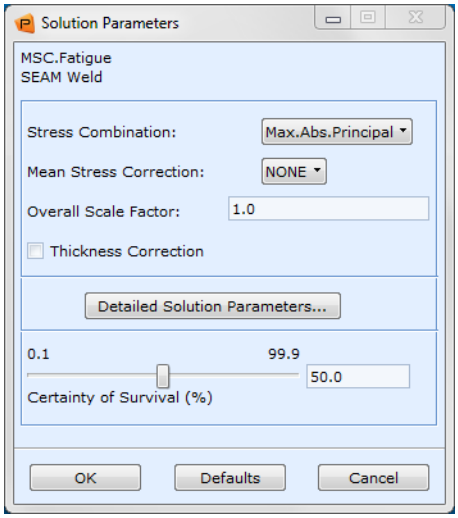


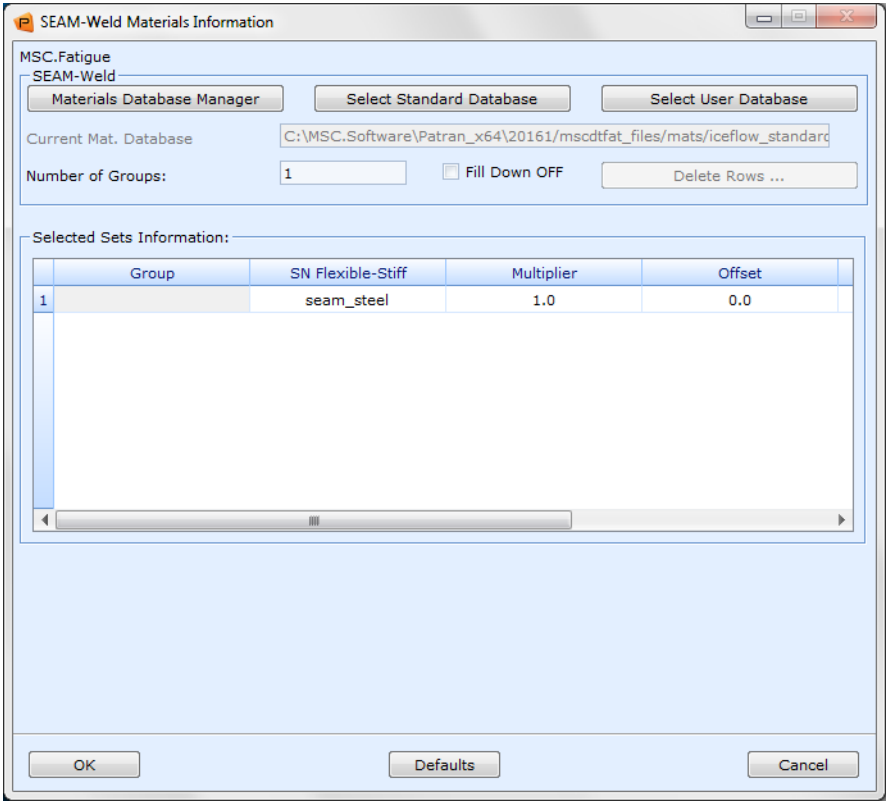
Figure 10-59 SEAM Weld Solution Parameters Form

Material Information

The Material Information form is similar to that for other analysis types and operates in the same fashion as other analysis types. The real major differences are the incorporation of a secondary GUI to create a nodal group that are shared by elements in both the weld and the parent sheet shell groups and the specification of two S-N curve for each analysis group. The first S-N curve (SN-Flexible) applies to joints where the analytical

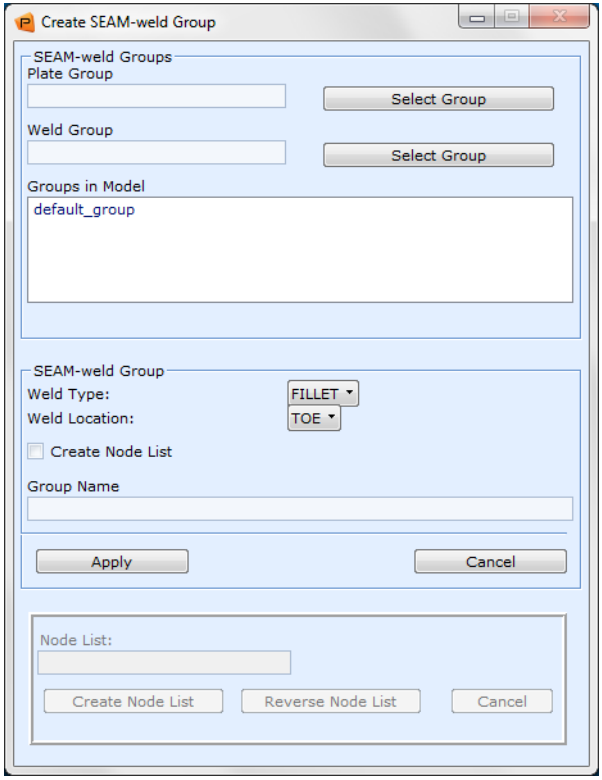
stress is predominantly due to bending and the second S-N applies to joints where the analytical stress is predominantly axial (SN-Stiff).

The cells of the spreadsheet are filled by first selecting one (making it active) with the cursor. A listbox, databox or a separate GUI will appear below the spreadsheet from which you make your selection. The data selected will then occupy the selected cell and the adjacent cell will become active for data input. A list of the items for the spreadsheet are explained here.



Group - Clicking on the first cell under Group will bring up an automated tool for defining seam welds in a secondary GUI as shown below. While the weld group must be set up in Patran, it is not necessary to create the plate group as the element group connected on the "toe" side of the weld is extracted automatically. In this case, the default group is selected for the plate group. If the group of elements representing the set of elements connected to the weld group plate exists, then that group may be selected for the plate group. For both cases, the resulting Seam-weld group will be identical. Groups in the model databox are selected by clicking on the group in the databox and clicking on the respective select group button. Specify the name of the nodal group that shares the common nodes in the selected weld and plate groups. If you wish to create a nodel list, click on Create Node List before clicking on the Apply button. Selecting Apply computes the common nodes between the plate and weld groups. The new group is stored in the database with a MW_groupname. Note that groupname cannot have any spaces, either leading, trailing or anywhere in between. There is no limit on the number of MW_groupname groups that may be created and analysis is

performed and results reported on the MW_groupnamegroups. If a MW_groupnamegroup exists in the database the group cell in the main form gets populated automatically.



Note that the weld could be considered to have two toes, one at each plate. Which toe is extracted is dependent on the definition of the plate group. the use may wish to define the plate group that extracts both toes.

The user can now create a node list for plotting various results by clicking in the Node list text box and selecting nodes from the display of the MW_group. The result plotted is in the order in which the nodes are selected, i.e. the x-axis is the node list. This list may be reversed by clicking on the reverse node list button.

Selecting Create Node List saves the node list to a `jobname.ent` file. Select Cancel to return to the main material form.

The second button is:
SN Flexible - when this cell is selected, a listbox of available seam weld S-N curves appears at the bottom of the form. Selecting one of these from the listbox will fill the active cell with the material name and make the adjacent cell active. A suitable flexible S-N material curve should be selected. There is a generic flexible material curve in the database that is applicable for most low carbon steel applications. If this not applicable for your analysis, you will have to create/provide your materials data with the materials database manager (PFMAT).

The third button is:
SN Stiff – This allows the user to pick the stiff S-N curve from the materials database. There is a generic stiff curve in the database that is applicable for most low carbon steel applications. If this is not applicable for your analysis, you will have to create/provide your materials data with the materials database manager (PFMAT). It should be noted that, in general, the weld geometry detail has a greater influence on the S-N curve than the parent metal.

The fourth button is:
M1/M2 Ratio – The default is 2.5. This ratio is used in computing the mean stress correction based on the Haigh diagram. For an explanation of how this is applied using both curves, please refer to General Procedure, 835.

The fifth button is:
Multiplier – The multiplier is used to scale the stress for the group of elements in the MW_Username group. User enters value in dialog box which fills the cell automatically – default = 1.0.

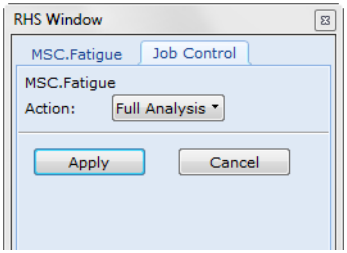
The sixth button is:
Offset – The offset is used to offset the stress for the group of elements in the MW_Username group. User enters value in dialog box which fills the cell automatically – default = 0.0. If offset is specified, the units must be stress as per the S-N curve. This is applied after the multiplier (stress=(stress*multiplier)+offset).

Loading Information

The loading is identical to that of the other analysis types. You are referred to Loading Information, 786 for more detailed information about this input.

Job Control

The Job Control form allows you to submit a Full Analysis. A Partial Analysis is not possible in SEAMW. Most other activities available from the Job Control form are as with the other analysis types. When running Interactive, the SEAMW program will be invoked. When running a Full Analysis, SEAMW will run in the background and progress maybe monitored.



Since control of the seam weld analysis job is so similar to any other analysis jobs you are referred to Job Control, 787.

The following files are created after a seam weld analysis is complete.

jobname.fin	Fatigue job parameter data.
jobname.fes	FE stress and fatigue input file.

jobname.fef	Fatigue results file.
jobname.msg	Message file.
jobname.sta	Job status file.
jobname.ent	Analysis node file (entity list).

Job Execution Status Messages

When a job is submitted it will pass through three phases. The Users will be informed through the status option of the progress of the job. Both success and error messages are displayed. The following list summarizes some of the typical, normal operation messages which the user may experience. Not all the messages will be displayed since the status file is updated very quickly in some cases. In certain cases, the status file may not be available in which case a "Try again" message will appear. When execution is through MSC's Analysis Manager, these messages appear in the Analysis Manager message window.

```
Phase 1
JOB jobname HAS BEEN SUBMITTED BUT HAS NOT STARTED EXECUTION
JOB HAS BEGUN EXECUTION
WRITING THE JOB (.FIN) FILE

Phase 2
PAT3FAT" reading the neutral file...
PAT3FAT" reading the.FIN file...
PAT3FAT" reading the FE results...
PAT3FAT" writing the.FES file...
PAT3FAT" terminated normally
or
FATTRANS" reading the neutral file...
FATTRANS" reading the.FIN file...
FATTRANS" reading the FE results...
FATTRANS" writing the.FES file...
FATTRANS" terminated normally

Phase 3
Fatigue analysis module loaded and running
Fatigue analysis completed successfully
```

In addition there may be other messages giving status of other aspects of the job. Error messages are also displayed via these status messages.

What To Do When a Job Stops

If the status message does not appear to be updating, it is possible that the job has halted due to an error. In many cases, that error message will be reported through the status facility. However, if it is not reported, you can investigate the problem by opening another window and examining the following file:

jobname.msg: This file will contain all the status messages for the job including any error messages.

Some hints on determining why a job has failed:

1. If the `jobname.fin` file and the `database` exist in your directory, but not the `.fes` file,try running the job interactively by typing:

pat3fat jobname

or by typing:

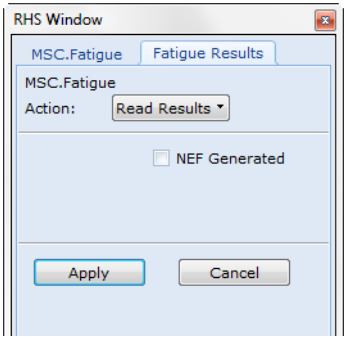
fattrans jobname

then checking the message file.

2. If the jobname.fes file exists, run the SEAMW program interactively and watch for error messages. Type **seamw** at the system prompt.
3. If the jobname.fes file exist, but SEAMW still gives errors, try running FEFAT. Type fefat at the system prompt. Use the Utilities menu to examine the file contents.

Results

A Patran element style .els file called jobname.fef which contains the usual MSC Fatigue information, plus a few additional pieces of information is created from the analysis and is read in to the database. The results file columns are: damage, life, life (repeats), log of damage, log of life, log of life in Repeats. The repeats may be specified on the Loading form.



Marker Plots in Insight

The best way of visualizing seam weld fatigue analysis results on the FE model is using Insight. This is done by selecting the Insight toggle from the main menu bar or MSC Fatigue Pre & Post or within Patran. Once the Insight tool has opened, set the action to Create and the tool to Marker, choose a name for the marker plot. Use the results selection button to select the results to be plotted. The results available are:

Damage

Log of Damage	Damage reported in log units which is interpreted to be 10 raised to the reported value. This is log(base10) of damage.
Life in Repeats	Life in repeats of the defined load history service.
Log of Life in Repeats	Life in log units which is interpreted to be 10 raised to the reported value. This is log(base10) of life.

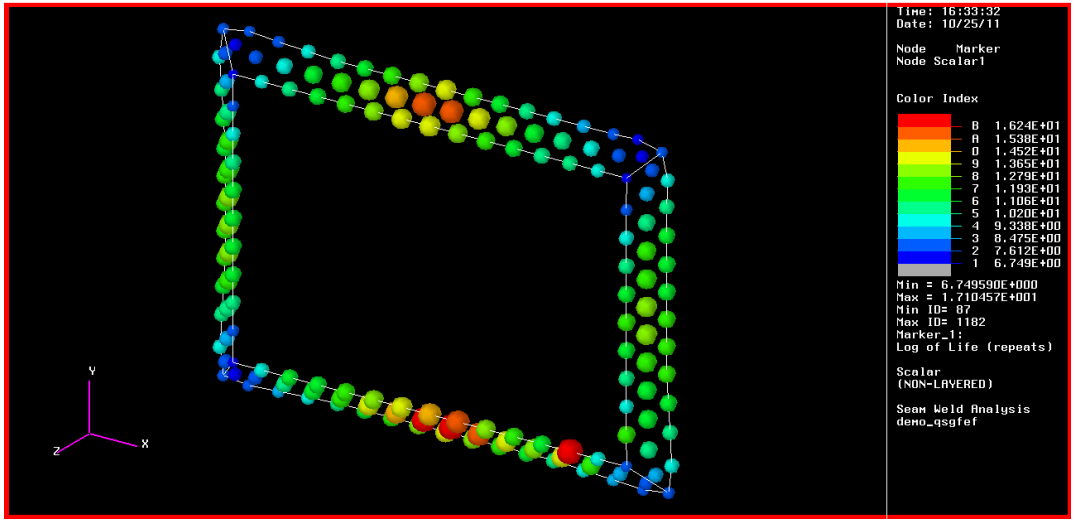
Life Life	The reciprocal of damage. The event units are reported with reference to those defined for the time history.
Log of Life	This is the life in log units. Log results generally give better looking fringe plots as the log units linearize the results that makes it easier to interpret the results.

Select the result to be plotted; one of the most useful to plot is Log of Life (equivalent units). Use the Marker Attributes button to choose the kind of marker to be applied to the model. The recommended choices are:

Set the target for posting the results to nodes and target only the MW_group that was analyzed.

- Color: Mapped
- Type: Sphere
- Scale: Screen
- Scale Factor 0.03

A typical insight plot from a seam weld analysis is shown below.



11 Rotating Structures Analysis

- Introduction 870
- Job Setup 871
- Wheels Analyzer (FEROT) 883

Introduction

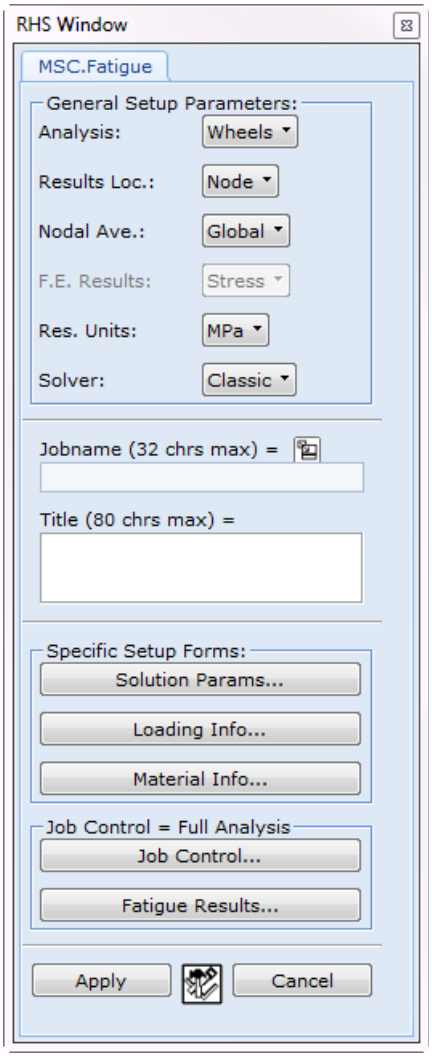
FEROT is a specialized MSC Fatigue module that analyzes the fatigue life of rotating structures using the S-N method.

In setting up a proper Wheels analysis, it is important to first have a sound understanding of the operation of MSC Fatigue within the Patran environment or within MSC Fatigue. It is suggested that you first read and understand Using MSC Fatigue - The Classic Modules (Ch. 3). Job setup for a spot weld analysis is very similar to job setup for other fatigue analysis types with only a few differences, as pointed out in this chapter.

Certain limitations or restraints exist in setting up a Wheels analysis. These are mainly the fact that the analysis MUST be set up within a preprocessor, and that results are surface resolved. These stresses can then be used to make fatigue life predictions on every surface node using a critical plane S-N (total life) method.

Job Setup

The setup starts in the MSC Fatigue forms within Patran or MSC Fatigue . You must set the analysis type to Wheels. There are three basic inputs just as for any other analysis types, those being Solution Parameters, Material and Loading Information. Results postprocessing and Job Control are also described in this section.



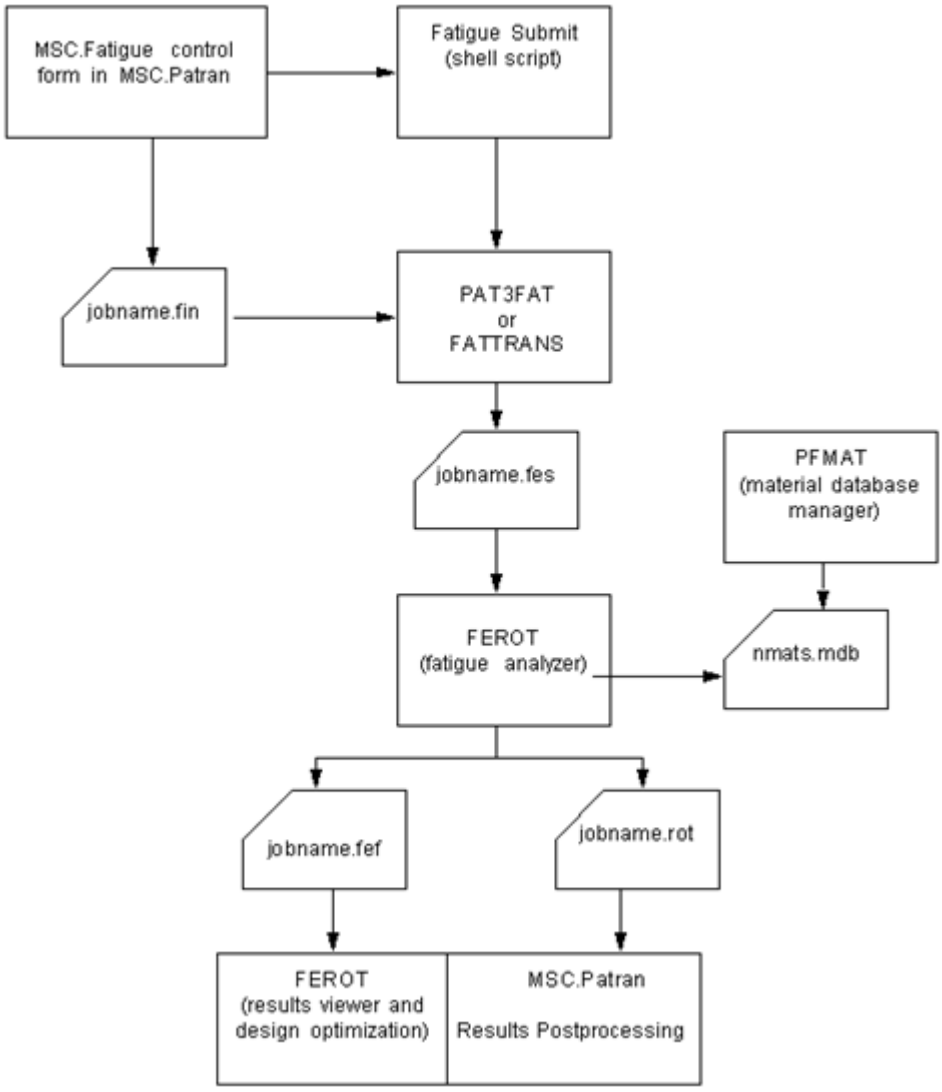


Figure 11-1 Wheels Analysis Submittal Schematic

Basic Information

All programs in the MSC Fatigue system may be executed by typing the names of the program at the system prompt. These programs may ask questions which are not normally presented to you since they are executed as batch jobs when called from MSC Fatigue or Patran. The programs normally used in a Wheels Analysis are listed below.

1. Data Preparation

PFMAT	Materials Database Manager
PVXMUL	A Peak-Valley Extraction Program for Reducing Lengthy Time Histories
MFD	A Multi-file Display Program

2. Global Wheels Analysis

PAT3FAT	Model database (Patran) to Fatigue Input Translator
FATTRANS	New model database (Patran) to Fatigue Input Translator
FEROT	Fatigue Preprocessor (rainflow cycle counting)

3. Results Postprocessing and Design Optimization

FEROT	Results Listings, Polar Plots
FEROT	Design Optimization Tools

4. General Utilities

FEROT	FES File ASCII/Binary Convertor
PFTRM	Terminal Driver
CONFIL	Binary to Binary File Convertor

Analysis Route

The actual programs needed to complete a global multi-node or element wheels analysis are:

FatigueSubmit	Shell script (necessary for submittal from MSC Fatigue Pre & Post or Patran)
PAT3FAT	Translator (creates the fatigue input file `filename.fes`)
FATTRANS	New Translator (creates the fatigue input file `filename.fes`)
FEROT	Fatigue Analyzer (life prediction)

The programs and options must be used in this order. The results may be reviewed using the results listing options in SPOTW or by inspecting the ASCII results files (`jobname.fef` and `jobname.spt`) using a text editor or by inspecting marker plots in Patran/Insight.

Necessary files

When a wheels fatigue analysis is set up using MSC Fatigue menus or Patran, these are the files necessary to run the analysis and the files that are created.

jobname.fin	This file contains all the analysis parameters that were defined in the main and subordinate MSC Fatigue forms, (e.g., material data file names). In addition, the analysis type and job titles are also defined in this file. A full description of this file is contained in The Job Information File (jobname.fin), 369.
Database	The groups of nodes or elements for which the fatigue life is to be calculated are contained in the database. For a wheels analysis, it is necessary to have carried out a finite element analysis in MSC.Nastran where each loaded segment on the wheel is a separate subcase.
Additional Files	Other files that are necessary to complete a successful fatigue is the materials database (`nmats.mdb` which is generally held in a central location and not necessary to be located in the user's local directory.

The first of these files is ASCII and may be edited using a standard text file editor. Although this method of defining the MSC Fatigue job parameters is not as automated as the MSC Fatigue menus in Patran, it does offer a simple and rapid method of changing a few parameters without the encumbrance of a menu structure.

File that may be created during an analysis run are summarized below:

File Name	Description
***jobname*.fin**	Job parameter file (ASCII).
***jobname*.fes**	MSC Fatigue Input file (Binary).
***jobname*.asc**	ASCII version of the *jobname*.fes file.
***jobname*.fpp**	MSC Fatigue intermediate results file (Binary).
***jobname*.msg/log**	MSC Fatigue message and log files (ASCII).
***jobname*.sta**	Job status file (ASCII).
***jobname*.fef**	Global multi-node/element results file (ASCII).
***jobname*.rot**	This file provides useful information on the stress contribution and damage for each load condition at the analysis nodes for each rotational increment of the structure and each surface angle increment.
***.dac, *.cyh**	Loading time history/rainflow matrix files (Binary).
***.xyd**	K solution XY data (ASCII).
***.tem**	Plotting format data (ASCII PCL file).
***_tmpl**	Results template files (ASCII).
***.adb/.mdb**	Materials database description files (ASCII/Binary/Binary).

After the translator has been run (described in The Translator (PAT3FAT or FATTRANS), 294) and the fatigue input file (`jobname.fes`) has been created, the Wheels Analyzer, FEROT, is run. There is no pre-processing stage in a Wheels Analysis. When run in interactive mode, this program asks for a number of input parameters

which are passed in through the `jobname.fes` file when run from the MSC Fatigue menus within Patran. A full description of file content is provided in Description of Files, 369.

Solution Parameters

The only parameters that can be set on this form are mean stress correction and surface angle. A Gerber, Goodman or no mean stress correction may be selected. The FEROT module performs a critical plane S-N analysis using surface resolved stresses at every surface node. The option to generate surface normals is available from the job control menu. Since the certainty of survival is not selectable in this module, the S-N curve is used without any modifications.

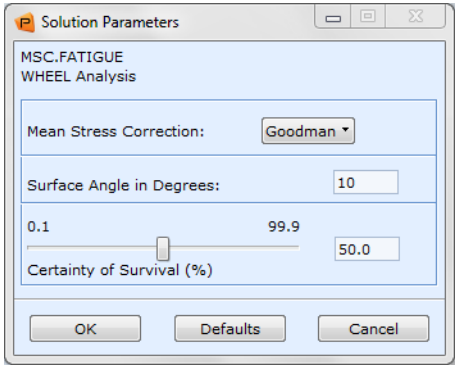


Figure 11-2 Wheels Solution Parameters Form

Materials Information

The materials form and functionality is identical to other analysis types. The user is referred to the apropriate section and page number for guidance

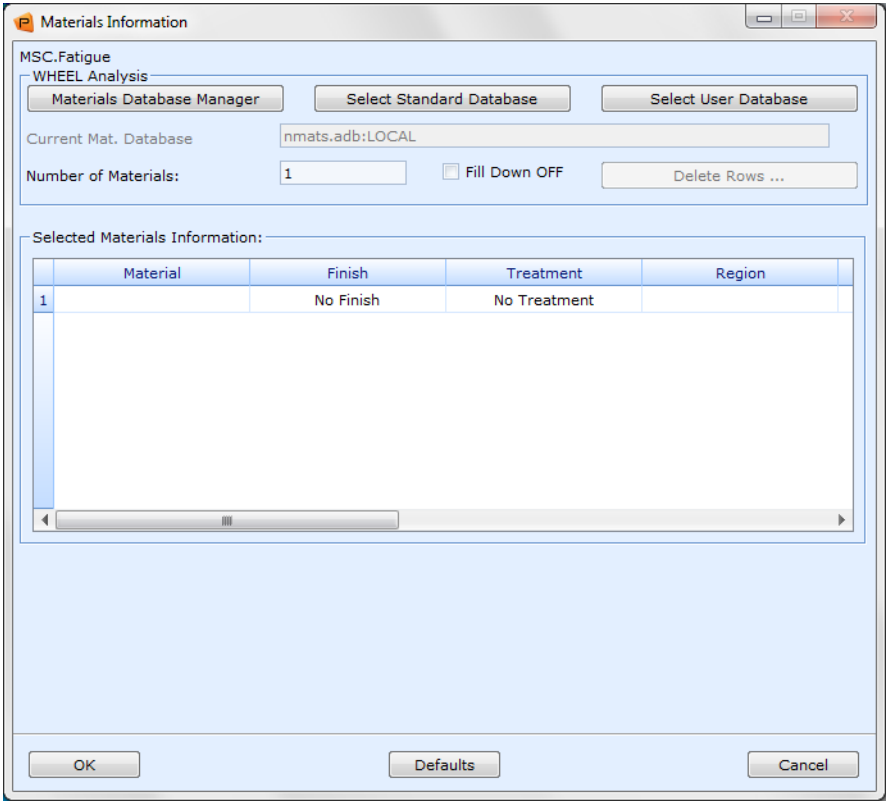


Figure 11-3 Materials Information Form

Loading Information

The loading information form is unique in that no load time history files are required. The form also provides the capability of defining load events and allows the specification of a design life to report a factor of safety.

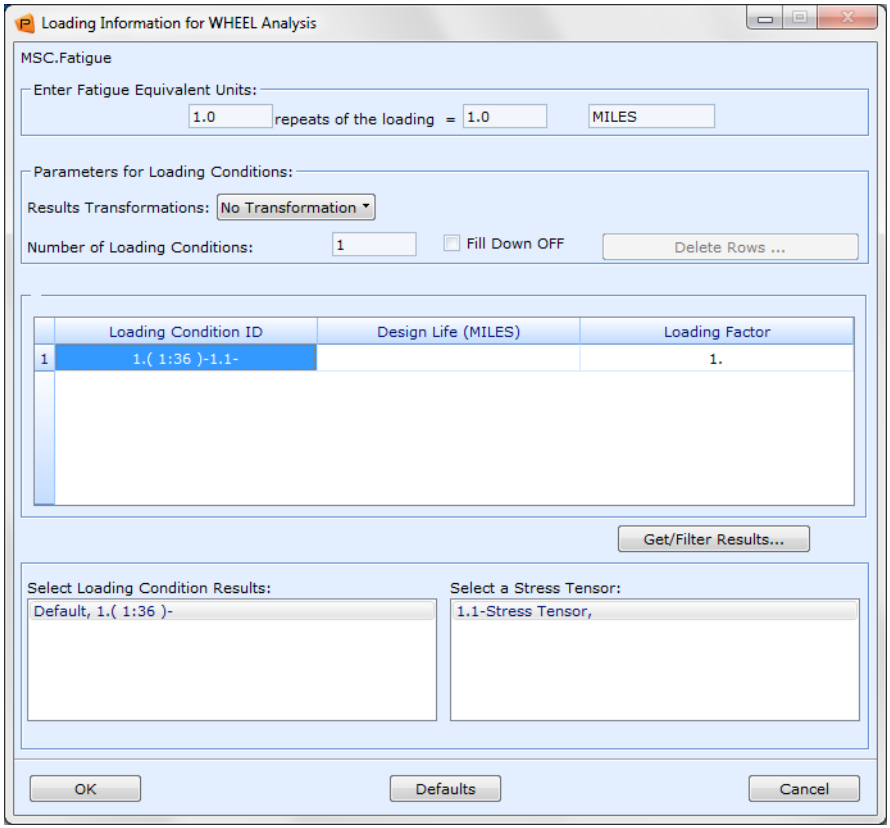


Figure 11-4 Main Loading form

A description and functionality of each field is explained below.

Field	Description
Fatigue Equivalent Units	This sets the equivalent units for reporting purposes. Life is reported in terms of the equivalent units e.g. in the example shown life will be reported in Miles.
Number of Loading Conditions	It is important to understand the concept of a load condition in this module. The load condition is not a Unit case - rather it is a sequence of subcases in a NASTRAN run. The load condition represents a particular type of loading e.g. the first load condition could be a sequence of subcases for one revolution of the wheel on a smooth surface, the second for one revolution of the wheel on a rough surface and so on. Clicking on the load condition cell displays a secondary GUI , shown in figure , that allows the user to select the subcases corresponding to the load condition. The descriptions of the fields and functionality of the secondary GUI are shown below.
Design life	This is the design life in the equivalent units. A factor of safety calculation is performed using this input e.g. if the design life input is 1000 miles and the calculated life is 1100 miles the factor of safety is 1100/1000 = 1.1
Loading Factor	Stress results for the load condition may be scaled by specifying a factor in this cell.

Secondary Loading GUI

The top window shows all subcases for all load conditions that exist in the database.

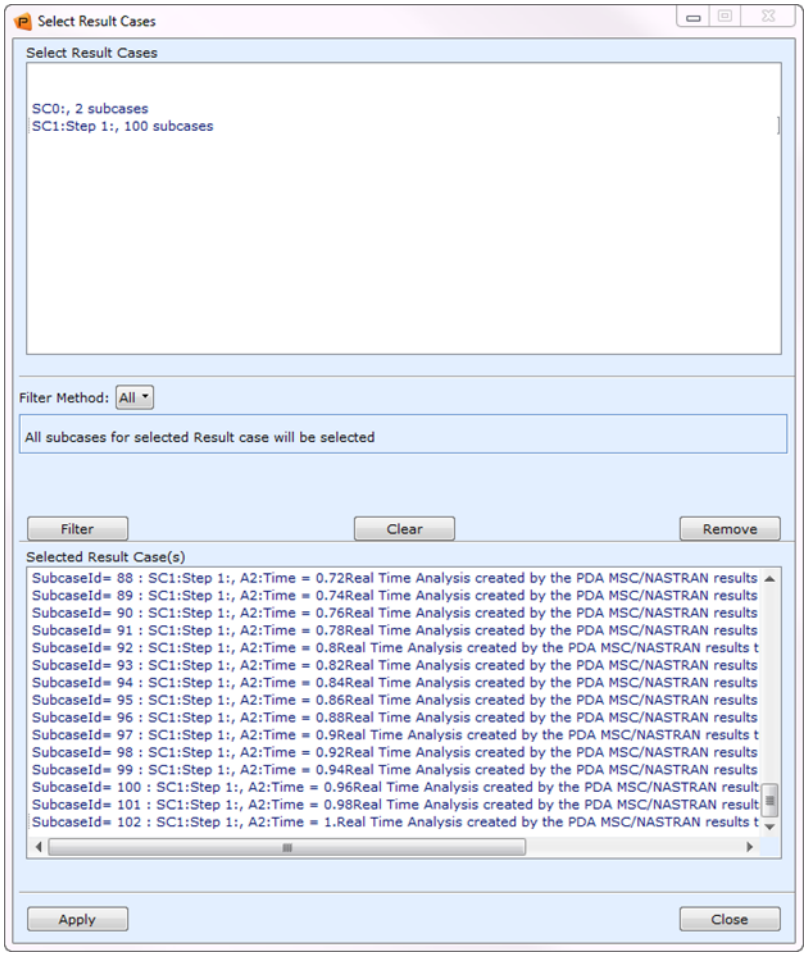
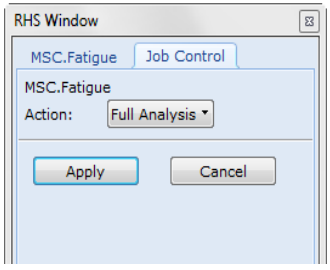


Figure 11-5 Secondary Loading form

A description and functionality of each field is explained below.

Field	Description
Select Results cases	Window displaying all the results cases in the database. Clicking on any one case will populate the lower window for subsequent filtering.
Filter Method	Subcases can be filtered on Subcase IDs by entering the appropriate IDs in the next field .
Subcase ID	Enter range of subcase ID's e.g. 1:30 or separate IDs (1 3 5) or incements (1:10:2 - means select subcases one through 10 by twos). Or use any combination of spaces and colons between subcase IDs to select as many as you wish.and click on Filter. The lower window will show the results of the filtering.
Filter	Click on this button to display the results of the range selected above.
Clear	Clears the lower window for the user to start over and specify a different range for filtering.
Remove	Removes a subcase that is clicked on in the lower window. The remove button is clicked after making the selection from the lower window.
Add	The filtered subcases are added to the select loading conditions text box of the main loading form (Figure 8-80). These subcases are subsequently selected with the appropriate stress tensor results and loaded into load condition cell (Figure 8-80) with the Fill cell button.
Overwrite	The overwrite button allows the user to overwrite the results of a previous Add operation. The results in the select loading conditions text box of the main loading form (Figure 8-80) are replaced with the results from the current filter operation.
Close	Close the window and return to the main loading form.

The Job Control form allows you to submit a Full Analysis. A Partial Analysis is not possible. Most other activities available from the Job Control form are as with the other analysis types. When running FEROT program will be invoked. When running a Full Analysis, FEROT will run in the background.



Since control of the Wheel analysis job is so similar to any other analysis jobs you are referred to Job Control, 787. The following files are created after a Wheels analysis is complete.

File name	Description
jobname.fin	Fatigue job parameter data.
jobname.fes	FE stress and fatigue input file.
jobname.fef	Fatigue results file.
jobname.msg	Message file.
jobname.sta	Job status file.
jobname.rot	This file provides useful information on the stress contribution and damage for each load condition at the analysis nodes for each rotational increment of the structure and each surface angle increment.

Job Execution Status Messages

When a job is submitted, it will pass through three to five phases. The user will be informed through the status option of the progress of the job. Both success and error messages are displayed. The following list summarizes some of the typical, normal operation messages which the user may experience. Not all the messages will be displayed since the status file is updated very quickly in some cases. In certain cases, the status file may not be available in which case a "Try again" message will appear. When execution is through MSC's Analysis Manager, these messages appear in the Analysis Manager message window.

Phase 1

```
JOB jobname HAS BEEN SUBMITTED BUT HAS NOT STARTED EXECUTION
JOB HAS BEGUN EXECUTION
WRITING THE JOB (.FIN) FILE
```

Phase 2

```
PAT3FAT" reading the neutral file...
PAT3FAT" reading the.FIN file...
PAT3FAT" reading the FE results...
PAT3FAT" writing the.FES file...
PAT3FAT" terminated normally
or
FATTRANS" reading the neutral file...
FATTRANS" reading the.FIN file...
FATTRANS" reading the FE results...
FATTRANS" writing the.FES file...
FATTRANS" terminated normally
```

Phase 3

```
Fatigue analysis module loaded and running
Fatigue analysis completed successfully
```

In addition, there may be other messages giving status of other aspects of the job. Error messages are also displayed via these status messages.

What To Do When a Job Stops

If the status message does not appear to be updating, it is possible that the job has halted due to an error. In many cases, that error message will be reported through the status facility. However, if it is not reported, you can investigate the problem by opening another window and examining the following file:

jobname.msg: This file will contain all the status messages for the job including any error messages.

Some hints on determining why a job has failed:

1. If the `jobname.fin` file and the `database` exist in your directory, try running the job interactively by typing:

   ```
   pat3fat jobname
   ```

or by typing:

   ```
   fattrans jobname
   ```

   then checking the message file.

2. If the `jobname.fes` file exists, run the FEROT program interactively and watch for error messages. Type **Ferot** at the system prompt.
3. If the `jobname.fes` file exist, but FEROT still gives errors, try running FEFAT. Type **fefat** at the system prompt. Use the Utilities menu to examine the file contents.

Also check the file called `batlog.lst,` if it exists, for any additional clues if none of the above helps.

Important:	If a job inadvertently quits, sometimes a `jobname.fpr` file is left in the directory. This file is created during submission to detect a running job so that inadvertent submissions while a job is in progress of the same jobname are detected. In some cases, it may be necessary to remove this file before resubmitting the job.

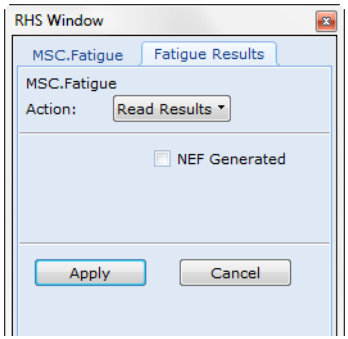
Error Messages

See Error Messages (App. 19) for a description of error messages and possible solutions.

Results

A Patran element style `.els` file called `jobname.fef` which contains the usual MSC Fatigue information, plus a few additional pieces of information is created from the analysis and is read in to the database. The results file columns are: damage, log-damage, life (repeats), log-life (repeats), life (equivalent units), log-life (equivalent units), failure location, angle, node, maximum force, and are apparent when accessing them from the postprocessor Results application.

The Log of Life in Repeats and the Log of Life (equivalent Units) are log (base10) of the life in repeats or user units respectively.



Also flat file called `jobname.rot` which contains damage and maximum and minimum stress is created which can be processed by the FEROT module itself.

As with other analysis types, these options remain basically the same for spot weld analysis, therefore you are referred to Postprocessing Results, 100. The differences are:

1. List Results, Re-Analyze, and Optimize will spawn SPOTW to the appropriate functionality requested
2. Factor of Safety is not available.

Wheels Analyzer (FEROT)

The MSC Fatigue analysis module FEROT performs a number of tasks including Global Analysis, Results Listing, and Time history extractions. Module operation of each of these tasks is described in detail in this section.

The operation of FEROT can be in three modes: by spawning from the MSC Fatigue or Patran environment, in stand alone mode by typing **ferot** at the system prompt or in batch mode. The only difference is that in stand alone mode, the user must supply the jobname. (In direct mode from Patran, these are passed to FEROT automatically). Once FEROT has been initiated in either of these modes, two windows will be presented with the Motif driver. The top, small form is a generic form and allows for general program control. This is discussed in detail in Module Operations (Ch. 18) for the Motif driver.

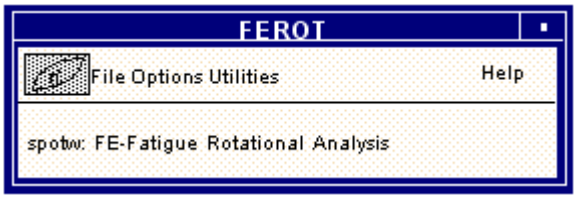


Figure 11-6 FEROT Utility Form

The main menu appears as follows. Each item is discussed in this section.

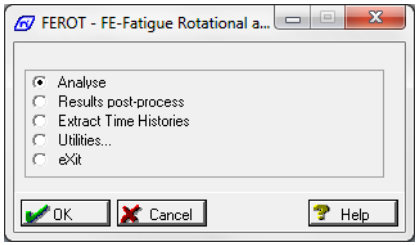


Figure 11-7 FEROT Main Menu Form

1. **Analyse** - This option will rerun the analysis interactively for you. If this it is a computationally intensive operation, you may find it more convenient to run in batch mode. See Analyze, 884.
2. **Results post-process** - This option will put you in another menu to allow for sensitivity studies and design optimization tasks.
3. **Extract Time Histories** - This option will read the `jobname.fes` file and display the results.
4. **Utilities** - This option allows for the binary MSC Fatigue input file (file `type.fes`) to be manipulated and converted.
5. **eXit** - This options will exit you from the program.

Analyze

Note that the fatigue analysis may take some time. It may be worth considering operating FEROT in batch. Batch operation is discussed in FEROT Batch Operation on page .

When the "Estimate Fatigue Life" option has been selected, the user will be presented with a number of questions. The first question asks for the input file name. Press the **OK** button once a file name (`jobname.fes`) has been selected. Use the List button to list all available input files. These files have been created by the PAT3FAT or FATTRANS translator. The default will be the last `jobname.fes` created.

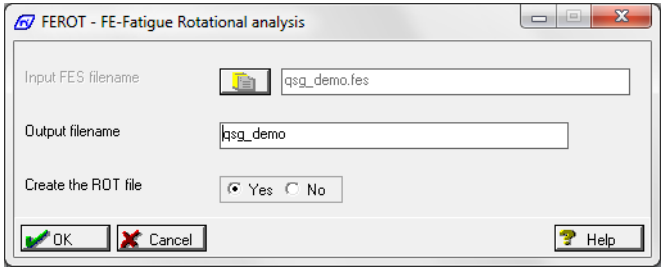


Figure 11-8 FEROT Global Analysis Form

The following table explains each entry on the previous form.

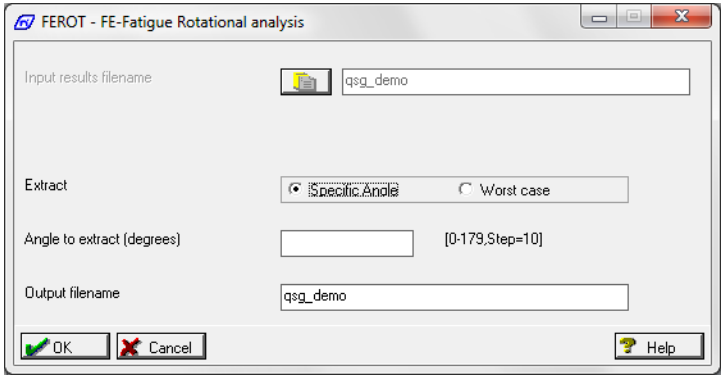
Field	Description
Input Filename	This is the fatigue input file name (`jobname.fes`) to be used in the fatigue pre-processing. The job must have already run at least through the PAT3FAT or FATTRANS translator to produce a `jobname.fes` file. This is achieved by carrying out a full or translate only submission from the job submit options in the MSC Fatigue menus of Patran, or by running PAT3FAT (or FATTRANS) in stand-alone mode (see The Translator (PAT3FAT or FATTRANS), 294). To select a jobname from a list of available jobs, use the List button. Once the file name has been supplied and the screen inputs accepted, the rest of the input options will be displayed.
Output filename	The default is the jobname. After processing `jobname.fef` will exist. You will be requested to overwrite any existing output file of the same name if one exists.
Create ROT file	If yes is toggled a jobname.rot will be created. You will be requested to overwrite any existing rot file of the same name if one exists.

When this form is complete, select **OK** and the analysis will run. While the analysis is running, a form appears which shows the progress of the analysis.

When the analysis is complete .

Results Postprocess

The following form is displayed when this option is selected:

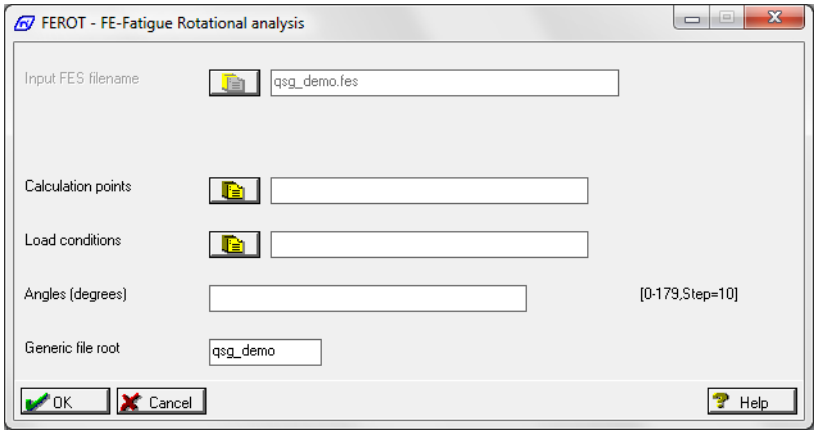


The results post-process is a utility for filtering the results from an analysis since the output can be quite extensive depending on the analysis set and the number of load conditions run. The following table explains each entry on the previous form.

Field	Description
Input results filename	This is the .rot file (`jobname.rot`) that must be available from a previous analysis (jobname.fes). To select a .rot file from a list of available jobs that have been run, use the List button. Once the file name has been supplied and the screen inputs accepted, the rest of the input options will be displayed.
Extract	**Specific Angle :** When this is selected, the Angle to extract text box is activated for entry. This angle is dependent on the surface angle entry on solution control form (page).Only an integer multiple of the surface angle may be specified e.g. if the surface angle for the critical plane analysis was specified as 10 degrees, results may only be extracted for angles in ten-degree increments up to 360 degrees. **Worst Case:** This extracts the worst (lowest life, maximum damage) results for all conditions including the worst case (accumulated damage from all load conditions)
Output Filename	Enter the name of the output file. This file will have a .fef extension and can either be listed or read back into the PATRAN database by specifying the same Jobname as this output file.

Extract Time Histories

This option allows the extraction of stress time histories (.DAC) files.



Field	Description
Input fes filename	Enter the jobname.fes filename. To select a .fes file from a list of available jobs that have been run, use the List button. Once the file name has been supplied and the screen inputs accepted, the rest of the input options will be displayed.
Calculation points	Enter the node number for extraction. To browse the nodes in the analysis set use the list button and enter the node number from the displayed list.
Load conditions	Enter the load condition number. To browse the available load conditions, use the list button and enter the load condition from the displayed list.
Angle (degrees)	Enter the surface angle at which the time history is desired. Remember this is dependent on the surface angle specified on the solution control form and only multiples of that angle may be specified.
Generic file root	The default output filename is the jobname_node_loadcondition_angle.dac.A plot of the extracted time history will be displayed automatically.

FEROT Batch Operation

Batch operation has the following batch keywords:

/INPut	The name of the input job (/INP=myjob)
/OUTput	The name of the output file (/OUT=newname)
/OPTion	This is the option as shown on the main screen (A,R,E.X) (/OPT=e)
/CRE	Create the ROT file (Y,N)
/NUMEQU	Number of Equivalent Units
/MINers	This is the Miner's Sum. Valid from 0.5 to 2.0 generally. (/MIN=1.0)
/NANGD	Number of Angles for Calculation
/NUGGET	Calculate Nugget? (Y or N)

/OVerwrite	Overwrite (yes or no) (/OV=yes)
/*	Output to.... (TT = screen, Filename = to file, None = None)

Example:

```
spotw/opt=e/inp=myjob/ov=y/*=tt
```

This batch line would run a spot weld analysis in batch where the first option is called out (Estimate Fatigue Life) and the jobname is myjob. All the defaults are used for the rest of the inputs such as matrix size and Miners sum. The results files will be called `jobname.fef` and `jobname.spt` by default.

12 Software Strain Gauges

- Introduction 890
- The Gauge Tool 892
- Software Strain Gauge Module (SSG) 906
- Stress-Strain Analysis (MSSA) 911

Introduction

The Software Strain Gauge (SSG) is a Test-Analysis Correlation Tool. This is a fatigue specialist tool for correlating measured responses with those extracted from FE analysis. Throughout this chapter, certain knowledge by the user of standard MSC Fatigue terminology, awareness, and usage is assumed. If you are unfamiliar with MSC Fatigue, it is suggested that you read chapters 1 through 5 before attempting to use this tool.

The software strain gauge is a software tool which allows a direct correlation to be made between measured strain histories obtained using resistance strain gauges and predicted strain histories from the surface of finite element models. This is achieved by allowing the application of simulated strain gauges to the surface of the FE model in the same positions as real strain gauges on the corresponding component.

The simulated gauges consist of one or more thin shell elements which are fitted to the surface of the FE model. The gauges can then be used to extract the results of previously carried out FE stress/strain analyses at the locations and in the orientations defined. Then, using MSC Fatigue and one of its modules called SSG it is possible to predict the stress or strain histories from the gauges in a way which is directly comparable with direct strain measurements.

In addition to its use as a correlation tool, the software strain gauge has other uses. For instance, the weld fatigue analysis techniques described in BS7608 require structural stresses on welds to be determined from finite elements by a process of extrapolating the stresses to the weld toe, using stresses up to a certain distance from the weld toe. It can be difficult to obtain this information in an automated way. The problem can be overcome by simulating the recommended experimental technique of placing a strain gauge of 3 to 6 mm in size at 0.1 x (sheet thickness) from the weld toe.

Users may also find the software strain gauge useful simply for obtaining static stress and strain results from particular locations within elements, or in particular directions.

In addition to the software strain gauge tool, this option of MSC Fatigue also includes a module (MSSA) for processing rosette data and creating outputs suitable for use by either the stress or strain-life fatigue analyzers MCLF or MSLF. It also provides an indication of the state of multiaxiality present and suggests possible processing routes through the fatigue analyzers.

Basic Information

The software strain gauge requires first that a FE stress/strain analysis or analyses be carried out, and that the model and results be read into the MSC Fatigue or Patran database. The software strain gauge supports both static and transient results. It is normally expected that the results be elastic.

Note:	You do not have to carry out an elastic-plastic FE analysis.

Having carried out the FE stress analyses, the following programs are normally used in the course of an analysis:

1. Data Preparation

PFMAT	Preparation of Material Data
PTIME	Preparation of Time Histories including an ASCII to Binary Convertor
MMFD	A Multi-file Display Program A Text Editor for adding new gauges to a gauge definition file may also necessary.

2. Strain Gauge Management

GAUGE TOOL	Position, Modification, and Deletion of Gauges and Extraction of Results

3. Time History Response Creation

PAT3FAT	Model database (Patran) to Fatigue Input Translator
FATTRANS	New model database (Patran) to Fatigue Input Translator
SSG	Creation of Stress or Strain Histories
MMFD	A Multi-file Display for Results Comparison Purposes with Measured Strains

4. General Utilities

FEFAT	FES File ASCII/Binary Convertor
PFTRM	Terminal Driver
MCONFIL	Binary to Binary File Convertor

Analysis Route

The basic steps in a software strain gauge analysis are as follows:

1. First run a finite element analysis. This can be either a transient analysis, or a set of static load cases, such as might be suitable for use in a standard MSC Fatigue quasi-static analysis. Then read the model and results into the Patran or MSC Fatigue database.
2. Set the Analysis Type to Soft S/G for software strain gauges in the main form. Select the Gauge Tool option. Use the Gauge Tool to apply the numbers and types of gauges required. See Creating and Modifying Gauges, 897.

   Gauges are selected from a gauge definition file `gauges.def` which is described in The Gauge Definition File, 893. The Gauge Tool allows gauges to be positioned, modified and deleted. Placement of gauges is described in more detail in the section entitled “Applying and Modifying Gauges and Extracting Results”. Note that if you want the software to calculate elastic-plastic strain histories, you should specify “plastic” on the Gauge Tool form.
3. When all the gauges are positioned as required, return to the Soft S/G menu and start up the Results Extraction form. Use this form to select the results cases to be extracted, and the gauges to which the results will be written. See Extracting the FE Results, 901.

4. Now, return to the MSC Fatigue main form and set up a crack initiation analysis in the normal way. Each gauge is a separate group in the analysis. It is necessary only to use the option Translate Only, 90 to produce a `.fes` fatigue input file. This process is explained in detail in Using MSC Fatigue - The Classic Modules (Ch. 3) and Total Life and Crack Initiation (Ch. 6).
5. Run SSG either by Pressing on "Rosette Analysis" on the Soft S/G menu, or by typing **ssg** at the system prompt using the newly created `.fes` fatigue input file. See Software Strain Gauge Module (SSG), 906.

The Gauge Tool

The Gauge Tool is an integral part of the MSC Fatigue and Patran systems accessible by setting the Analysis Type in the main MSC Fatigue form to Soft S/G. There are two basic step that must be followed in order to use the tool properly.

1. Create (or Modify) the gauges.
2. Extract results from the FE model to the gauge elements.

This section explains how to define gauges by way of The Gauge Definition File, 893. It explains how to then physically create the gauges (Creating and Modifying Gauges, 897) in the proper locations on the finite element model. And finally it describes how to extract the FE results (Extracting the FE Results, 901) such that they are associated with and in the same coordinate system as the gauge elements.

Once this is done you can translate the results from these gauge elements to the fatigue input file, typically called `jobname.fes` as if you were going to run a normal crack initiation or S-N analysis. This simply creates the necessary input file to feed into the Software Strain Gauge Module (SSG), 906 analyzer.

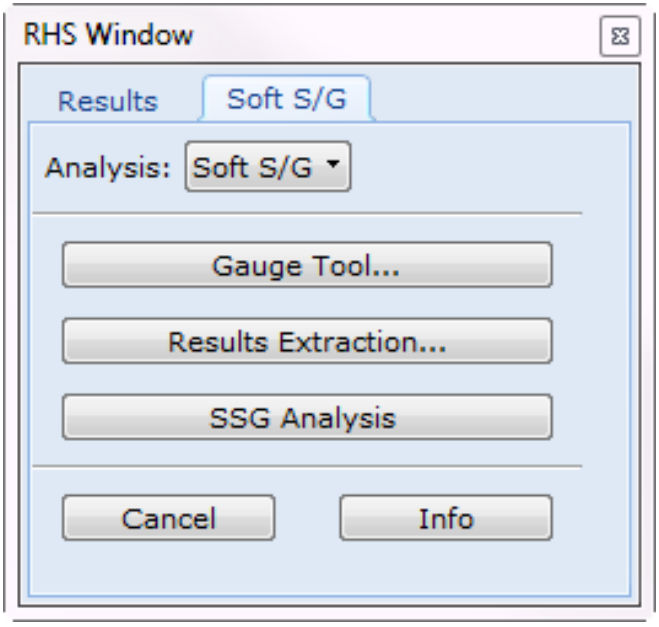


Figure 12-1 Accessing the Gauge Tool

The Gauge Definition File

The Gauge Definition file is an ASCII file that contains the definition of the specifications of the available strain gauges. The file is called `gauges.def` and by default resides in the installation directory (e.g. `<install_dir>/ mscfatigue_files/ gauges.def`). If desired, the user can also make a copy in his home directory or in the local working directory. The program will automatically look first in the local, then the home and then the central directory, and will use the first `gauges.def` file found.

The gauge definition file can include definitions of up to 35 gauges. The software is delivered with a gauge file containing 8 commonly used gauge types. The file can easily be edited to include new gauge types. Each gauge in the file is represented by the following block of information (comments are represented by #):

TYPE NUMBER=	#1-9 or A-Z unique identifier for each gauge type
NAME=	#any string here (used in picklist)
COMMENTS=	#any string here
GAUGE TYPE =	#1=single, 2=tee, 3=rosette
S/P=	#0=NA, 1=stacked, 2=planar
CONFIGURATION=	#0=NA, 1=rectangular, 2=delta, 3=other
KTS=	#transverse sensitivities - 0,1,2, or 3 numbers, comma separated (in %)
POISSON=	#*gauge* poisson ratio, typically 0.285

UNITS=	#1=mm, 2=m, 3=inches, 4=feet
COORD1=	#4 x,y pairs of co-ordinates describing the
COORD2=	#locations of the corners of each element
COORD3=	#leave blank if not used
#	# comment line

Each gauge has a unique type number. Type numbers can be 1-9 or A-Z which restricts the total number of different gauges used in a definition file to 35. The software supports single gauges, tees and rosettes, both stacked and planar in configuration. Rectangular rosettes have included angles of 45 degrees between the elements. Gauges typically have some transverse sensitivity (sensitivity to transverse strain). The transverse sensitivity is typically defined as the ratio (%) of the transverse and longitudinal gauge factors. Some gauge types have up to three different transverse sensitivities. See the manufacturers information for more details. Since transverse sensitivities are rarely much more than 1%, in practice these make little difference to the results.

The Poisson's Ratio is the Poisson's ratio of the material on which the gauge was calibrated, not that of the material to which it was applied. This is typically 0.285.

Each Gauge, whether it be a single, tee or rosette, has its own coordinate system. The X-axis is always parallel to gauge 1 and the Z-axis is the outward surface normal. The three sets of coordinates allow the locations of the four corners of each gauge to be defined in the gauge's own coordinate system. The program expects eight numbers, comma separated, representing the X-Y coordinates of the corners of each element: X1,Y1,X2,Y2,X3,Y3,X4,Y4. If there are only one or two elements in the gauge. Leave the remaining lines blank.

The standard gauge definition file delivered with the software is given below in Figure 12-2 and Figure 12-3. The file contains eight gauges. Note that there is a small file header, and that each gauge is separated by a blank comment line.

```
#MSC.Fatigue GAUGE DEFINITION FILE
#VERSION 1.0
#
TYPE NUMBER=1
NAME=MM-125RD          #any string here
COMMENTS=3 ELEMENT RECTANGULAR PLANAR ROSETTE
GAUGE TYPE =3          #1=single, 2=tee, 3=rosette
S/P=2                  #0=NA, 1=stacked, 2=planar
CONFIGURATION=1        #0=NA, 1=rectangular, 2=delta, 3=other
KTS=                   #normally contains 1,2, or 3 numbers, comma separated
POISSON=0.285
UNITS=1                #1=mm, 2=m, 3=inches, 4=feet
COORD1=2.25,-0.785,5.43,-0.785,5.43,0.785,2.25,0.785
COORD2=1.68,0.57,3.927,2.817,2.817,3.927,0.57,1.68
COORD3=-0.785,2.25,-0.785,5.43,0.785,5.43,0.785,2.25
#
TYPE NUMBER=2
NAME=MM-125AC          #any string here
COMMENTS=GENERAL PURPOSE HIGH RESISTANCE GAUGE
GAUGE TYPE =1          #1=single, 2=tee, 3=rosette
S/P=0                  #0=NA, 1=stacked, 2=planar
CONFIGURATION=0          #0=NA, 1=rectangular, 2=delta, 3=other
KTS=                   #normally contains 1,2, or 3 numbers, comma separated
POISSON=0.285
UNITS=1                #1=mm, 2=m, 3=inches, 4=feet
COORD1=-1.59,-1.59,1.59,-1.59,1.59,1.59,-1.59,1.59
COORD2=
COORD3=
#
TYPE NUMBER=3
NAME=MM-125BZ            #any string here
COMMENTS=NARROW HIGH RESISTANCE GAUGE
GAUGE TYPE =1          #1=single, 2=tee, 3=rosette
S/P=0                  #0=NA, 1=stacked, 2=planar
CONFIGURATION=0        #0=NA, 1=rectangular, 2=delta, 3=other
KTS=                   #normally contains 1,2, or 3 numbers, comma separated
POISSON=0.285
UNITS=1                #1=mm, 2=m, 3=inches, 4=feet
COORD1=-1.59,-0.785,1.59,-0.785,1.59,0.785,-1.59,0.785
COORD2=
COORD3=
#
TYPE NUMBER=4
NAME=MM-120WR          #any string here
COMMENTS=RECTANGULAR STACKED ROSETTE
GAUGE TYPE =3          #1=single, 2=tee, 3=rosette
S/P=1                  #0=NA, 1=stacked, 2=planar
CONFIGURATION=1        #0=NA, 1=rectangular, 2=delta, 3=other
KTS=                   #normally contains 1,2, or 3 numbers, comma separated
POISSON=0.285
UNITS=1                #1=mm, 2=m, 3=inches, 4=feet
COORD1=-1.525,-1.015,1.525,-1.015,1.525,1.015,-1.525,1.015
COORD2=1.80,0.36,0.36,1.80,-1.80,-0.36,-0.36,-1.80
COORD3=-1.015,-1.525,-1.015,1.525,1.015,1.525,1.015,-1.525
#
```

Figure 12-2 Gauge Definition FIle

```
TYPE NUMBER=5
NAME=MM-062EN              #any string here
COMMENTS=GENERAL PURPOSE GAUGE
GAUGE TYPE =1              #1=single, 2=tee, 3=rosette
S/P=0                      #0=NA, 1=stacked, 2=planar
CONFIGURATION=0            #0=NA, 1=rectangular, 2=delta, 3=other
KTS=                       #normally contains 1,2, or 3 numbers, comma separated
POISSON=0.285
UNITS=1                    #1=mm, 2=m, 3=inches, 4=feet
COORD1=-0.785,-0.785,0.785,-0.785,0.785,0.785,-0.785,0.785
COORD2=
COORD3=
#
TYPE NUMBER=6
NAME=MM-125UW              #any string here
COMMENTS=GENERAL PURPOSE GAUGE
GAUGE TYPE =1              #1=single, 2=tee, 3=rosette
S/P=0                      #0=NA, 1=stacked, 2=planar
CONFIGURATION=0            #0=NA, 1=rectangular, 2=delta, 3=other
KTS=                       #normally contains 1,2, or 3 numbers, comma separated
POISSON=0.285
UNITS=1                    #1=mm, 2=m, 3=inches, 4=feet
COORD1=-1.59,-2.285,1.59,-2.285,1.59,2.285,-1.59,2.285
COORD2=
COORD3=
#
TYPE NUMBER=7
NAME=MM-120WY              #any string here
COMMENTS=60 DEGREE STACKED ROSETTE
GAUGE TYPE =3              #1=single, 2=tee, 3=rosette
S/P=1                      #0=NA, 1=stacked, 2=planar
CONFIGURATION=2            #0=NA, 1=rectangular, 2=delta, 3=other
KTS=                       #normally contains 1,2, or 3 numbers, comma separated
POISSON=0.285
UNITS=1                    #1=mm, 2=m, 3=inches, 4=feet
COORD1=-1.525,-1.015,1.525,-1.015,1.525,1.015,-1.525,1.015
COORD2=-1.64,-0.81,0.117,-1.83,1.64,0.81,-0.117,1.83
COORD3=-0.117,-1.83,1.64,-0.81,0.117,1.83,-1.64,0.81
#
TYPE NUMBER=8
NAME=MM-120WT
COMMENTS=2-ELEMENT 90 DEGREE TEE STACKED ROSETTE
GAUGE TYPE=2
S/P=0
CONFIGURATION = 0
KTS =
POISSON= 0.285
UNITS=1
COORD1= -1.525,-1.015,1.525,-1.015,1.525,1.015,-1.525,1.015
COORD2= -1.015,-1.525,1.015,-1.525,1.015,1.525,-1.015,1.525
COORD3=
```

Figure 12-3 Gauge Definition File (Continued)

Creating and Modifying Gauges

The gauge tool allows you to create, modify and delete software strain gauges from your model. The gauges are simply made up of rectangular shell elements placed on the finite element model in the orientation of the actual strain gauge as defined in the gauge definition file (`gauges.def`).

Creating a gauge is quite simple. Just follow these steps.

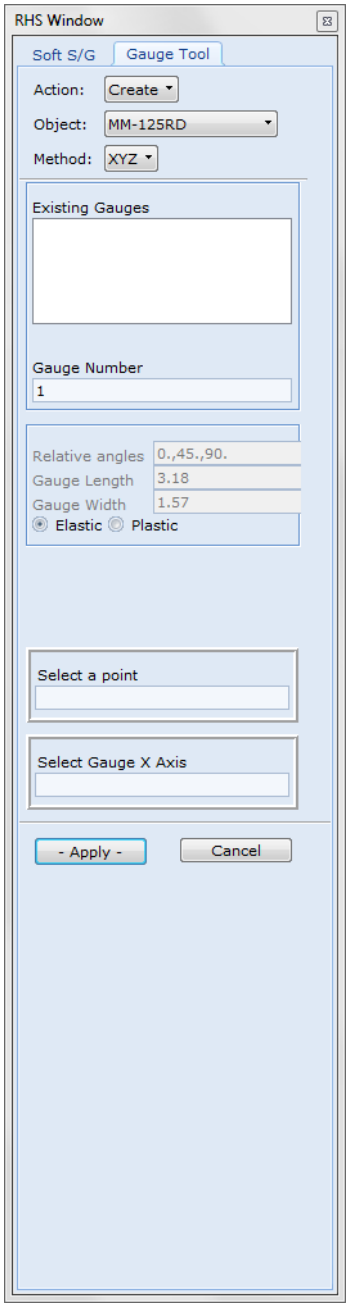


Figure 12-4 Create Software Strain Gauge Form

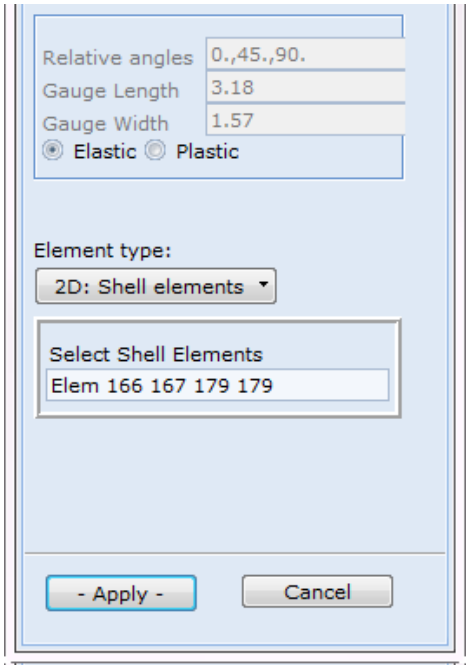


Figure 12-5 Create Software Strain Gauge Form (Continued)

The following hints and recommendations are made when trying to create gauges:

1. The elements selected to define the surface where the gauge will be placed must describe an area larger than the gauge footprint, otherwise the creation will fail and an error message will be issued.
2. Do not try to place gauges on top of "sharp" geometric features such as corners. The surface where the gauge is to be placed should have a radius of curvature which is large relative to the gauge dimensions.
3. Select as few elements as possible, but sufficient to define an area larger than the gauge. Selection of extra elements will result in longer creation times.

Note:	The gauge outward normal is calculated as the average outward normal of the element/faces selected.

Modify a Gauge

You may modify a gauge's position by translating and rotating it about the point where it is has been placed. The gauge is essentially recreated but you do not have to go through all the same steps you did to create the gauge with the exception of selecting some shell elements or solid element faces again.

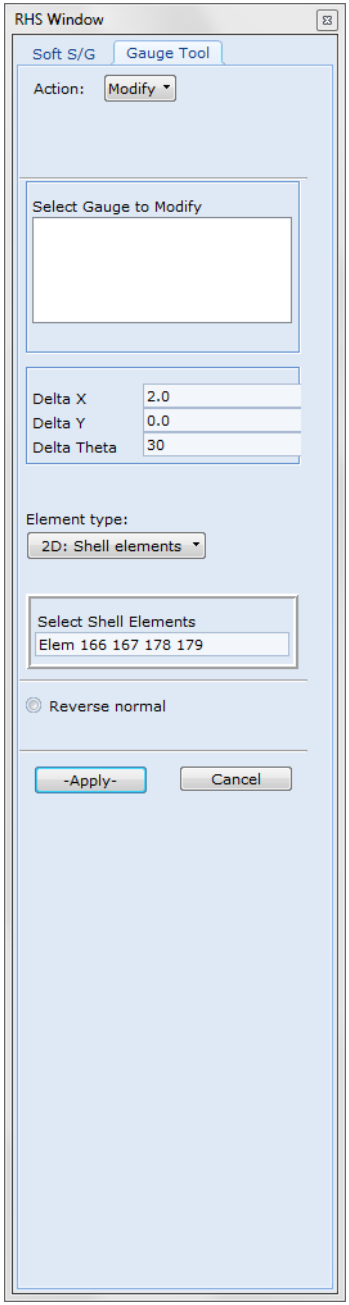


Figure 12-6 Modify a Gauge Form

Note:	Do not translate or rotate the gauge further that the surface you have defined by selecting the elements. Otherwise an error message will result.

Delete a Gauge

Deleting a gauge is simple. Simply select the gauge number and press the Apply button when the Action is set to Delete. The gauge, its elements, and the corresponding group will be deleted.

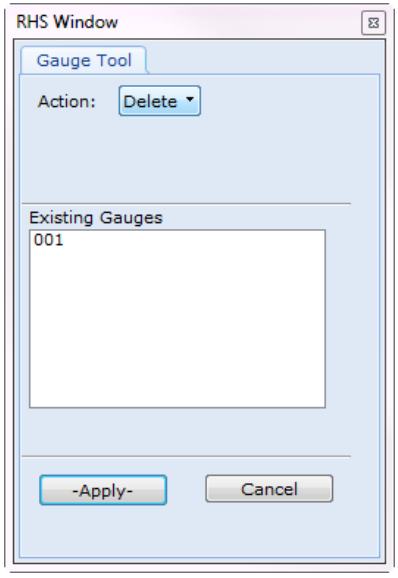


Figure 12-7 Delete a Gauge Form

Extracting the FE Results

The process of Creating and Modifying Gauges described in the previous section results in the generation of a number of new groups in the database, one for each strain gauge.

Each group represents a gauge, which may have 1, 2 or 3 elements. The group names take the form:

dms_n_m_oo_ppp

- **dms** is the German abbreviation for strain gauge
- **n** is **t** or **b** indicating whether the results set is extracted from the top or bottom of underlying shell elements making up the software strain gauge
- **m** is the gauge type number (its unique type identifier)
- **oo** is either **el** or **ep** indicating elastic or elastic-plastic
- **ppp** is the number of the gauge applied to the model, e.g. 001, 002, 003 etc.

Two new result types are created after the results extraction is complete for each stress analysis load case selected. Under each selected set of load case results, the two new subcases are:

Gauge Stress, Average

and

Gauge Stress, Centroidal

The results in the "Gauge Stress, Average" subcase are, for each element, the average of the results from the four corners of the element and the element centroid. For "Gauge Stress, Centroidal", they are the results from the origin of the gauge coordinate system.

For a rosette or tee gauge, the Average results will in general be a little different for each element, depending on the stress field in which the gauge is placed.

For Centroidal results, the same results should be written to all the gauge elements.

In all cases the gauge results are written to both the top and bottom surfaces of the gauge elements.

To actually extract the results the following form is used when you press the Results Extraction button.

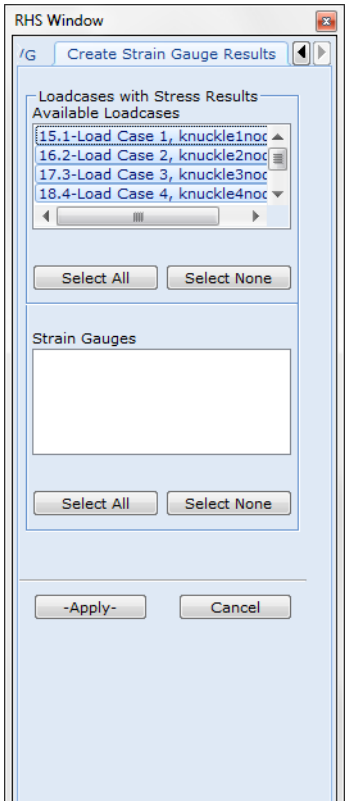


Figure 12-8 Extracting Results Form

The following notes are made to extract results successfully:

1. The results of the extraction are two new result types for each selected results case as explained above. When setting up the fatigue input file in the next section, you will reference one of these newly created result types.
2. Results extraction only works on MSC.Nastran Stress Tensor results. If these results do not exist the extraction will fail.
3. For each load case selected which contains a stress tensor result, an average and centroidal result is calculated. The result is obtained by interpolating the 3D stress field to provide a tensor value at each gauge location (corner and center). The tensor is then converted to the gauge coordinate system.

Creating a Fatigue Input File

The next step in the process is to prepare to run the MSC Fatigue Software Strain Gauge Module SSG which requires only a `.fes` input file. This file is created by running the PAT3FAT or FATTRANS translator, so a full MSC Fatigue analysis is not necessary. The manner in which the MSC Fatigue job should be set up and

run is briefly described here. A more detailed description can be found in Using MSC Fatigue - The Classic Modules (Ch. 3) and Total Life and Crack Initiation (Ch. 6).

Main Form

On the main form (Figure 12-1) make the following selections:

1. Analysis: Crack Initiation
2. Results Location: Element
3. F.E. Results: Stress
4. Results Units: Whatever they are (usually MPa)
5. Jobname:
6. Title:

Solution Parameters Form

1. Analysis Method: Any (this selection is ignored)
2. Plasticity Correction: Choose from Neuber, Mertens-Dittmann or Seeger-Beste methods according to which elastic-plastic correction method you wish to use in the software strain gauge. If you select Mertens-Dittmann or Seeger-Beste you will be required to enter shape factors on the material information form. Analysis for any gauges which are elastic will ignore this selection. If you want to estimate elastic-plastic strains, you should ensure that the gauges have "ep" included in the group name.
3. Run Biaxiality Analysis: This should be set to "on". There will be no need to execute the "Calculate Normals" option because the stress analysis results written to the gauges are already in a surface resolved coordinate system.
4. Biaxiality Correction: Select the option that you want to use in the analysis of your elastic-plastic gauges. Select either "None" or "Hoffmann-Seeger"; "Material Parameter" is not allowed. Analysis for any gauges which are elastic will ignore this selection.
5. The rest of the information on the Solution Parameters form is ignored, so it is alright to accept the defaults.

Materials Information Form

The materials information form is used to assign material and other information to the individual gauges. The strain gauge software requires that each gauge be a group, consisting of 1 shell element for each leg. Valid group names take the form "dms_m_n_oo_pp" as described previously.

The strain gauge software will assume for a tee gauge or a rosette that gauges 1-3 or 1-3 are in numerical order of elements. Gauges are numbered in an anticlockwise direction. The following figure is a typical planar rosette. It is a rectangular rosette, meaning that the elements are 45 degrees apart. Note the numbering of the gauge elements, and the gauge coordinate system.

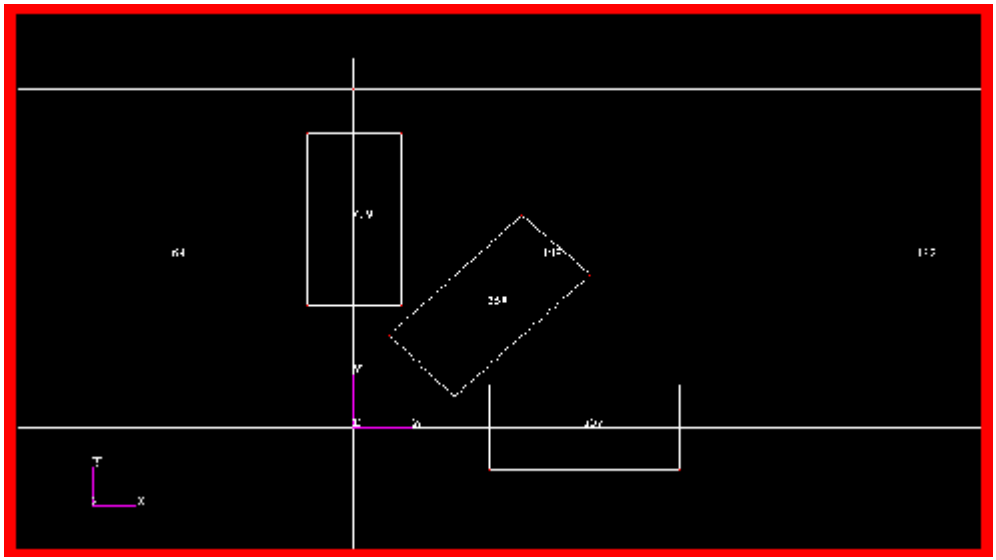


Figure 12-9 A Planar Rosette Software Strain Gauge

The Materials Information form should then be filled in as follows:

1. Number of Materials: This should be set to the number of strain gauges that are to be processed. There is a limit of 100. If more that 100 are to be processed you will have to break the analysis up into multiple analyses.
2. Then, on the spreadsheet, make one line for each gauge or rosette.
3. Material: This is the material on which the gauge is positioned.
4. Finish and Treatment: These are not used, so any setting is acceptable.
5. Region: Select here the name of the group defining each gauge.
6. Kf: This is a surface finish correction factor and is not used by the strain gauge software.
7. Shape factor: This is the shape factor (Formzahl) or plastic strain concentration factor required for the Mertens-Dittmann and Seeger-Beste methods. Valid values are greater than 1. Zero (0) can also be used, and this is interpreted as infinity. In this case both methods reduce to the Neuber method.
8. Multiplier and Offset: These should not be used.

Loading Information Form

The loading information form should be used in the same way as for any other MSC Fatigue job, with two restrictions: it should only be used for results from the database, and there should be no results transformation.

The results can be from a transient analysis (time step analysis) or from a set of static load cases which will be associated to time variations in the normal way (which are defined using PTIME).

When selecting actual results, the user should choose one of the following result types for the extracted strain gauge results:

1. Gauge Stress, Average (for the stresses averaged from the four corners and the centroid of each strain gauge element),

   or

2. Gauge Stress, Centroidal (for the stresses at the centroid of each gauge element).

If neither of these exist, it means you haven't extracted results to the software strain gauges from the FE model.

Job Control

On the job control form, the action "Translate Only" should be selected and applied. This will save the MSC Fatigue job (`.fin`) file and run the PAT3FAT or FATTRANS translator to produce the intermediate (`.fes`) file required by SSA. There is of course no reason why you should not run a full analysis, but there is nothing to be gained from this, unless you are using the software strain gauge to support a BS7608 welded analysis.

When the translation is complete you are now ready to run SSG.

Software Strain Gauge Module (SSG)

This section explains the operation of SSG to extract stress or strain histories once soft gauges have been created, the FE results extracted and an appropriate fatigue input file (`jobname.fes`) has been created.

The program SSG (Software Strain-Gauge) can be started by clicking on the "SSG Analysis" item on the Soft S/G menu, or from the system prompt (by typing **ssg**) or by including a suitable line into a batch script.

Interactive Operation

When run interactively, SSG may be used with a Motif or Mask interface, according to user preference. The figures used here depict the Motif interface. This interface works according to the standard of all other MSC Fatigue modules as explained in Module Operations (Ch. 18). When the program is started, the screens appears as shown below for the Motif interface.

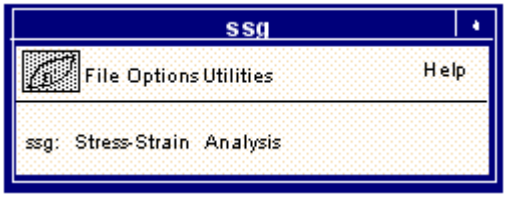


Figure 12-10 The SSG Utilities Menu

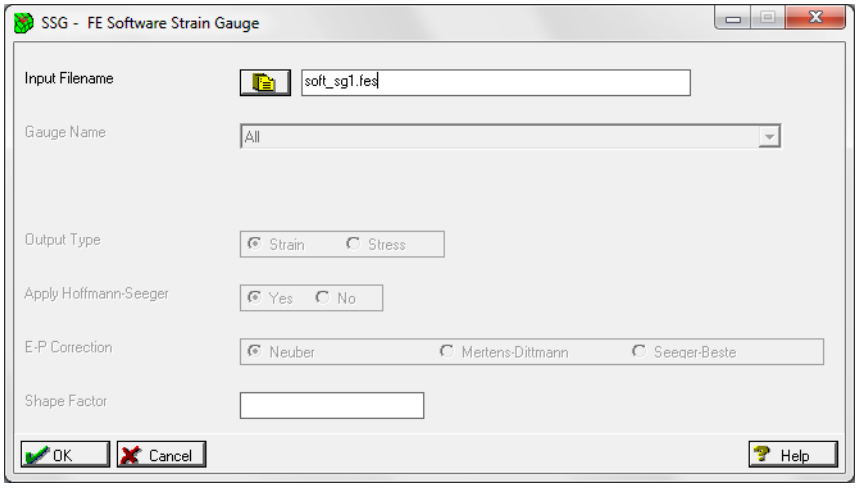


Figure 12-11 Software Strain Gauge (SSG) Main Form

Enter the name of the input (`.fes`) file, or select one using the List option. Once an input file is selected in Figure 12-11 the rest of the form becomes active.

The rest of the fields are described below:

Field	Description
Gauge Name	Any one gauge can be selected, or the default, which is ALL. If the default is selected, all the gauges included in the `.fes` file will be processed, according to the information set up on the Strain Gauge Creation and MSC Fatigue forms. The default analysis route is to calculate the strain histories for all the gauges, giving either elastic histories (el) or elastic-plastic histories (ep) estimated using the notch correction procedure specified in the MSC Fatigue job file. Each gauge will produce strain histories with names: `<jobname>xxxyy.dac` where `xxx` is the gauge number, and `yy` is the channel number, referring to the gauge element (001, 002 or 003).
Gauge Type	Alternatively, the user may select only one of the gauges from the toggle list. There are then two possibilities: ■ If the strain gauge is specified as an elastic gauge, i.e. the gauge name includes "el", for example "dms_t_1_el_057", the gauge type will be displayed, and the user is prompted to ask for either elastic stress or elastic strain. ■ If the strain gauge is specified as an elastic-plastic gauge, i.e. the gauge name includes "ep", for example "dms_t_3_ep_135" the gauge type will be displayed, and the user is prompted to ask for either stress or strain, followed by all the other available elastic-plastic options.
Output Type	Select either stress or strain, according to whether you need stress output files or strain output files.
Apply Hoffmann-Seeger	If Hoffmann Seeger is set to "No", each strain gauge leg will be treated as if it is in a uniaxial stress field, i.e. no correction will be made for the state of biaxiality. If "Yes" is selected, the Hoffmann-Seeger method will be applied, treating each gauge leg independently, and assuming proportional loading, i.e. that the biaxiality ratio is constant (the mean value is used) and the orientation of principal stress axes is fixed (orientation of absolute maximum principal is taken from the most popular bin).
E-P Correction	If either the Mertens-Dittmann or the Seeger Beste method is selected, the user is prompted for a shape factor which will be applied to all three gauge legs.
Shape Factor	The shape factor to be applied for E-P Correction.

The result of running the software will be a number of `.dac` files with the naming convention "`jobnameyyyzz.dac`" where `yyy`=gauge i.d. and `zz`=gauge leg (e.g. 01, 02, and 03 for a rosette). These can be post-processed using any appropriate MSC Fatigue module.

Technical Details

Elastic Strain and Stress Calculation

The elastic strains and elastic stresses output from the strain gauge are the direct strains and stresses parallel to the axes of gauges 1-3, calculated using the full 3D stress tensor, the Young's modulus E and the Poisson's ration ν.

Elastic-plastic Strain and Stress Calculation

Elastic-plastic strains and stresses are calculated using either the Neuber method or a modified version of the Hoffmann-Seeger method. Both these methods can also be combined with the Mertens-Dittmann or Seeger-Beste methods.

If the Neuber method is chosen (i.e. Hoffmann-Seeger = No), the elastic-plastic stresses and strains will be calculated using the Neuber method, based on the elastic value of the direct strain, and taking no account of the multiaxial state of stress - i.e. using the uniaxial material properties.

If the Hoffmann-Seeger method is selected, the mean value of the biaxiality ratio and the most popular orientation of the absolute maximum principal stress are calculated for each gauge in exactly the same way as they are in MSC Fatigue biaxiality analysis. An equivalent (von Mises) elastic strain history is then calculated on the basis of the direct strain along the axis of the gauge, the mean biaxiality ratio and the most popular angle, assuming that the ratio and angle are constant, i.e. proportional loading.

The Neuber correction is then carried out on the equivalent strain, and the angle and biaxiality ratio are used again to predict the direct stress and strain in the gauge. This process is only really valid for proportional loadings, but the implementation does not preclude its use for non-proportional loadings. There are a number of philosophical problems with using this method for non-proportional loadings, e.g. that the equivalent stress may be different for each gauge, and also be a function of gauge orientation. In the near future a non-proportional notch correction procedure will be implemented in this program, but this is not sufficiently well validated for inclusion at this stage.

Correlation with Test

Once the strain histories have been generated from the FE model, they may readily be compared with the corresponding measured strains from the real component. Here are a few possible methods for comparing them:

- Using the Multi-File Display module MMFD to overlay or cross-plot the data
- By comparing the statistics of the signals, max, min, RMS etc.
- Using the strain gauge rosette analysis option in MSSA

Correlation is a very important aspect of reliable durability calculations. If a correlation exercise indicates that there is poor qualitative and quantitative correlation between predicted and measured strain histories, any fatigue calculations are also likely to give poor results. Likely causes of poor correlation are:

1. Errors in setting up the MSC Fatigue job, particularly in matching the correct channels to the correct load cases with the correct scaling factors
2. Errors in calculating the loading histories

3. Poor definition of the loads and boundary conditions, or missing loads
4. Inadequate meshing
5. Inaccurate strain gauge placement
6. Inappropriate analysis (e.g. quasi-static when the problem is dynamic)
7. Poor materials
8. Non-Proportional Loadings together with high levels of plasticity.

SSG Batch Operation

The program can also be operated in batch mode, in the same way as any other MSC Fatigue modules such as FEFAT. The syntax required of each batch line is:

```
<program name> /<keyword>=<value>/<keyword>=..... etc.
```

or

```
<program name> @<filename> /<keyword>=<value> etc.
```

where <filename> is the name of a file containing keyword/value pairs, with one pair to a line:

```
/<keyword>=<value>
/<keyword>=<value>
```

etc.

If a keyword/value pair is not specified, the default value for that keyword will be used. A typical batch line might be:

```
ssg /inp=test.fes/gauge=dms_t_1_ep_003/hofseg=y/ov=y/\*=tt
```

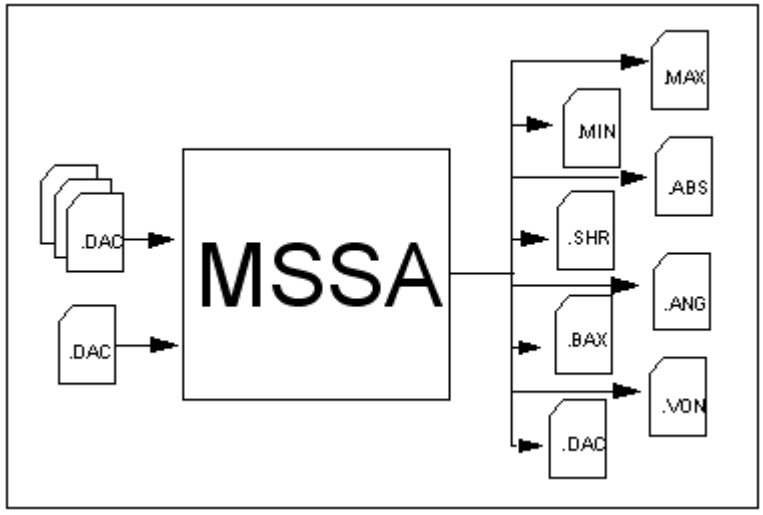
meaning run SSG with an input file = `test.fes`, use gauge dms_1_ep_003, use the Hoffmann-Seeger method, overwrite any existing files, echo output to the screen and default the rest of the questions.

The batch keywords, meanings, and possible values relevant to the Software Strain Gauge are (defaults in bold):

/INP	Input filename: `jobname.fes`
/GAUGE	Software strain gauge i.d.: e.g. dms_t_3_ep_047 or "all"
/OTYPe	Output data type: Stress or Strain
/HOFSEG	Use Hoffmann-Seeger method: Y, N
/EPTYPE	Elastic-Plastic Correction: Neuber, Mertens-Dittmann, Seeger-Beste
/SHAPe	Shape factor, a number - 0 or 1 < n < *
/OV	Overwrite: Y,N
<*>	Output to tt (screen), <filename>, none

Stress-Strain Analysis (MSSA)

MSSA processes rosette data and creates outputs suitable for use by either the stress or strain-life fatigue analyzers. It also provides an indication of the state of multiaxiality present and suggests possible processing routes through the fatigue analyzers.



Additionally, MSSA can be used to convert elastic-plastic strain records, measured on one material, to that of another material. It can also convert elastic-plastic strain records to equivalent fully elastic ones and visa-versa.

MSSA processes data from either rectangular, delta or TEE rosettes. In addition, it can transform elastic-plastic strain records, measured on one material, to that of another material. It can also convert elastic-plastic strain records to equivalent fully elastic ones and vise-versa.

Strain Gauge Rosettes

Nomenclature for strain gauge rosette analysis:

Geometrically different, but functionality equivalent configurations of rectangular and delta rosettes are shown in Figure 12-12 and Figure 12-13.

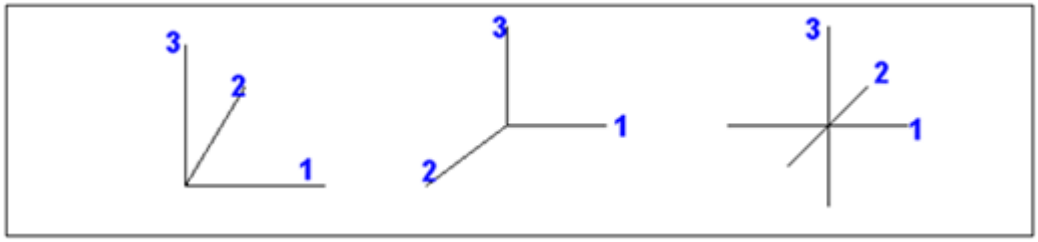


Figure 12-12 Rectangular Rosettes

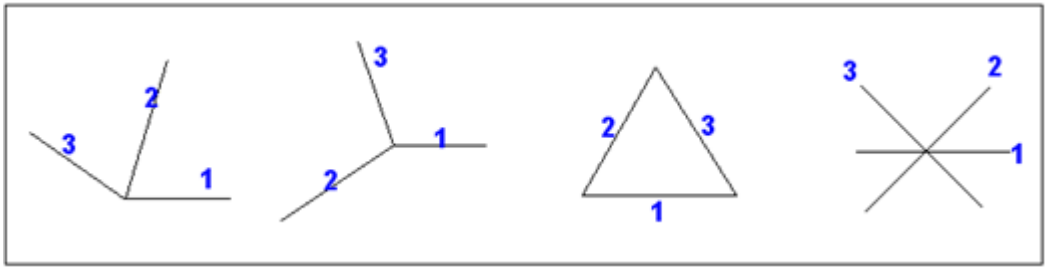


Figure 12-13 Delta Rosettes

By convention gauge numbering, 1,2,3 is counter-clockwise.

Rectangular Rosette

ε_1	First strain gauge
ε_2	Strain gauge at 45 to ε_1
ε_3	Strain gauge at 90 to ε_1

Delta Rosette

ε_1	First strain gauge
ε_2	Strain gauge at 60 to ε_1
ε_3	Strain gauge at 120 to ε_1

Transverse Sensitivity

ε_1^1	Gauge (1) strain corrected for Poisson's strain
ε_2^1	Gauge (2) strain corrected for Poisson's strain
ε_3^1	Gauge (3) strain corrected for Poisson's strain
Kt	Transverse sensitivity factor
$\mathbf{Kt_1}$	Transverse sensitivity factor for gauge (1)

$\mathbf{Kt_2}$	Transverse sensitivity factor for gauge (2)
Kt_3	Transverse sensitivity factor for gauge (3)
υ	Poisson's ratio
ε_p	Maximum principal strain
ε_q	Minimum principal strain
γ **max**	Maximum shear strain
ϕ **p**	The angle from grid 1 to the max. principal
ϕ **pq**	The angle from grid 1 to either the max. or min. principal
σ **p**	Maximum principal stress
σ **q**	Minimum principal stress
τ **max**	Maximum shear stress
σ **vm**	Von Mises stress

Calculation of Principal Strains & Stresses

The following equations are used in the calculation of principal stress and strain from the component strains.

Rectangular Rosette

$$\varepsilon_{pq} = \frac{\varepsilon_1 + \varepsilon_3}{2} \pm \frac{1}{\sqrt{2}}\sqrt{\{(\varepsilon_1 - \varepsilon_2)^2 + (\varepsilon_2 - \varepsilon_3)^2\}} \tag{12-1}$$

$$\varepsilon_{pq} = \frac{E}{2}\left(\frac{\varepsilon_1 + \varepsilon_3}{1 - \upsilon} \pm \frac{1}{1 + \upsilon}\sqrt{\{(\varepsilon_1 - \varepsilon_2)^2 + (\varepsilon_2 - \varepsilon_3)^2\}}\right) \tag{12-2}$$

Delta Rosette

$$\varepsilon_{pq} = \frac{\varepsilon_1 + \varepsilon_2 + \varepsilon_3}{3} \pm \frac{\sqrt{2}}{3}\sqrt{\{(\varepsilon_1 + \varepsilon_2)^2 + (\varepsilon_2 + \varepsilon_3)^2 + (\varepsilon_3 + \varepsilon_1)^2\}} \tag{12-3}$$

$$\varepsilon_{pq} = \frac{E}{3}\left(\frac{\varepsilon_1 + \varepsilon_3 + \varepsilon_3}{1 - \upsilon} \pm \frac{\sqrt{2}}{1 + \upsilon}\sqrt{\{(\varepsilon_1 - \varepsilon_2)^2 + (\varepsilon_2 - \varepsilon_3)^2 + (\varepsilon_1 - \varepsilon_3)^2\}}\right) \tag{12-4}$$

Tee Rosette

$$\varepsilon_p = \varepsilon_1 \text{ and } \varepsilon_q = \varepsilon_2 \tag{12-5}$$

$$\sigma_p = \frac{E}{1 - \upsilon^2}(\varepsilon_1 + \upsilon\varepsilon_2) \tag{12-6}$$

$$\sigma_q = \frac{E}{1 - \upsilon^2}(\varepsilon_2 + \upsilon\varepsilon_1) \tag{12-7}$$

Maximum Shear Strain

$$\gamma_{\max} = \varepsilon_p - \varepsilon_q \tag{12-8}$$

Maximum Shear Stress

$$\tau_{\max} = \frac{\sigma_p - \sigma_q}{2} \tag{12-9}$$

Von Mises Stress (under conditions of plane stress)

$$\sigma_{VM} = \sqrt{\sigma_p^{\,2} - \sigma_p \sigma_q + \sigma_q^{\,2}} \tag{12-10}$$

Angle Between Grid 1 and the Maximum Principal Strain

Rectangular Rosette

$$\phi_{pq} = \frac{1}{2}\tan^{-1}\left[\frac{2\varepsilon_2 - \varepsilon_1 - \varepsilon_3}{\varepsilon_1 - \varepsilon_3}\right] \tag{12-11}$$

Delta Rosette

$$\phi_{pq} = \frac{1}{2}\tan^{-1}\left[\frac{\sqrt{3}(\varepsilon_2 - \varepsilon_3)}{2\varepsilon_1 - \varepsilon_2 - \varepsilon_3}\right] \tag{12-12}$$

Corrections for Transverse Sensitivity

Stacked Tee Rosette

$$\varepsilon_1' = \frac{(1 - \upsilon_0 K_t)(\varepsilon_1 - K_t \varepsilon_2)}{1 - K_t^{\,2}} \tag{12-13}$$

$$\varepsilon_2' = \frac{(1 - \upsilon_0 K_t)(\varepsilon_2 - K_t \varepsilon_1)}{1 - K_t^{\,2}} \tag{12-14}$$

Planar Tee Rosette

$$\varepsilon_1' = \frac{\varepsilon_1(1 - \upsilon_0 K_{t1}) - K_{t1}\varepsilon_2(1 - \upsilon_0 K t_2)}{1 - K t_1 K t_2} \tag{12-15}$$

$$\varepsilon_2' = \frac{\varepsilon_1(1 - \upsilon_0 K_{t2}) - K_{t2}\varepsilon_1(1 - \upsilon_0 K t_1)}{1 - K t_1 K t_2} \tag{12-16}$$

Stacked Rectangular Rosette

$$K_{t1} = K_{t2} = K_{t3} = K_t \tag{12-17}$$

$$\varepsilon_1' = \frac{1 - \upsilon_0 K_t}{1 - K_t^2}(\varepsilon_1 - K_t\varepsilon_3) \tag{12-18}$$

$$\varepsilon_2' = \frac{1 - \upsilon_0 K_t}{1 - K_t^2}[\varepsilon_2 - K_t(\varepsilon_1 + \varepsilon_3 - \varepsilon_2)] \tag{12-19}$$

$$\varepsilon_3' = \frac{1 - \upsilon_0 K_t}{1 - K_t^2}(\varepsilon_3 - K_t\varepsilon_1) \tag{12-20}$$

Planar Rectangular Rosette

When the transverse sensitivities of the orthogonal gages (1) and (3) are nominally the same.

$$K_{t1} = K_{t2} = K_{t13} \tag{12-21}$$

$$\varepsilon_1' = \frac{1 - \upsilon_0 K_{t13}}{1 - K_{t13}^2}(\varepsilon_1 - K_{t13}\varepsilon_3) \tag{12-22}$$

$$\varepsilon_2' = \frac{(1 - \upsilon_0 K_{t1})(1 + K_{t13})\varepsilon_2 - K_{t1}(1 - \upsilon_0 K_{t13})(\varepsilon_1 + \varepsilon_3)}{(1 + K_{t13})(1 - K_{t2})} \tag{12-23}$$

$$\varepsilon_3' = \frac{1 - \upsilon_0 K_{t13}}{1 - K_{t13}^2}(\varepsilon_3 - K_{t13}\varepsilon_1) \tag{12-24}$$

Rectangular Rosette

In the case where all three transverse sensitivities are dissimilar.

$$\varepsilon_1' = \frac{\varepsilon_1(1 - \upsilon_0 K_{t1}) - K_{t1}\varepsilon_3(1 - \upsilon_0 Kt_3)}{1 - Kt_1 Kt_3} \tag{12-25}$$

$$\varepsilon_2' = \frac{\varepsilon_2(1 - \upsilon_0 K_{t2})}{1 - K_{t2}} - \frac{K_{t2}[\varepsilon_1(1 - \upsilon_0 K_{t1})(1 - K_{t3}) + \varepsilon_3(1 - \upsilon_0 Kt_3)(1 - K_{t1})]}{(1 - Kt_1 Kt_3)(1 - Kt_2)} \tag{12-26}$$

$$\varepsilon_3' = \frac{\varepsilon_3(1 - \upsilon_0 K_{t3}) - K_{t3}\varepsilon_1(1 - \upsilon_0 Kt_1)}{1 - Kt_1 Kt_3} \tag{12-27}$$

Stacked Delta Rosette

In this case $K_{t1} = K_{t2} = K_{t3} = K_t$ (12-28)

$$\varepsilon_1' = \frac{1 - \upsilon_0 K_t}{1 - K_t^2}\left[\left(1 - \frac{K_t}{3}\right)\varepsilon_1 + \frac{2}{3}K_t(\varepsilon_2 + \varepsilon_3)\right] \tag{12-29}$$

$$\varepsilon_2' = \frac{1 - \upsilon_0 K_t}{1 - K_t^2}\left[\left(1 - \frac{K_t}{3}\right)\varepsilon_2 + \frac{2}{3}K_t(\varepsilon_3 + \varepsilon_1)\right] \tag{12-30}$$

$$\varepsilon_3' = \frac{1 - \upsilon_0 K_t}{1 - K_t^2}\left[\left(1 - \frac{K_t}{3}\right)\varepsilon_3 + \frac{2}{3}K_t(\varepsilon_1 + \varepsilon_2)\right] \tag{12-31}$$

Planar Delta Rosette

When the transverse sensitivity of two of the gauges, (1) and (3) are nominally the same

$$K_{t1} = K_{t3} = K_{t13} \tag{12-32}$$

$$\varepsilon_1' = \frac{\varepsilon_1(1-\upsilon_0 K_{t13})(3-K_{t2}-K_{t13}-K_{t13}K_{t2}) - 2K_{t13}[\varepsilon_2(1-2K_{t2})(1-\nu_0 K_{t13}) + \varepsilon_3(1-\upsilon_0 K_{t13})(1-K_{t2})]}{2K_{t13}^2 K_{t2} - 2K_{t13}^2 - 2K_{t13}K_{t2} - 2K_{t13} - K_{t2} + 3}$$

(12-33)

$$\varepsilon_2' = \frac{\varepsilon_2(1-\upsilon_0 K_{t2})(3+K_{t13}) - 2K_{t2}[(\varepsilon_1-\varepsilon_3)(1-\nu_0 K_{t13})]}{K_{t13} - 3K_{t13}K_{t2} - K_{t2} + 3} \tag{12-34}$$

$$\varepsilon_3' = \frac{\varepsilon_1(1-\upsilon_0 K_{t13})(3-K_{t2}-K_{t13}-K_{t13}K_{t2}) - 2K_{t13}[\varepsilon_1(1-\nu_0 K_{t13})(1-K_{t2}) + \varepsilon_2(1-\nu_0 K_{t2})(1-K_{t13})]}{3K_{t13}^2 K_{t2} - 2K_{t13}^2 - 2K_{t13}K_{t2} - 2K_{t13} - K_{t2} + 3}$$

(12-35)

Delta Rosette

In the case where all three transverse sensitivities are dissimilar.

$$\varepsilon_1' = \frac{\varepsilon_1(1-\upsilon_0 K_{t1})(3-K_{t2}-K_{t3}-K_{t2}K_{t3}) - 2K_{t1}[\varepsilon_1(1-\nu_0 K_{t2})(1-K_{t3}) + \varepsilon_3(1-\nu_0 K_{t3})(1-K_{t2})]}{3K_{t1}K_{t2}K_{t3} - K_{t1}K_{t2} - K_{t2}K_{t3} - K_{t1}K_{t3} - K_{t1} - K_{t2} - K_{t3} + 3}$$

(12-36)

$$\varepsilon_2' = \frac{\varepsilon_2(1-\upsilon_0 K_{t2})(3-K_{t3}-K_{t1}-K_{t3}K_{t1}) - 2K_{t2}[\varepsilon_1(1-\nu_0 K_{t3})(1-K_{t1}) + \varepsilon_1(1-\nu_0 K_{t1})(1-K_{t3})]}{3K_{t1}K_{t2}K_{t3} - K_{t1}K_{t2} - K_{t2}K_{t3} - K_{t1}K_{t3} - K_{t1} - K_{t2} - K_{t3} + 3}$$

(12-37)

$$\varepsilon_3' = \frac{\varepsilon_3(1-\upsilon_0 K_{t3})(3-K_{t1}-K_{t2}-K_{t1}K_{t2}) - 2K_{t3}[\varepsilon_1(1-\nu_0 K_{t1})(1-K_{t2}) + \varepsilon_2(1-\nu_0 K_{t2})(1-K_{t1})]}{3K_{t1}K_{t2}K_{t3} - K_{t1}K_{t2} - K_{t2}K_{t3} - K_{t1}K_{t3} - K_{t1} - K_{t2} - K_{t3} + 3}$$

(12-38)

Elastic-plastic to Elastic Conversion

Using successive elastic-plastic strain ranges, $\Delta\varepsilon$, the corresponding stress range, $\Delta\sigma$ is calculated from either,

$$\varepsilon = \frac{\sigma}{E} + \left(\frac{\sigma}{K'}\right)^{1/n'} \tag{12-39}$$

or,

$$\Delta\varepsilon = \frac{\Delta\sigma}{E} + 2\left(\frac{\sigma}{2K'}\right)^{1/n'} \tag{12-40}$$

The equivalent elastic strain range, Δe, is calculated from

$$\Delta e = \sqrt{\left\{\frac{\Delta\sigma\Delta\varepsilon}{EK_t^2}\right\}} \tag{12-41}$$

Elastic to Elastic-plastic Conversion

Using successive elastic strain ranges, the corresponding equivalent elastic-plastic stress ranges are calculated from either,

$$Ee^2K_t^2 = \frac{\sigma}{E} + \left(\frac{\sigma}{K'}\right)^{1/n'} \tag{12-42}$$

or,

$$E\Delta e^2K_t^2 = \frac{\Delta\sigma}{E} + \left(\frac{\sigma}{K'}\right)^{1/n'} \tag{12-43}$$

The equivalent elastic-plastic strain range, epsilon>, is calculated from

$$\Delta\varepsilon = \frac{E\Delta e^2K_t^2}{\Delta\sigma} \tag{12-44}$$

Elastic-plastic to Elastic-plastic Conversion

This conversion uses the procedures described above to first convert the elastic-plastic strains to fully elastic ones using the first set of material properties, and then to convert these elastic strains to elastic-plastic strains using the second set of material properties.

MSSA Module Operation

The MSSA module can be run in one of the following three modes:

- From the MSC Fatigue menu driven system.
- In stand alone mode by typing **mssa** at the system prompt.
- By incorporating the MSSA commands in a batch or macro operation

The first two modes are interactive. Once running in interactive mode, the MSSA module will display the following main menu screen.

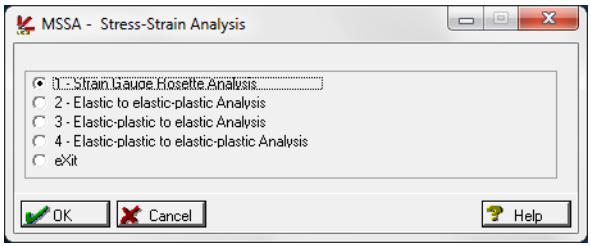


Figure 12-14 The First MSSA Screen

Select option 1 of this menu to set up rosette calculations, and the other options are for strain conversions. See the technical overview for a definition of these terms.

- Note that if rosette calculations had been previously carried out, then it is possible to go straight to the postprocessing menu (see Figure 12-18) via the option 1 sub menu - Analyze or Display. The following text assumes that an analysis is being carried out.
- Note also that Strain Gauge Rosette Analysis = Display uses graphics modules MMFD and MQLD to display plots

Each menu option will be explained, starting with option 1 - Strain Gauge Rosette Analysis.

1 - Strain Gauge Rosette Analysis

When option 1 is selected, the following screen is displayed. On it the user must define the type of rosette calculation to be undertaken.

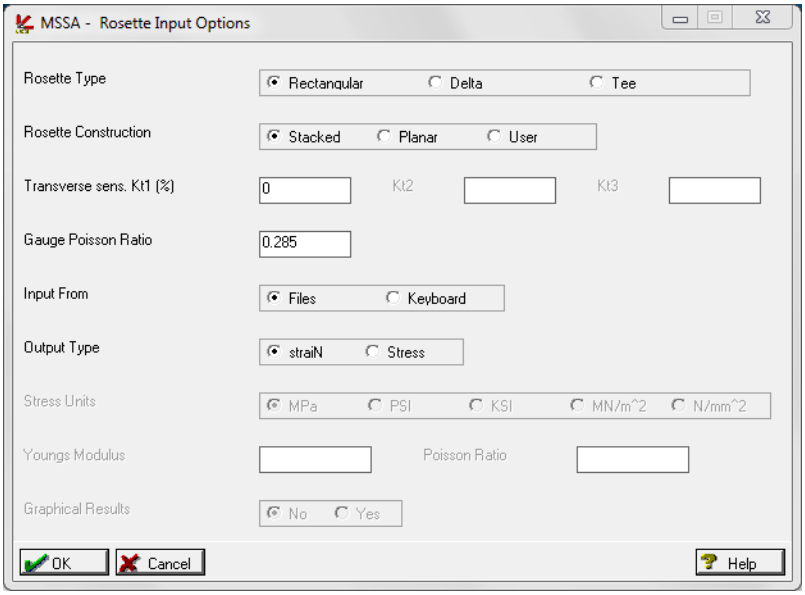


Figure 12-15 Specifying the Rosette Calculation

The fields are as follows:

Field	Description
Rosette Type	MSSA supports three types of rosette configuration: rectangular, delta and tee. Tee rosettes consist of two mutually perpendicular grids. Rectangular rosettes have three grids, with the second and third grids angularly displaced from the first grid by 45 and 90 degrees, respectively. Delta rosettes have three grids, with the second and third grids 60 and 120 degrees away, respectively, from the first grid.
Rosette Construction	Strain gauge rosettes are manufactured in two forms: planar, and stacked. Stacked rosettes have the three grids layered on top of each other. Planar rosettes have all three grids lying in the same plane. All the gauges in a stacked rosette have the same gauge factor and transverse sensitivity whilst the grids of a planar rosette will have slightly differing values of these properties. In the latter case, the transverse sensitivities of grids 1 and 3 are generally the same and grid 2 is different. The 'User' field allows for the case when all three transverse sensitivities are different. The rosette manufacturer's data sheet should indicate the type of rosette that the analysis is being set up for. The number of transverse sensitivities (see below) depends upon the choice made in this field.
Transverse Sens. Kt1 (%)	The transverse sensitivity coefficient, Kt, is defined as the ratio of the transverse and axial gauge factors. It is used to correct measured strains for the effects of transverse Poisson's strains. It should not to be confused with the elastic stress concentration factor of the same name. The number of values to be entered is: ■ One Stacked Rosette Configuration ■ Two Planar Rosette Configuration ■ Three User Defined Rosette Configuration Enter the value of transverse sensitivity coefficient as a percentage. Please refer to the rosette manufacturer's data sheet for the value(s) of this factor.

Field	Description
Gauge Poisson Ratio	Errors due to transverse sensitivity and the effect of Poisson's strains in the calculation of stress from strain in anything other than a uniaxial field, may be corrected by using the Poisson's ratio of the material on which the gauge or manufacturer's factor was measured. For steel this value is usually, 0.285. Please refer to the rosette manufacturer's data sheet for the required value if different to the default.
Input From Files/Keyboard	MSSA can calculate principals, shears, and angles from strains measured by rosettes. These measured strains may be submitted as discrete values entered from the keyboard, or from data files containing any number of values. Select whether strains are to be entered manually, or read from a disc file. Note that entering strains manually will allow a Mohr's circle representation of the principal strains, or stresses to be displayed as shown in Figure 12-16. **Note:** The Keyboard option requires the user to specify the stress units, and then type the micro-strain values from the rosettes. MSSA will calculate the principles, angles, and shears in the normal way.
Output Type	This field only appears if Input From is set to Files. MSSA uses rosette strains to calculate principals and shears in terms of either stress or strain. The conversion from strain to stress values takes into account biaxiality caused by significant Poisson's strains.
Stress Units	MSSA uses rosette strains to calculate principals and shears in terms of either stress or strain. The stresses can be calculated in a number of different unit systems. The conversion MPa = 6.895 KSI is used Note that the full units string, e.g. N/mm^2 must be used in conjunction with the batch keyword.
Young's Modulus	The value of E is used in the principal stress option to convert from strain to stress.
Poisson Ratio	The value of mu is used in the principal stress option to convert from strain to stress. It is NOT the same value entered for the Gauge Poisson Ratio.
Graphical Results	When discrete Rosette Strains are entered from the keyboard, SSA offers the option of displaying the Rosette, the principals and the angle between the maximum principal and grid 1 in terms of a Mohr's circle.

Graphical Operations

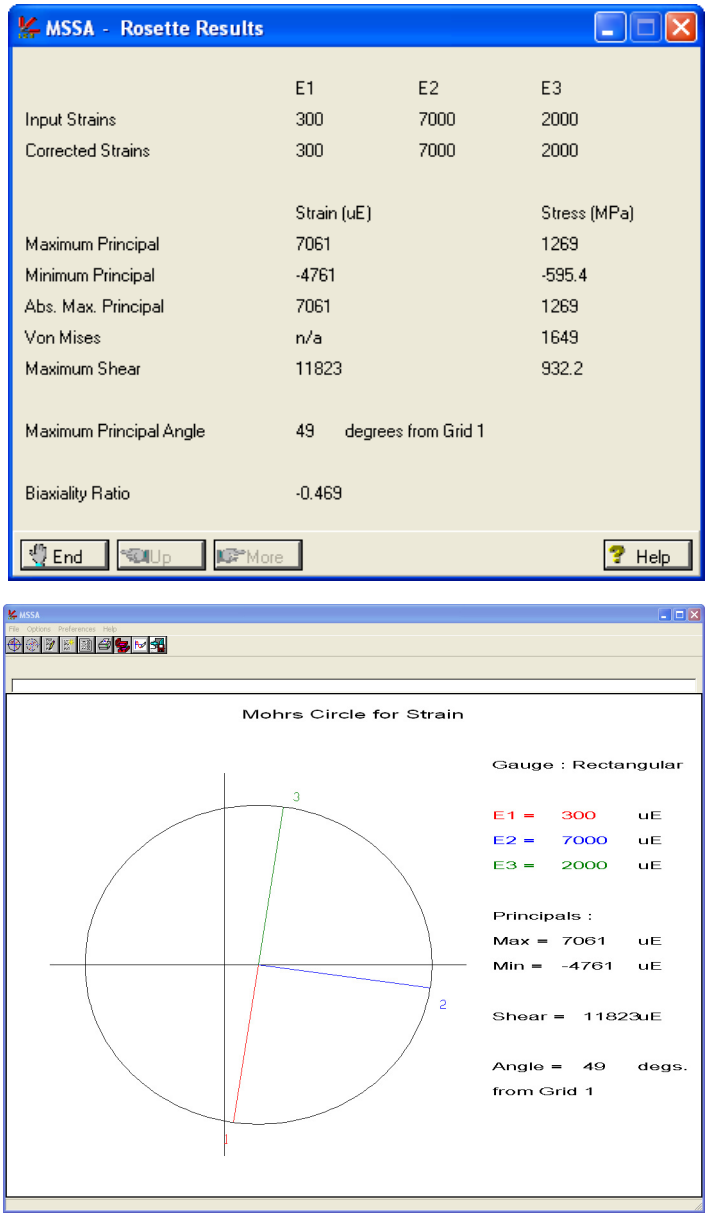


Figure 12-16 Tabular and Graphical Results from Keyboard input (dependent upon Input Settings and Graphical Results Setting on Figure 12-15)

Note:	The statistics shown on the graphical display are rounded to the nearest integer. Values accurate to 3 significant figures are shown on the tabular display.

Operation of the graphics menu system is mostly standard MSC Fatigue operations described elsewhere and in Module Operations (Ch. 18). However menu options of particular interest to MSSA are shown in the table below. The commands can be executed from a menu or at the command line. The menu options and command lines are briefly described below.

Option	Description
General commands:	
Strain\Stress circle	This toggles the display between the Mohr's circle for stress and the Mohr's circle for strain.
List Values	Lists all the values for stress and strain to maximum accuracy (3 significant figures).
New Options	Returns the user to the Input Options screen.
Exit	Will QUIT MSSA and return to the operating or menu system.
Graphics options in prompt mode:	
HC	Hard copy (optionally HC=filename)
EX	Exit the program
PL	Replot. In batch, switches to interactive.
MCS	Show Mohr's Circle for Stress.
MCE	Show Mohr's Circle for Strain.
LV	List values numerically.
NV	Enter new values.
NOP	Modify options.
CU	Sets cursor mode.
Keys available in this mode and in menu mode:	
P	Replots
H	Hardcopy of entire screen (including menu)
V	Display coordinate value
Q	Quits cursor mode
#	Go from menu mode to prompt mode

Input Files Screen

When all of the above parameters have been defined, the input file(s) screen must be completed.

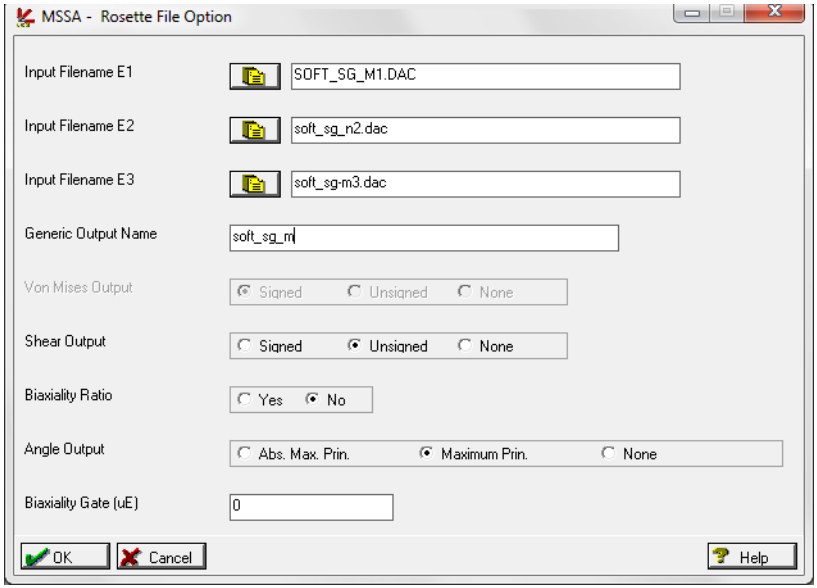


Figure 12-17 The Rosette File Option screen

This screen defines the input and output files to be used for rosette calculations. There are three different input files required for rectangular and delta rosettes or two for a tee rosette. Up to 7 output files will be created. Use the options on this screen to limit or configure the outputs.

Field	Description
Input File Names, E1, E2, and E3	MSSA calculates principals, angles and shears from strains measured by a rosette. Enter the name of the file that contains the rosette output, in MICROSTRAIN, from grids 1, 2, and 3 of a stain gauge rosette. Note that for rectangular rosettes, grids 2 and 3 are angularly separated from grid 1 by 45 and 90 degrees respectively. For delta rosettes grids 2 and 3 are separated from grid 1 by 60 and 120 degrees respectively. Alternatively, select the required file from the pick list of all `.dac` files present in the current directory by pressing the F3 key, or by clicking the LIST button with the mouse. Files in other directories can be listed by specifying the appropriate path name. The required file can be selected by tagging with either the arrow keys or the mouse.
Generic Output Name	A number of output files, which contain the calculated stresses and strains will be created. These files will assume the generic name specified here and are distinguished from each other by different file extensions. The following file extensions will be applied to the generic file name: `.max`The maximum principal stress or strain. `.min`The minimum principal stress or strain. `.abs`The absolute maximum principal stress or strain. `.shr`Either the absolute maximum or maximum shear stress or strain. `.ang`The angle between grid 1and either the absolute maximum principal, or maximum principal, stress or strain. `.bax`The biaxiality ratio of minimum to maximum principal. `.von`The Von Mises stress.

Field	Description
Von Mises Output	MSSA can calculate the signed and unsigned Von Mises stresses directly from the principals. The unsigned value is taken to be the positive root of the Von Mises expression. The signed value is taken to be a positive root with the sign of the absolute maximum principal given to it. Either signed or unsigned Von Mises stress may be used directly in stress-based fatigue analyses. If the None option is selected, then the Von Mises stress will not be calculated.
Shear Output	MSSA can calculate the maximum shear stress (or strain) from the two principal stresses (or strains) respectively. The unsigned maximum shear strain is given by the difference between the two principal strains. The unsigned maximum shear stress is given by half the difference of the two principle stresses. The signed maximum shears are calculated as above, BUT with the sign of the absolute maximum principal assigned. Either the signed or unsigned maximum shear stresses and strains can be used directly in stress and strain-based fatigue calculations respectively. If the None option is selected, then no shear values will be calculated.

Field	Description
Biaxiality Ratio	The biaxiality ratio is the ratio of the smaller in-plane principal stress to the larger (the absolute maximum principal). See Multiaxial Fatigue (Ch. 7) for more information. It can take on values which range from -1 (pure shear), through zero (uniaxial), to +1 (equi-biaxial) loading. As a rough guide, if the biaxiality ratio is greater than about +0.25 then fatigue calculations should employ a plane strain correction and use the absolute maximum principal strain as input. In the band between +0.25 and -0.25, (basically uniaxial loading), fatigue calculations should use the absolute maximum principal as input. For biaxiality ratios less than about -0.25, the absolute maximum principal can still be used or otherwise the maximum shear strain together with shear strain life fatigue properties. If the No option is selected, then the biaxiality ratio will not be calculated.
Angle Option	MSSA can calculate the angle between the orientation of grid 1 of the rosette and either the maximum principal or the absolute maximum principal stress or strain The variation of this angle with time through a loading history provides a valuable insight into the nature of the loading environment. It is sometimes useful to carry out a time at level analysis of the angle file in order to gauge angular stability. See also the graphical output options presented in the MSSA postprocessing menu. If the angle is relatively stationary then the loading is said to be proportional, and simple uniaxial or shear models can be used in fatigue calculations. If the angle fluctuates markedly, particularly at high strain amplitudes, then the loading is probably non-proportional and so fatigue analysis must use a critical-plane approach together with kinematic hardening models. If the none option is selected, then the angle will not be calculated. See Multiaxial Fatigue (Ch. 7) for more information.

Field	Description
Biaxiality Gate	This value will be used to exclude any values of biaxiality for points whose absolute maximum principal strain is less than the gate.

When all of the above fields have been completed (or their defaults accepted), MSSA will process the input files according to the user's specification. It will then display the Rosette Postprocessing menu which allows the results to be plotted.

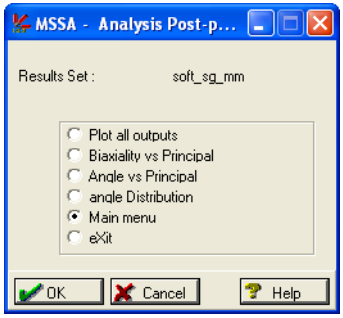


Figure 12-18 The Rosette Postprocessing Menu

At this stage the required output files have been created from the rosette strains and it is possible to display the results graphically.

- Time series plots of all files can be plotted on a single display.
- Crossplots of the biaxiality ratio or angle to grid 1 vs. absolute maximum principal can be also be displayed.
- Finally, an amplitude distribution, time at level, can be created and plotted.

Note: MSSA uses the graphics module Multi File Display (MMFD) to plot the files. See Multi-File Display (MMFD), 251. From MMFD the user will always be returned to the Analysis Postprocessing menu.

2 - Elastic to Elastic-plastic Analysis

The purpose of this function is to convert a strain time history of elastic only data to an equivalent elastic-plastic strain time history file (assuming uniaxial loading).

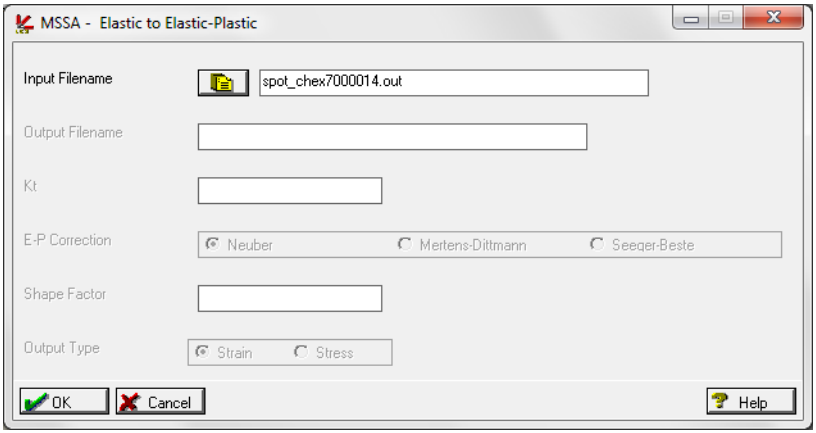


Figure 12-19 The Elastic to Elastic-Plastic Option Screen

The fields are as follows:

Field	Description
Input Filename	This is the name of the input file of elastic strain data. It is normally a single parameter file, e.g. `test.dac`.
Output Filename	Specify the name of the strain time history to create.
K_t	Specify the scaling factor to be used on the linearized data. This value is simply a strain multiplier or divisor. The stress concentration factor, K_t, may be used here to convert from nominal to local and vice-versa. When converting from elastic-plastic to elastic, this value is used as a divisor.
E-P Correction	This field allows you to choose the plastic stress correction type.
Shape Factor	Used in the Mertens-Dittmann and Seeger-Beste corrections. It is the ratio of the plastic limit load to the yield load.
Output Type	Select whether the output file will be in units of stress or strain.

When the above fields have been filled the Input Material Data Screen is displayed.

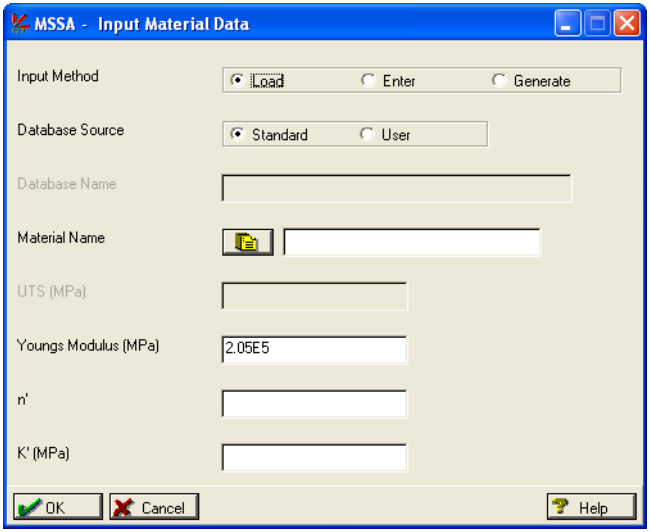


Figure 12-20 The Input Material Data Screen

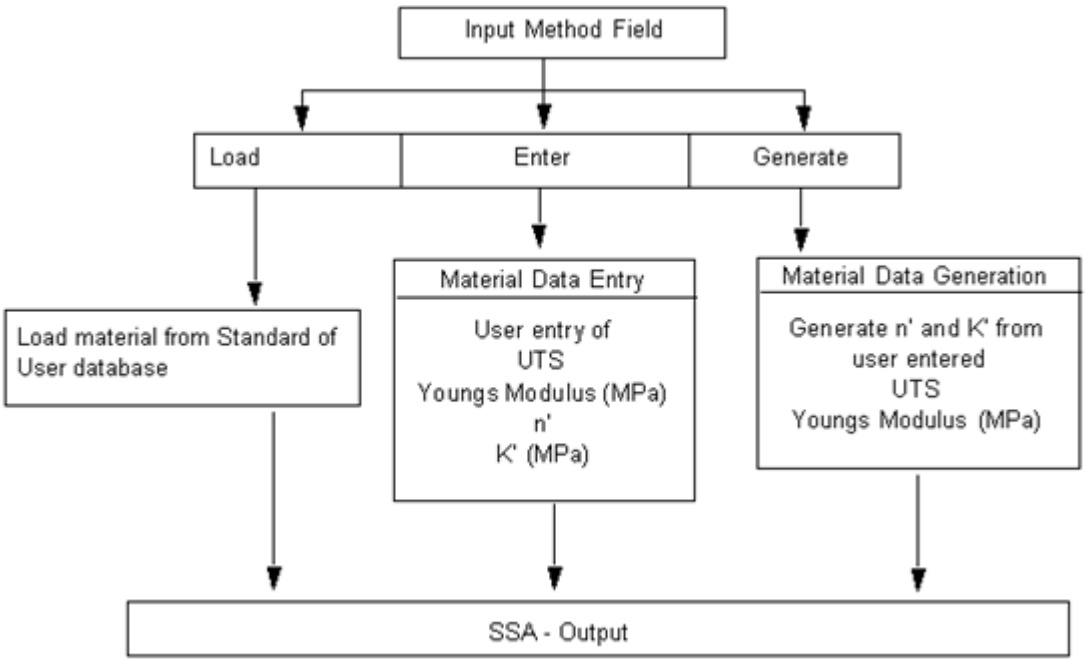


Figure 12-21 The Materials Data Options

There are three ways of inputting materials data.

- load from the materials database using the material name or picking from the F3/List pick list. The properties can be edited. PFMAT is the materials data source.
- entered directly into the Young's modulus, Cyclic exponent, and Cyclic coefficient fields.
- generated from the material ultimate tensile strength (UTS) and Young's modulus.

Specify the method to use. Depending upon the method chosen, one or more of the following fields will appear.

Field	Description
Material Name	All materials within PFMAT that have values for Young's modulus and UTS are available. The List facility is probably the easiest way of scanning the database.
UTS (MPa)	The Ultimate Tensile Strength. The value of UTS is used to generate n' and K' (below). This field does not appear if Load or Enter are selected.
Young's Modulus (MPa)	The value of E is used in the generation option to determine basic material type. If the data is loaded from the database this field will be modified each time the name is changed.
n'	n' is the Cyclic Hardening Exponent. If the data is loaded from the database, or generated from the UTS and E, this field will be modified each time the name is changed or UTS/E are modified.
K' (MPa)	K' is the Cyclic Strength Coefficient (not to be confused with cyclic exponent above). If the data is loaded from the database, or generated from the UTS and E, this field will be modified each time the name is changed or UTS/E are modified.

When all of the above fields have been filled and accepted, the correction will be made and a results screen displayed. A sample results screen is shown below.

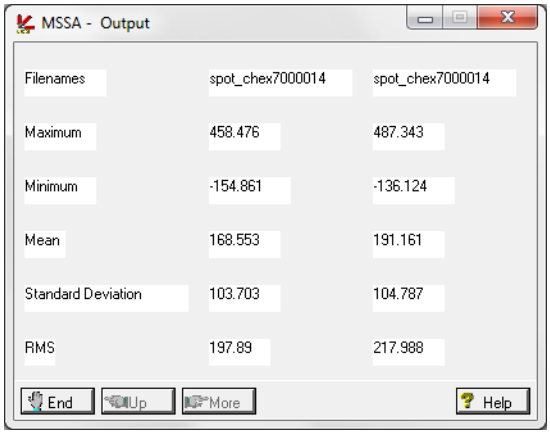


Figure 12-22 An Elastic to Elastic-plastic Results Screen

3 - Elastic-plastic to Elastic Analysis

This option performs the opposite conversion to that described in option 2 - Elastic to Elastic-plastic Analysis, 927 above. However, all the screens and fields are the same as already described. Therefore, for usage instructions please refer to the option above.

4 - Elastic-plastic to Elastic-plastic Analysis

This option converts elastic-plastic strain records obtained with material A into an equivalent set of results that would have be achieved by material B. The basic steps are:

- Name the input strain file (a single parameter data file), and the output file which will be created
- Specify the input material data
- Specify the output material data
- The following fields need filling:
- Input File Name name (see text for Figure 12-19 above)
- Output File Name (see text for Figure 12-19 above)
- Note that no Kf value is needed. The following need supplying for both input and output materials;
- Input Method (see text for Figure 12-20 above)
- Material Name (see text for Figure 12-20 above)
- Young's Modulus (see text for Figure 12-20 above)
- n' (see text for Figure 12-20 above)
- K' (see text for Figure 12-20 above)

The post-analysis results screen is the same as for Figure 12-18 above.

MSSA Batch Operation

MSSA can be run in batch mode.

An example batch or macro command line is shown below.

```
ssa /ov=y/opt=1/type=r/kt1=10/inp1=test101/inp2=test10
/inp3=test103/out=sumall
```

Which will carry out a rectangular (TYPE=R) strain gauge rosette analysis (OPT=1) with a stacked configuration (CON=S), transverse sensitivity (KT1) of 10%, and three INPut files. The family of output files will have the root name `sumall`, e.g. `sumall.max`, `sumall.min`, `sumall.abs`, `sumall.shr`.

Note that defaults were accepted for the other inputs.

A list of MSSA's batch keywords:

/OPTion	The main option (1-4). /OPT=2
/INPut	The input filename. /INT=TEST1.DAC
/OUTput	The output filename. /OUT=RESULT.DAC
/KT	The scaling factor for conversion. /KT=1.5
/METH	Material entry method for input material, L, E, G. /METH=L
/MATname	Material name for input material. /MAT=RQC100
/UTS	UTS for input material. /UTS=900
/YM	YM for input material. /YM=2E5
/NP	n' for input material. /NP=0.15
/KP	K' for input material. /KP=1000
/METHOUT	Material entry method for output material. /METHOUT=G
/MATOUT	Material name for output material. /MATOUT=
/UTSOUT	UTS for output material. /UTS=900
/YMOUT	YM for output material. /YMOUT=2E5
/NPOUT	n' for output material. /NPOUT=0.3
/KPOUT	K' for output material. /KPOUT=1100
/MPOIS	Gauge Manufacturer's Poisson's Ratio. /MPOIS=0.285
/TYPe	Rosette Type, R, D, T. /TYP=D
/CONfig	Rosette Configuration, S, P, U. /CONF=P
/KT1 ,/KT2 ,/KT3	Transverse sensitivity factors. /KT1=1.3
/INFRom	Input from File F or Keyboard K. /INFR=K
/OTYPe	Output Type - Stress S or straiN N. /OTYP=N
/CONversion	Stress Conversion - Linear or Non linear
/STSUNI	Stress Units (MPa,KSI,PSI). /STUNI=MPA

/GRAph	Whether to plot Mohr's Circle Y, N. /GRA=Y
/E1	First strain value. /E1=1E3
/E2	Second strain value
/E3	Third strain value
/INP1	File for gauge 1. /INP1=FILE1.DAC
/INP2	File for gauge 2. /INP2=FILE2.DAC
/INP3	File for gauge 3. /IINP3=FILE3.DAC
/OUT	Generic output name for rosette calculations. //OUT=FILES.
/VON	Von Mises output option signed S, unsigned U, none N. //VON=S
/SHEar	Shear output option signed S, unsigned U, none N. //SHE=S
/BIAX	Biaxiality ratio output yes Y, no N. /BIAX=Y
/ANGle	Angle output; maximum principle M, absolute maximum principle A, none N. /ANG=A
/POPTion	Post-processing option P, B, A, D, M, X (see main menu hot keys)

13 Duty Cycle Analysis

- Introduction 936
- Duty Cycle Setup (dtfat) 936
- dcbuilder 939
- Interface (Classic Modules) 945
- Duty Cycle Setup 947
- Sequence Information 949
- Event Information 949
- Job Submit 951
- Review Results 952
- Conclusion 953

Introduction

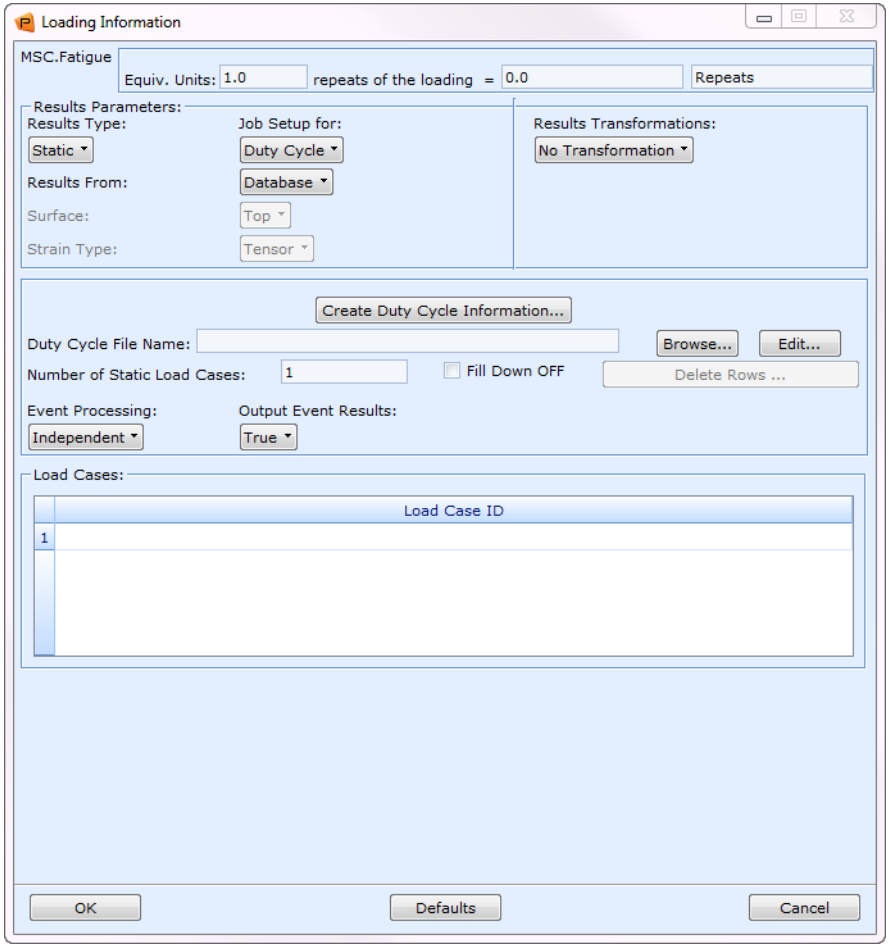
The duty cycle analyzer computes damage from multiple sequences and events, through an intuitive interface. A typical usage sequence may be a car running at a certain gross weight and subjected to various events such as turning, braking, and traveling over potholes. It is very useful for the analyst to identify which events contribute significant damage in a usage sequence or which usage sequence (e.g., different weight configurations) cause significant damage. In either case, the Duty Cycle capability in MSC Fatigue allows the user to simulate usage profiles (sequences) consisting of multiple events.

Duty Cycle Setup (dtfat)

The duty cycle analyzer in the DTLib-based solution is more sophisticated than the duty cycle tool for the Classic modules. Rather than running a separate analysis for each event, computing damage, and then summing damage for all events, the DTLib-based duty cycle uses only a single run, and all event results can be available for viewing. Complex events, which are themselves duty cycles can also be constructed.

The DTLib-based duty cycle approach can be used with S-N, E-N, Spot Weld, and Seam Weld analysis. Currently, the DTlib-based duty cycle analysis is not supported for vibration or for transient analysis.

The Loading Information form as configured for a Duty Cycle analysis is shown below. The


Loading Information
MSC.Fatigue
Equiv. Units: 1.0
repeats of the loading = 0.0
Repeats
Results Parameters:
Results Type:
Static
Job Setup for:
Duty Cycle
Results From:
Database
Surface:
Top
Strain Type:
Tensor
Results Transformations:
No Transformation
Create Duty Cycle Information...
Duty Cycle File Name:
Browse...
Edit...
Number of Static Load Cases:
1
Fill Down OFF
Delete Rows ...
Event Processing:
Independent
Output Event Results:
True
Load Cases:
Load Case ID
1
OK
Defaults
Cancel

The settings on the Loading Information form are described in the table below.

Table 13-1 Loading Information Form Settings

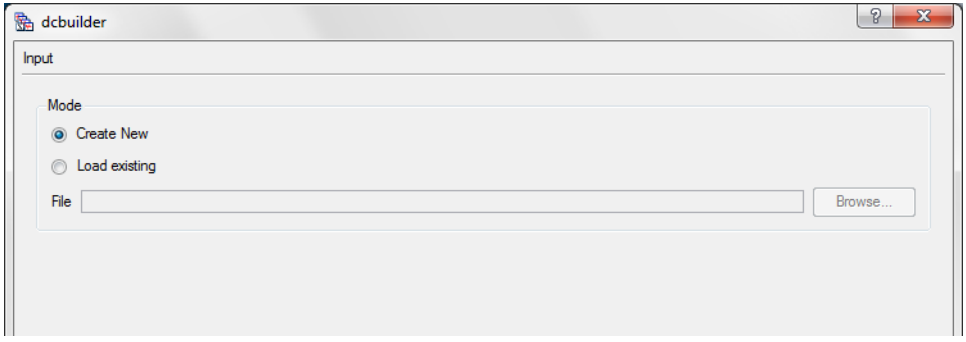
Table Entry	Option	Description	
Results Type		There are two basic FE result types that are used for most fatigue analysis: Static and Transient.	**Static**
	Static	FE results used in the fatigue analysis is static results cases.	
	Transient	FE results used in the fatigue analysis is transient time step results cases. This is currently not supported for DTLib-based duty cycle analysis.	
Job Setup	**Single**	Job is configured for a single loading specification.	**Single**
	Duty Cycle	Job is configured for a multi-event duty cycle specification.	
	Load Spectrum	Job is configured for a multi-event mission-type loading specification.	
Results Transformation		This option is generally obsolete, since FE results are transformed to a consistent coordinate system before nodal averaging.	**No Transformation**
	No Transformation	No transformation of FE results is performed. DTLib assumes that FE results for vibration fatigue are 2-D, and thus only considers the Sxx, Syy, and Sxy components of the results tensor. This option should be used for vibration fatigue analysis.	
	Transform to Basic	FE results are transformed to the basic coordinate system.	
Duty Cycle Name	**Value**	Name of the duty cycle file used for this job setup.	**No Default**
Number of Static Load Cases	**0 < Value**	The number of static results cases which are to be associated with a loading event.	**1**
Event Processing		A duty cycle is a sequence of events to be processed through the analysis engine. Duty cycles can be processed by the analysis engine in three different ways:	**Independent**
	Independent	Each unique event in the duty cycle is processed separately, and the damage from each is multiplied by the number of times it appears in the complete sequence. The total damage is the sum of the damage from all the different events. This is fast, but may miss significant cycles that cross different events.	

Table 13-1 Loading Information Form Settings

Table Entry	Option	Description	
	Combined Fast	An intermediate method that cycle counts events individually but also captures the cycles that cross events. It will in general be almost as accurate as the Combined Full option and almost as quick as the Independent mode.	
	Combined Full	All the events in the duty cycle are concatenated, including all repeats of each event, and the resulting stress history is processed as if it was one long event. This is accurate, but is computationally inefficient and could take a long time to process.	
Output Event Results	**True, False**	If this property is set to True, the accumulated damage (and other calculated results) associated with each individual event from the duty cycle will be output in addition to the total damage. If either of the "Combined" options is being used, some cycles may cross events. The damage associated with these cycles is split equally between the two events with which it is associated.	**True**

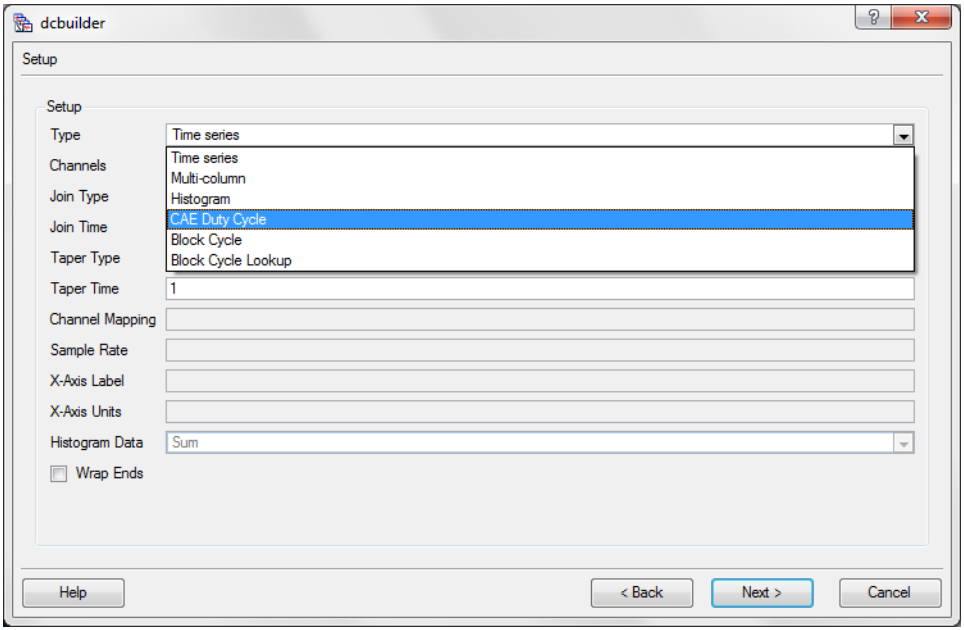
dcbuilder

A utility called **dcbuilder** is used to construct duty cycles for analysis using dtfat. This utility can be run from the command line, as $p3_home/bin/dcbuilder, or it can be launched from the **Loading Information** form, by clicking the **Create Duty Cycle Information** button.



To create a new duty cycle file, select **Mode** = **Create New**. To edit an existing duty cycle file, select **Mode** = **Load Existing** on the dcbuilder opening page.

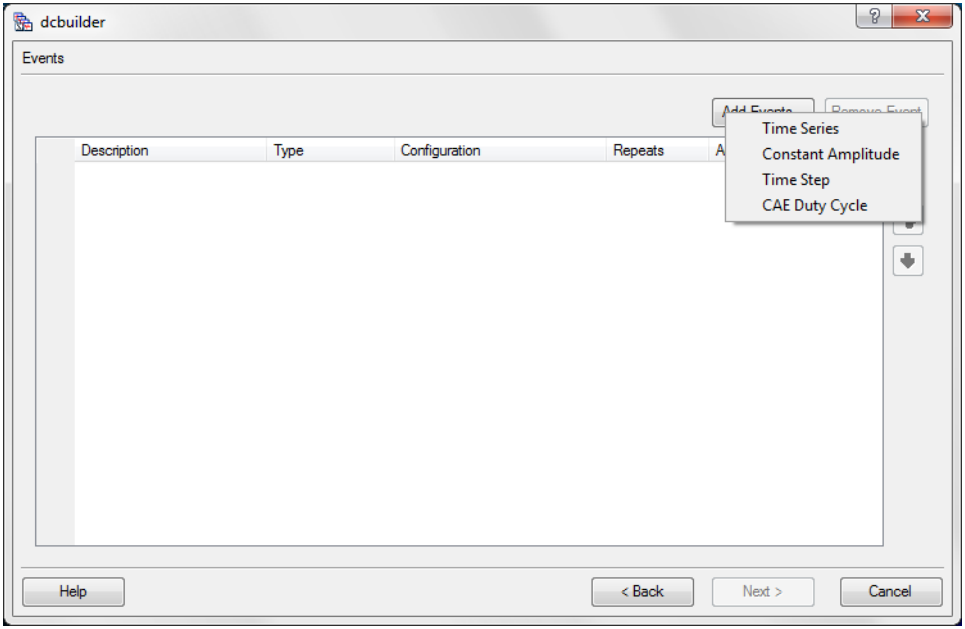
In most cases, a **CAE Duty Cycle** is to be created, although there are other alternatives listed in the figure below. The **CAE Duty Cycle** file is composed of one or more type of event, which are repeated a specific number of times. Duty cycle files may be composed of events that are themselves duty cycles, allowing for a quite expressive medium.



The next step is generally to create events, which model the loading regime experience by the users model.

As an example, suppose there is a situation in which three static results cases model the stress due to a reference load in three different coordinate directions. Suppose also that three different multi-channel files contain the time history of loading for each direction corresponding to three different loading events. Suppose further these events are labeled Event_A, Event_B, and Event_C. Finally, suppose that channel 1 of each time history corresponds the loading in coordinate direction 1, and so forth.

To add the loading event Event_A, click the Next key on the duty cycle construction wizard and then click **Add Events**,. in this case a **Time Series** event,

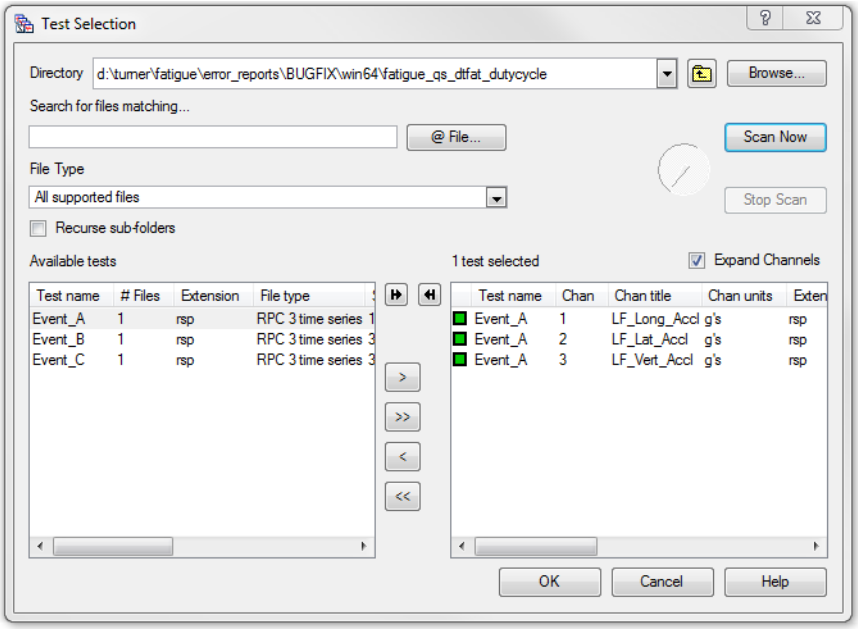


The resulting dialog allows the user to select a time history (or time histories) that correspond to this event. In this particular case, the first event is modeled with time history Event_A, which is a multi-channel rsp file with channels 1 - 3 corresponding to coordinate directions 1 - 3. The check-box **Expand Channels** can be used to view more details concerning any given time history.

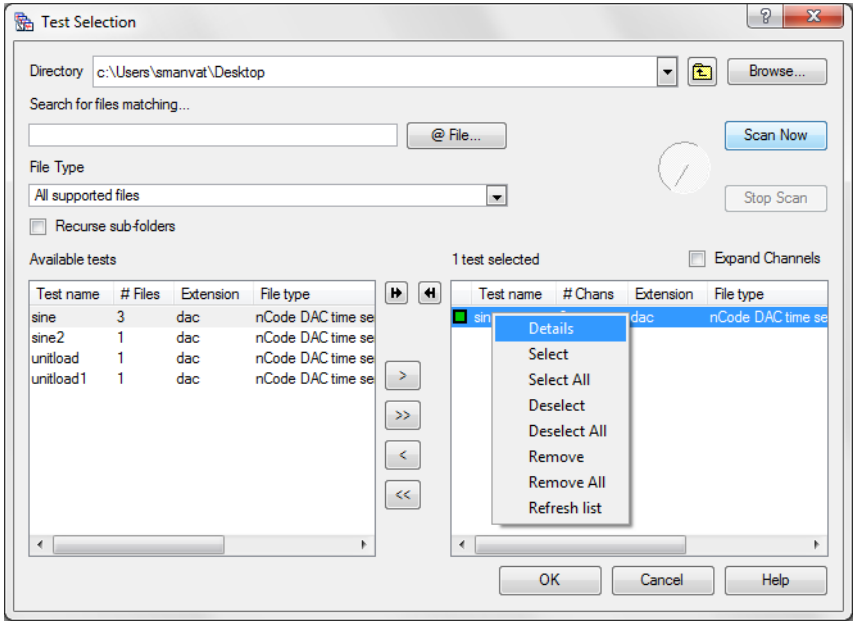
To select a time history, click on the time history name in left-hand pane, and then click the “>” key to select. The selected time history (or histories) appear in the right-hand pane. To de-select a time history, click on the time history in the right-hand pane and click the “<“ button to remove if from the right-hand pane.

To select more than one time history, select the desired time history files in the left-hand pane and click the “>” button. To select or de-select all, click the “>>” or “<<“ buttons.

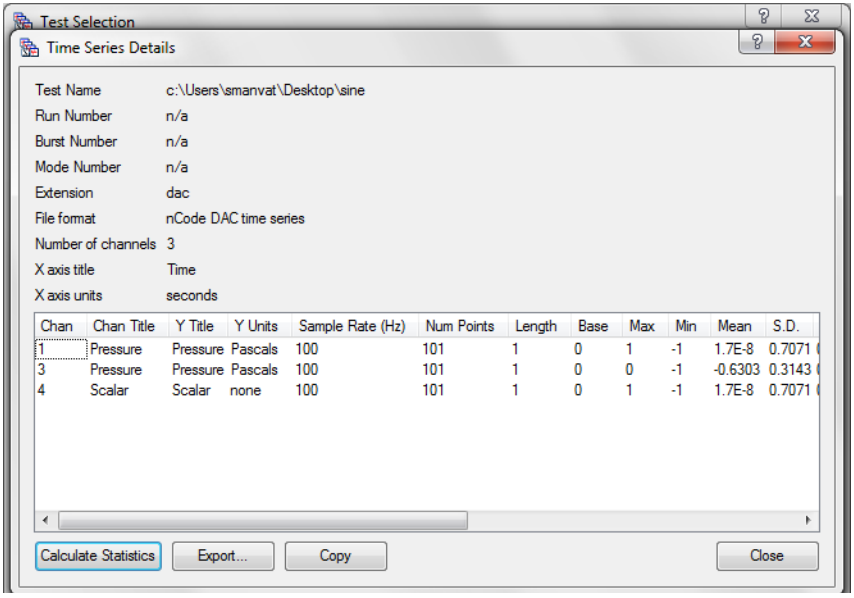
Keep in mind that a time history channel as they appear in the list on the right-hand pane will be associated with each load case selected on the main **Loading Information** form. This will be true for each event in the duty cycle. If any load case should have a zero loading during an event, it is the responsibility of the user ensure that a time history which is zero at all time points is available for this type of usage.



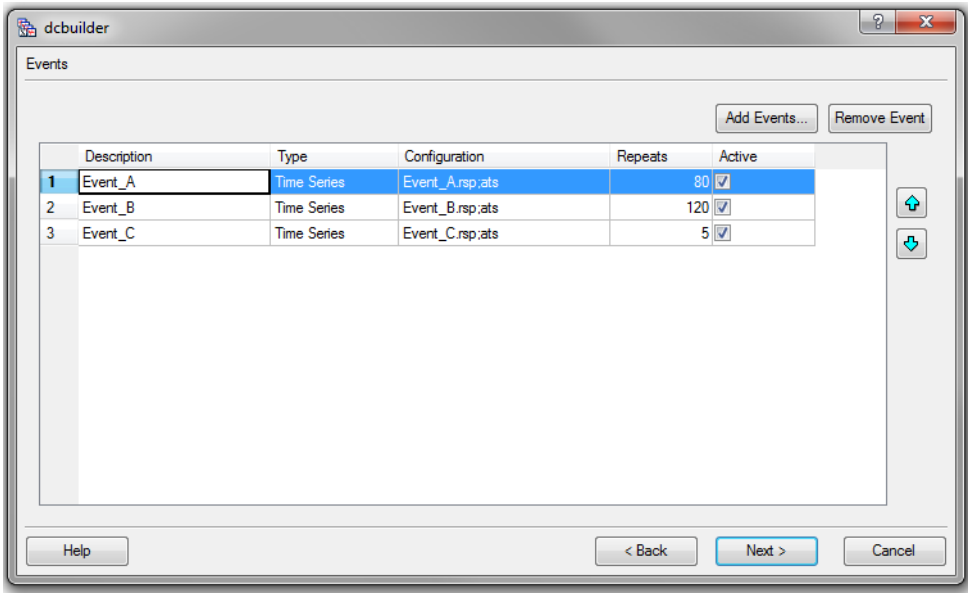
A time history details function is available in dcbuilder, which displays the statistics for a given time history file. This is available by clicking the right mouse button while a given time history file is selected, as shown below.



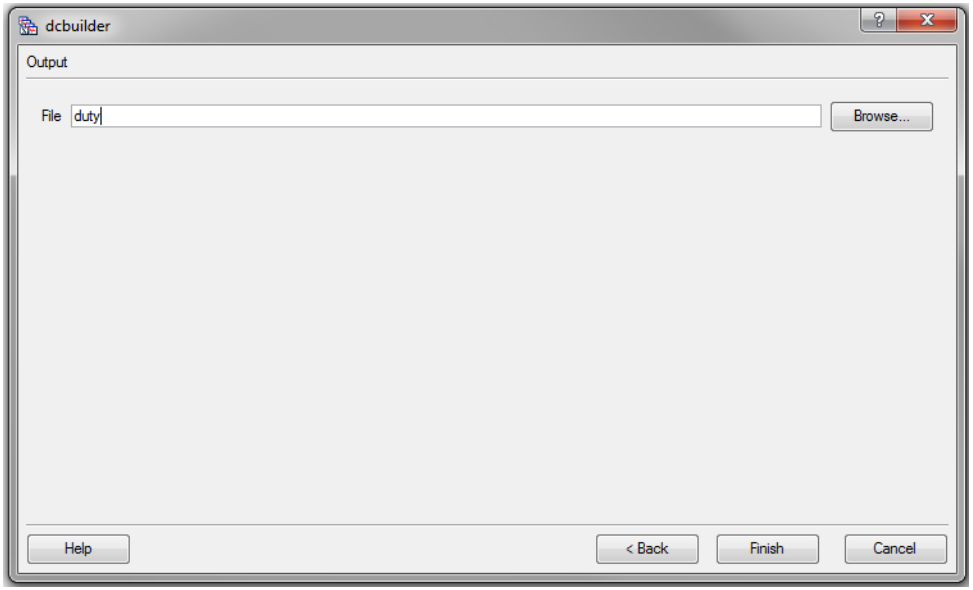
From the resulting pull-down menu, clicking on Details gives the available information on the selected time history file as shown below.



After defining the time histories associated with each event, the user can adjust the number of repeats of each event, by changing the **Repeats** field on the **Events** spreadsheet.



Finally, the duty cycle should be saved to a file. This file can assigned later on the Loading Information form. After assigning a name, the duty cycle creation wizard can be closed by clicking the **Finish** button at the bottom of the dcbuilder dialog.



The duty cycle file is a human-readable file in XML format. This file can be edited by the user, but caution should be used since the formatting is important. The duty.dcy file created with the above actions is shown below.

It can be seen that the file is composed of **Events** and a **Sequence** of Events.

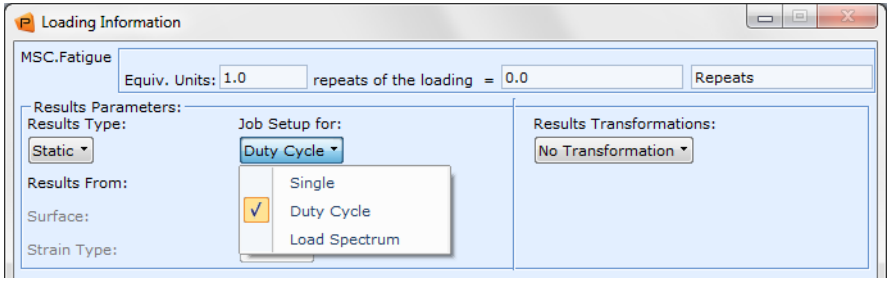
```
<DutyCycleDefinition Version="2">
<Events>
<Event Name="Event_A" Type="TimeSeries" Testname="Event_A.rsp;ats"/>
<Event Name="Event_B" Type="TimeSeries" Testname="Event_B.rsp;ats"/>
<Event Name="Event_C" Type="TimeSeries" Testname="Event_C.rsp;ats"/>
</Events>
<Sequence>
<Item EventName="Event_A" RepeatCount="80"/>
<Item EventName="Event_B" RepeatCount="120"/>
<Item EventName="Event_C" RepeatCount="5"/>
</Sequence>
</DutyCycleDefinition>
```

Interface (Classic Modules)

The duty cycle analyzer computes damage from multiple sequences and events, through an intuitive interface. A typical usage sequence may be a car running at a certain gross weight and subjected to various events such as turning, braking, and traveling over potholes. It is very useful for the analyst to identify which events contribute significant damage in a usage sequence or which usage sequence (e.g., different weight

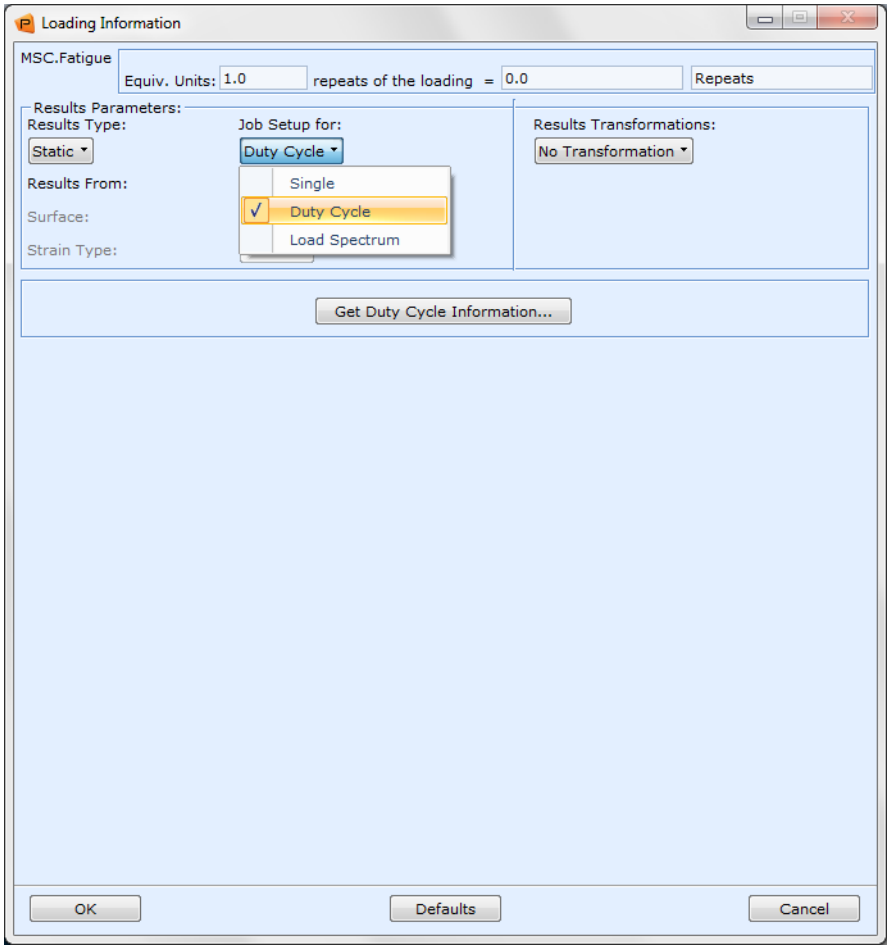
configurations) cause significant damage. In either case, the Duty Cycle capability in MSC Fatigue allows the user to simulate usage profiles (sequences) consisting of multiple events.

Clicking on the Job Setup button in the Loading Information Form allows the user to select the Duty Cycle Analysis, as shown below.



Depending in the selection made, the **Load Information** form will change to present the user with the correct options for the type of analysis selected.

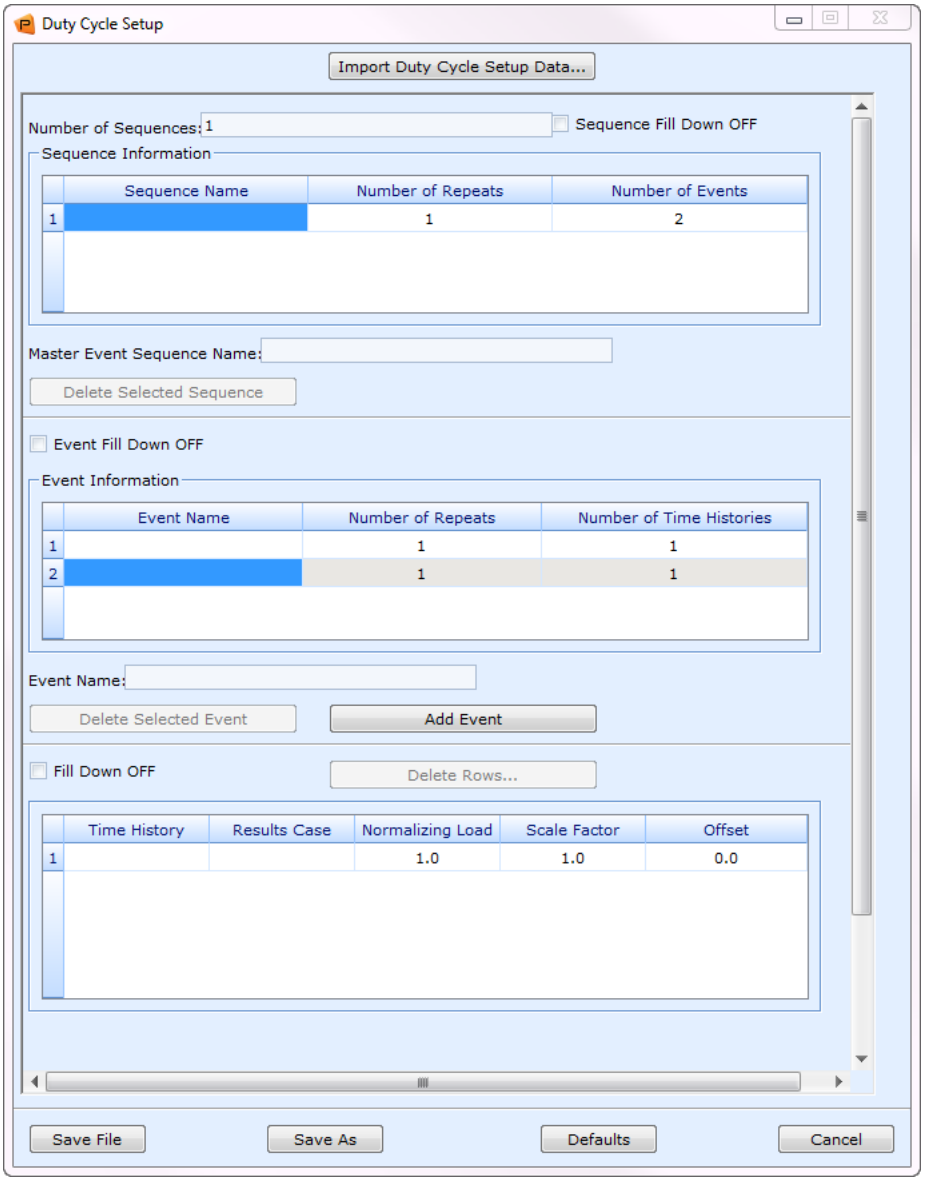
Click on **Duty Cycle** to setup your Job for a Duty Cycle Analysis.



Clicking on the Get Duty Cycle Information button brings the Duty Cycle Setup form, shown below.

Duty Cycle Setup

Data for the Duty Cycle form may be filled out sequentially, i.e. complete definition of a sequence followed by the next sequence or added randomly. However, in the latter case the user has to exercise caution to ensure that the data being entered is for a particular sequence and event; e.g. if the user has 3 sequences, and the user wishes to enter data for sequence 3, one of the cells in the Sequence Information spreadsheet for Sequence 3 must be active. The same logic applies to entering Time History data for a particular event. In the example below, we shall use the random method and point out the focus requirement as necessary.



Import Duty Cycle Setup Data: Imports a previously saved Duty Cycle Setup file named <jobname>.cfg. The first sequence and event information from this file is loaded to the form.

Sequence Information

Number of Sequences: This is either entered or displayed from the imported setup file. This number also sets the number of rows to display in the spreadsheet for entering the sequence information. The maximum number of sequences allowed is 10. For this exercise, we will have 2 sequences. Enter 2 in the number of sequences and enter the sequence definitions as defined below.

Sequence Name Cell: Enter or display the names of the sequence in the imported file. Enter sequence names wt1 and wt2 by clicking in the respective sequence name cells.

Number of Repeats Cell: Enter or display the number of repeats of the current sequence in the imported file (Default =1). Enter 1 for sequence wt 1 and 2 for sequence wt2.

Number of Events Cell: Enter or display the number of events for the sequence in the imported file. This number also sets the number of rows to display in the Event Information spreadsheet. The maximum number of events per sequence is 100 and the default is 1. Enter 3 for sequence wt1 and 4 for sequence wt2. This signifies that the first sequence wt1 shall have 3 events and the second have 4 events.

Delete Sequence: Deletes the sequence and associated information (event, time histories, etc) associated with the sequence. The delete function allows single or multiple deletes.

Sequences may be added by incrementing the number of remaining sequences. We shall proceed to defining the events specified for each sequence.

Event Information

The event information for each sequence is added by clicking on the ***sequence name*** cell or anywhere in the row for the sequence and defining the event information for the selected sequence in the event information spreadsheet. Since we defined 3 events for the first sequence, 3 rows will be displayed for sequence wt1 and 4 for the second sequence.

Event name Cell: Enter or display the names of the event in the imported file for a selected sequence. With the focus on the first sequence, enter Push_back for event 1, Taxi for event 2 and flight for event 3. For the second sequence, reestablish the focus and click on the Sequence name 2 and add the same information as sequence wt1. However, since we defined an extra event for wt2, enter Land as the name for the fourth event.

Number of Repeats Cell: Enter or display the number of repeats of the current event in the imported file (Default = 1). We shall assign 1 repeat for every event except the flight event in both sequences, where we will define 2 repeats. With the focus set appropriately for each sequence, enter 2 for the number of repeats for Sequence wt1, event flight and Sequence wt2, event flight.

Number of Time Histories for this Event: Enter or display the number of Time Histories or other load types from the imported file. This number also sets the number of rows in the load association spreadsheet. In this exercise, we will use a single time history for each event (see table below).

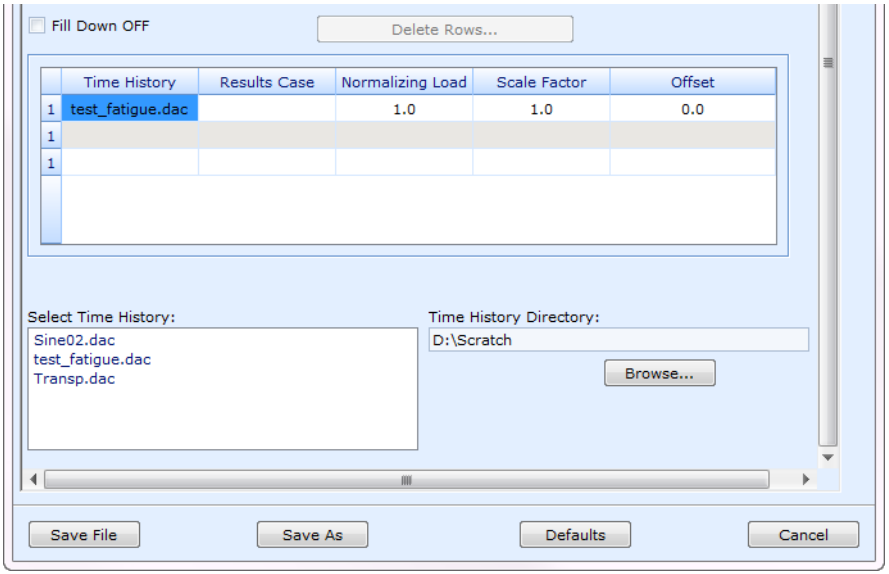
Delete Selected Event: Deletes a selected event and associated information with the event. The delete function allows single or multiple deletes.

Add Event: Clears the widgets and allows information to be added for a new event after the selected event. If no selection is made, the event is added at the top.

Entering Time-History data

Fill Down OFF: Checking this will set Fill Down to ON, where a user can take advantage of filling the spreadsheet if time histories and load cases have been named in a logical fashion. This function will enable Users to retain some of the functionality of the previous Multiple Analysis Tool.

Time History Cell: Pushing this button will display a browser dialog to pick the .dac files from any directory. Selecting a dac file from the directory will load up the first cell and a list box shall display all the .dac files in the current directory for subsequent picking (see image below).



The time histories can be used by browsing to the central location (p3_home/mscfatigue_files/ptime) and picking up Sine01.dac for Sequence wt1, event push_back. The list box should show all time histories in this directory from which the following will be used for both sequences as defined below.

Sequence	Event	Time History
Wt1	Taxi	Saetrn.dac
Wt1	Flight	Saebrkt.dac
Wt2	Push_back	Sine01.dac
Wt2	Taxi	Saetrn.dac
Wt2	Flight	Saebrkt.dac
Wt2	Land	Saetrn.dac

Results Case: Pushing this button will display, in a list box, the load and results types in the Patran database. Selecting the load and results type will load the root results case identifier (i.e. the load case and results type)

into the cell. Notice the absence of a layer selection - this has been automated and the correct layer information will be extracted based on the results type.

Normalizing Load, Scale Factor and Offset: The default values are 1.0, 1.0 and 0.0 respectively. The max/min value in the entered time history will be displayed to allow the user to normalize the time history. For this example, the following normalizing values shall be applied to every occurrence of the time histories in the table above.

Saetrn.dac	999.
Saebrkt.dac	738.

Save File: Saves the current information that has been either loaded from an existing load setup file (existing_jobname*.cfg) as existing_jobname*.cfg. A warning message will be issued advising if a .cfg file exists in the directory to prevent accidental overwriting of a previous setup. If a user does not wish to overwrite an existing .cfg file, the Job name may be changed on the Main form to force a save to the changed Job name.

Save your setup file and cancel out of the Duty Cycle setup form.

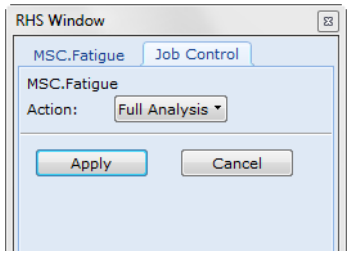
Note:	Messages will be echoed warning the user if problems are encountered in generating the .cfg file. The user should pay attention to these messages and correct the problems by making a note of the messages and ensuring that all cells in the row identified are reloaded and saved..

Defaults: Restores the form to the default state for the current sequence and event.

Cancel: Discard all inputs and exit out of the form.

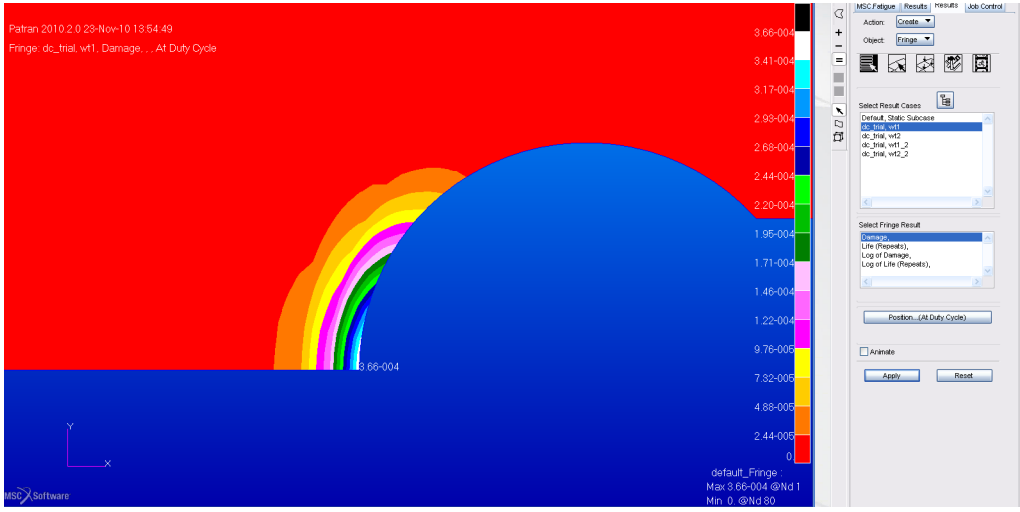
Job Submit

From the Job Control form submit a Full analysis, and a Duty Cycle analysis will be submitted automatically.



Review Results

Monitor the progress of the job and at completion, read the results from the analysis using Read results from the Main form. Notice again the absence of the Job Type button as in the previous versions of MSC Fatigue. A Duty Cycle job is automatically detected in the run directory and results are read accordingly. With the Setup described above, the results form should display the results for the 2 sequences, wt1, and wt2 as shown below:



The result shown above is the Duty Cycle result at the bottom surface for sequence wt1..

Sequence	Event	Damage (Node 1)	Duty Cycle
Wt1	Push_back	2.34e-6	
Wt1	Taxi	1.18e-5	
Wt1	Flight	2.7e-4	
			2.84e-4
Wt2	Push_back	2.34e-6	
Wt2	Taxi	1.18e-5	
Wt2	Flight	2.7e-4	
Wt2	Land	1.18e-5	
			5.92e-4

Note:	The damage results for event Flight in both sequences take into account the number of repeats (2) applied to this event. The Duty Cycle results are, as expected, the sum of the damage in each sequence. Note that the Duty cycle result for sequence wt2, takes into account the repeat factor specified for this sequence.

Conclusion

The Duty Cycle analysis provides a powerful and flexible tool for setting up and analyzing complex sequences or conditions. The user has complete flexibility in setting up sequences, events consisting of uneven time histories in each event and the ability to navigate to the source of time history data.

14 Fatigue Utilities

- Fatigue Preprocessing 956
- Utilities 959
- Advanced Loading Utilities 963
- Advanced Fatigue Utilities 1083
- Graphics Display Utilities 1143
- File Conversion Utilities 1178

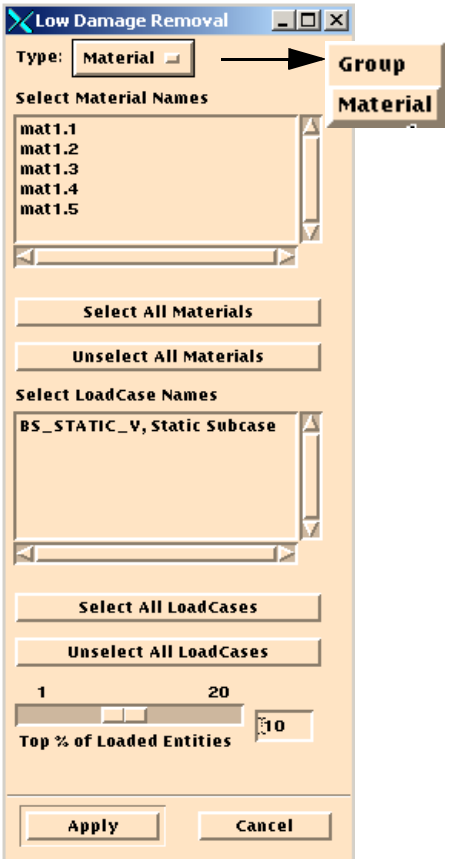
Fatigue Preprocessing

MSC Fatigue has the following preprocessing modules:

1. Set Fatigue Enviromental Variables
2. Low Damage Removal
3. Calculate Stress Gradient

Low Damage Removal

The Low Damage Removal utility is designed to extract areas of the model that are likely to sustain high damage. These areas are then grouped based on user selection of existing user created groups or automatically extracted material groups resident in the Patran database. In the example shown on the right the user may select all or pick one or more of the automatically extracted material groups. The Von mises stress or strain results from all selected loadcases are scanned and for each material group, a group is created that contains the top user-selectable percentage of loaded entities in that group. The solution sequences that utilize stress or strain results can benefit from this extraction. The only exception to this is Seam weld and although this utility can be used to identify the critical areas, users will have to exercise caution in using the extracted groups as the Seam weld module performs an extraction of the seam off the weld group and it is possible that the weld may not be a part of the extracted group.



For the force based Spotweld solution sequence critical Spot weld elements are extracted based on the magnitudes of the translational and rotational components. Either the material group associated with the Spot weld elements or a group containing only the Spot weld elements (recommended) is used for the extraction.

The groups extracted by this utility may then be used in a subsequent fatigue analysis for a quick assessment of the high damage areas of the model.

Note:	This extraction performed by this Utility does not take into account the phasing from Superpositioning of the results for multiple channel loading. FASTAN should be used in this case as proper account of the phasing is taken into account to extract high damage areas.

This Low Damage Removal utility can be accessed by selecting the option from the Fatigue Preprocessing pulldown menu under **Tools | MSC Fatigue** (for Patran) or under **Tools | Fatigue Utilities** (for).

From the low damage removal form:

1. Select the Type (either Group or Material).
2. Select either the material names or group names that are of interest.
3. Select the loadcases.
4. Select the percentage of loaded entities desired. The default value is 10 but you can select a value anywhere from 1 to 20. (i.e., If you select 15 then the top 15 percent of damaged entities will be added to the new group)
5. Hit Apply. The results data for each selected material name or group name is processed and the top requested percent of elements with the worst damage are put into a group using the following naming convention: **HD_*material name*** or **HD_*group name*.** (ie. If you selected mat1.1 and mat1.2 then two new groups would be created. One called HD_mat1.1 and the other called HD_mat1.2)
6. These new groups can then be used in the Materials Information form to quickly identify the "Fatigue hot-spots" in the Model.

Calculate Stress Gradients

If fatigue calculations are made based on either local elastic stresses calculated from FE, or based on nominal stresses and Kt values, over conservative life predictions may result. This is particularly a problem with many engine and powertrain components where combinations of sharp stress concentrations, small fillet radii and notch-insensitive cast materials can lead to the effect being quite large. The stress gradient correction adopted in MSC Fatigue is based on a method described in the FKM Guideline "Analytical Strength Assessment of Components in Mechanical Engineering" 5th edition, 2003, English version, Tr. E. Haibach published by VDMA, Frankfurt. ISBN 3-8163-0425-7. This method which considers only normal and shear cases, and positive stress gradients, is an endurance limit approach. It does not allow corrections at shorter lifetimes or specify how it might be applied to a general multiaxial case.

In MSC Fatigue we are effectively applying the same factor to the strength at all lifetimes. This means predictions at shorter lifetimes may still be somewhat conservative, but not as conservative as predictions made with no stress gradient correction.With the application targeted for such components, the stress gradient correction has only been implemented for solid element FE models and users should not use this for shell element models.

The FKM method considers only normal and shear cases, and does not allow for more complex loadings, or for cases where the stress gradient may be negative. In the absence of better information, the FKM method in MSC Fatigue has been adapted by considering only the normal stress gradient and making some reasonable assumptions with recognition to the fact that the stress gradient is a second-order effect on fatigue life.

In general we have 6 (3D) or 3(2D) stress gradients, which can all vary with time and in relation to one another. We have to define some way of determining a single stress gradient, which can be used to determine the appropriate correction factor. Possibilities are:

1. Determine a separate stress gradient for each component for each moment in time and correct each component independently.
2. Determine the gradient of stress corresponding to the damage parameter being used, so if the damage parameter is abs-max-principal, we also determine the gradient of abs-max-principal stress.

3. Be more consistent and use the gradient of the von Mises stress. Since von Mises strain is proportional to von Mises stress, we could use this method equally well with stresses or strains.

There are arguments for and against all three options. MSC Fatigue uses option 3, defining the normalized stress grdient G_σ as:

$$G_\sigma = \frac{\frac{\partial}{\partial z'}\left(von\,Mises\left(\sigma_{ij}\right)\right)}{\left(von\,Mises\left(\sigma_{ij}\right)\right)}$$

The correction factors n_σ are calculated using the FKM guidelines as follows:

$$\text{For } G_\sigma \le 0, \quad n_\sigma = 1$$

$$\text{For } 0 < G_\sigma \le 0.1, \quad n_\sigma = 1 + G_\sigma \cdot 10^{-\left(a_G - 0.5 + \frac{R_m}{b_G}\right)}$$

$$\text{For } 0.1 < G_\sigma \le 1, \quad n_\sigma = 1 + \sqrt{G_\sigma} \cdot 10^{-\left(a_G + \frac{R_m}{b_G}\right)}$$

$$\text{For } 1 < G_\sigma \le 100, \quad n_\sigma = 1 + \sqrt[4]{G_\sigma} \cdot 10^{-\left(a_G + \frac{R_m}{b_G}\right)}$$

The dimensions are in mm, and the stress gradients in mm^{-1}. Rm is the UTS in MPa. The required constants are material dependent, and come from the following table:

Material	Stainless steel	Other steel	GS	GGG	GT	GG	Wrought Al alloy	Cast Al alloy
nCode mat type no.	23-25	13,14,16-22,26-99	9-12,15	5-8	2-4	1	100-105	106
a_G	.4	.5	.25	.05	-0.05	-0.05	0.05	-0.05
b_G	2400	2700	2000	3200	3200	3200	850	3200

The correction factor n is calculated on a point-by-point basis, and the stress or strain history is then corrected as:

$$^{corrected}\sigma_{ij} = \frac{\sigma_{ij}}{n_{\sigma}}$$

The Calculate Stress Gradient utility can be accessed by selecting the option from the **Fatigue Preprocessing** pulldown menu under **Tools | MSC Fatigue** (for Patran) or under **Tools | Fatigue Utilities** (for).

Utilities

The following advanced utilities are available as separate modules of MSC Fatigue. These modules are viewed as additional capabilities to the basic functions of MSC Fatigue. It is therefore assumed that a good working knowledge of MSC Fatigue is understood. Please familiarize yourself with text and graphical modes of operation as explained in Module Operations (Ch. 18). The modules can be group into several categories:

Type	Description
Advance Loading Utilities	
Arithmetic Manipulation - (MART), 963	MART is a module which arithmetically manipulates standard loading data files. Files may be operated on in ways similar to that offered by a hand held calculator. Complex mathematical procedures can be built up by "chaining" together sets of MART commands. If necessary, the arithmetic manipulation can be applied to a section or window in the data file, leaving the remainder of the data unaffected.
Multi-Channel Editor - (MCOE), 970	MCOE is an interactive alphanumeric editor that allows both the creation of new and the editing of existing time series data files. MCOE operates within a highly featured spreadsheet environment that allows data to be manipulated within the spreadsheet (points inserted, moved, etc.).
Rainflow Cycle Counter - (MCYC), 980	MCYC is used to process a time series signal by extracting fatigue cycles according to the rainflow cycle counting algorithm. MCYC is useful because it allows the user to count cycles using the same parameters for comparing and assessing various time signals.
Formula Processor (MFRM), 984	MFRM is an arithmetic and logical module which can be used to process formulae of varying complexity.
File Cut and Paste - (MLEN), 1016	MLEN is used to extract a portion of data from one file or several files, and load the extracted portions into a new output file. It can be used to concatenate (merge), individual data sets into a single output file. It can also be used to delete a selected portion of data from files.

Type	Description
Multi-File Manipulation - (MMFM), 1024	MMFM is an arithmetic and logical module which can be used to process formulae of varying complexity. The formulae themselves are defined by means of a command language which MMFM interprets and executes. The command language is, in effect, a simple programming language that gives users access to all parts of loading files, i.e. header, data area, and extra details keywords.
Peak-Valley Extraction - (MPVXMUL), 1031	MPVXMUL extracts turning points from single parameter files such as **.dac** and RPC multiple data - channel files. The MPVXMUL extraction process maintains synchronous phase by writing corresponding data values to all the output files whenever a turning point is found in any channel.
Simultaneous Values Analysis DAC/RPC - (MSIMMAX), 1032	MSIMMAX performs simultaneous values analysis on either multi-channels in a single RPC file or multiple DAC files from the same test.
Amplitude Distribution - (MADA), 1034	MADA calculates the probability density distribution and other function of a time signal.
Auto Spectral Density - (MASD), 1035	MASD performs a frequency analysis of a time signal to determine frequency content.
Fast Fourier Filter - (MFFF), 1036	MFFF creates a finite impulse response (FIR), filter by using the window method. After creation, MFFF automatically removes unwanted frequency components from time series data. High-pass, low-pass, band-pass or band-reject can be created.
Butterworth Filtration - (MBFL), 1043	This program takes a signal file and passes it through a Butterworth filter to produce an output signal file. The filter characteristic can be a low pass, high pass, band pass or band-stop. The filter order can be from 1 to 8 poles which will give a cutoff of between 6db and 48db per octave.
Frequency Response Analysis - (MFRA), 1049	The frequency response analysis, MFRA, analyses the response of a single input, single output linear system. Six files of statistics are generated as a result of this analysis including the cross-correlation.
Statistical Analysis - (MRSTAT), 1061	MRSTATS analyses a time signal and produces a number of statistics about that signal. RSTATS works by breaking the input time signal into segments, and statistically analyzing each segment. Each statistic is fed into an output signal file.
Header/Footer Manipulation - (MFILMNP), 1071	File manipulation (MFILMNP) allows both header and extra details manipulation. In addition it can check that a header conforms to MSC Fatigue conventions. MFILMNP can list and edit data file headers, and control the extra details data that can be written to files. The files types that can be edited include single parameter files e.g. `.dac` files and 2 parameter files e.g. `.mdf`. Three parameter histogram files can also be processed.

Type	Description
Advanced Fatigue Utilities	
Single Location S-N Analysis - (MSLF), 1083	The MSLF program models fatigue life in software to predict durability based on SN-curves derived from constant amplitude test results for specimens or components. It is the equivalent of MSC Fatigue's stress based analyzer, FEFAT, but accepts measured stresses as input from a single location.
Single Location e-N Analysis - (MCLF), 1092	MCLF is the equivalent of MSC Fatigue's crack initiation module, FEFAT, but in the testing world where the input is a measured strain history from a single location.
Cycle and Damage Analysis - (MCDA), 1111	MCDA calculates and displays cycles and damage distributions so that different test conditions may be compared and the reasons for variations in fatigue damage may be determined. Displays may be as histograms, continuous curves, or exceedance plots.
Cycles File Lister - (MCYL), 1116	MCYL is used to numerically list the contents of a histogram or cycle file. MCYL can also display a summary of the information resident in the file header region. MCYL also accesses the cycle results files generated by the MSC Fatigue fatigue analyzers FEFAT, MCLF and MSLFMSLF and create a rainflow matrix of cycles according to user specified engineering units and limits.
Time Correlated Damage - (MTCD), 1121	MTCD is a fatigue analyzer that can be used to pin-point fatigue damage within a loading history. It uses the local stress-strain approach to track local stresses and strains by means of a single-pass algorithm. An estimate of the total damage accrued by one pass through the load history is made and displayed graphically.
Single Location Vibration Fatigue - (MFLF), 1128	MFLF is a single location, stress-based fatigue analysis module that accepts stress response PSDFs as input.
Stress-Strain Analysis - (MSSA), 1130	MSSA processes rosetta data and finite element data from MSC Fatigue, including software strain gauges. MSSA creates outputs suitable for use by either the stress or strain-life fatigue analyzers.
Multi-Axial Life Analysis - (MMLF), 1130	MMLF is a single location multiaxial fatigue analyzer based on Crack Initiation. MMLF requires three strain input signals which typically come from strain gauge rosettes.
Crack Growth Data Analysis - (MFCG), 1132	MFCG calculates the Paris Law coefficient and exponent from actual raw test data obtained under constant amplitude loading conditions.
Kt/Kf Evaluation - (MKTAN), 1133	MKTAN stores and retrieves values for stress concentration factor (K_t) solutions for geometric details, and calculates K_t and K_f. It allows users without finite element analysis (FEA) software rapid and convenient access to K_t values for a range of common component geometries. The K_t values can be used in MTCD to predict the fatigue life of an engineering component.
Graphical Display Utilities	
Graphical Editing - (MGED), 1143	MGED is the multi-channel interactive graphical editor for time series data allowing online manipulation of a signal.

Type	Description
Multi-File Display - (MMFD), 1144	MMFD displays single parameter data files.
Quick Look Display - (MQLD), 1145	MQLD displays single channel data files.
Two Parameter Display - (MTPD), 1146	MTPD displays pared (X-Y) data files. For .mdf files.
, 1147	MPOD displays pared (X-Y) data files. For .pod files.
Three Dimensional Display - (MP3D), 1148	MP3D is the histogram and waterfall display module.
Linux Based Plotting Utility - (MQPLOT), 1149	MQPLOT is used to view or print plot (.plt) files. Up to 2000 plot files can be loaded onto MQPLOT and viewed on screen and/or printed or plotted as hard copy. A large number of output devices are supported. MQPLOT can also be used to convert plot files into different formats, for example .plt to .eps (encapsulated postscript).
Windows-Based Plotting Utility - (MWNPLOT), 1152	MWNPLOT is equivalent to MQPLOT but for the Windows environment.
Printer/Plotter Definition Module - (MPLTSYS), 1159	The MSC Fatigue system can produce graphical hardcopy on a wide range of output devices. These include pen plotters such as the wide range of HP devices, laser printer-plotters and dot matrix printers. Before any hardcopy can be produced, the plotters that are available on the system must first be defined. The plotter definition system controls all aspects of defining the available plotters, and their setup (plot size, position etc.). MPLTSYS allows definitions to be created, modified deleted and listed.
Plot/Pen Colors Utility - (MNCPENS), 1177	MNCPENS is a simple utility to allow users to change graphic colors on any plot.
File Conversion Utilities	
Binary/ASCII Convertor - (MDTA/MATD), 1178	The signal to ASCII module, MDTA, converts a single parameter, X-Y, or histogram binary file into ASCII format. It can also write or omit header details, and write an ASCII file as single or multiple column. MDTA is complimentary to module PTIME which converts ASCII files to binary files. The MADT module also converts ASCII files to binary, operates in batch mode, and converts multi-channel data to multi-files automatically.

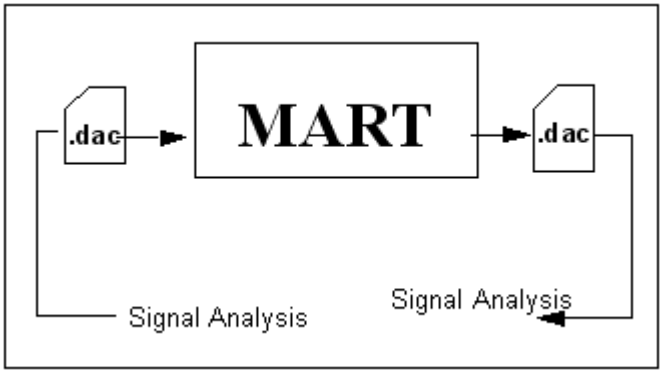
Type	Description
Signal Regeneration - (MREGEN), 1195	MREGEN can regenerate a single parameter signal file (.dac extension) from a three parameter range-mean cycles histogram file (.cyg type), regenerate a single parameter signal file (.dac extension) from a three parameter maximum-minimum cycles histogram file (.cyh type), regenerate a single parameter signal file (.dac extension) from a three parameter Markov Matrix (.mkh type), and generate a Gaussian series from a user supplied irregularity factor and save it as a .dac file.
RPC to DAC - DAC to RPC - (MREMDAC/MDACREM), 1202	This program extracts channels of data from MTS RPC^{tm} remote parameter (RPC) files, and creates a single .dac file for each channel of the RPC file. It is possible to selectively extract one or more channel numbers. A time window within the RPC file may be selected which will apply to all channels extracted. Both RPC II and RPC III files may be processed. Full details of the extraction and creation process can be saved to a report file. The reverse of this is MDACREM.
Cross-Platform Conversion - (MCONFIL), 1213	MCONFIL is a binary to binary file conversion program for transfer of files across multiple platforms.
Waterfall File Create - (MWFLCRE), 1216	WFLCRE creates a three parameter waterfall file from multiple single parameter files.

Advanced Loading Utilities

The following sections describe advanced loading utilities. There are several advanced modules for manipulating loading history files of format .dac and other formats (.cyh). It is assumed that the user has a good working knowledge of the loading capabilities of MSC Fatigue. Please review Loading Management (Ch. 5) first, if this is not the case.

Arithmetic Manipulation - (MART)

MART is a module which arithmetically manipulates standard MSC Fatigue data files. Files may be operated on in ways similar to that offered by a hand held calculator. Complex mathematical procedures can be built up by “chaining” together sets of MART commands. If necessary, the arithmetic manipulation can be applied to a section or window in the data file, leaving the remainder of the data unaffected.



MART carries out arithmetic operations on MSC Fatigue data files which have either the time-series, X-Y, or histogram formats, (1, 2 or 3 parameter files). MART supports the following arithmetic operations:

- Multiply or Divide every data value in a file by a constant.
- Add or Subtract a constant from every data value in a file.
- Normalize a data file to a specific mean.
- Raise every data value to a specified power.
- Extract the SIN, COS or TAN of every value in a data file.
- Take the Absolute value of every value in a data file.
- Linear transformation of the type Y = MX + C for every value in a data file.
- Logarithmic and Anti-Logarithmic transformations to base 10 or base e.

Throughout the course of the above manipulations the following conventions are observed:

- The tangent of 90 and 270 degrees is taken to be zero.
- Data files which contain negative values can only be raised to integer powers.
- Zero values are taken to be real numbers in the range: $-1.0^{-10} < n < 1.0^{-10}$
- Zero values raised to a negative power are set to zero.

Module Operation

The MART module can be run in one of the following 3 ways:

- From the Advanced Loading Utilities pulldown menu under Tools | MSC Fatigue (for Patran) or under Tools | Fatigue Utilities (for).
- In stand alone mode by typing **mart** at the system prompt.
- By incorporating the MART commands in a batch operation.

Modes 1 and 2 above are interactive. Once running in one of the interactive modes MART will display the main screen which has various options. Select an option, then supply a file name and the data necessary for the selected option.

Option	Description
Multiply **Divide** **Add** **Subtract**	MART will ask for the number by which the user wishes to multiply, divide, add, or subtract the data in the input file. The defaults are 1, 1, 0, 0, for Multiplication, Division, Addition, and Subtraction respectively for safety i.e., to produce an output the same as the input if the default is accepted accidentally. In addition to entering an explicit numeric value, it is possible to enter the name of an extra details keyword. See the section on data file formats, and the documentation associated with the File Manipulation module, Peak-Valley Extraction - (MPVXMUL), 1031, for more information on the extra details area. After accepting a keyword, MART will search the extra details area of the input file and assign the value associated with it to the arithmetic constant to be used. For example, suppose that an input data file contains an extra details keyword called "MODULUS" which has a value 205000 associated with it. Entering the word MODULUS in response to the above prompt will cause MART to use the value 205000 as the arithmetic constant. If the keyword cannot be found an error message will result.
Selecting Normalize to a new mean	If this option is selected, MART will ask for the new mean value. The current mean of the data file is offered as the default, selecting this default will produce an output with the same mean as the input. Extra details keyword input is also supported as explained for the first options.
Selecting Raise to a power	Any valid numeric value may be entered as the required power. Extra details keyword input is also supported as explained for the first options. Data files that contain negative values can only be raised to INTEGER powers.
Selecting Trigonometric functions	If this option is selected, MART will display a submenu alongside the main menu which allows the user to specify whether the conversion function will be Sine, Cosine, or Tangent. The user must specify the units of the input file (using File type), the amount of input file to be processed (From and To), and its Y-axis labels and units. Note that the TANGENT of 90 or 270 degrees is taken to be ZERO.

Option	Description
Selecting Absolute value of data	Selection of this option will cause MART to assign a positive sign to every negative value found in the input data file and write it out to the output file. Positive values will be written without change.
Selecting Y = MX + C	This option allows the input signal to be multiplied by one constant, M, and then to have a second constant, C, added to the result. They are incorporated into the well-known equation for a straight line (linear transformation). The value for M is entered in field 'Y = ' and the C value is entered in field 'X +'. Any number consistent with the arithmetic operation to be undertaken, non-zero values for division for example, may be entered. Note that the constant is added AFTER the multiplication by M. Extra details keyword input is also supported as explained for the first options.
Selecting Logarithmic functions	If this function is selected, MART will display a submenu to the main menu. Suboptions 1 and 2 will take the natural and base 10 logarithms of the input signal respectively and write the result to the output file. Options 3 and 4 carry out the reverse transformations i.e. e^x and 10^x respectively. If the input data file contains NEGATIVE or ZERO values, MART will return an error message. Also, a file cannot be Anti-logged if the resultant file would contain values greater than 1E30. After selecting a logarithmic function, MART will (as is always the case) ask for the name of the input and output file, and will then process the named input file.

If the NOTEBOOK keyword is defined within the local environment an entry of the information generated during the operation(s) of MART will be made within the MSC Fatigue notebook. See the Modifying the MSC Fatigue Environment (MENM), 1463 for details of the environment.

All of the main menu options use the same input screen, as shown in Figure 14-1 below. The field that changes according to the option is the third field down (marked **Option**). The other fields may or may not be grayed out depending upon whether they are necessary for that option.Whichever option is chosen from the main menu, the first field that the user must fill is the name of the input file.

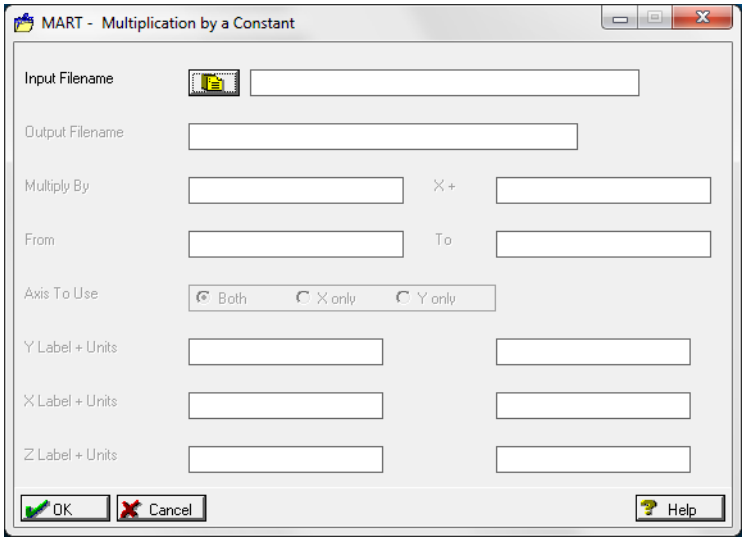


Figure 14-1 Input File Name and Options Screen

Each data field is explained below:

Option	Description
File Names	MART can process data files which are in either a time series, X-Y, or histogram format. By default it expects the input file to have a time series format and to have a file extension of .dac. If such a file is to be processed it is only required to enter its name, the .dac extension may be omitted. Files with the correct internal format but different file extensions must have their names entered in full, i.e. including the file extension. By default, MART expects to find the input data files to be resident in the user's directory. However, other directories can be accessed. The processed data will be written to an output file. As a default, this file will be created within the current directory and the extension of the input file is appended to the name entered. If another extension or destination is required then it must be entered in full. The name of the input file is offered as the default output file name. Acceptance of this default means that input data will be overwritten and so MART will prompt for confirmation to proceed. In addition, if the name of a file which already exists is specified, MART will also prompt for confirmation to overwrite it. When the input and output files have been named then some or all of the following fields will have to be completed.
From and To	Input files are of a finite length and not all of a file has to be processed. A time segment of a file may be processed. A time segment of a file can be specified by From (start time) and To (finish time). The start of analysis 'From' option supports keywords (START, extra details etc.) and calculations such as START+5 (seconds). If the file is not a time history then enter the appropriate unit e.g. Hz for an .asd file. Data before the start time is not changed in any way, or deleted. The finish time of analysis 'To' also supports keywords such as END, extra details, END-5. If the file is not a time history then enter the appropriate unit, e.g. Hz for an .asd file. The 'To' value must be greater than the start time of the analysis. The default is the end of the file. Data after the 'To' time is not changed in any way, or deleted.

Option	Description
Y-axis label + units	The Y-axis label is a text string which is stored in the header area of the data file and is used to describe the parameter that is represented on the Y-axis. The text entered will automatically appear on any plots generated by any suitable graphics module such as MQLD or MMFD. The Y-axis units are represented by a text string which is stored in the header area of the data file. The text entered will automatically appear on any plots generated by any suitable graphics module such as MQLD or MMFD. In the case of a file with a histogram format (`.cyh` or `.mkv` for example) MART will additionally prompt with:
Z-axis label + units	The Z-axis label is a text string which is stored in the header area of the data file and is used to describe the parameter that is represented on the Z-axis. The text entered will automatically appear on any plots generated by any suitable graphics module such as P3D. The text string must not be longer than 12 characters. The Z-axis units are represented by a text string which is stored in the header area of the data file. The text entered will automatically appear on any plots generated any suitable graphics module such as MP3D. The text string must not be longer than 10 characters.
2 parameter files	If X-Y or 2 parameter files (often with the `.mdf` extension) are being processed then, MART prompts with the following fields:
Axis to use **Both ** **X only** **Y only**	This refers to which axis is going to be arithmetically operated upon. For example if multiplying only the X-axis, then the Y-axis data remains the same etc. Note that if only one axis is being operated upon then only the label and units for that axis can be user specified.

Batch Operation

```
mart /opt=l/inp=test/out=test1/ov=y/mul=10.1/con=-123
```

In this example, MART will process an input data file called `test.dac` by multiplying every value contained therein by 10.1 and then subtracting 123 from the result. The output will be written to another data file called `test1.dac`. If a file with this name already exists then it will be overwritten.

As another example of the use of simple arithmetic operations to significantly alter the nature of a time history consider the sine wave illustrated in Figure 14-2 below.

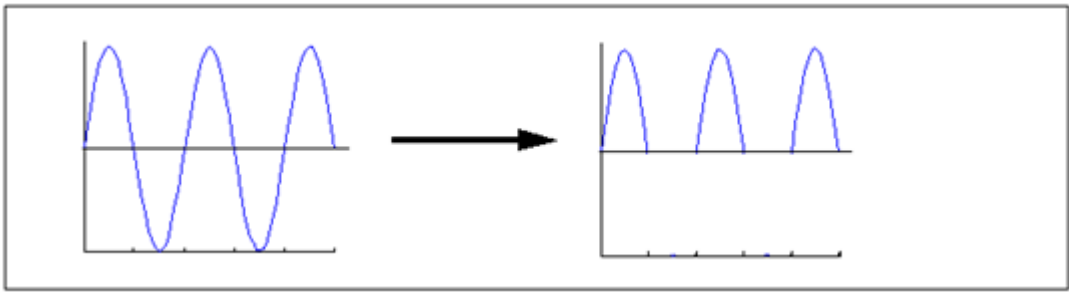


Figure 14-2 Fully Reversed Sine Wave With a Zero Mean Converted

Suppose that, for some reason, it was required to modify the data in such a way that all the negative values were set to zero. To achieve this the following strategy could be adopted.

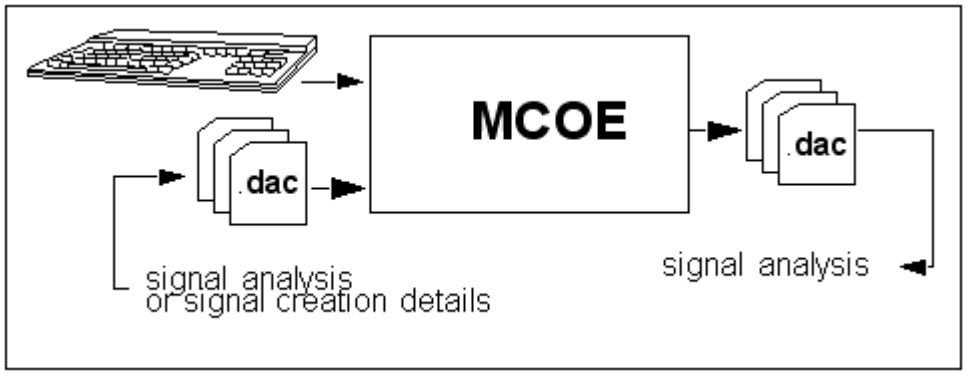
- Using MART's option Absolute value take the absolute value of the input data file and save the output in a scratch file.
- Use the multi-file manipulation module, MMFM, to add the contents of the original data file to that contained in the newly created scratch file and store the result within an output file. Remember that the result of this addition cannot be stored within either of the two input files.
- Finally, use MART again to divide the contents of the output file by 2 and save the result within the input file.

The batch command lines to achieve these operations are given below:

```
mart /opt=B/inp=sine/out=temp/ov=y/*=tt
mmfm /opt=1/inp=sine,temp/out=sine1/ov=y/*=tt
mart /opt=D/inp=sine1/out=sine1/ov=y/*=tt/val=2
```

Multi-Channel Editor - (MCOE)

MCOE is an interactive alphanumeric editor that allows both the creation of new and the editing of existing time series data files. MCOE operates within a highly featured spreadsheet environment that allows data to be manipulated within the spreadsheet (points inserted, moved, etc.).



MCOE can be used to create or edit MSC Fatigue data files which have a single parameter “time series” format. The time series structure is used to describe single channel data where the increment between samples is constant.

MCOE can create or edit multi-channel sets of files provided they have the same base offset and sample rate in every file in the set. Data is displayed on the screen in columns, and in response to MCOE commands, many functions are available to perform column operations.

Module Operation

The MCOE module can be run in one of the following ways:

- From the **Advanced Loading Utilities** pulldown menu under **Tools | MSC Fatigue** (for Patran) or under **Tools | Fatigue Utilities** (for).
- In stand alone mode by typing **mcoe** in response to the system prompt.

There are 3 modes of operation of MCOE:

Option	Description
Browse	Allows existing data to be viewed but not edited.
Edit	Allows existing data to be viewed and edited.
Create	Allows new files to be created and edited.

All three modes use similar screen layouts and share a large number of capabilities. The differences are that each mode has capabilities appropriate to its purpose. For example, in Browse mode it is impossible to change data. Therefore, capabilities such as being able to enter new data values via the edit window are disabled. In Edit and Create modes these functions do operate.

Most of the differences occur between what is, and what is not available on the Options menu. Rather than largely duplicate an explanation for each mode of operation, it is simpler to highlight the differences between them, as shown in the table below.

After selecting Browse, Edit or Create, the first stage of a run of MCOE involves the specifying of the file or files upon which it will operate.

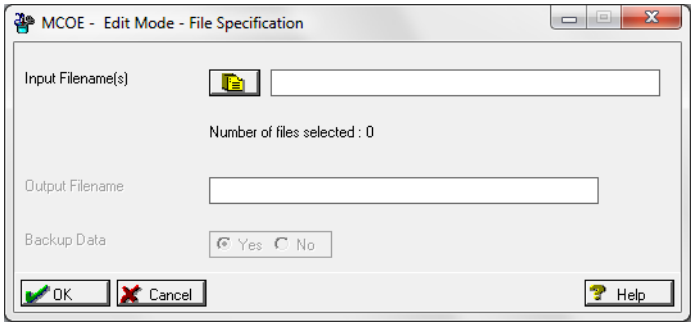


Figure 14-3 Specifying the Input File Name

Editing an existing file

If two or more input files are entered, then MCOE offers the option to preserve the original data in a backup file. The name of this file is always the same as that of the input file but with a `.bak` extension. However, if only one input file is to be edited, then a specific output file name is asked for (rather than automatically reusing the input file name).

Up to 256 files can be specified here. The Pick List facility is probably the easiest way of specifying multiple file entries. All input files must have the same sample rate.

MCOE will expect the input to have an extension of `.dac`. Files with the correct internal format but different file extensions, such as frequency spectra (`.psd`), or joint probability density distributions (`.jpd`), must have their file extensions entered explicitly.

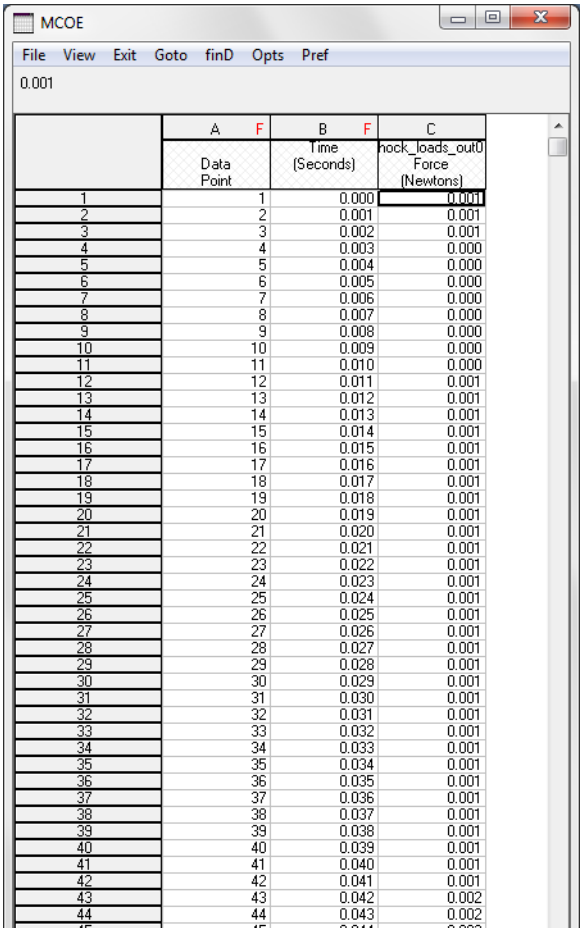
Creating an all new file(s)

This option will cause MCOE to ask for the files Sample Rate and Base Offset (as per Figure 14-3 above). Up to 256 new file names can be entered in the input file name field, possibly by using quick entry naming conventions such as `test(1-10)` which will create `test01.dac, test01.dac... test10.dac`

Option	Description
Sample Rate and Base Offset	The Sample Rate is the number of points per second of the new files which is any number greater than zero. The Base Offset is the starting point of the sample relative to the zero line of the X-axis, i.e. if a Base Offset of 200 is entered and the units are seconds, then the created file will be assigned a start time of 200 seconds rather than the default of zero seconds. If it is intended to create an another type of data set, such as a spectrum for example, then the "sample rate" must be used to define the increment between samples. For example, consider the creation of a spectrum with a maximum (Nyquist) frequency of 250 Hz and consisting of 128 data values. These parameters imply an inter-sample separation, spectral width, of 250 / 128 or 1.9531 Hz and so, in this case, the "sample rate" the would be 1 / 1.9531 or 0.512.
X and Y axes Labels and Units	The axes labels fields ask for the text string which will be assigned to the X and Y-axis labels of all the files to be created. The axes units fields ask for the text string which will be assigned to the X and Y-axis units of all the files to be created. Labels and units may be also defined and edited within the spreadsheet by selecting 'View Data' from the options menu.

Spreadsheet Screen

When the input files have been specified and the screen accepted, MCOE will move to its spreadsheet screen, which is shown below.



	A F	B F	C
	Data Point	Time (Seconds)	hock_loads_out0 Force (Newtons)
1	1	0.000	0.001
2	2	0.001	0.001
3	3	0.002	0.001
4	4	0.003	0.000
5	5	0.004	0.000
6	6	0.005	0.000
7	7	0.006	0.000
8	8	0.007	0.000
9	9	0.008	0.000
10	10	0.009	0.000
11	11	0.010	0.000
12	12	0.011	0.001
13	13	0.012	0.001
14	14	0.013	0.001
15	15	0.014	0.001
16	16	0.015	0.001
17	17	0.016	0.001
18	18	0.017	0.001
19	19	0.018	0.001
20	20	0.019	0.001
21	21	0.020	0.001
22	22	0.021	0.001
23	23	0.022	0.001
24	24	0.023	0.001
25	25	0.024	0.001
26	26	0.025	0.001
27	27	0.026	0.001
28	28	0.027	0.001
29	29	0.028	0.001
30	30	0.029	0.001
31	31	0.030	0.001
32	32	0.031	0.001
33	33	0.032	0.001
34	34	0.033	0.001
35	35	0.034	0.001
36	36	0.035	0.001
37	37	0.036	0.001
38	38	0.037	0.001
39	39	0.038	0.001
40	40	0.039	0.001
41	41	0.040	0.001
42	42	0.041	0.001
43	43	0.042	0.002
44	44	0.043	0.002

Figure 14-4 The MCOE Spreadsheet Environment

There are three main areas on a spreadsheet screen.

- The **menu bar** across the top of the screen which contains the buttons for manipulating the data and navigating around the columns and rows.
- The **data area** itself which consists of rows and columns of data cells.
- The **status bar** along the bottom of the form. It contains temporary or volatile data such as the data value currently highlighted or being edited and the data file's name.

Each area will be explained separately, starting with the data area.

Option	Description
The Data Area	Figure 14-4 shows the editing of a .dac file, as specified in the input field of Figure 14-4. All columns from column C onwards (maximum 256) contain data values. **Column A** contains a simple data point/row label. It can contain numerical designations for billions of rows. The labels can NOT be edited. Note that the numbers are not actually stored as part of the .dac file; they are present in MCOE for purposes of identification, and are implied rather than stated in the post MCOE .dac file **Column B** contains the X-axis values of the file data. If a scale factor has been applied to the range column then the values appearing in this column will have been multiplied by this factor and the scale factor number is displayed above the column. Values in column B can not be edited. **Column C** and all subsequent columns contain data values, i.e. the Y-axis values. The letter F that appears in columns A and B means that they are fixed, cannot be edited, and will be on-screen at all times. A letter G means that a column is grouped (see the Options menu for details).
The Edit Window	This is an edit window. The contents of whichever cell is highlighted will be displayed here. Any highlighted Y-axis data value can be edited simply by typing a new number. As soon as typing starts the edit window becomes active and values can be entered in the usual way. If a value is entered beyond the end of the current column, then all values between the previous end - and the new column end positions - are linearly interpolated.
The Header Bar	The menu options contain what are probably the most easily usable methods of navigating around the spreadsheet and manipulating its contents which are explained below.

The File Menu

Option	Description
OK	Exits MCOE completely.
Back	Goes back to MCOE's main menu.

The View menu

Option	Description
Up\ Page Up key	Scrolls up 1 screen of data values.
Down\ Page Down key	Scrolls down 1 screen of data values.
Home/ End	Goes to row 1 or the last row.
Left/ Right	Move left or right n columns (if there are columns to move to).
First/ Last	Goes to column 1 or the last column.
Goto option	Goes to a user-specified row.

The Find menu

The FIND menu offers a data search facility. It is offered by clicking the mouse on the 'Find' option.

All searches operate in the current column starting at the current row. If the column end is reached and no match found the search resumes from the first row. A message is displayed if no match is found.

Option	Description
Find nearest match	A target value is input. All column data is compared against it and the *closest* match reported. Any other values equally close are found using the NEXT option
Less than value	A value is entered and the column is searched for the point at which the column date descends *below* the threshold value.
Greater than value	A value is entered and the column is searched for the point at which the column data rises *above* the threshold value
Between 2 values	Two values which form the target range are entered. The first value found within this range is reported.

The Options menu

This menu offers the principal tools for editing and manipulating the data. When a multi-column edit operation is selected a question is asked which determines the scope of the operation. The possibilities are: **Current column**, **Grouped columns**, **All columns**. The default is current column until MCOE is set differently.

Option	Description
Gap	Allows data to be repetitively appended to the end of the current column. The user must supply the number of points in the gap from which the equivalent number of points, NPTS, is determined. MCOE then moves NPTS on from the column's end and waits for the user to enter a number that will fill the new end cell. The gap between the previous and the new column end positions is bridged by a linear ramp.
Delete	Allows a block of data to be deleted from the columns. The block is copied to the clipboard and may be inserted elsewhere in the spreadsheet using the Paste option. The user is asked for the start and end positions of the block. If the block extends beyond the end of any of the nominated columns the data cannot be copied to the clipboard and a warning question offers the option to cancel the operation. Deleted data may be restored to any row position but multi-column data deletions can only be restored to their original columns. Delete is a multi-column operation.
Copy	Allows a block of data to be copied to the clipboard without it being deleted from the spreadsheet. The data may be pasted elsewhere in the spreadsheet. The user is asked for the start and end positions of the block. If the block extends beyond the end of any of the nominated columns the operation is aborted and a warning issued. Copy is a multi-column operation.
Paste	Allows the clipboard's contents to be inserted into the spreadsheet. A question is asked to determine the point *before* which the data is to be inserted. Data cannot be pasted to a position starting beyond the end of the column into which it is being inserted. Multi-column paste always returns data to the columns from which it was taken. Single column paste offers the option of returning data to its original column or to the current column (if different). Paste is a multi-column operation.
Cycle	Allows a sequence of saw-tooth ramps to be appended at the end of a column. The user must supply the following: ■ The start point of a cycle (Maximum or Minimum) ■ The cycle range (from Peak to Trough) ■ The number of cycles to be appended ■ The mean value of the first half-cycle ■ The mean value of the last half-cycle The cycles are then appended to the end of the current column. Cycle is a single column operation.

Option	Description
Insert	Allows a linear ramp to be inserted into a column. The user must supply the following: ■ The duration of the data to be inserted ■ The first and last values to be inserted ■ The point *before* which the data is to be inserted The first and last values are then inserted into the column and all intermediate points are linearly interpolated. Insert is a single column operation.
Append	Allows data to be added to the end of a column. The user must supply the following: ■ The duration of the data to be inserted ■ The first and last values to be inserted The first and last values are then inserted into the column and all intermediate points are linearly interpolated. Append is a single column operation.
Join	Allows the data between two existing points in the columns to be joined by a simple linear ramp. Questions are asked to determine the two points between which the columns are to be joined by a linear ramp. A warning is issued if a join point is beyond the end of the data. Join is a multi-column operation.
Export	This writes column data to a TAB separated ASCII text file with a user supplied name and a `.txt` extension. The exported file may then be imported into other commercially available spreadsheets (e.g. MicroSoft EXCEL). Export is a multi-column operation.
Plot	Allows the contents of the columns to be viewed graphically by loading them into the MSC Fatigue module Multi File Display (Multi-File Display (MMFD), 251). Plot is a multi-channel operation.
Rescale and offset	Performs a simple linear re-scaling of the columns according to the equation: Y = FACTOR * X + CONSTANT. Questions are asked to determine the following: ■ The start and end points of the rescaling region ■ The value of FACTOR and SCALE in the above equation If the specified region extends beyond the end of any column to which the operation is being applied a warning is issued. Rescale is a multi-column operation.
View file Header	Toggles the spreadsheet between the data editing mode and the file header information editing mode.

Option	Description
View extra details:	The input files may have additional information about themselves appended to their extra details areas. You may also add new extra details area keywords to the input files with this option.
Format Columns	Columns can be locked onto the screen, formed into groups, and hidden from view. **Fixed:** A fixed column will remain visible on the screen at all times regardless of how the spreadsheet viewing windows is maneuvered. A highlighted letter F appears in the banner of columns which are fixed. **Grouped**: Specific columns of interest may be grouped together and submitted, as a group, to multi-file manipulation options (e.g. Plot). A highlighted letter G appears in the banner of grouped columns. **Hidden:** When a column is hidden it is no longer displayed in the spreadsheet window. The column still forms part of the spreadsheet and is still subject to multi-file data manipulation commands.
Exit	Accepts all edits/modifications made during the current session and exits the program. Confirmation is requested before this operation is carried out.
Quit	Rejects all edits/modifications made during the current session and exits the program. Confirmation is requested before this operation is carried out.

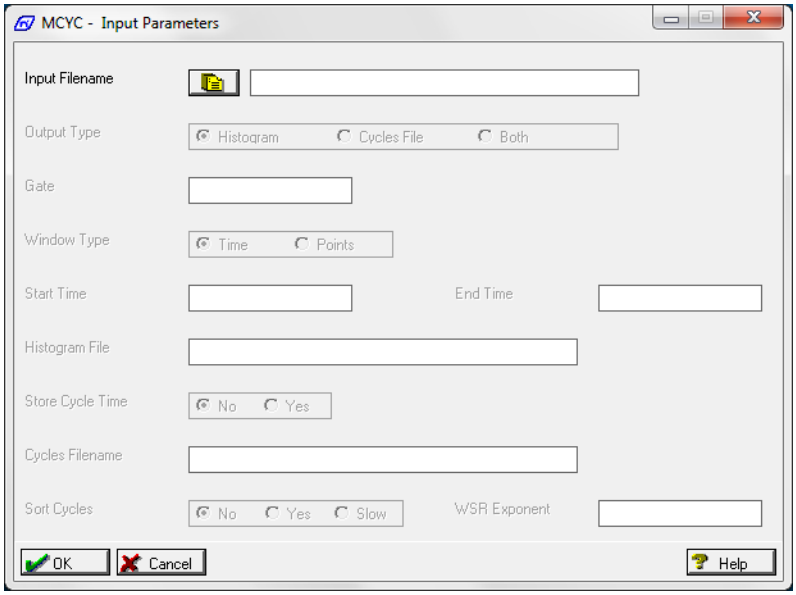
The Preferences menu

The spreadsheet environment is customized by the preferences menu. When MCOE is exit or quit the current preferences are stored as the defaults for the next run of the program.

Option	Description
Point pick	The row specification method. When an option such as delete cells is carried out MCOE can ask for the location to delete in terms of point number (column A), X values (column B) or by pointing and clicking on the required rows. Click on cells: All edit operations prompt for rows to be specified by clicking on the required rows. The column in which the mouse was last clicked becomes the current column for the operation. Point numbers: All edit operations determine the required row or row range by asking for row numbers. X values range: All edit operation determine the required row or row range by asking for X values. Values which are entered at other than the discrete positions at which they occur are rounded to the nearest match.
X-value format	The number of decimal places in the X-values column (B).
Data format	The number of decimal places in the data column (from C onwards). For both range and data format, if a value is so large that the number of places is not possible the display reverts to Exponential format.
Scale X-values	Allows X-values to be scaled by a factor. For example, setting the scale factor to 1000.0 allows a signal originally sampled in seconds to be displayed in milliseconds.
Export options	The export options capability allows the binary `.dac` file currently loaded to be saved as a text file suitable for import into commercial spreadsheet and database applications. When selected, a form appears asking for the specification of what data is to be written and column separation delimeters, such a tabs, commas, etc.

Rainflow Cycle Counter - (MCYC)

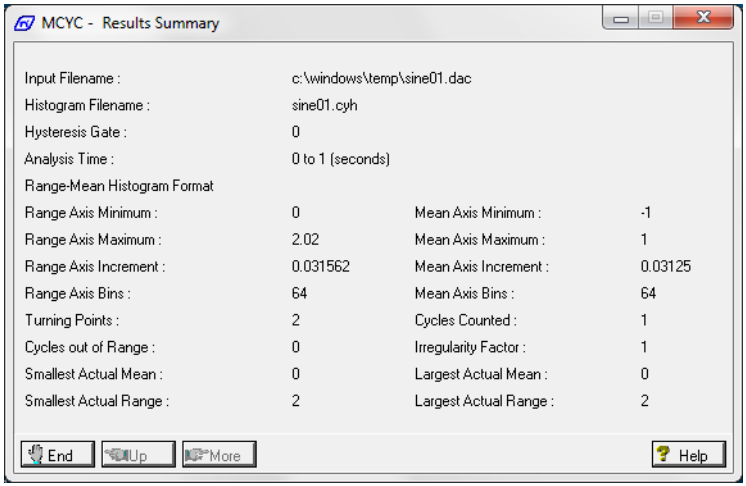
The rainflow cycle counter, mCYC, processes a time series signal, by extracting fatigue cycles according to the rainflow cycle counting algorithm. The utility is useful as it allows the user to count cycles using the same parameters (gate, range, bin width) for comparing and assessing various time signals. The results are presented in the form of a range-mean or a max-min matrix which can be displayed or used as input to mCLF or MSLF. In addition, a file containing a description of each cycle can be generated. If the time of each cycle can be stored, this file may also be used in crack growth analysis.



For example, use PTIME to Copy from central the time history, **SAETRN.**

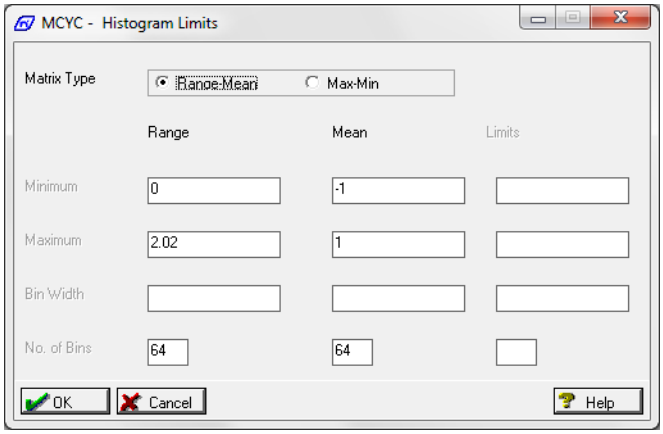
1. Invoke MCYC by typing **mcyc** from the system prompt or select the Rainflow Cycle Counter option from the **Advanced Loading Utilities** pulldown menu under **Tools | MSC Fatigue** (for Patran) or under **Tools | Fatigue Utilities** (for).
2. Select input file. By default, mCYC expects the input data to be a standard .dac file but files with the correct internal format but different file extensions must have their names entered in full e.g. filename.pvx.
3. Select Output Type - **Histogram**, **Cycles Files,** or **Both**. The layout of the lower part of the above screen and next screen (shown here) will depend on the selection made here. If Histogram or Both is selected the Gate, Histogram Filename, Range Parameters and Mean Parameters are prompted for in the screen as shown on page 416.
4. Enter Gate to filter cycles (e.g. 75 which is approximately 10% of the max indicated value 747). The value entered here must be in physical units (usually microstrain) and greater than zero. If the gate value is more than half the size of the largest cycle in the input file, an error message will be issued. All cycles bigger than the gate will be counted.
5. Enter Window Type - **Time** or **Points**. Selecting one or the other changes the next input to time or points.
6. Specify Start Time and End Time or Start Points and End Points. (e.g. start+6 -start 6 seconds or points from start and end-100 - end 100 seconds or points from end). Default is **start** and **end**.
7. Specify histogram filename if the Output Type selected is Histogram or Both.
8. Store Cycle Time and Cycles Filename are only activated if Cycles File or Both are selected above. If **yes** is selected, a time based cycles file (.tcy) is generated that can be used in crack growth analysis. If **no** is selected, a .cyc file is generated that stores the ranges and means from the largest cycle onwards.

9. Specify Cycles Filename if the Output Type selected is Cycles File or Both. The .tcy file can be reordered if desired using the Sort Cycles. If **no** is selected the cycles are sorted in order of size. Note that if sorting is not carried out, it is possible that a crack growth analysis will be wrong since the order of the cycles is important in crack growth analysis. If **yes** is selected the cycles are sorted in order of time. **Slow** selects an old method for sorting which requires less disk space than the current faster method. If disk space is not a concern, do not use this option.
10. The WSR component field is for the exponent on which to base the weighted stress range (range 2-10). The value of 2 would give a rms of stress ranges, the default of 3 gives a root mean cube used typically with welded joint S-N curves. For crack growth, the Paris Law exponent, m, should be used.
11. Pressing **OK** takes you to the next screen or, if only Cycles files was selected, a results summary is displayed as shown below.



Histogram Limits

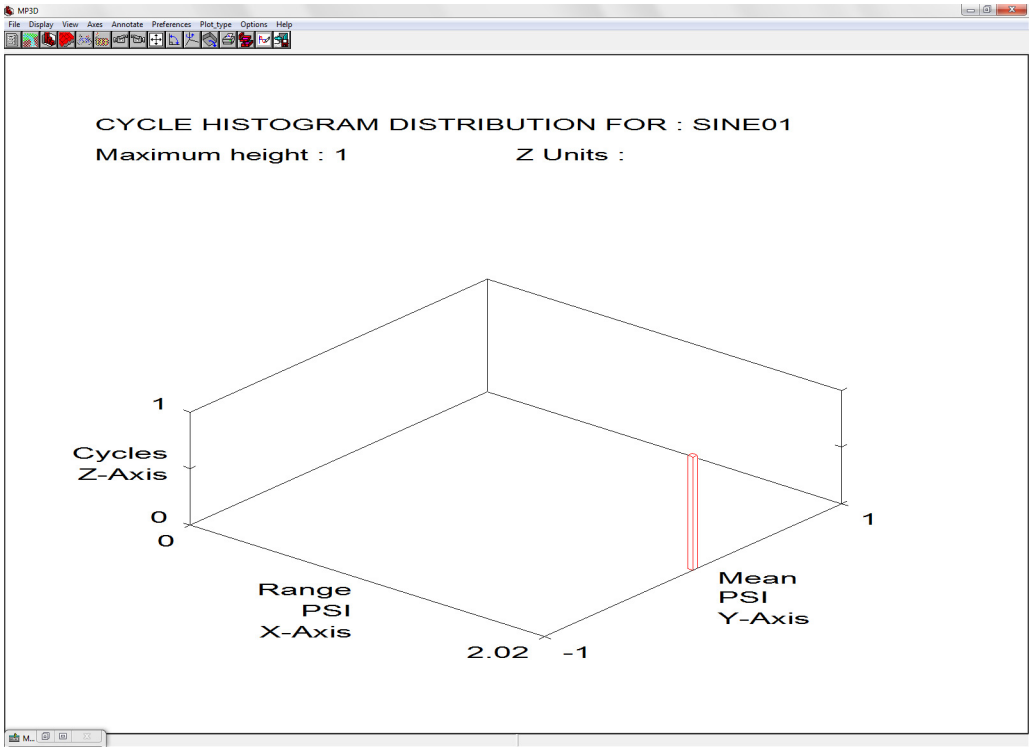
The following form is displayed if both or Histogram is selected above:



The fields that are activated depend upon whether the environment keyword HISTFORM is set to MINMAX (use the full range from minimum to maximum values) or BINSIZ (specify on the minimum and the bin width).

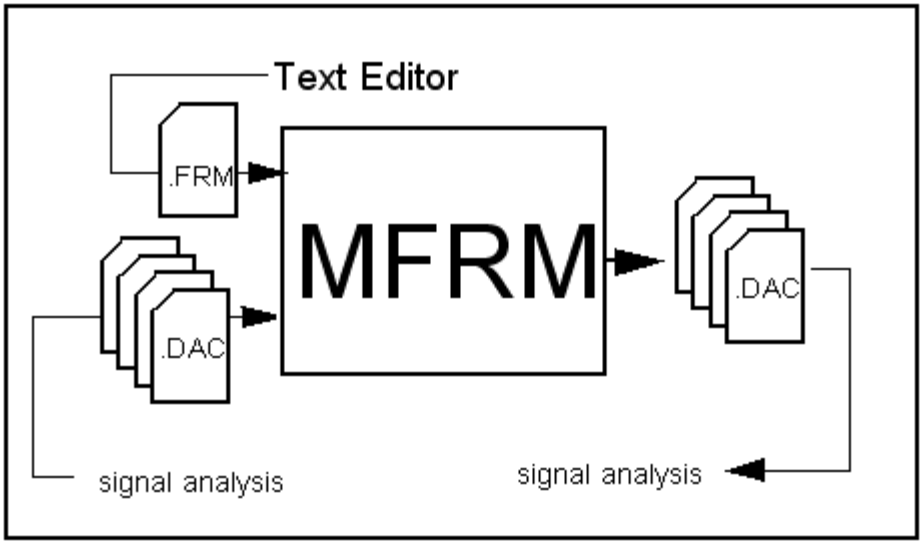
1. Min (Range) - For the purposes of scaling the histogram, the range of the smallest cycle to be represented in the histogram must be entered in physical units. If there are any cycles smaller than the minimum range specified, then those cycles will be excluded from the histogram.
2. Max (Range) - For the purposes of scaling the histogram, the range of the largest cycle to be represented in the histogram must be entered in physical units. If there are any cycles larger than the maximum range specified, then those cycles will be excluded from the histogram.
3. No. of Bins - To scale the histogram, specify the number of bins into which to classify the cycle ranges. Any integer up to a maximum of 128 may be entered.
4. Min (mean) - For the purposes of scaling the histogram, the smallest mean value to be represented must be entered in physical units. If there are any cycles whose mean values are smaller than the value specified, then those cycles will be excluded from the histogram.
5. Max (mean) - For the purposes of scaling the histogram, the largest mean value to be represented must be entered in physical units. If there are any cycles whose mean values are greater than the value specified, then those cycles will be excluded from the histogram.
6. No. of Bins - To scale the histogram, specify the number of bins into which to classify the cycle means. Any integer up to a maximum of 128 may be entered.
7. If the environment variable HISTFORM=BINSIZ (set in mENM) then this field is displayed. The format (size and shape) of the histogram can be set by specifying the bin width. By default, the program calculates the bin width needed to include the maximum values in the input file but the user can enter a smaller or larger width.
8. The max-min toggle will plot a histogram based on a count of cycles between the between the maximum and minimum cycle. To scale the histogram, specify the number of bins into which to classify these cycles. Any integer up to a maximum of 128 may be entered.

The output histogram using the range-mean option is shown below.



Formula Processor (MFRM)

MFRM is an arithmetic and logical module which can be used to process formulae of varying complexity. The formulae themselves are defined by means of a command language which FRM interprets and executes. The command language is, in effect, a simple programming language that gives users access to all parts of MSC Fatigue files, i.e., header, data area, and extra details keywords. MFRM supports the following functions:



- Addition
- Subtraction
- Multiplication
- Division
- Exponentiation
- SIN and arc SIN
- COS and arc COS multiplication
- TAN and arc TAN
- Degrees to Radians Conversion
- Radians to Degrees Conversion
- Logs to the base 10
- All can be written into an MFRM commands file.

MFRM interprets and executes commands which are stored in a formula command file. Individual commands, when grouped together, make up the required formula and so MFRM may be regarded as a formula processor. For example, the following formula command file could be used to prompt for the names of two input files which need to be added together, and the output loaded into a third file called `result.dac`

```
;
; --- Define all the files ---

%QYFIL /FILE=F1 /PROMPT="Enter first filename" /TYPE=INPUT
%QYFIL /FI=F2 /PR="Enter second filename" /TY=IN
%DFFIL /FILE=F3 /NAME=RESULT /TYPE=OUTPUT
;
; --- Carry out the addition ---
;
```

```
%BEGIN
;
F3 = F1 + F2
;
%END
;
; --- Finish ---
```

The template file must have an ASCII format. It can be created or edited through the use of a text editor or word processor. However, care must be taken to ensure that no control characters are output from the word processor (WP). Control characters cannot be interpreted and so will cause MFRM to stop processing and display an error message.

Note: Save the MFRM file as an ASCII file, not in the WP's proprietary format. Give it the extension `.frm`

The above command structure has two important regions.

The first is a constant, or file definition, region in which all the constants and file types for both input and output are defined.

The second is the arithmetic region in which all the arithmetic operations to be carried out are specified. The arithmetic region must always be embedded within the definition region and start with the %BEGIN command and finish with the %END command. The %BEGIN and %END commands themselves should be considered as belonging to the definition region.

From the above example, it is useful to note that, in the definition region, file specification is through keyword switches which may be expressed in their full form such as /FILE= or in a more compressed form such as /FI=. For more details of the keyword switches associated with each of the file definition commands are provided in the section Syntax of Commands in the Definition Region, 986.

Operational Constraints

Data Files

All data files which are input for processing by MFRM must have the following properties, otherwise they cannot be processed.

- All input files must physically exist on the path specified
- All input files must have the same length
- All input files must have the same sampling rate
- They must be single parameter files (normally with the `.dac` file extension
- The default output data file is single parameter with a `.dac` file extension
- The default output list file extension is `.lst`

The following pages contain details of the MFRM command language and syntax to be used in the MFRM command files.

Syntax of Commands in the Definition Region

A command in the definition region is made up of up to three sections:

- A command mnemonic
- One or more mandatory switches
- Some optional switches

As an example, consider the %QYFIL command:

```
Command Mnemonic    Mandatory Switches            Optional Switch
%QYFIL              /FI=F1 /PR="INPUT"/TYP=IN     /LA="Pounds"/OV=Y
```

> **Note:** Only one command may be issued per line and all questions and definitions must be declared before the %BEGIN command. All arithmetic, trigonometric or conversion commands must appear between the %BEGIN and %END commands.

All commands and switches can be written in either upper or lower case letters.

Switches are interpreted from a maximum of 3 characters, any further characters are ignored so that to define the /FILE= switch it is sufficient to write /FI =.

The following command mnemonics are supported in the definition region:

Command Mnemonic	Description
%QYFIL	Prompt for the entry of a filename.
%DFFIL	Define a file with a given filename.
%QYCON	Prompt for the entry of a constant.
%QYINT	May be used instead of %QYCON or GET_VARIABLE to define a variable that will only accept integers.
%DFINT	This command which may be used instead of %DFCON or DEFINE_VARIABLE to define a variable which will only take integer values.
%DFCON	Define a value for a constant.
%DLOOK	Define a lookup table.
%PRINT	Print a string to screen or file
%BEGIN	Define the start of the arithmetic processing region.
%END	Define the end of the arithmetic processing area.

The %QYFIL Command

This command causes MFRM to prompt for a filename. When using this command, three further definitions must be made. Firstly, the filename must be allocated an internal MFRM file number. Secondly, the prompt string must be defined followed by the nature of the file, input or output, specified. These additional definitions are made through the use of the mandatory command switches detailed below:

Command Switch	Description
/FI=Fn	Allocates a file number to the named file. /FI=F1 allocates the code F1. Up to a maximum of 64 files may be allocated, i.e. /FI=F32.
/PR="string"	String is a character string representing the required prompt. The string must be enclosed by quotation marks.
/TY=	Defines the type of the file. /TY=IN specifies an input file. /TY=OU specifies an output file. /TY=LI specifies a list file. If /TY=VA, then a variable length output file may be specified. Any files created in this way will only have data written to them when the file is specified on the left hand side. For example, if F2 is a variable file, then %IF (F1 > 20) F2 = F1

%ENDIF will cause only the values of F1 greater than 20 to be written to F2.

/DE is the default switch and may now specify an environment keyword. To do this, do not enclose the string in quotes. i.e., /DE="TEST101" will set the default to TEST101/DE=.DAC will set the default to the last file written to the .DAC environment keyword.

Examples of valid %QYFIL commands:

```
%QYFIL /FILE=F1 /PROMPT="Enter input filename" /TYPE=INPUT
%QYFIL /FI=F2 /PR="Enter output filename" /TY=OU
```

The following optional switches are available for use with files which are specified as output files:

Command Switch	Description
/LA="string"	String is a character string representing the Y-axis label.The string must be enclosed by quotation marks.
or /LA=$Fn	Where Fn is a defined input file. The label is copied from the Y label of file Fn.
/UN="string"	String is a character string representing the Y-axis units. The string must be enclosed by quotation marks.
or /UN=$Fn	Where Fn is a defined input file. The label is copied from the Y units of file Fn.
/OV=option	Whether to overwrite the output file or not. If option is Y, overwriting is automatic. If N, the file will not be overwritten, if Q then the user will be asked whether the file should be overwritten.

Example use of the %QYFIL command to define an output file:

```
%QYFIL /FI=F1 /pr="Enter results filename" /ty=ou
/la="Distance" /un="mtrs."/ov=q
```

The LA and UN parameters are ignored if the output type is LIST.

The %DFFIL Command

This command defines the name of a data file explicitly. When using this command the file must be allocated an internal MFRM file number, the filename defined, and the nature of the file, be it an input or output, specified. These definitions are made through the use of the mandatory command switches detailed below:

Command Switch	Description
/FI=Fn	Allocates a file number to the named file.
/FI=F1	allocates the code F1. Up to a maximum of 64 files may be allocated, i.e., /FI=F32.
/NA=FNAME	Where FNAME is the filename. If no extension is specified DAC will be assumed.
/TY=	Defines the type of the file. /TY=IN specifies an input file. /TY=OU specifies an output file. /TY=LI specifies a list file.

Examples of valid %DFFIL commands:

```
%DFFIL /FILE=F1 /NA=file1 /TYPE=INPUT
%DFFIL /FI=F2 /NA=result /TY=OU
%DFFIL /FI=F2 /NA=data /TY=LIST
```

As a result of the first command line, MFRM will try to access an input file called `file1.dac`. From the second line, it will attempt to write out an output file called `result.dac`. If `file1.dac` does not exist or `result.dac` does, then an error condition will arise which will cause MFRM to cease processing. The third line will create an output file called `data.lst` to which the %PRINT command can write.

The following optional switches are available for use with files which are specified as output files:

Command Switch	Description
/LA="string"	String is a character string representing the Y-axis label. The string must be enclosed by quotation marks
or /LA=$Fn	Where Fn is a defined input file from which the variable is copied.
/UN="string"	String is a character string representing the Y-axis units.The string must be enclosed by quotation marks.
or /LA=$Fn	Where Fn is a defined input file from which the variable is copied.
/OV=option	Whether to overwrite the output file or not. If option is Y, overwriting is automatic. If N, the file will not be overwritten, if Q then the user will be asked whether the file should be overwritten.

Example use of the %DFFIL command to define an output file:

```
%DFFIL /FI=F1 /NA=RESULT/ty=ou /la="Distance" /un="mtrs."
```

The command DEFINE_FILE may be used as well as %DFFIL. The new /TY=VA keyword as described above is also supported.

The %QYCON Command

This command causes MFRM to prompt for the value of a constant. When using this command, it is necessary to assign the constant to an internal MFRM number and also to define the prompt string. These definitions are made through the use of the mandatory command switches detailed below:

Command Switch	Description
/CO=Cn	Allocates an internal number to the constant.
/CO=C1	allocates the code C1 to the required constant. Up to a maximum of 1000 constants may be allocated.
/PR="string"	String is a character string representing the required prompt. The string must be enclosed by quotation marks.
/DE="Default"	Gets the default string for the constant

Examples of valid %QYCON commands:

```
;
; -- Ask for constants --
;
%QYCON /CONSTANT=C1 /PROMPT="Enter amplitude"
%QYCON /CO=C2   /PR="Enter frequency value"/DE="5"
```

The command GET_VARIABLE may be used as well as %QYCON. A new keyword /NA=NAME may be used to define a name to represent the variable. For example,

```
GET_VARIABLE    /CO=C3 /PR="Enter speed" /NA="SPEED"
DEFINE_VARIABLE    /CO=C4 /NA="DISTANCE"
```

(see %DFCON below for DEFINE_VARIABLE). Then instead of

```
C4=C3*5.75
```

one may use

```
DISTANCE=SPEED*5.75
```

Output files do not have to be declared before being created

%QYINT or GET_INTEGER

This command may be used instead of %QYCON or GET_VARIABLE to define a variable which will only take integer values.

The %DFCON Command

This command defines the value of a constant explicitly. When using %DFCON, it is necessary to assign the constant to an internal FRM number and also to define its value. These definitions are made through the use of the mandatory command switches detailed below:

Command Switch	Description
/CO=Cn	Allocates an internal number to the constant.
/CO=C1	allocates the code C1 to the required constant. Up to a maximum of 1000 constants may be allocated.
/VA=n	Defines the value of the required constant. The value may be integer or decimal.

Examples of valid %DFCON commands:

```
;
; -- Define the constants --
;
%DFCON /CONSTANT=C1 /VA=10
%DFCON /CO=C2  /VA=5.123
```

Optionally, %DFCON can be used to define header or extra details and store them in a constant. If the command /FILE=<file ID> is added, it will assume that the /VAL switch defines either a header or an extra details keyword, the data coming from the file specified by <file ID>. For example:

```
%DFCON /CO=C6 /FI=F1 /VAL=MEAN
%DFCON /CO=C7 /FI=F3 /VAL=EXTRA
```

The first example extracts the mean from the file defined as F1. In the second example, the value of the extra details keyword EXTRA will be stored in C7. Clearly the file specified in /FI= must be previously defined in a %QYFIL or %DFFIL, and must be of type INPUT. Some header keywords are:

TOTPTS	The number of points in the file.
SRATE	The sample rate of the file.
BASE	The base X-value of the file.
INCREM	The X-axis increment.
MEAN	The mean value of the data points.
SDEV	The standard deviation of the data points
YMAX	The maximum data value in the file.
YMIN	The minimum data value in the file.
RMS	The root mean square of the file
MAXLOC	The X location of YMAX
MINLOC	The X location of YMIN

The command DEFINE_VARIABLE may be used as well as %DFCON.

The %WTVAL command

This command writes to the extra details area of a specified file. It can write constants, file values, and temporary variables. The %WTVAL can only be used inside a %BEGIN and %END construction. The /VA switch can be: /VA=Cn constants, /VA=Fn a file value, /VA=Tn temporary variable.

An example of a line using /WTVAL is shown below:

```
%WTVAL /FI=F2/KW=SPOINT/VA=C2
(command) (file to write to) (EDA keyword) (value of keyword)
```

A more comprehensive example is shown below. It should be used in conjunction with a signal and trigger file. The trigger start and end points are put into the signal file's extra details area.

```
;
; Get trigger start and end point numbers
; and put them in the signal file extra
; details area
```

```
;
%DFFIL /FI=F1 /NA=SIGNAL /TY=INP
%DFFIL /FI=F1 /NA=TRIGGER /TY=INP
%QYCON /CO=C1 /PROMPT="ENTER TRIGGER VALUE"
%DFCON /CO=C2 /FI=F1 /VAL=POINT
%DFCON /CO=C3 /FI=F1 /VAL=TIME
;
%BEGIN ;-- Initialise temporary variables on point 1 only --
%IF (C2 = 1)
T1 = 0.0
%ENDIF
;-- trigger start --
%IF (F2 = C1)
%IF (T1  1.0)
%WTVAL /FI=F1 /KW=SPOINT /VA=C2
%WTVAL /FI=F1 /KW=STIME /VA=C3
T1 = 1.0
%ENDIF
%ELSE
;-- trigger end --
%IF (T1 = 1.0)
%WTVAL /FI=F1 /KW=EPOINT /VA=C2
%WTVAL /FI=F1 /KW=ETIME /VA=C3
T1 = 0.0
%ENDIF
%ENDIF
%END
```

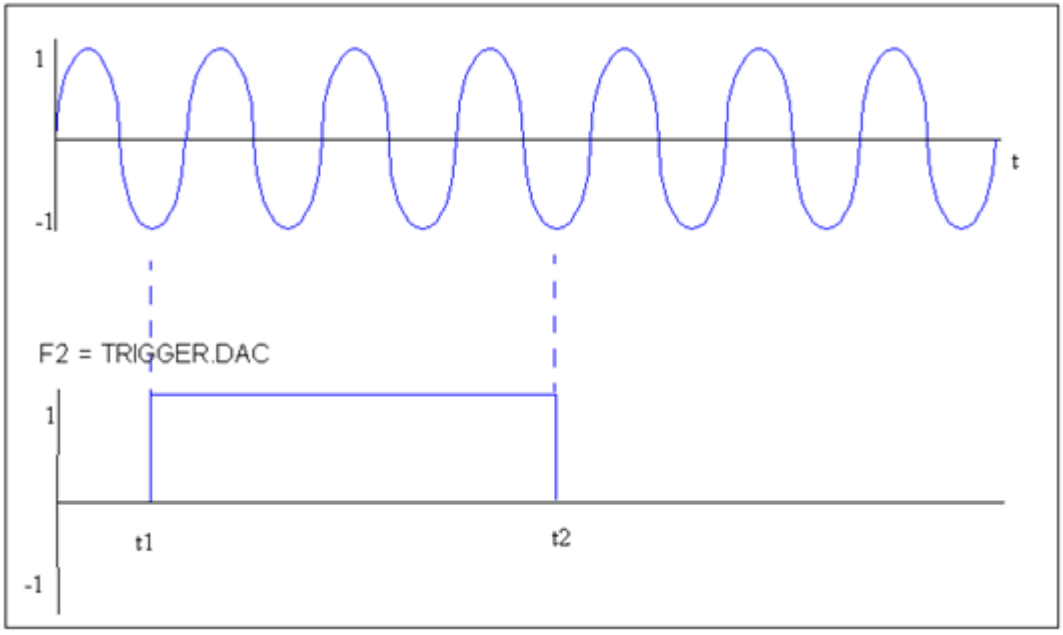


Figure 14-5 Diagram Showing `signal.dac` and `trigger.dac`

Points t1 and t2 are put in the EDA of `signal.dac`

SPOINT=Y-axis value of `signal.dac` at t1
STIME= time at t1
EPOINT=Y-axis value of `signal.dac` at t2
ETIME= time at t2

Special keywords

In addition, two special keywords are provided.

POINT	Stores the current point being processed.
TIME	Stores the time of the point being processed.

The values of these 2 keywords are modified dynamically throughout the %BEGIN...%END processing loop.

Any other keywords are expected to be found in the extra details area, and their value must convert to a single valid real number.

The %DLOOK Command

Defines a lookup table to be used by the LUT<n> function. Up to 10 lookup tables may be defined, either in the MFRM file as pairs of X-Y points or as a DAC file, or as a paired (X-Y) file. To define the lookup table, it must be given an ID, and the type must be specified. If the values are defined in the MFRM file, the number of values must be given. The maximum number of pairs of values in each lookup table is 100. The definition is made using the switches below.

Command Switch	Description
/TA=n	Allocates an internal number to the table.
/TY=VALUES	Defines the table as a set of values stored in the MFRM file.
/NVALS=N	Defines the number of values.
/TY=FILE	Defines the table as being stored in an external file
/NA=name	Defines the filename. No assumption is made about the file extension.

If TYPE=VALUES, the next NVALS lines of the MFRM file must contain a lookup table as X,Y pairs, space delimited. The values of X must be increasing. For example:

```
%DFLOOK /TA=1 /TY=VALUES /NVALS=5
1 100
3 200
7 300
9 600
10 665
```

If TYPE=FILE, the file must be a DAC or MDF format. The number of pairs must not exceed 100. In the case of the MDF, X must be increasing.

The command DEFINE_LOOKUP may be used as well as %DLOOK.

The %PRINT Command

The %PRINT command writes data to a list file or the screen. If it is used outside of the %BEGIN...%END loop, file names and constants may be displayed in addition to text. The following switches are used:

Command Switch	Description
/MEssage=string	The string to be written, including file names and constants (see below)
/FF	Will issue a form feed command to the printer which will cause it to scroll up 1 page
/FI=Fn	An output file to write to. Must be defined as type LIST. If omitted, the data is written to the screen.

A message string has the following format:

```
/ME="String"+5X+F1+30T+" another string "+C4
```

Entities must be separated by the + symbol. Strings must be enclosed in quotation marks. The entity Fn will cause the name of the file to be displayed at that point. The entity Cn will cause the value of constant Cn to be displayed at that point. The entity nX causes n spaces to be displayed. The entity nT causes the next entity to be displayed in column n (used for tabulation). n must be greater than the length of the string to that point. Examples:

```
%PRINT /FI=F4/ME=F1+25T+F2+50T+F3 %PRINT /ME=25T+"This is a file

heading"+3X+"mean = "+C5
```

The command PRINT may be used as well as %PRINT.

The %BEGIN Command

The %BEGIN command marks the start of the arithmetic operations region; it informs MFRM that until it encounters the %END command, all command lines are to be treated as arithmetic operations. The arithmetic operations region contains all the arithmetic, trigonometric and conversion commands required for execution of the formula.

Note:	All file names and constants MUST be defined before the %BEGIN command is issued. Each formula template file MUST contain a %BEGIN command.

The command BEGIN may be used as well as %BEGIN.

The %END Command

The %END command marks the end of the arithmetic operations region. It informs MFRM that no further arithmetic operations will be required. Only comment lines can follow the %END command. The command END may be used as well as %END.

Note: The %END command cannot be issued before a %BEGIN has been defined. Each formula template file MUST contain a %END command.

Example of the use of %BEGIN and %END:

```
;
; --- Definition Region ---

%QYFIL /FILE=F1 /PROMPT="Enter first filename" /TYPE=INPUT
%QYFIL /FI=F2 /PR="Enter second filename" /TY=IN
%DFFIL /FILE=F3 /NAME=RESULT /TYPE=OUTPUT /OV=Q
%DFCON /CO=C1 /VA=5.123
;
; --- Carry out the addition ---
;
%BEGIN
;
T1 = F1 * C1
F3 = T1 + F2
;
%END
;
; --- Finish ---
```

In this example file F1 is multiplied by constant C1, the result stored in a temporary file called T1 added to file F2. The final result is loaded into file F3.

Syntax of Commands in the Arithmetic Region.

Commands in the arithmetic operations region are made up of variables and manipulative operands. Operands are themselves subdivided into arithmetic, trigonometric, and conversion functions. The syntactical relationship between variables and operands is given below:

```
{ output variable } = { input variable 1 } { operand } { input
variable 2 }
```

Note: Spaces between variables, operands and the equals sign are allowed.

The only restriction concerning the placement of variables on either side of the equals sign is that any file which has been declared as an INPUT, through the use of the /TY=INPUT switch within either the **%QYFIL** or **%DFFIL** commands, cannot be treated as an output variable, i.e., it cannot appear on the LHS of the equals sign. Therefore the following would NOT be allowed:

```
;
%DFFIL /FILE=F1 /NAME=RESULT /TYPE=INPUT
%DFCON /CO=C1 /VA=5.123
;
%BEGIN
F1=F1*C1
F1=T1*F1
```

Temporary variables and files declared as OUTPUTS, through the use of the /TY=OUTPUT switch within either the **%QYFIL** or **%DFFIL** commands, can appear on both sides of the equality, e.g., consider the following fragment:

```
;
%DFFIL /FILE=F1 /NAME=RESULT /TYPE=OUTPUT /OV=Y
%DFCON /CO=C1 /VA=5.123
;
%BEGIN
T1=F1*C1
T1=T1*F1
F1=F1*C1
```

Syntax of Variables

Three type of variables are used to represent values within the arithmetic operations region:

Variable Type	Value
Files declared by the user	Denoted by Fn where n is the file number (1-64), e.g. F25.
Constants declared by the user	Denoted by Cn where n is the number of the constant (1-1000), e.g. C10.
Temporary file variables	Denoted by Tn where n is the temporary user variable number (1-1000), e.g. T27. Variables can also be given names (see %DFCON and %QYCON).
%QYINT or GET_INTEGER	This command which may be used instead of %QYCON or GET_VARIABLE to define a variable which will only take integer values.

Syntax of Arithmetic Operands

The table given below details the symbols required to achieve specific arithmetic functions within the arithmetic region:

Arithmetic Function	Symbol
Addition	+
Subtraction	-
Multiplication	*
Division	/
Exponentiation	^

All the above arithmetic operations require two input variables and one output variable. Operations, may be carried out on files, temporary variables and constants. Upper or lower case lettering is acceptable and spaces can be inserted if required.

Examples usage of arithmetic operands:

```
;
; -- Add two files together --
;
F1 = F2 + F3
; -- Subtract two files --
;
F1 = F3 - F4
;
; -- Divide two files --
;
F1 = F3 / F4
;
; -- Raise a file by a power --
;
F1 = F1^C2
;
; - Multiply a file by a constant and place in a temporary
variable
;
T5 = F1*3.141592
```

Assignments

A file or temporary variable appearing on the left of an equation may be simply assigned a value, with no operator. For example:

```
;
; Assign a file value
;
T3=F1
;
; Assign a numerical value
;
T3=3.141592
;
; Assign a constant value
;
T3=C1
```

%LOOP or LOOP

This is a command which may be used as the start of the arithmetic region. All commands between LOOP and BEGIN will be executed only once at the beginning of the processing. ENDLOOP or %ENDLOOP should then be used to terminate the loop. All commands after END and before ENDLOOP will be executed only once at the end of processing.

Example:

```
%BEGIN
%PRINT /ME="At the start of processing"
%LOOP
;Do the calcs.....
%ENDLOOP
%PRINT /ME="At the end of processing"
%END
```

The two PRINT statements will appear only once.

Arithmetic functions

The following arithmetic functions are available:

Trigonometric Function	Symbol
log base 10	LOG
absolute value	ABS
maximum of two values	MAX
minimum of two values	MIN
degrees to radians	RAD
radians to degrees	DEG

The LOG function requires one input variable, in rounded brackets, and one output variable.

Note: The argument of the LOG function must be greater than zero.

Examples:

```
;
; -- Log of a file --
;
F2 = LOG(F1)
;
; -- Log of a temporary variable --
;
F3 = LOG(T4)
;
; -- Log of a constant --
;
F3 = LOG(3.141592)
```

The ABS, RAD and DEG functions also takes one argument. The MIN and MAX functions take two arguments, of any type. Examples are:

```
;
F1 = ABS (F1) ;
F2 = MAX (F1,F3) ;
F2 = MIN (F1,0)
;
F3 = MIN (C3,50)
;
```

Syntax of Trigonometric Functions

The table given below details the symbols required to achieve specific trigonometric functions within the arithmetic region:

Trigonometric Function	Symbol
sine	SIN
arc sine	ASIN
cos	COS
arc cos	ACOS
tan	TAN
arc tan	ATAN or AT2

All trigonometric functions, except AT2, require one input variable, in rounded brackets, and one output variable. The inputs must be files rather than constants.

The input arguments required for the SIN, COS and TAN functions must be in **RADIANS**, each being treated modulo 2*PI.

AT2 is a real floating point arc tangent with two parameters, i.e. it represents the $\tan^{-1}(a/b)$ and so it requires TWO input variables, in round brackets (a,b) and one output variable. The output from this function is as follows:

a/b = 0	(a=0)	: angle = 0
a/b = ∞	(b=0)	: angle = π/2
a/b = ∞	(a=0,b=0)	: angle undefined.

The input arguments for ASIN, and ACOS must be less than or equal to 1.0.

Note: The outputs from the ASIN, ACOS, ATAN and AT2 functions is always in RADIANS.

In all instances upper or lower case letters are acceptable, and spaces can be inserted if required. Examples:

```
;
; -- Sine of a file --
;
F2 =  SIN(F1)
;
; -- Tangent of a temporary variable --
;
F4 = TAN(T3)
;
; -- Real arc TAN function ------
;
F3 = AT2 (T1,T2)
```

Lookup Table Functions

The function LUTn(value) can be used on the right hand side of an equation. The lookup table n must have been defined in a **%DLOOK** command. The value must lie within the bounds of the lookup table, otherwise a fatal error will result. Example:

```
%DFFIL /TYP=INP/FI=F1/NAM=INPUT
%DFFIL /TYP=OUT/FI=F2/NAM=OUTPUT/OV=Q
%DLOOK /TA=1/TYP=VALS/NVALS=5
0 1
1 3
2 9
3 19
4 33
;
%BEGIN
F2=LUT1(F1)
%END
```

Get Point Value Functions

The function FGET(FNO,POINT) gets the value at point number POINT from any input file FNO.

%IF, %ELSE and %ENDIF

Logic can be performed within the **%BEGIN...%END** loop. The equations within the **%BEGIN...%END** are performed on each data point in the file, in sequence.

The commands **%IF**, **%ELSE** and **%ENDIF** can be used to control which equations are performed. The **%ELSE** is optional, and the structures can be nested up to a maximum of 5 deep.

The %IF command is structured as follows:

```
%IF ( value operator value )
```

where value may be a file, e.g. F1, a temporary variable, e.g. T4, a constant e.g. C3 or a number, e.g. 1000.

The operator may be one of the following:

Operator	Meaning
=	Equal to
<>	Not equal to
>	Greater than
>=	Greater than or equal to
<	Less than
<=	Less than or equal to

The **%ELSE** and **%ENDIF** commands have no arguments.

A typical structure is:

```
%BEGIN
%IF ( F2 = 0 )
```

```
   F3 = 0
%ELSE
   F3 = F1/F2
%ENDIF
%END
```

The spacing is optional. A nested example is:

```
%BEGIN
%IF ( F2 = 0 )
%IF ( F1 > 0 )
F3 = 1E10
%ELSE
%IF ( F1 > 0)
F3 = -1E10
%ELSE
F3 = 0
%ENDIF
%ENDIF
%ELSE
F3 = F1/F2
%ENDIF
%END
```

The commands **%EXIT** and **%ABORT** can be used to end the calculation if an error in logic occurs. They have the syntax:

```
%EXIT /ME=string
%ABORT /ME=string
```

%EXIT will stop the calculation and save the output files up to the point where the error occurred. **%ABORT** will delete all output files and terminate. For example,

```
%BEGIN
%IF ( F2 = 0 )
%ABORT /ME="Cannot divide by zero, sorry!"
%ELSE
F3 = F1/F2
%ENDIF
%END
```

The message format is the same as the **%PRINT** command (see below). The **%ABORT** and **%EXIT** commands cannot appear outside an **%IF..%ENDIF** structure, since the program would always terminate at the first point.

The **%PRINT** command, which was used in the definition section, may also be used within the calculation section. The syntax is the same, except that Fn will display the value of file n, and Tn will print the value of a temporary variable. For example,

```
%DFFIL /FI=F1/NAM=INPUT/TYP=INP
%DFCON /CO=C1/FI=F1/VAL=POINT
%BEGIN
%IF (F1 > 100 )
%PRINT /ME=C1+25T+F1
%ENDIF
%END
```

will print to the screen the point number and value of all points where the value exceeds 100.

```
%DFFIL /FI=F1/NAM=INPUT/TYP=INP
%DFFIL /FI=F2/NAM=LIST/TYP=LIST/OV=Y
%DFCON /CO=C5/FI=F1/VAL=TIME
%BEGIN
%IF ( F1 = 0 )
%PRINT /FI=F2/ME="Cannot invert at time "+C5
%ELSE
T3=1/F1
%PRINT /FI=F2/ME=C5+25T+T3
%ENDIF
%END
```

will print different messages to a file called LIST.LST depending on the value of the file.

The **%BREAK** command breaks the calculation for this data point and moves to the next. Effectively jumps to the **%BEGIN**.

```
%PAUSE or PAUSE
```

This command causes the program to pause and wait for a key press at any point during the processing loop.

Synonyms for %IF, %BREAK, etc.

%IF , %ELSE and %ENDIF	These may be replaced by IF, ELSE and ENDIF.
%EXIT and %ABORT	These may be replaced by EXIT and ABORT
%BREAK	...may be replaced by BREAK.

Error Codes and Messages

The following pages contain a list of the possible errors that can be produced by illegal commands or conditions in MFRM. The error numbers that correspond to each error condition are also given. For some errors MFRM will display the section in the command file that is causing the error. Suggested remedies are given for each error code.

ERROR	Meaning	Suggested remedy
1	Incorrect keyword	An illegal % command has been found in the template file. Edit the file and correct the command.
2	Illegal command file instruction.	A syntax error has occurred in the template file. Edit the command file and correct the syntax.
3	Invalid switch.	An illegal switch has been incorporated in the % command that is shown with the error code. Edit the template file and correct the switch.
4	No such input file.	The input file declared by the %DEFIL command, or the input file entered by the user does not exist in the specified directory. Edit the template file and define the correct filename or enter the correct filename when prompted.
5	Output file already exists.	The output file declared by the %DFFIL command, or the output file entered by the user already exists. Either define or enter another filename, or delete the existing version.
6	Missing switches in command line.	There is a required switch missing from the % command that is shown with the error code. See the sections above for details of the correct switches and edit the template file accordingly.
7	Syntax error in switch.	There is a syntax error in the switch that is shown with the error code. See the sections above for details of the correct switches and edit the template file accordingly.
8	File assignment number greater than 64.	FRM only allows 64 files to be defined in any one template file. Limit the number of files in the command file to 64.
9	Multiple file numbers.	There is more than one file defined with the same file number. Edit the template file and define each file with a unique file number.
10	There are more than 1000 constants defined.	MFRM only allows 1000 constants to be defined in any one template file. Use fewer constants in the command file.
11	Multiple constant numbers.	There is more than one constant defined with the same constant number. Edit the template file and define each constant with a unique number.
13	Too many temporary variables are defined.	FRM only allows 1000 temporary variables to be defined in any one template file. Use fewer temporary variables by overwriting unwanted variables.

ERROR	Meaning	Suggested remedy
14	Error in default option.	
15	Undefined variable appears on RHS of = sign.	A temporary variable has been placed on the right hand side of the equals sign without it being assigned a value. Edit the template file and either change the temporary variable or assign it a value.
16	Undefined file.	A file number has been used in the template file, but there is no file assigned to it. Edit the template file and either change the file number, or assign a file to the file number using the %DFFIL or %QYFIL commands.
17	Undefined constant.	A constant number has been used in the template file, but there is no value assigned to it. Edit the template file and either change the constant number, or assign a value to the constant number using the %DFCON or %QYCON commands.
18	Illegal tangent function.	A calculation has occurred that involves taking the tangent of a value in the range 1.570796326 to 1.570796327 or the tangent of a value in the range 1.570796326 to 1.57079632.
19	Illegal arc cosine function.	A calculation has occurred that involves taking the arc cosine of a value greater than 1, or of a value less than -1.
20	Illegal arc sine function.	A calculation has occurred that involves taking the arc sine of a value greater than 1, or of a value less than -1.
21	Illegal log function.	A calculation has occurred that involves taking the log of a value less than zero.
22	Input files have different lengths.	The input files must all have the same length. Rerun FRM with files that have the same number of data points in each file.
23	Input files have different sample rates.	The input files must all have the same sample rate. Rerun FRM with files that have the same sample rate.
24	Corrupted template file.	The template file has become corrupted and cannot be interpreted. Try and edit the template file and then attempt rerun FRM again. If this is unsuccessful it may be necessary to recreate the template file.
25	Attempt to raise a negative number to a non-integer power.	An error has occurred in the course of a ^ calculation
26	No such command file.	
27	Cannot open command file. (Check if file is locked).	An error has occurred reading the template file
30	Invalid lookup table ID.	The lookup table Id must be in the range 1-10.

ERROR	Meaning	Suggested remedy
31	Lookup table already defined.	An attempt has been made to define two lookup tables with the same ID.
32	Invalid lookup type. Use /TY=File or /TY=Values.	
33	The number of lookup values must be in the range 2-100.	
34	Lookup table file open or read error.	
35	Number of lookup table values does not match /NV command.	
36	Lookup table out of sequence.	
37	There must be two real values on a lookup table entry.	The above errors may occur when defining lookup tables.
38	Invalid overwrite command, use /OV=N, /OV=Y or /OV=Q.	
39	Lookup table not defined.	
40	Input value out of lookup table range.	The above errors occur when using a lookup table.
41	Number of constants exceeds 100.	Too many numbers have been used in the calculation loop.
42	Equation must contain an = sign.	
43	Equation must have a RHS expression.	An invalid equation structure has been encountered.
44	Invalid command encountered inside {%BEGIN %END}	Only IF,ENDIF,ELSE,PRINT,ABORT and EXIT are allowed after %
45	%IF structures nested too deeply.	
46	%ELSE encountered outside %IF structure.	
47	Consecutive %ELSE commands encountered.	

ERROR	Meaning	Suggested remedy
48	%ENDIF command has no matching %IF.	
49	%EXIT or %ABORT encountered outside %IF structure!	
50	Missing "(" or ")" in %IF command.	
51	Missing operator in %IF command.	
52	Invalid or missing LHS of %IF command.	
53	Invalid or missing RHS of %IF command.	
54	Incomplete %IF structure - missing %ENDIF.	The above errors indicate invalid IF..ELSE..ENDIF structures
55	String too long.	
56	Internal string table has exceeded 1024 characters.	The /ME parameters have exceeded internal limits.
57	Invalid constant value or missing file in %DFCON	
58	File not defined.	
59	Attempt to get details from an output file.	
60	Error opening file to get DFCON keyword.	
61	Error reading file header to get DFCON keyword.	
62	Unable to convert DFCON keyword to a value.	
63	Unable to find DFCON keyword.	The above errors occur when using DFCON to load header or extra details into a constant.
64	Attempt to use a list file in calculations.	
65	Attempt to %PRINT to a non-list file.	
66	Error writing to list file.	

ERROR	Meaning	Suggested remedy
67	Invalid format for message string.	
68	Expanded message string is too long.	
69	Too many spaces for message string.	
70	Temporary variables cannot be displayed outside { %BEGIN ... %END } structure.	
71	Invalid tabulation in message string.	
72	Cannot tabulate backwards.	The above messages occur when trying to %PRINT.
74		Checks that there is at least 1 valid input file. If there is NOT then error 74 is issued and the program terminates

Module Operation

The MFRM module can be run in one of the following three ways:

- From the **Advanced Loading Utilities** pulldown menu under **Tools | MSC Fatigue** (for Patran) or under **Tools | Fatigue Utilities** (for).
- In Stand alone mode by typing **mfrm**
- By incorporating the MFRM commands in a batch operation

Once initiated in either of the interactive modes one or two MFRM will display the following screen.

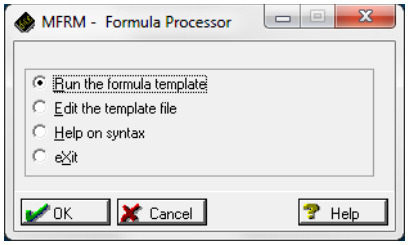


Figure 14-6 Screen One of MFRM

Additional menu options will become available when a formula template file has been named. An overview of the module structure is shown below in Figure 14-7.

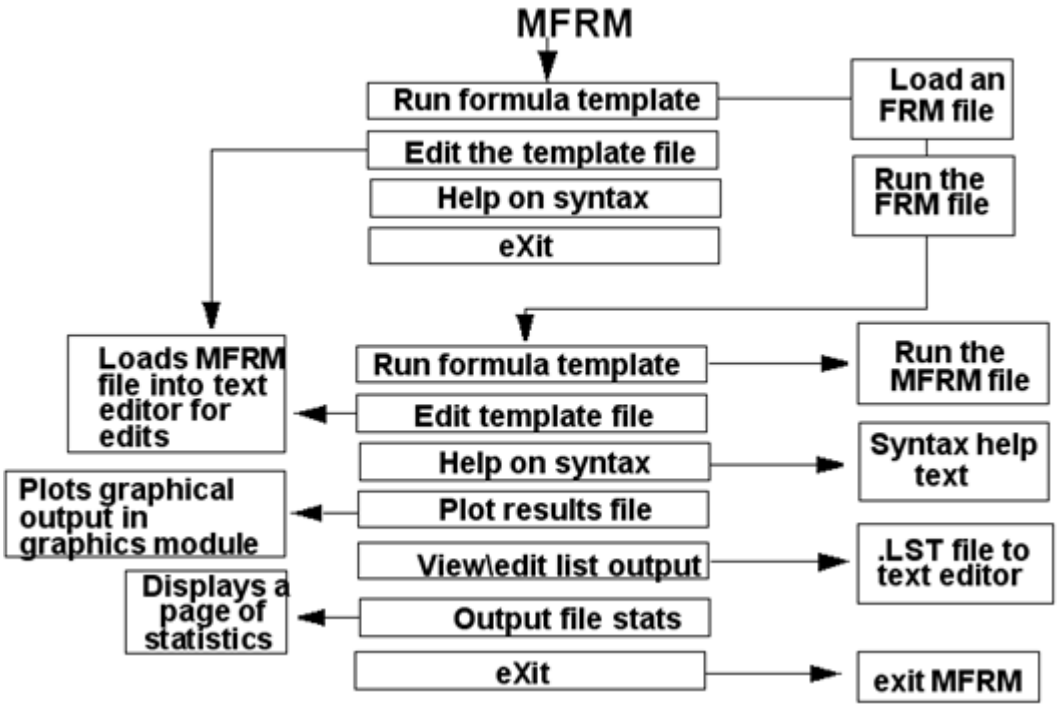


Figure 14-7 The Overall Module Structure of MFRM

The menu options on screen one are as follows:

Option	Description
Run the formula template	An MFRM template file must be named. If selected a file input screen appears. A command Filename (template file) must be entered in this field. Probably the easiest way of naming a file is by using the F3/List facility which brings up the file browser. The template file is an ASCII file such as might be created using a text processor. If it is created in a word processor then it must be saved as an ASCII file; a word processors own format usually contains hidden formatting characters embedded in the text, and these will confuse MFRM. By default the template file must have a `.frm` extension, although the filename can be anything as long as MFRM can interpret it's contents. MFRM manipulates data files which are in either a time series or histogram format. The `.dac` extension may be omitted. Files with the correct internal format but different file extensions such as spectra or histograms must have their names entered in full, i.e., including the file extension. **Note**: The input data files being processed, must all be of the same length, i.e., contain similar numbers of samples and must have the same sample rate. If these conditions are not met, MFRM will issue an error message. The MLEN PTIME modules can be used to ensure conformity with these conditions. When a valid file has been entered and accepted MFRM will carry out whatever instructions it contains. At this point, if no errors are detected, a message will be displayed to indicate that arithmetic processing has commenced. The appearance of subsequent screens depends very much upon the instructions in the template commands file.
Edit the template file	This option loads the MFRM file into a user nominated text editor. The text editor's name can be user supplied, or it may be stored in the local environment by the keyword $EDITOR (see module MENM for environment manipulation).
Help on syntax	This option calls 8 pages of explanation and help about the syntax and usage of FRM commands. It is a concise version of the Technical Overview in this document.
eXit	Quits FRM. Work is not *automatically* saved.

After a template file has been run, MFRM will re-display the menu shown above, but with additional options which are designed to enable the results of processing to be plotted or listed as appropriate.

The post-processing menu is shown below.

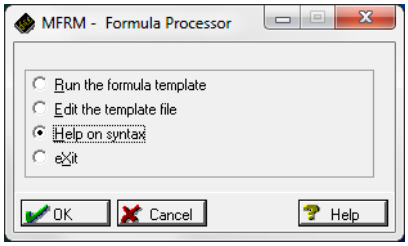


Figure 14-8 The Extended MFRM Postprocessing Menu

The additional menu options are as follows:

Option	Description
Plot results file	When selected, this option will run the graphics module Multi File Display (MMFD) which allows the results the output files to be viewed. When MMFD is quit the user is returned to MFRM. If no plot file was produced then this option is superfluous.
View/edit list output	This option will load the text editor (discussed in Editing the Template File above) and display the ASCII list (`.lst`) file produced when the template was run. If no list file was produced then this option is superfluous.
Output file statistics	This option will display a screen of statistics about the results of the most recent run of an MFRM template.

Batch Operation

MFRM can be run in all the standard batch modes as with most other MSC Fatigue modules. A list of batch keywords:

/OPTion	Main menu option [Run,Edit,Plot,eXit
/INP=	The name of the template commands file to process. /INP=TEMP1
/FILnn=	The name of any input data file to manipulate, or output file to create. This keyword must be issued the same number of times as the %QYFIL command has been used. The sequence of file names must also correspond i.e., the first /FIL= keyword will be matched with the first %QYFIL command etc. /FILnn=FILE1/FILnn=FILE2
/DIVide	Divide by zero value
/ZERo	Special case of 0/0 (which is replaced by zero, one
/AZERo	Arctan (x=0, y=0) value

/OV=	Whether to overwrite an output file
/CONnn=	The value of any constant to be used. This keyword must be issued the same number of times as the %QYCON command has been used. The sequence of constants must also correspond i.e.the first /CONnn= keyword will be matched with the first %QYCON command etc. /CONnn=5.123

Example MFRM Command Template Files.

The three examples that follow have been provided in order to illustrate the use of the MFRM formula processor.

Simple Arithmetic Test

In this example, MFRM will be used to calculate the following formula in which the variables " a " and " b " are represented by data files and the variable " c " is a user-defined constant:

$$\div \left\{ 2\,a^2 + (b / c)^2 \right\}$$

```
;
; --- Template (FRM1.FRM) for the calculation of the function :
;
;( 2.0 * A ** 2.0 + (B / C) ** 2) ** 0.5
;
; --- Prompt for the two input filenames ---
;
%QYFIL /FI=F1 /PR="Enter first input filename (a)" /TY=IN
%QYFIL /FI=F2 /PR="Enter second input filename (b)" /TY=IN
;
; --- Define the name of the output file ---
;
%DFFIL /FI=F3 /NA=RESULT.NEW /TY=OUT /LABEL="RESULT"
/UN="WIDGETS"/OV=Q
;
;--- Prompt for the constant " c " ---
;
%QYCON /CO=C1 /PR="Enter the constant (c)"
;
; --- Define the other two fixed constants ---
;
%DFCON /CO=C2 /VA=2
%DFCON /CO=C3 /VA=0.5
;
; --- Now start the calculation ---
;
%BEGIN
;
T1=F1*F1
T1=C2*T1
T3=F2/C1
T4=T3^C2
T5=T4+T1
F3=T5^C3
```

```
%END
;
; --- Finished ---
```

Try executing together with data file `65.dac` as file A and `102.dac` as file B, both DC levels with amplitude 65 and 102 units respectively and using a value of 5 as the input constant. After processing, use MQLD to check the maximum value calculated in file `result.new`. It should be 94.1.

Generation of a Square Wave

In this example, a square wave function will be generated by first creating and then adding together a number of individual sine waves of appropriate amplitude and frequency. In effect the series, to five terms, being solved is:

$$Y = a_0 sin(t) + \frac{a_0 sin(3t)}{3} + \frac{a_0 sin(5t)}{5} + \frac{a_0 sin(7t)}{7} + \frac{a_0 sin(9t)}{9}$$

The methodology for calculating this function is based on the use of a ramp or sawtooth function going from 0 to 360 degrees. If a ramp is used then only a single cycle of square wave form will be produced and if a sawtooth is used, then the same number of cycles as teeth will result. Figure 14-9 below illustrates the sawtooth function, `sawtooth.dac`.

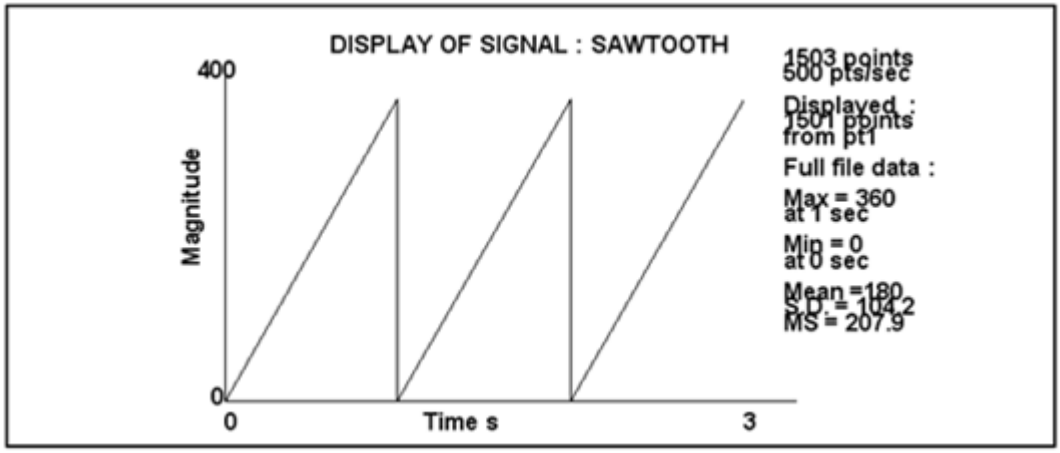


Figure 14-9 The Sawtooth Function `sawtooth.dac`

The MFRM template detailed below converts the seed input file (`sawtooth.dac`) to RADIANS and stores the result in a temporary variable file called T50. This file is then scaled and each of the five terms of the series outlined above calculated and stored in variables T1, T3, T5, T7 and T9. Finally, all the terms are added and the result loaded into the output file `square.dac`.

```
;
; -- Get the name of the input seed file --
;
%QYFIL /FILE=F1/PROMPT="Enter name of seed file [Degrees]"/TYPE=
INPUT
;
; -- Define the output file name --
;
%DFFIL /FILE=F2/ NAME=square/ TYPE=OUTPUT /OV=Q
;
; -- Ask for  and define constants constants --
;
%QYCON /CONSTANT=C1 /PROMPT="Enter amplitude"
;
; -- Start the calculations --
;
%BEGIN
;
; --- Convert seed to RADIANS and create basic sine wave ---
;
T50 = RAD(F1)
T1 = SIN(T50)
;
; -- Calculate the first term in the series, T1 --
;
T1 = C1*T1
;
; -- Calculate the second term in the series, T3 --
;
```

```
T2 = 3*T50
T3 = SIN(T2)
T3 = T3/3
T3 = C1*T3
;
; -- Calculate the third term in the series, T5 --
;
T2 = 5*T50
T5 = SIN(T2)
T5 = T5/C
T5 = C1*T5
;
; -- Calculate the fourth term in the series, T7 --
;
T2 = 7*T50
T7 = SIN(T2)
T7 = T7/7
T7 = C1*T7
;
; -- Calculate the fifth term in the series, T9 --
;
T2 = 9*T50
T9 = SIN(T2)
T9 = T9/9
T9 = C1*T9
;
; -- Add all the terms to produce the final square wave ---
;
F2 = T1+T3
F2 = F2+T5
F2 = F2+T7
F2 = F2+T9
%END
;
; --- Finish ---
;
```

Turning Points in an Input File

The algorithm below calculates the irregularity factor γ which gives an indication of the spread of frequencies in a signal according to this equation.

$$\gamma = \frac{E[0]}{E[P]}$$

where E[0] and E[P] which are statistical parameters that express the number of zeros and peaks per second according to:

$$E[0] = \left[\frac{m_2}{m_0}\right]^{1/2}$$

$$E[P] = \left[\frac{m_4}{m_2}\right]^{1/2}$$

The M_n values can be calculated from:

$$M_n = \int_0^\infty f^n G(f) df$$

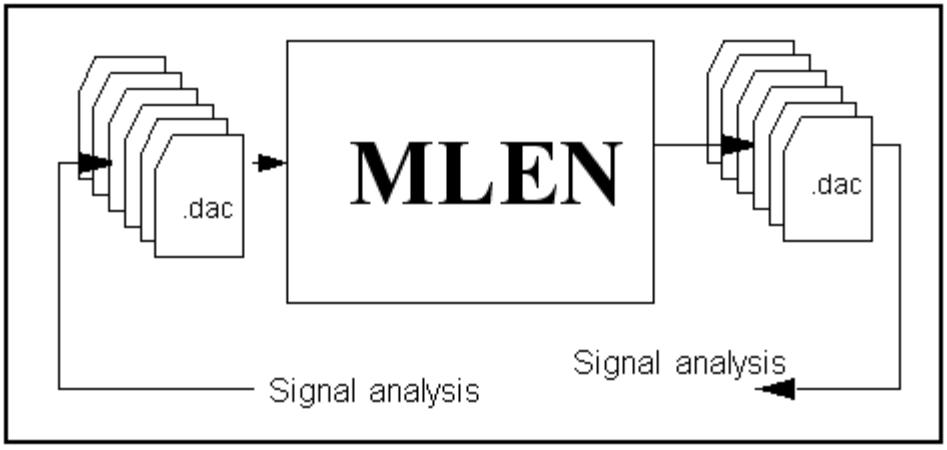
The γ value can vary from 1.0 (one dominant frequency) to 0.0 (equal energy at all frequencies).

```
; template calculates moments of PSD and irregularity factor
;-----------------------------------------------------------------
;
; Get the input filename and the output ASCII filename
;
%QYFIL /FI=F1 /TYP=INPUT /PR="Input Spectra Filename (.PSD)"
%DFFIL /FI=F2 /TYP=LIST /NA=irrfact.lst /OV=Y
;
; Define constants
;
%DFCON /CO=C2 /VA=2.0
%DFCON /CO=C3 /VA=TOTPTS /FI=F1
%DFCON /CO=C4 /VA=4.0 %DFCON /CO=C10 /VA=0.0
;
; Counter variables
;
; Current Point
;
%DFCON /CO=C9/VA=POINT/FI=F1
; Current time = frequency
;
%DFCON /CO=C11/VA=TIME/FI=F1
;
; Frequency Increment
;
%DFCON /CO=C12/VA=INCREM/FI=F1
;
; Initialise Summation Counters
;
%DFCON /CO=C13/VA=0.0
%DFCON /CO=C14/VA=0.0
%DFCON /CO=C15/VA=0.0
;
%BEGIN
;
C5=C11^C2
C6=C11^C4
C7=C11^C10
T1=F1*C5
T2=F1*C6
T3=F1*C7
T4=T1*C12
T5=T2*C12
T6=T3*C12
C13=C13+T4
C14=C14+T5
C15=C15+T6
```

```
;
; Check for end of file and write out information
;
%IF (C9 = C3)
   %PRINT /FI=F2 /ME="Moment"+15T+"Value"
   %PRINT /FI=F2 /ME="----------------------------"
   %PRINT /FI=F2 /ME="0"+15T+C15
   %PRINT /FI=F2 /ME="2"+15T+C13
   %PRINT /FI=F2 /ME="4"+15T+C14
  %PRINT /FI=F2 /ME="----------------------------"
;
;  Calculate expected number of zeros per second, E(0)
;   C20=C13/C15   C21=C20^0.5
;
;  Calculate expected number of peaks per second, E(P)
;
  C22=C14/C13
  C23=C22^0.5
;
;  Calculate irregularity factor, Gamma
;   C24=C21/C23
;
;  Write out expected values and irregularity factor to ASCII
file
;
  %PRINT /FI=F2 /ME="E(0)  ="+15T+C21
  %PRINT /FI=F2 /ME="E(P)  ="+15T+C23
  %PRINT /FI=F2 /ME="Gamma ="+15T+C24
;
;  Write the irregularity value to extra details area of the PSD
;
  %WTVAL /FI=F1 /KW=IRRFACT /VA=C24
%ENDIF
%END
```

File Cut and Paste - (MLEN)

MLEN is used to extract a portion of data from one file or several files, and load the extracted portions into a new output file. It can be used to concatenate (merge), individual data sets into a single output file. It can also be used to delete a selected portion of data from files.



The process of data extraction is carried out by cutting portions of data from an input data file and copying them to a named output file. The limits of the extraction, i.e. the start and end of the data to be cut, may be defined in terms of X-axis parameter, usually time, or data point numbers; the original input data file remains unaltered. The process of concatenation consists of joining together a number of input data sets, with the same sampling rate, into a single output file.

MLEN automatically recalculates the appropriate header details for the resultant output file. If any of the input data files have extra details associated with them, then these will be attached to the output file accordingly. If any of the input files have extra details, then details from all files to be specified for concatenation will be attached to the output file. This will be done in the order the files are specified. If there are duplicate names then subsequent files will overwrite existing values.

Module Operation

The MLEN module can be run in one of the following three ways:

- From the **Advanced Loading Utilities** pulldown menu under **Tools | MSC Fatigue** (for Patran) or under **Tools | Fatigue Utilities** (for).
- In stand alone mode by typing **mlen** at the system prompt.
- By incorporating the MLEN commands in a batch operation.

Once running in interactive mode, a screen appears showing the main menu selections.

Extract Single File

This option displays a form shown in Figure 14-10. The name of the data file from which data are to be extracted must be entered. By default, MLEN expects this file to have a time series format and a `.dac` file extension. If a `.dac` file is being processed then the `.dac` extension may be omitted. If another file type with the correct internal format, such as spectra is being processed, then it must have its name entered in full, i.e. including the file extension.

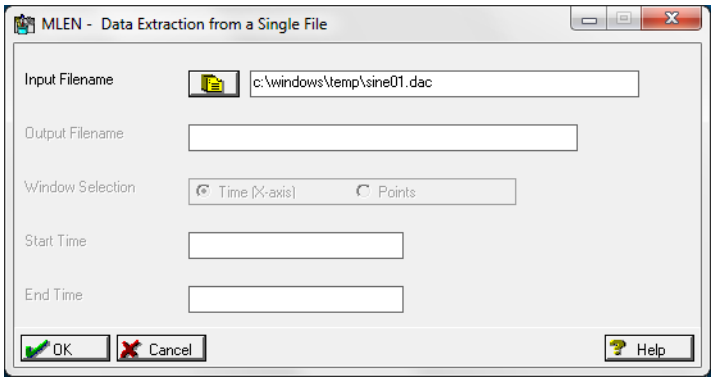


Figure 14-10 Data Extraction from a Single File

Option	Description
Input File	MLEN looks for the input file in the current directory. However, other directories can also be accessed if the complete file specification of path name and file name is entered. Pressing the pick list button will display a list of available files, or allow the user to choose other files/directories, using the Path option.
Output File Name	The extracted data will be written to an output file with a `.dac` file extension. As a default, this file will be created within the current directory. If another file extension and/or destination are required then these must be specified in full. At this stage it is required to specify the method by which the limits of the extraction will be defined. This is done by selecting either Time or Points on the two horizontal radio buttons.

Extraction according to X-axis parameter (Time)

The fields **Start** and **End** become active. For the start time, any X-axis value within the range defined between the start and end of the data set may be entered. Extraction will commence at the data point nearest and LESS than the value of X-axis parameter specified. For example, for a signal sampled at 50Hz, specification of a start time of 0.11 seconds will cause MLEN to commence extraction at 0.1 seconds.

The default for the start of extraction is the beginning of the data. In addition to entering an explicit numeric value at which extraction is to commence, it is possible to enter instead the name of an extra details keyword, see the documentation associated with the Extra Details Manipulation module, Peak-Valley Extraction - (MPVXMUL), 1031, for more information on the extra details area.

After accepting a keyword, MLEN will search the extra details area of the input file and assign the value of the X-axis parameter for the start of extraction. For example, suppose that the input file referred to above contains an extra details keyword called T1 which has a value 0.11 associated with it, then entering the word T1 in response to the above prompt will cause MLEN to use the value 0.11 for the start time for extraction. If the keyword cannot be found an error message will result.

For the end time, any X-axis time value within the range defined between the start point specified above and end of the data set may be entered. The default for the end of extraction is the end of the data.

Note: MLEN analyzes the file selected and displays the range of start and end times/points on the screen. The numbers displayed will depend upon the file selected.

Extraction according to data point number

The fields **Start Point** and **End Point** become active. For the start point, any point number within the range defined by the data set may be entered. Extraction will include the data point specified.

The default for the start of extraction is the first data point; this default may be selected by pressing the ENTER key. Typing START would have the same effect. For the end point, any point number within the range defined by the data set may be entered (even a number less than the start point). Extraction will include both the data points specified.

The default for the end of extraction is the last data point; this default may be selected by pressing the ENTER key, or by typing END.

When the user has filled all the fields, and accepted them, MLEN will proceed to extract data according to the defined limits. On completion it will present a summary table of the operations carried out.

Extract Multiple Files

Multiple File Extraction allows the same portion of data to be extracted from a selection of one or more input files, and written to other output files, or files with the same names as the input files. There are two ways of selecting the input files:

- Test Name + channels
- Separate File names

Test name + channels

Option	Description
Test Name	Test Name is the generic file name, i.e. if the user were to enter `data` as the file name in this field then the program would look for `data01`, `data02`, `data03` and so on up to the number of channels. The concept of a generic name for a test represents a quick method of entering channel names for processing. The methodology is based on the fact that associated data (derived from the same test) are usually demultiplexed after data acquisition into individual channel files. The structure of these consists of a generic base name, normally the name of the multiplexed data file itself, with the appropriate channel numbers appended. By default MLEN expects to process standard MSC Fatigue time series data files with a file extension of `.dac`. Furthermore, the above generic test name convention only applies to `.dac` files.
Channels	MLEN will prompt for the numbers of the channels to process. The individual channel numbers can be entered as a string of numbers separated by commas, or grouped by hyphens, or as a combination of the two. Start and End points and SRATE do not have to be equal. Each channel can have a different sample rate and signal length.If using extra details keywords to identify the start and end then the values may well differ from file to file.
Output File Name	The Output File Name is the generic test name used to create the output files, with the channel numbers selected appended to it. For example, if `test` was entered as the Output File Name name then the files; `test01`, `test02`, `test03`, etc., dependent on the channel numbers requested, would be created.

As with the Extract Single File option, it is required to specify the method by which the limits of the extraction will be defined. This is done by selecting either Time or Points on the horizontal bar question. The start and end points are entered in the same manner as they are for Single File Extraction (see above). After completion of all the entries, MLEN will proceed to extract data according to the defined limits and will display a scrolling list of file names and the number of points being processed as the extraction takes place on that file. On completion it will present a summary table of the operations carried out.

Separate File Names

Option	Description
Input File Names	MLEN allows up to fifty files per input line to be processed, which means a total of 200 files can be extracted at any one time. The four input file fields allow files to be selected in one of three ways: ■ A string containing a list of file names separated by commas, e.g. `sine01`, `sine02`. ■ A string containing a group of file names separated by hyphens, i.e. entering `data(1-3)` would select `data01`, `data02`, and `data03`. ■ A list of file names selected using the pick list button. If this method is used then a directory listing of .`dac` type files in the selected directory are displayed. Start and End points and SRATE do not have to be equal. Each channel can have a different sample rate and signal length.If using extra details keywords to identify the start and end then the values may well differ from file to file.
Output File Options	The output files for a multiple extraction, by default will overwrite the input files unless a new extension for the output files is specified. This is achieved by selecting the Modify Extension option on the horizontal bar question.
New Extension	On pressing ENTER a new prompt will appear requiring a new extension to be entered. MLEN allows up to three characters for this extension.

As with the two extraction options described earlier, the user must specify the method and the limits of the extraction. This is achieved by selecting Time or Points on the horizontal bar question. The start and end points are entered in the same manner as they are for single file extraction (see earlier). On accepting this information an option is made available for editing the combined list of input files in case you made a mistake or wish to tag certain of them for processing and exclude others without redefining the input file lists. By using the cursor key or the mouse, files in the list can be tagged or untagged as desired. If OK is selected as in the above example, then MLEN carries out the extraction as per user specifications.

Delete Section of Single File

Single File Deletion enables a portion of data to be removed from a file and the remaining data to be written to a single output file. The method of input for deletion is the same as for extraction. See Extract Single File, 1017.

> **Note:** Defaults for the start and end limits for deletion are not given as they are for extraction, and entering start and end limits which both equal the start and end limits of the data set will result in an error message.

Delete Section of Multiple Files

Multiple File Deletion allows the same portion of data to be deleted from a selection of one or more input files and written to these or other output files. The method of input for deletion is the same as for extraction. See Extract Multiple Files, 1019.

> Note: Defaults for the start and end limits for deletion are not given as they are for extraction. Entering start and end limits which equal the start and end limits of the data set will result in an error message.

Concatenation

Concatenation allows the user to join several files into a larger single file. Joining is nose to tail. MLEN allows up to fifty files to be processed by using either the pick list or explicit file naming via the keyboard.

If you have selected a list of files and wish to change it, add to it, otherwise operate on that list, simply click on the pick list button a second time. A menu will appear allowing for several options to operate on the file list.

The File Options menu

> Note: The order in which file names are entered is important. The sample rate associated with the first data set entered is used by MLEN as the reference rate. The sample rate of all the other data sets to be concatenated must be equal to this reference; if they are not them an error message will be output. It is up to the user to ensure that sample rates are compatible and, if necessary to use the PTIME module to adjust sample rates accordingly (Sample Rate Adjust Option, 241).

The merged data will be written to an output file. As a default, this file will be created within the current directory and, irrespective of the input file extensions, the extension `.dac` appended to the name entered. If another extension or destination are required then these must be entered explicitly. On completion it will present a summary table of the operations carried out. If the NOTEBOOK keyword is defined within the local environment, (see Modifying the MSC Fatigue Environment (MENM), 1463 for more details) an entry of this information will be made within the MSC Fatigue notebook.

Start-End Smoothing

The questions on this page determine the method by which the start and end of a signal are to be smoothed. Input fields are explained below.

Option	Description
Input File Names	The file name can be typed in directly or can be selected from a pick list. The default extension is .dac.
Joining Function	The end point may be joined to the first point using a half-sine, a linear ramp or by tapering a section at the end of the signal such that it blends into the first point. Alternatively, if the end point does not require joining to the first, the taper options may be used to smooth the start and end sections of the signal. The taper options may also be used in conjunction with the join options to smooth the start and end sections of the signal before appending the ramp or half-sine to join the signal start and end.
Join Window	The Join Window is the length of signal over which the edited ends will be joined.
Taper Function	The edited signal can be tapered using one of two functions: ■ Ramp function ■ Half sine function If a function is selected then the duration over which the signal is to be tapered will be requested. If 'None' is selected, the end points are simply joined using the curve selected in the join function. If tapering is not used, the signal may exhibit hard transitions at the intersections of the joining curve.
Taper Window	The Taper Window is the length of signal over which the edited ends will be tapered together.
Output File Name	This is the name of the smoothed file. Its default extension is .dac.

Reverse File

This option allows the user to reverse a .dac file to produce a .dac file whose first point is the last point of the input file, and whose last point is the first point of the input file, etc. This can be an aid when filtering certain .dac files. There are only two fields to fill (the name of the input file and the name of the reversed order output file.

Batch Operation

MLEN can be run in non-interactive batch mode. In this mode of operation complex procedures for creating signals can be created. A typical batch command line is:

```
mlen /opt=1/inp=data01/out=data02/ov=y/type=t/sta=0.75/end=12.0
```

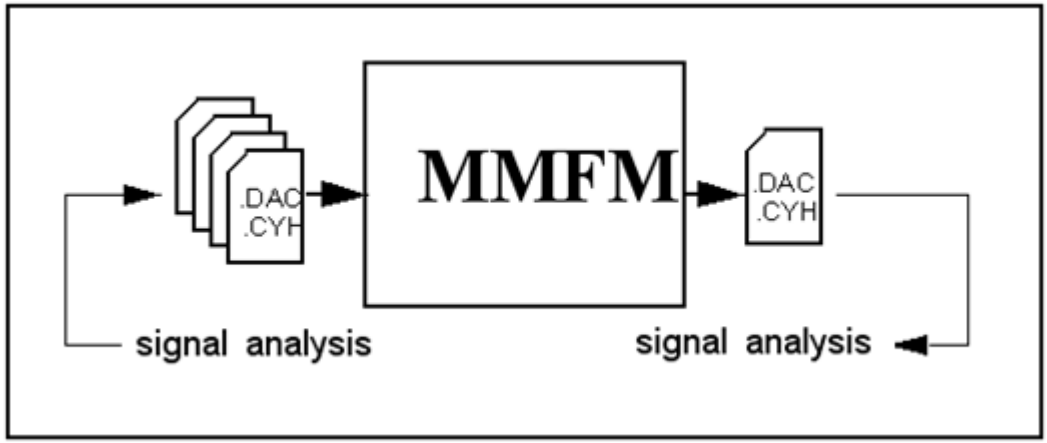
In this example, MLEN will extract a portion of data from an input file called `data01.dac`, resident in the current directory and write it to an output file called `data02.dac`. If a file with this name already exists MLEN will overwrite it. The extraction will be according to X-axis units, time in this case, and will commence at 0.75 seconds and end at 12.0 seconds.

A list of MLEN batch keywords:

/OPT=	=1 Single file extraction, =2 Multiple file extraction, =3 Single file deletion, =4 Multiple file deletion, =5 Concatenation, =6 Start-End smoothing, =7 Reverse File
/OUT=	The name of the output file.
/OV=	Whether to overwrite an existing output file, OV=Y.
/FILTYPe=	Multiple file option. FILTYP=T test name + channels, specifies a generic input file name (only applicable when OPT=2 or 4)
/INPut=	Input file list for single or multiple file /INP= FILE1,FILE2, etc. up to 50 times, processing. Input can be used up to and this structure can be used up to 4 times, each time naming up to 50 times making a total of 200 files (i.e. up to 200 files can be processed)
/MODify=	Whether to modify file extensions, = O - Overwrite input files for output files, = M - Modify extensions
/EXT=	Extension to append to output files EXT=EXTENSION
/GEN=	Generic input test name.
/CHA=	Channels to process CHA=(1-6)
/OUT=	Generic output name
/TYPE=	In the case of extraction and /TYPE= T - Selects X-axis units, (default) deletion, specifies the method for defining the start and end points /TYPE= P - Selects point numbers.
/STA=	Start point for the Extraction/Deletion STA= 0.5, STA= 3
/END=	End point for the Extraction/Deletion /END= 12.4, END= 25

Multi-File Manipulation - (MMFM)

MMFM is a module which can be used to manipulate standard MSC Fatigue data files. Files can be added, subtracted, multiplied, divided, or added according to the rules of vector addition. Complex mathematical procedures can be built up by chaining together sets of MMFM commands in a batch process.



MMFM supports the following arithmetic operations:

- Addition of a number of files.
- Subtraction of one or several file(s) from another.
- Multiplication of one or several file(s) by another.
- Division of one file by one or more files.
- Vector addition of two or three files.

Throughout the course of the above manipulations the following conventions are observed:

- The largest and smallest numbers which can be processed are respectively 1.0^{10} and 1.0^{10}. Some computer types may be able to process numbers beyond this range, however, results may be unpredictable.
- Zero values are taken to be real numbers in the range: $1.0^{-10} < n < 1.0^{10}$
- The vector addition processor assumes that the input files X, Y, and Z are orthogonally related.
- The vector addition processor accepts either two or three input files. Each point in the output file is calculated on the basis of the following combination of values from the input files:

  OUTPUT = $\sqrt{(X^2 + Y^2)}$ or OUTPUT = $\sqrt{(X^2 + Y^2 + Z^2)}$

Operations on single parameter data files (time series type, .dac)

Depending on the specific operation to be carried out, MMFM can accept an unlimited number of input files to process. These files can be of unequal length and unequal sampling rate. The length of the file which contains the result of the operation, the resultant output file, is calculated on the basis of the smallest and largest values of X-axis parameters found in the set of input data files. The sample rate of the resultant output file may take any value and is not limited to any of the input sample rates. For example, in Figure 14-11 below, data file `s1.dac` had a sample rate of 1000 Hz and a duration of 1 second, whilst file `s2.dac` had a sample rate of 750 Hz and a duration of 2 seconds. The file resulting from the addition of these two data sets has a sample rate of 500 Hz and is given in file `s12.dac`.

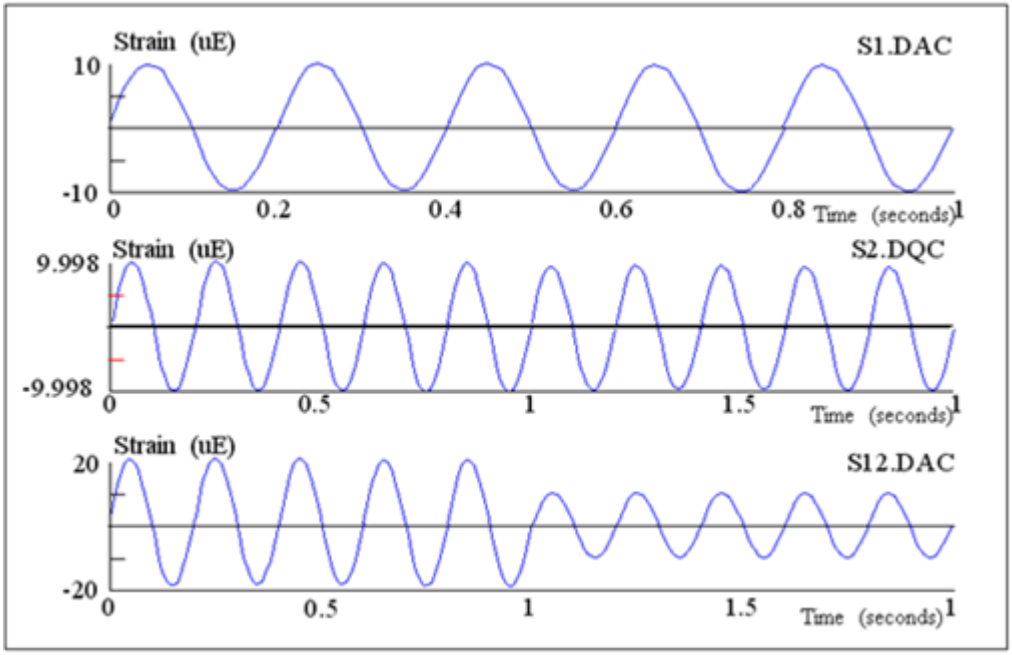


Figure 14-11 Addition of Files of Different Sample Rate and Duration

Operations on three parameter data files (histogram type)

In addition to processing time-series type data files, MMFM can also process files which have the standard MSC Fatigue histogram format. In the general case, MMFM can process the following histogram file types:

- Rainflow cycle (`.cyh`)
- Fatigue damage (`.dhh`)
- Range mean (`.rmn`)
- Range pair (`.rph`)
- Markov (`.rph`)

The ability to process histograms in this way can be very useful, particularly in situations where the data, if left in the form of time-series, would consume an inordinate amount of disk space if concatenated for some

reason. A significant amount of disk space could be saved if each history were converted to an appropriate rainflow matrix and then the three histograms added together. The fatigue analysis could then be carried out using the matrix rather than the time series fatigue processors.

Note:	Ensure that histograms are compatible prior to manipulation. They must have the same dimensions, i.e. the same start point and bin size on both axes.

Module Operation

The MMFM module can be run in one of the following three ways:

- From the **Advanced Loading Utilities** pulldown menu under **Tools | MSC Fatigue** (for Patran) or under **Tools | Fatigue Utilities** (for).
- In stand alone mode by typing **mmfm** at the system prompt.
- By incorporating the MMFM commands in a batch operation.

The first two modes are interactive. Once running in one of the interactive modes MMFM will display the following menu of options:

- Addition
- Subtraction
- Multiplication
- Division
- Vector Addition

Regardless of the option you select you will be prompted for a list of input signal names than may be entered one per line, or as a string of names separated by commas, or as ranges e.g. `saetrn`, `saesus`, `saebrakt`, or `data (1-3)`. The last example (`DATA (1-3)`) would load `data01.dac`, `data02.dac` and `data03.dac`.

Note:	If histograms are being manipulated it is up to the user to ensure that they are compatible (see the technical overview).

Probably the easiest method of entering input file names is to use the pick list facility. Once a set of files has been selected you can always press the pick list button again to add or change the list of selected files.

Note:	The order in which file names are entered is important especially the first one.

If file names are entered as:

```
file1,file2,file3,..., filen
```

MMFM will process them in the following ways:

Option	Description
Addition	Selection of the addition option above will cause MMFM to add the input data files according to: `file1 + file2 + file3 +... + filen`
Subtraction	Selection of the subtraction option above will cause MMFM to subtract data files according to: `file1 - {file2 + file3 +... + filen}`
Multiplication	Selection of the multiplication option above will cause MMFM to multiply data files according to: `file1 x file2 x file3 x... x filen`
Division	Selection of the division option above will cause MMFM to divide data files according to: `file1 / {file2 x file3 x... x filen}`

Vector addition	Selection of the vector addition option above will cause MMFM to add data files according to:
	
	`(file12 + file22)1/2 or`
	`(file12 + file22 + file32)1/2`
	
	depending on the number of file names entered.

The second input screen looks like Figure 14-12.

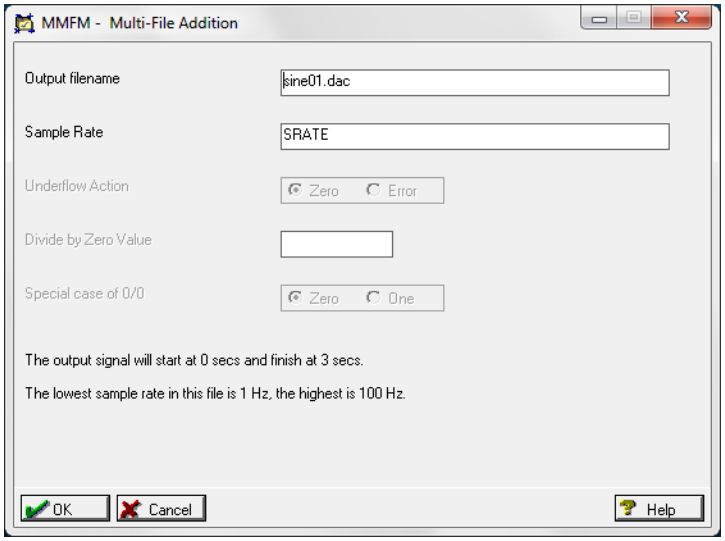


Figure 14-12 The Addition Screen

Option	Description
Output File Name	The output file name defaults to the standard `.dac` type. If another extension or destination are required then these must be entered explicitly; this is particularly important when working with histograms. The name of the first input file entered is offered as the default output name. Selection of this default means that this data file will be overwritten and so MMFM will prompt for confirmation to proceed.
Divide by Zero Value	Division by zero is illegal, so a value to replace the divide by zero calculation is needed. Whenever a division by zero is encountered during processing, the value entered will be used for the answer. A zero value is defined within the software as any value less than 1E-10.
Divide Zero by Zero Value	A division of 0/0 is a special case. The result of a 0/0 division could logically be zero or one, depending upon the application. Whenever a 0/0 division is encountered during processing, the value entered will be used for the answer.
Sample Rate	In the case of time-series data structures, the resultant output data file can have any sample rate. By default, MMFM offers the lowest sample rate found in the input data files; this may be selected by clicking OK. Any other value, even outside the limits of the minimum and maximum values found may be entered; MMFM will automatically interpolate to provide the desired values. Note that decreasing the value may cause features to be missed.

MMFM will scan the input files for length and sampling rate. If the input files are of different length or sample rate a warning message will be output. Under all circumstances the following messages will be displayed.

```
The output signal will start at S and finish at F.
The lowest sample rate in these files is L samples/sec,
the highest is H samples/sec.
```

S	Is the smallest X-axis value found in the input files.
F	Is the largest X-axis value found in the input files.
L	Is the lowest sample rate found in the input files.
H	Is the highest sample rate found in the input files.

Signals with different sample rates will be correctly handled. Vector addition takes either two or three signals, the other options take up to 50 input files.

MMFM will handle MSC Fatigue XYZ files, such as histograms, but only if the X and Y-axis limits (max and min values) are the same for each input file. The multi-file manipulation is performed only on the Z axis.

Environment Keywords

A list of environment keywords and meanings:

$SIGNAL	The Output File Name
.DAC	The Output File Name if the input file was a `.dac` file.

Batch Operation

MMFM can be run in non-interactive batch mode. In this mode of operation complex arithmetic formulations can be created. For example:

```
mmfm
/opt=a/inp=test1,test2,test3/out=outfile/ov=y/sam=300
```

In this example, MMFM will add together the contents of files `test1.dac`, `test2.dac` and `test3.dac` and write the result to a file called `outfile.dac` at a sample rate of 300 samples per second. If a file `outfile.dac` already exists in the current directory it will be overwritten. Keywords are:

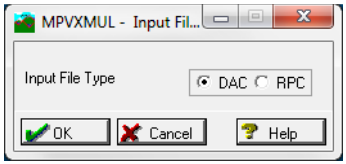
/OPTion=	The number of the required arithmetic option where /opt=A is Addition, =S Subtraction, =M Multiplication, =D Division, =V Vector Addition. /OPT=A
/INPut=	The name of the data file(s) to process. /INP=`file1,file2`
/OUTput=	The name of the resultant output file. /OUT=`output`
/OVerwrite=	Whether to overwrite the existing output file. /OV=Y
/SAMple=	The new sample rate for the output file. /SAM=500
/ZERODIVide	The value of the output where a divide by 0 is attempted. /ZERODIV=1000
/ZEROZERO	When 0 is divided by 0 the result can be 1 (1) or 0 (Z). /ZEROZERO=Z

Peak-Valley Extraction - (MPVXMUL)

MPVXMUL extracts turning points (maxima and minima or “peaks” and “valleys”) from single parameter files such as **.dac** and RPC multiple data - channel files. The peak valley extraction process maintains synchronous phase by writing corresponding data values to all the output files whenever a turning point is found in any channel. Facilities for gating out small peak valley pairs by absolute value or by percentage of range, on each channel, are available.

1. Invoke MPVXMUL by typing **mpvxmul** at the system prompt or select the Peak-Valley Extraction option from the **Advanced Loading Utilities** pulldown menu under **Tools | MSC Fatigue** (for Patran) or under **Tools | Fatigue Utilities** (for).
2. Select the Input File type of **DAC** or **RPC**. Press **OK**.
3. Select the generic Input Filename, the Channels, the Output Filename, and whether or not to write a time file. Click **OK**. See the *MSC Fatigue User’s Guide* for file naming conventions and other information.

4. Either the range of cycles from rainflow analysis can be used as the gate in the completion of your analysis. This information along with other relative data is entered on the analysis setup form that is displayed. See the *MSC Fatigue User's Guide* for more information.



Note: Input .dac files exist as families of files with a common generic name but with different numbers appended to the name which denotes the channel number (i.e., test01.dac, test02.dac, etc., where test is the generic name).

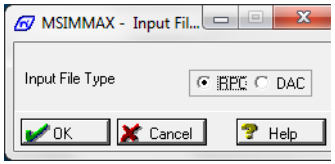
See Peak-Valley Extraction (MPVXMUL), 254 for a more detailed description of this utility.

Simultaneous Values Analysis DAC/RPC - (MSIMMAX)

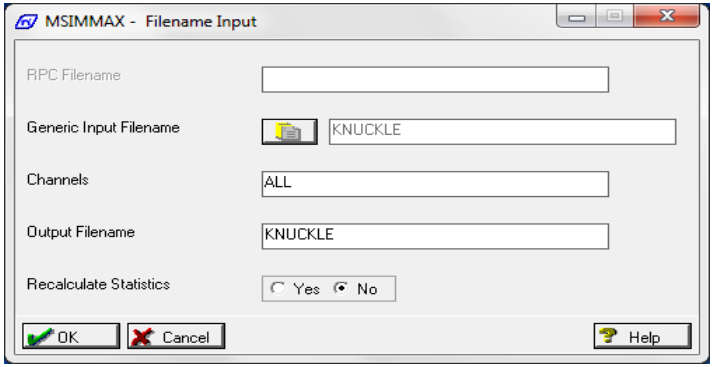
MSIMMAX performs simultaneous values analysis on either multi-channels in a single RPC file or multiple DAC files from the same test. Two analysis methods are available. The first uses a "control" channel, from which turning points are extracted and scanned for the highest peaks, the lowest valleys or the highest absolute maxima. Up to 50 of these events may be saved, with their positions in the data. The simultaneous values of all the other channels at these positions are saved into the output files. The second method scans each of the input channels for the single largest maximum, minimum or absolute maximum. For each channel, the simultaneous values of all other channels at the position of the largest event is saved into the output files.

The output file created is a tab-separated ASCII file suitable for input to a spreadsheet or word processing package.

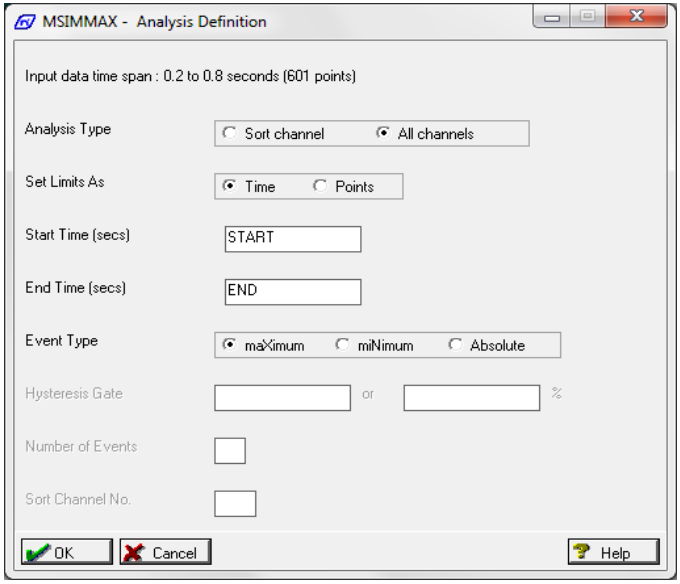
1. Invoke MSIMMAX by typing **msimmax** at the system prompt or select the Simultaneous Values Analysis DAC/RCP option from the **Advanced Loading Utilities** pulldown menu under **Tools | MSC Fatigue** (for Patran) or under **Tools | Fatigue Utilities** (for).
2. Select the Input File type of DAC or RPC. Click **OK**.



3. The next form is the Filename Input form which allows the names of the input and output files to be specified. For an input file type of **RPC** the **RPC Filename** field appears and for an input file type of **DAC** the Generic Filename field appears. Select the RPC Filename or the **Generic Filename**, the Channels, the Output Filename, and whether or not to recalculate the statistics. Click **OK**. See the *MSC Fatigue User's Guide* for file naming conventions and other information.



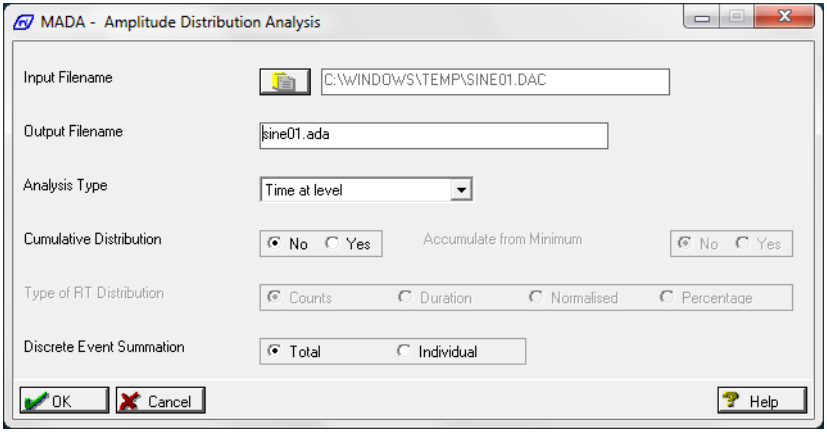
4. The final form for MSIMMAX is the Analysis Definition form. This form allows you to choose between **Sort Channel** analysis and **All Channels** analysis. It is here that the limits are set and the event type is selected. This information along with other relative data is entered on the form that is shown below. See the *MSC Fatigue User's Guide* for more information.

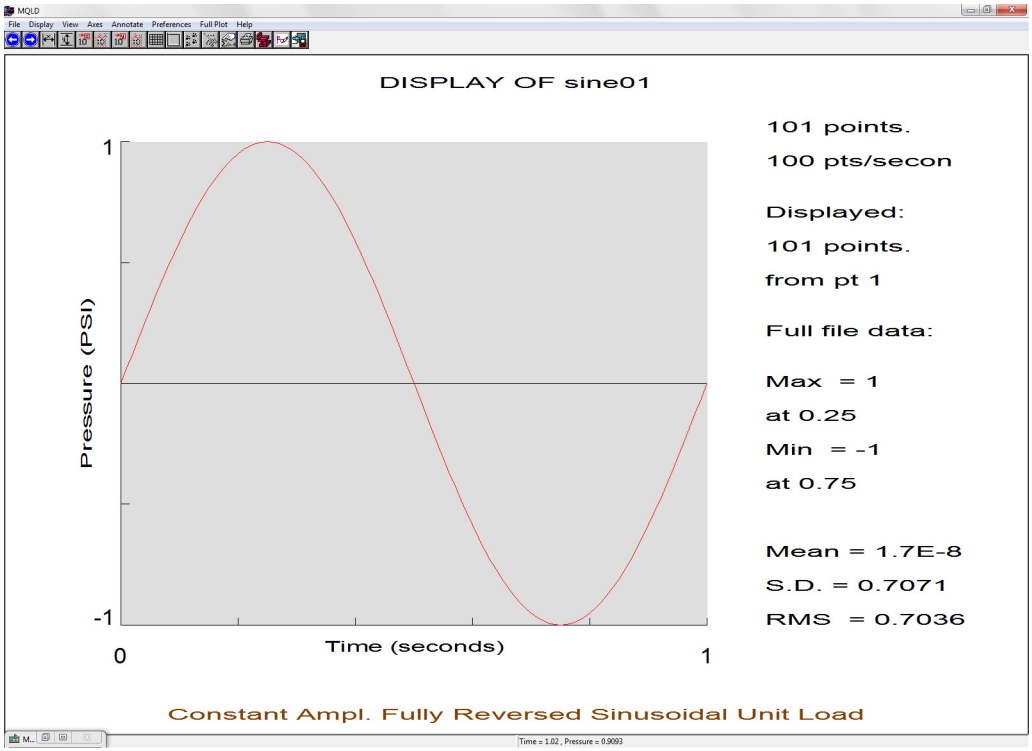


Amplitude Distribution - (MADA)

MADA, amplitude distribution analysis, calculates the probability density distribution (which defines the probability of finding a value of a particular magnitude within the population of measured values) and other function of a time signal.

For example if you use **saetrn.dac** as input to MADA and set the Analysis Type to **Prob. Distribution**, it will output a file **saetrn.ada** shown here which is the probability density function of Y-values.



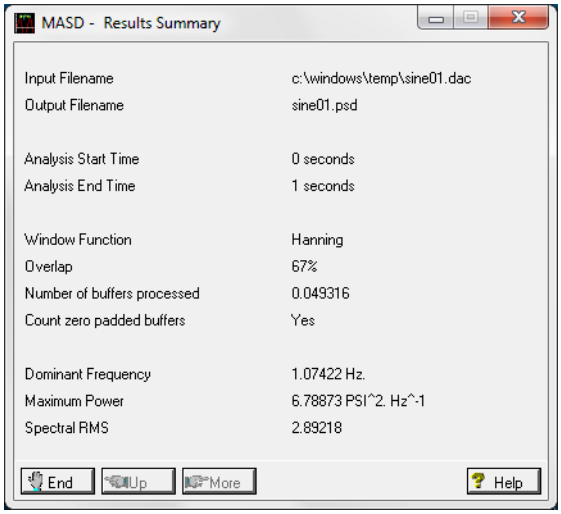


Auto Spectral Density - (MASD)

MASD performs a frequency analysis of a time signal to determine frequency content. Various output types are available which are beyond the scope of this text. Perhaps the best use of this module comes in vibration fatigue problems for converting time signals into power spectral density functions (PSDFs).

As an example, let us convert the time signal SAETRN, used in many of the previous sections in this chapter into a PSDF, which will quickly show us the frequency content of the signal.

1. Invoke MASD by typing **masd** at the system prompt. It can also be invoked directly from PTIME under **Add an entry | creaTe psd from time** or select the Auto Spectral Density option from the **Advanced Loading Utilities** pulldown menu under **Tools | MSC Fatigue (for Patran) or under Tools | Fatigue Utilities** (for).
2. Select the file **saetrn.dac** as the Input Filename.
3. Make sure the Output Type is **Power Spectral Density** and accept the defaults for all other inputs.
4. Click the **OK** button until the conversion takes place.

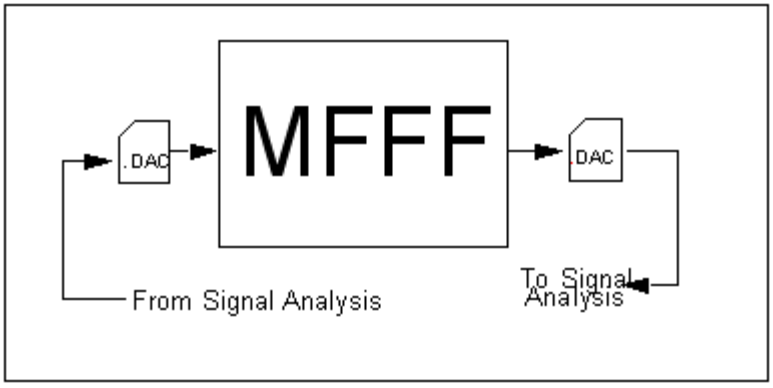


A display of the resulting PSDF is shown above. Note that only one predominate frequency is present in the signal at around 1/2 Hz plus a DC component at zero Hz. This PSDF could be used as input to a vibration fatigue analysis.

Note: See Auto Spectral Density (MASD), 260 for a more detailed description of this utility.

Fast Fourier Filter - (MFFF)

MFFF creates a finite impulse response (FIR), filter by using the window method. After creation, FFF automatically removes unwanted frequency components from time series data. High-pass, low-pass, band-pass or band-reject can be created.



MFFF calculates a set of filter coefficients, {h(n)}, which represent the ideal digital filter response {h(n)} modified by a Kaiser (I_o-Sinh) window, {w(n)}. The windowed digital filter can be represented as:

$$h(n) = w(n)h(n) \quad 0 \le n \le N-1$$

$$h(n) = 0 \text{ otherwise}$$

In the time domain, the ideal filter response, h(n), is calculated for each of the four filter types: low-pass, high-pass, band-pass, and band-reject from the definitions of the cut-off frequencies and a {SIN(X) / X} type function. Figure 14-13 illustrates such a function for a high-pass filter with a normalized cut-off of 0.35.

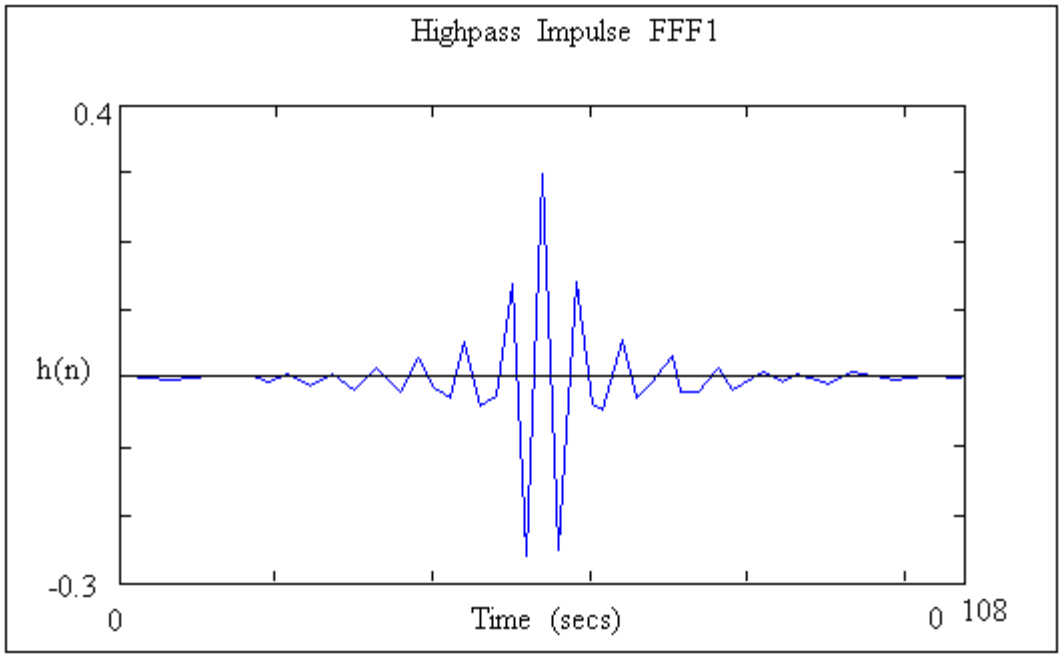


Figure 14-13 Ideal Impulse Response for a High-Pass Filter, $f_c = 0.35$

The Kaiser (I_o-Sinh) window function, w(n), is calculated from:

$$w(n) = I_o(b\sqrt{1-[2n/(N-1)]^2})/I_o(\beta)$$

$$-\left(N-\frac{1}{2}\right) \le n \le \left(N-\frac{1}{2}\right)$$

and

N		the number of values in the Kaiser window.

β		a constant that specifies a frequency response trade-off between the peak height of the side lobe ripples and the width or energy of the main lobe β is given a value of 9.620 which assures a stop-band attenuation of -96 Db.
$I_o(x)$		Is the modified zero-order Bessel function.

The modified zero-order Bessel function is calculated from the power series expansion of:

```
Io(x) = 1 + Å [(x/2)k/k!]2
```

k		summed from 1 to 25.

Having calculated the required impulse response, h(n), MFFF automatically realizes the filter, i.e., filters the specified data file.

A digital filter transfer function can be realized in either one of two ways, recursively or non-recursively. For a recursive realization, the functional relationship between the input {x(n)} and output {y(n)} sequences can be written as:

```
y(n) = fn [y(n-1), y(n-2), . . . . , x(n), x(n-1), . . .]
```

In this case, the current output sample, y(n), is a function of past output samples, y(n-1), y(n-2), as well as past and present input samples, x(n), x(n-1). . ..

For a non-recursive realization, the functional relationship between the input {x(n)} and output {y(n)} sequences can be written as:

```
y(n) = fn [ x(n), x(n-1), x(n-2), . . . ]
```

In this case, the current output sample, y(n), is a function of only past and present input samples, x(n), x(n-1). . ..

MFFF realizes the calculated filter function in a non-recursive way. If {h(n)} represents the set of, N, calculated filter coefficients then for a non-recursive realization the following difference equation may be written:

```
y(n) = h(0)x(n) + h(1)x(n-1) + h(2)x(n-2) + . . . + h(N-1)x(n-
N+1)
```

which may be rewritten as

```
y(n) = Σ h(m) x(n-m)
where m is summed from m = 0 to m = (N-1) the number of filter
coefficients.
```

The above equation represents a linear convolution of x(n) and h(n), therefore, the process of "filtering" the time sequence {x(n)} involves convolving it with the filter function {h(n)}. Convolution may be carried out wholly in the time domain, i.e., by explicitly solving the difference equation or by direct multiplication of the filter response function and the time series in the frequency domain. Because the procedure in the time domain involves many more arithmetic operations it is called slow convolution whilst the frequency domain approach is called fast convolution. MFFF uses fast convolution.

Module Operation

The MFFF module can be operated in one of the following three ways:

- From the **Advanced Loading Utilities** pulldown menu under **Tools | MSC Fatigue** (for Patran) or under **Tools | Fatigue Utilities** (for).
- In Stand alone mode by typing **mfff**
- By incorporating the MFFF commands in a batch operation.

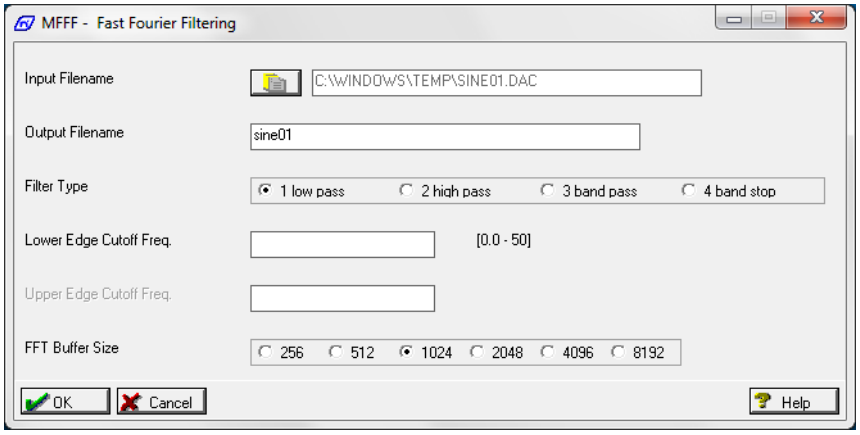


Figure 14-14 MFFF's Only Data Input Screen

The fields are as follows:

Option	Description
Input Filename	MFFF filters the contents of a file containing a single channel of data. By default MFFF expects this file to be a standard time series with a file extension of .dac. Therefore, if a time series is to be processed, it is only necessary to enter the filename, the .dac extension may be omitted. Files with the correct internal format but different file extensions, must have their names entered in full, i.e,. including the file extension. The name offered in the input field is the current default filename. To select this default it is only necessary to press the RETURN key or the F1 key. MFFF expects to find the input data files to be resident in the user's directory, however, other directories can be accessed if the complete file specification i.e,. path name and filename are entered. Alternatively use the pick list facility.
Output Filename	The filtered signal is written to a standard time series file with an extension of .dac. The default for this output file is the input file, i.e., the input file itself being filtered. This default is presented in the output file name field and to select it is only necessary to press the RETURN key. If the default is selected, or if the filename of an existing file is entered, MFFF will prompt for confirmation to overwrite it. The output file containing the filtered signal is written back to the user's directory. If it is required to place it in some other directory, then this must be specified explicitly.
Filter type	**1 - low pass, 2 - high pass, 3 - band pass, 4 - band stop** MFFF can filter data according to one of the following four filter types: Low-pass - that which passes lower frequencies but excludes higher frequencies. High-pass - which passes higher frequencies but excludes lower frequencies. Band-pass - which passes a contiguous band of frequencies but excludes all other frequencies. Band-stop - which excludes a contiguous band of frequencies but passes all other frequencies. Figure 14-15 below illustrates the characteristics of the four filter types. The two frequencies, f_1 and f_2, define the respective limits (cutoffs) of each filter type. Note that f_n is the Nyquist frequency, i.e., half the sampling rate - 0.5 f_s. It is only necessary to enter the appropriate option number (1 to 4) to select the desired filter type. Depending upon the nature of the filter required, MFFF will prompt for the following inputs.

Option	Description
Low-Pass Option 1	Lower edge cut-off freq: Enter the cut-off frequency for the pass-band, i.e. frequency f_1 in Figure 14-15. This frequency will define the limit of the pass band. Any frequency between zero and f_n, the Nyquist frequency, may be entered.
High-Pass Option 2	Lower edge cut-off freq: Enter the cut-off frequency for the stop-band, i.e. frequency f_1 in Figure 14-15. This frequency will define the limit of the stop-band. Any frequency between zero and f_n, the Nyquist frequency, may be entered.
Band-Pass Option 3	Lower edge cut-off freq: Enter lower cut-off frequency of the pass-band, i.e. frequency f_1 in Figure 14-15. This frequency will define the limit of the first stop band. Upper edge cut-off freq: Enter the cut-off frequency of the pass-band, i.e. frequency f_2 in Figure 14-15. This frequency will define the upper limit of the pass-band. Any frequency, between f_1 and f_n, the Nyquist frequency, may be entered.
Band-Reject Option 4	Lower edge cut-off freq: Enter the lower cut-off frequency of the stop-band, i.e. frequency f_1 in Figure 14-15. This frequency will define the limit of the first pass-band. Upper edge cut-off freq: Enter the cut-off frequency of the stop-band, i.e. frequency f_2 in Figure 14-15. This frequency will define the upper limit of the pass band. Any frequency, between f_1 and f_n, the Nyquist frequency, may be entered.
FFT Buffer Size	Specify the size of the FFT buffer to use in the analysis. The bigger the buffer size, the higher the resolution of the FFT. The FFT buffer sizes allowed are 256, 512, 1024, 2048, 4096, or 8192. At this stage MFFF will proceed to create the desired filter and then use it to filter the input data file. On completion, it will display a page of results such as that shown below.

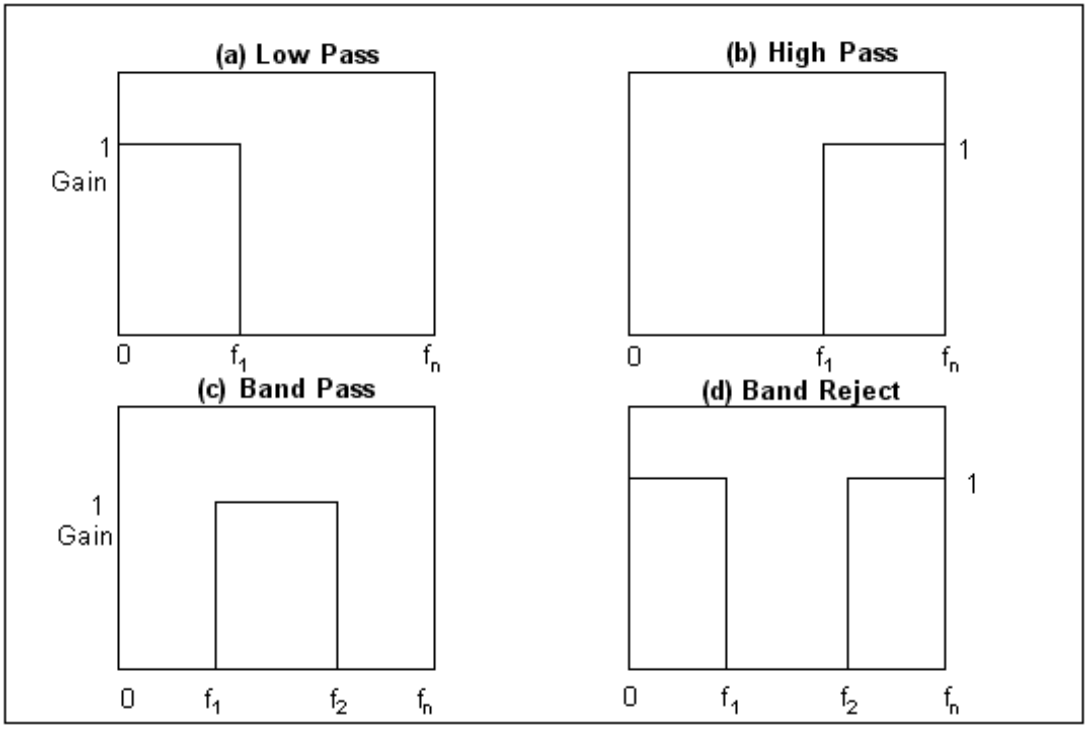


Figure 14-15 Characteristics of the Four Filter Types

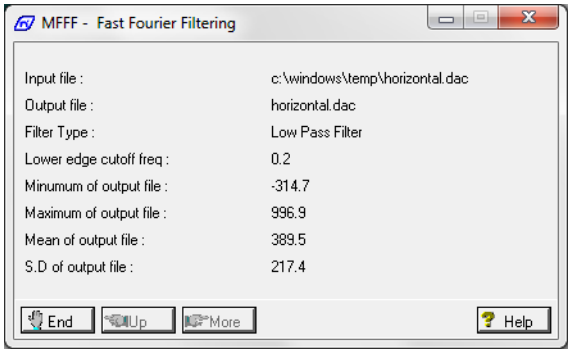


Figure 14-16 A Results Screen

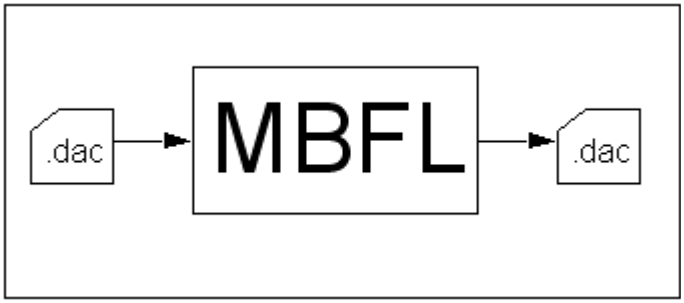
Batch keywords

A list of MFFF's batch keywords:

/INPut	The name of the data file to process, /INP=INDATA
/OUTput	The name of the of the output data file, /OUT=OUTFIL
/OVerwrite	Whether or not to overwrite an existing data file, /OV=Y
/TYPe	The type of filter to use in the analysis. /TYP=1 selects a low-pass filter., /TYP=2 selects a high-pass filter., /TYP=3 selects a band-pass filter, /TYP=4 selects a band-stop filter
/LOcut	Lower edge cut-off frequency, f1 (Hz), /LO=5
/HIcut	Upper edge cut-off frequency, f2 (Hz), /HI=50
/FFT	Size of FFT buffer, 256-8192., /FFT=1024

Butterworth Filtration - (MBFL)

This program takes a signal file and passes it through a Butterworth filter to produce an output signal file.



The filter characteristic can be a low pass, high pass, band pass or band-stop. The filter order can be from 1 to 8 poles which will give a cutoff of between 6db and 48db per octave.

Module Operation

The MBFL module can be run in one of the following three modes:

- From the **Advanced Loading Utilities** pulldown menu under **Tools | MSC Fatigue** (for Patran) or under **Tools | Fatigue Utilities** (for).
- In Stand alone mode by typing **mbfl**
- By incorporating the MBFL commands in a batch operation.

The first two modes are interactive. Once running in interactive mode, the MBFL module will display the following screen:

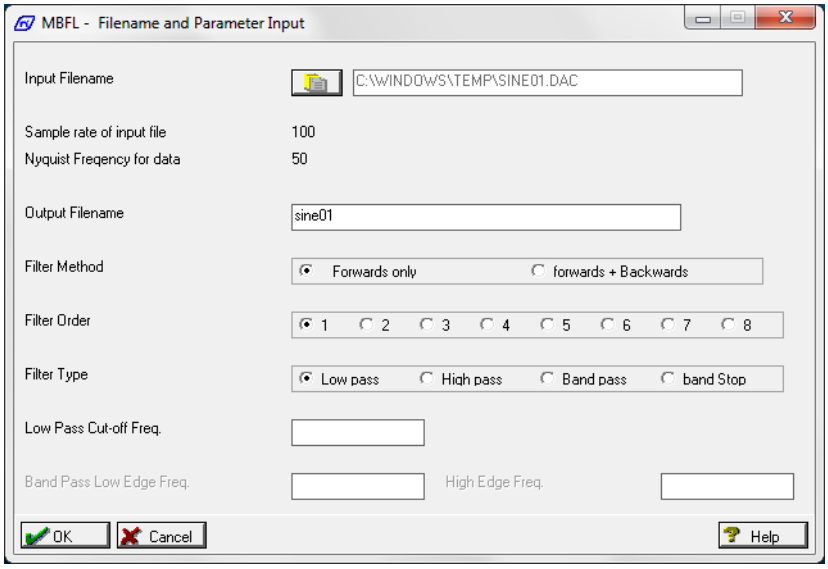


Figure 14-17 The First MBFL Screen

The purpose and usage of each field is explained below. Until Input Filename is completed and confirmed (by pressing F1) then none of the other fields will appear.

Option	Description
Input Filename	In this field the user should type the name of an input file (a single parameter `.dac` file). By default MBFL assumes a `.dac` file extension but if a file with a different file extension is to be processed then enter the file name plus extension in full. MBFL will expect to find the input data files resident in the user's directory, however, other directories can also be accessed if the complete file specification i.e., path name and filename are entered. Probably the easiest way to name an input file is to use the F3/List facility.
Output Filename	This is the file into which the filtered data is placed. It is a standard single parameter file, and will be given a `.dac` extension, although the user can enter a different extension. This file can also be created in a different drive/directory by specifying the full path name of the file. If the file exists the user will be asked for permission to overwrite it.
Filter Method Forwards only / forwards & Backwards	In forwards mode the data is passed through the filter from *tstart* to *tend*. This emulates a normal analogue hardware Butterworth filter and induces phase shift characteristics. In forwards/backwards mode the data is first passed through the filter from *tstart* to *tend* as above. The data is then passed again through the filter but this time from *tend* to *tstart*. This has the effect of correcting any phase shift introduced in the forward pass. **Note:** Passing the data through forwards and backwards will double the attenuation, i.e., a 4 pole forward and back filter gives the equivalent attenuation to an 8 pole forward only filter.

Option	Description
Filter order [1-8]	The filter order determines the rate at which frequencies are rejected as the cut-off frequency is exceeded. For a Butterworth filter this is 6 dB per octave, or 20 db per decade (see Table 14-1). If the cutoff frequency is set at a Hz (amplitude = 1) then by the time b Hz is reached (which is 1 Octave higher than a the amplitude is 1/256, i.e., it is reduced by 256 times. Lower orders will produce shallower filter slopes and a lesser reduction in signal amplitude per octave. What this actually means is shown in the diagram below. It illustrates an 8th order low pass filter **Note:** This means that data at and before the cut-off frequency is reduced in amplitude. The maximum reduction is always 0.7071 at the cut off frequency and the effect in the pass band is dependent on the order of the filter. This is a characteristic of all filters, both analogue and digital. The cutoff frequency defines a 3dB drop in amplitude
Filter type	Here you are being prompted to enter which filter type you wish to use on the input signal, there are four choices: L - Low pass (frequencies above the cut-off will be attenuated). H - High pass (frequencies below the cut-off will be attenuated). B - Band pass (frequencies below the lower cut-off and above the higher cut-off will be attenuated). S - Band stop (frequencies between the lower cut-off and the higher cut-off will be attenuated). **Note:** For the Band pass, and Band stop filter, unpredictable results can occur if the upper and lower cut-off frequencies are close. The problem becomes worse the higher the filter order. The fields that appear beneath Filter type depend upon the setting of filter type.
Cutoff frequency setting fields	The Figure 14-19 flowchart determines which fields appear depending upon the setting of Filter type.

Table 14-1 Filtration per Octave

ORDER	Filter slope per octave (db)	Filter slope per decade (db)	Reduction/Octave
1	6	20	1/2
2	12	40	1/4
3	18	60	1/8
4	24	80	1/16
5	30	100	1/32
6	36	120	1/64
7	42	140	1/128
8	48	160	1/256

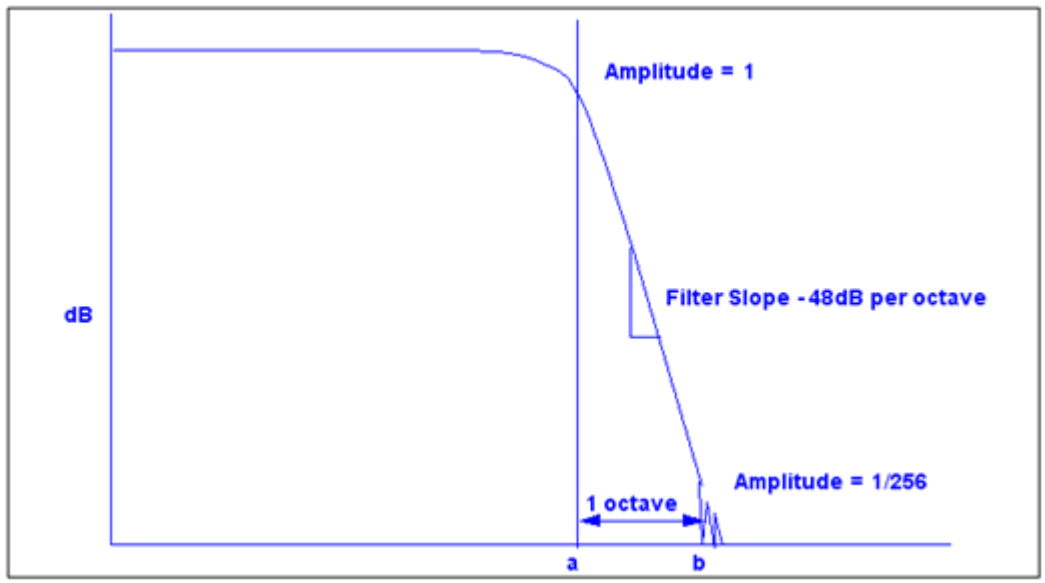


Figure 14-18 An 8th Order Low-Pass Filter

If Filter type is...

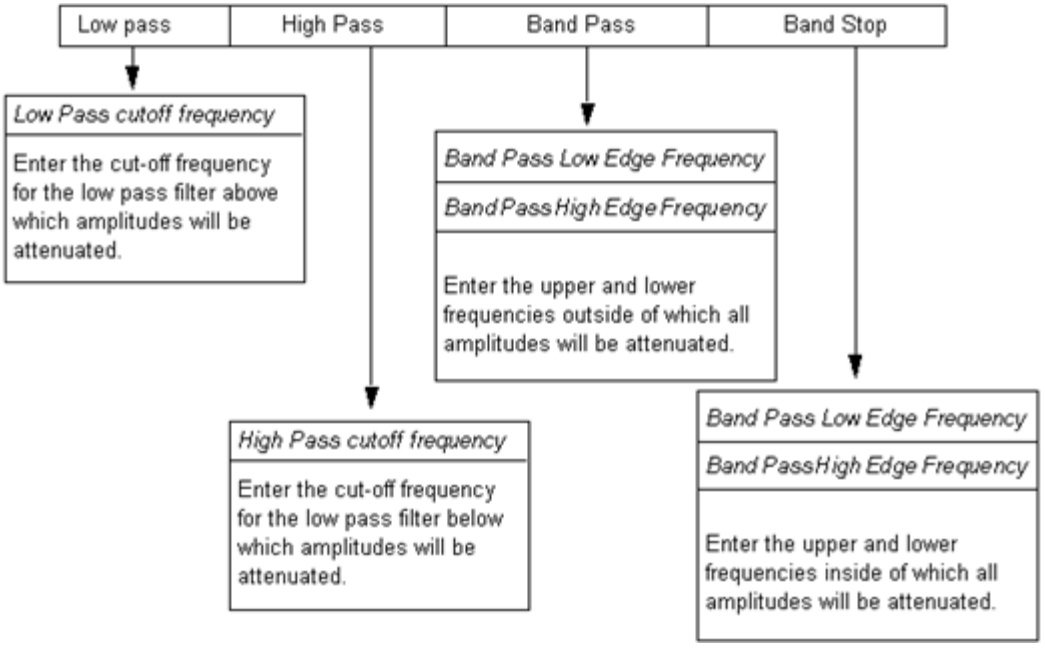


Figure 14-19 Flowchart of Cutoff Frequency Settings

Note:	The time taken to perform a filter will be longer for band pass and band stop. It will also be longer the higher the order of the filter.

When all the above fields are set, pressing or clicking OK will cause MBFL to create the output file, and then exit MBFL.

Batch Operation

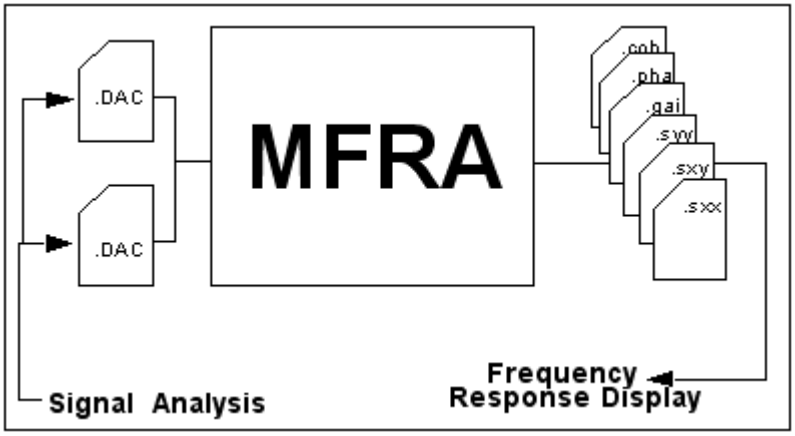
A list of MBFL's batch keywords:

/INPut	The name of the input time series to be filtered, /INP=XFILE
/OUTput	The name of the output file that will contain the filtered data, /OUT=OFILE
/OVerwrite	Whether to overwrite existing output files, /OV=Y
/METHod	Filter method, forward or backward (F or B)
/TYPE	The type of filter to use, /TYPE=BP
/ORDER	The order of the filter [1-8], /ORDER=8

/FRQ1	The filter cut-off frequency for low, high pass or for the lower edge of the band pass and band stop, /FRQ1=100
/FRQ2	The filter cut-off frequency for the higher edge of the band pass or band stop, /FRQ2=150

Frequency Response Analysis - (MFRA)

The frequency response analysis, MFRA, analyses the response of a single input, single output linear system. Six files of statistics are generated as a result of this analysis.



The program analyses the response of a single input, single output linear system.

MFRA uses the following terms (ASD=auto spectral density, CSD=cross spectral density):

$S_{xx}(\omega)$ as the ASD of x(t)

$S_{yy}(\omega)$ as the ASD of y(t)

$S_{xy}(\omega)$ as the CSD of x(t) with y(t)

$S_{yx}(\omega)$ as the CSD of y(t) with x(t)

These are complex values having a real and imaginary part

e.g., $S_{xx}(\omega) = C_{xx}(\omega) + Q_{xx}(\omega)$

where $C_{xx}(\omega)$ is the real part and $Q_{xx}(\omega)$ is the imaginary part.

The GAIN of the system is defined as

$$\frac{[Cxy(\omega)^2 + Qxy(\omega)^2]^{\frac{1}{1/2}}}{Sxx(\omega)}$$

The PHASE relationship between x(t) and y(t) is

$$TAN^{-1}[\frac{Qxy(\omega)}{Cxy(\omega)}]$$

The COHERENCE is

$$[Sxy(\omega) \bullet Syx(\omega)]/[Sxx(\omega) \bullet Syy(\omega)]$$

The TRANSFER FUNCTION of the system is a complex set of values where.

Real Part =

Complex Part = $\frac{Qxy(\omega)}{Qxx(\omega)}$

The program uses Fourier Transforms to calculate the Auto and Cross Spectral Density files as complex values. Using these values it generates all the results files, then saves the Spectra as real parts only.

The following files are generated with each run of the program.

ASD of x(t)	`.sxx`
ASD of y(t)	`.syy`
CSDs	`.sxy and syx`
GAIN	`.gai`
PHASE ANGLE	`.pha`
COHERENCE	`.coh`

Module Operation

The MFRA module can be run in one of the following three modes:

- From the **Advanced Loading Utilities** pulldown menu under **Tools | MSC Fatigue** (for Patran) or under **Tools | Fatigue Utilities** (for).
- In stand alone mode by typing **mfra**
- By incorporating the MFRA commands in a batch operation.

The first two modes are interactive. Once running in interactive mode the MFRA module will display the following screen:

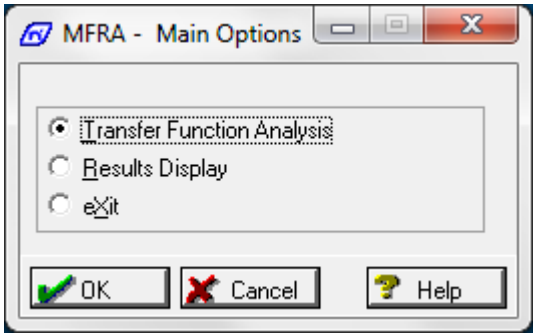


Figure 14-20 MFRA's Main Menu

The routes through MFRA are shown below.

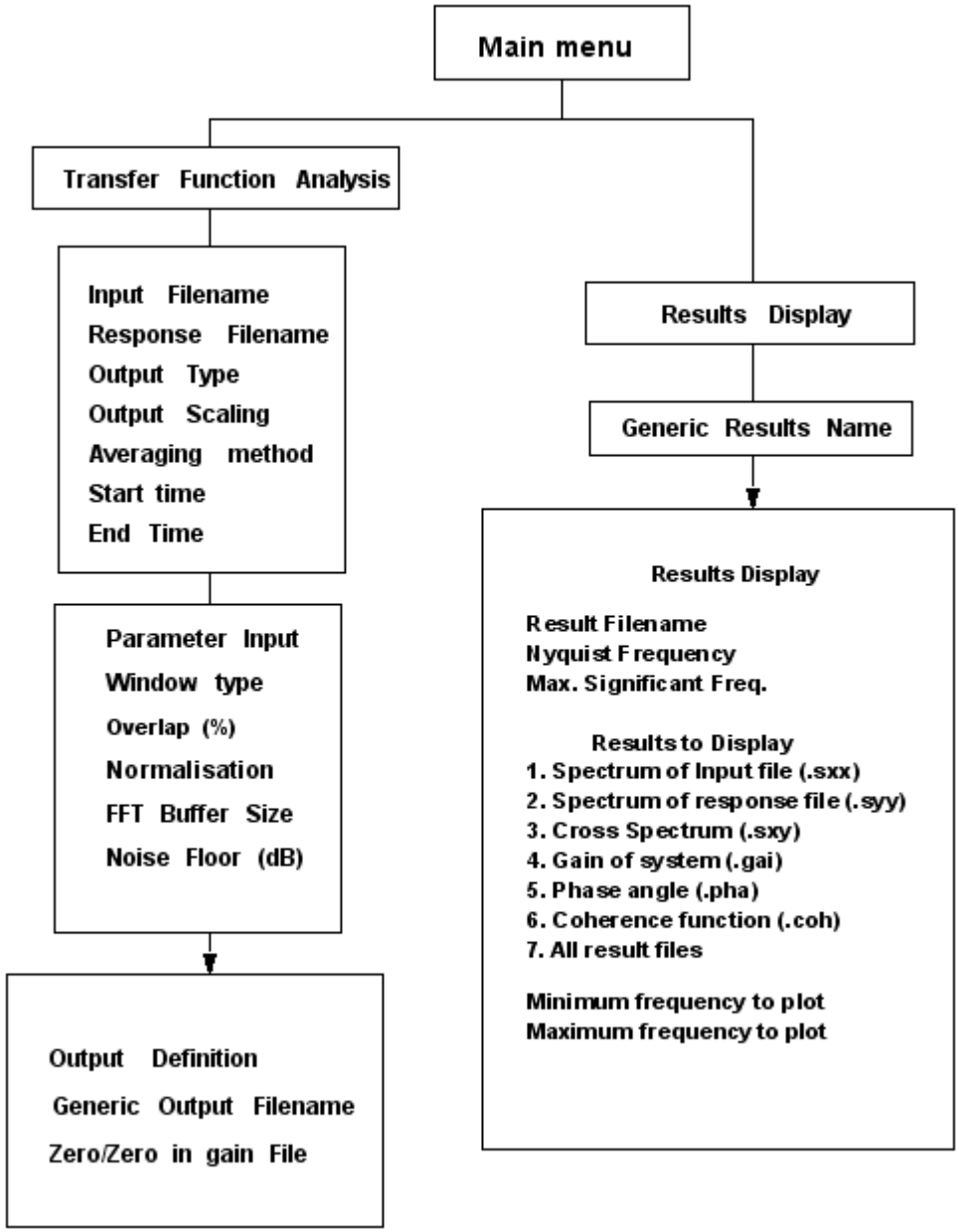


Figure 14-21 The Routes Through MFRA

OPTION 1 : Analysis of signal - the Transfer Function Analysis (TFA)

The first screen in this option is:

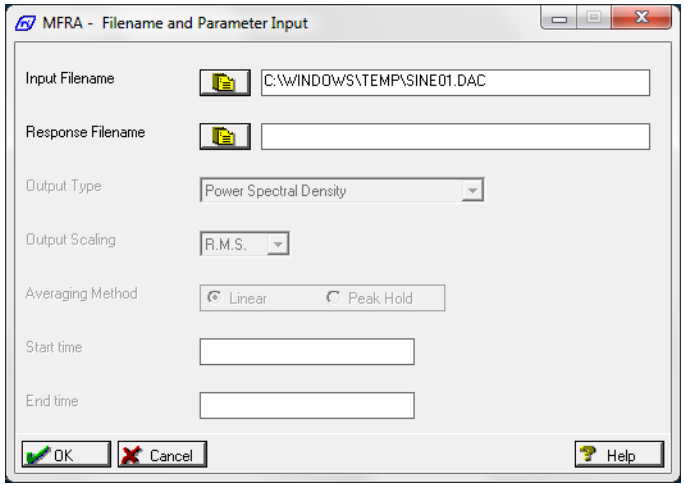


Figure 14-22 The First TFA Screen

Option	Description
Input Filename	By default, MFRA expects the input data file to be a standard time series with a file extension of .dac. Files with the correct internal format but different file extensions must have their names and extensions entered in full. MFRA will expect to find the input data files resident in the user's directory, however, other directories can be accessed by using the pick list facility.
Response Filename	This is the measured response of the system. Again, by default, the response data file is assumed to be a standard time series with a file extension of .dac. The same condition apply to this question as for the input signal. **Note:** BOTH input files must have the same number of data points and sample rates. If they do not, then using the graphical editor MGED, and Sample Rate Adjuster (in PTIME), may rectify the situation.
Output Type	Power or Energy Spectral Density, Amplitude Spectrum This option can be used to specify the nature of the spectral output required from the analysis. The different outputs arise from the way in which the complex FFT coefficients are postprocessed. If the magnitude of the FFT is scaled, squared and then divided by the spectral width, then the Power Spectral Density, or PSD, results. If the magnitude of the FFT is scaled to reflect the amplitude of the original data, the Amplitude Spectrum is produced. If the PSD is multiplied by the analysis time the so-called Energy Spectral Density or ESD is produced.
Output Scaling	Peak/RMS Normally the magnitude of the FFT components is scaled such that the amplitude will be the RMS of the frequency component. For example, a sine wave of amplitude 1 has an RMS of 0.7071. To calculate and display the true amplitude of the frequency components, select the Peak option.

Option	Description
Averaging Method	When multiple FFT buffers are calculated, the spectral estimates for each buffer can be handled in several ways. One simple way is to linearly average the component values over the number of buffers calculated. The disadvantage of this method is that fast, high amplitude events which occur in a single buffer are lost when the average of a large number of buffers is taken. In this case the alternative peak hold method, in which the largest FFT magnitude for each component is retained, may be more suitable.
Start time	This option allows the specification of a time window for analysis. As a default, the whole input time series will be analyzed. Specific times for the start and end of analysis my be entered directly or alternatively the keywords START and END may be used, together with a single arithmetic operation. For example, START+5 will specify that analysis should commence 5 seconds into the data and END-3.1 will specify a halt to analysis 3.1 seconds before the end of file. Extra details keywords may also be used to define the extent of the analysis time window.
End time	This option allows the specification of a time window for analysis. As a default, the whole input time series will be analyses. Specific times for the start and end of analysis my be entered directly or alternatively the keywords START and END may be used, together with a single arithmetic operation. For example, START+5 will specify that analysis should commence 5 seconds into the data and END-3.1 will specify a halt to analysis 3.1 seconds before the end of file. Extra details keywords may also be used to define the extent of the analysis time window. Note that the end of the analysis window is defined by the first occurrence of the last data point within an FFT buffer.

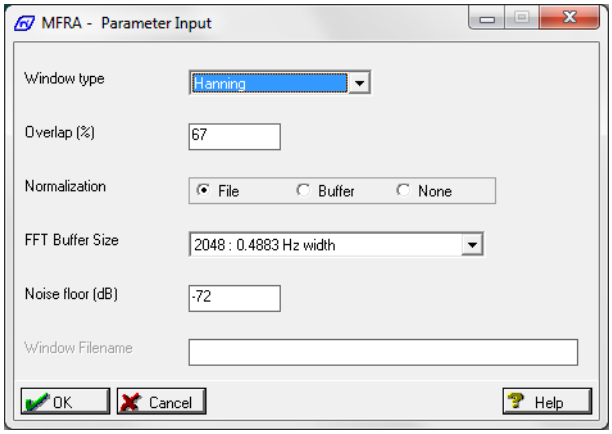


Figure 14-23 Further Process Specification

Option	Description
Window type	Each buffer of data processed by MFRA is tapered using a window function. A window improves the accuracy of the FFT since it reduces the magnitude of the data at the ends of each FFT buffer in a gradual manner thus avoiding discontinuities at the extremities of the block of data. The following window functions are supported: Hanning, Kaiser-Bessel, Triangular, Cosine bell, User defined, and Rectangular.
Overlap (%)	The purpose of overlapping spectral windows is to minimize the effects of "leakage" at the edges of each window. The greater the overlap the smaller will be the effect of this leakage, however, the processing time will increase. An optimum value for the overlap, which minimizes attenuation between successive buffers, and providing reasonable computational times would be 67%.

Option	Description
Normalization	File\Buffer\None A time series which has an overall mean significantly away from zero is said to possess a DC offset. This offset will generate a component in the frequency domain close to zero Hz. which can swamp the spectrum. The effect of a DC component can be minimized by eliminating it prior to transformation into the frequency domain. This can be done in two ways: Firstly, the mean of the whole file can be subtracted from each value in successive FFT buffers prior to transformation. Secondly, the mean of each buffer can be calculated individually and that value subtracted. Note that these normalization procedures do not permanently affect the input time series itself.
FFT Buffer Size	The FFT buffer size defines the resolution of the power spectrum. The buffer must be a power of 2, and the longer the buffer, the higher the resolution of the spectral lines. To calculate the resolution divide the Nyquist frequency by half the FFT buffer size. e.g., if nyquist = 178 Hz and the FFT buffer size selected is 1024, then the spectral lines are 178 / (1024 / 2) = 0.347 Hz apart. Another use of a smaller buffer size is for short data files as these cannot be adequately analyzed with a big buffer, since there may not be enough data to give a good spectral average. Using a smaller buffer size could give a better spectral average at the expense of spectral line width.
Noise floor (dB)	When the amplitude of a particular frequency component is small, the FFT coefficients become vanishingly small and may cause computational difficulties or distort results; for example, the phase calculation is particularly affected. In order to overcome these difficulties, a value may be defined which represents an effective zero. This cut-off point is normally specified in dB down from the maximum magnitude. The default is -72dB. Note that -20dB is one order of magnitude less than the maximum.

The final screen of the process setup is shown below:

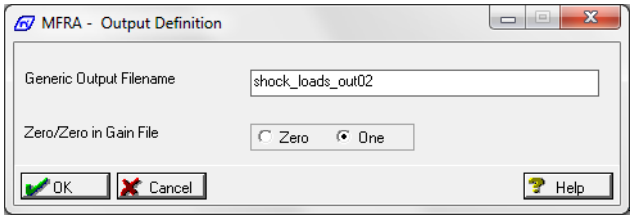


Figure 14-24 The Last Setup Screen

Option	Description
Generic Output Filename	The output results from MFRA are: ■ ASD of x(t) - (.sxx) ■ ASD of y(t) - (.syy) ■ CSD - (.sxy) ■ GAIN - (.gai) ■ PHASE ANGLE - (.pha) ■ COHERENCE - (.coh) The name you enter here will have the above file name extensions.
Zero/Zero in Gain File	When calculating the gain, the software must decide what to output when there is no significant data in the FFT of both the input and response files. The two alternatives the user may select are Zero (result is not considered to be important) or One (input and output are identical).

OPTION 2: - Results Display

This form allows input of the generic results filename and some output parameters. The generic output filename will be used to form the stem of the six output filenames to be plotted.

The user will then be offered a list of plot options which will allow the plotting of one or all of the available results files.

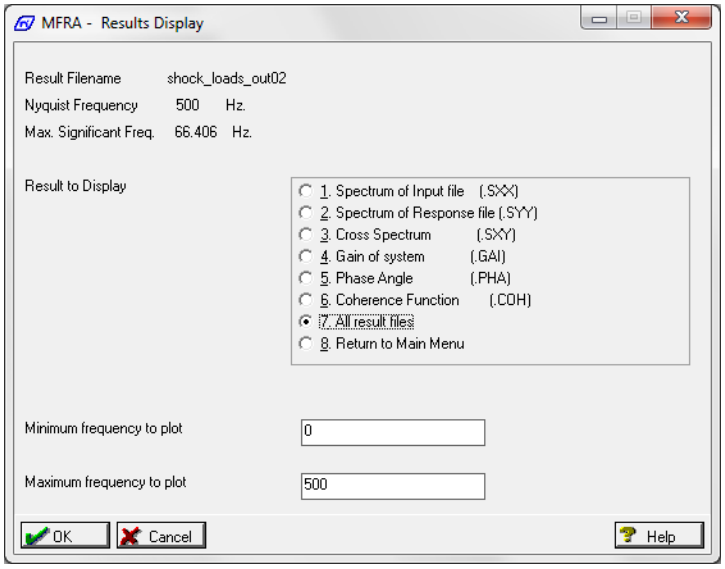


Figure 14-25 The Results Display Screen

The fields are as follows:

Option	Description
Result to Display	Choose any one, or choose all, or the results files to plot. MFRA will then load the graphics module Quick Look Display (MQLD) and plot the results file. When QLD is exited the user will be returned to this screen. Multi File Display (MMFD) is used when '7. All results files' is selected.
Minimum frequency to plot	Specify the minimum frequency to plot in Hz. The value must not be negative and must be less than the Nyquist frequency (the Nyquist frequency for this case is displayed in the statistics area). This question is not so important for interactive users since the limits may be changed when the data is displayed graphically but is very useful for batch operation.
Maximum frequency to plot	Specify the maximum frequency to plot in Hz. The value must greater than the specified Nyquist frequency (the Nyquist frequency is displayed in the statistics area). This question is not so important for interactive users since the limits may be changed when the data is displayed graphically but is very useful for batch operation.

Batch Operation

A typical batch command line would be:

```
mfra /opt=1/inp1=data01/inp2=data02/fft=4/win=3
/gen=fraexamp/ov=y/norm=n/opt=2/disp=5/pltnam=fred
```

This batch command line will carry out a frequency response analysis using `data01` as the input signal and `data02` as the response. The results will be stored under the default generic name of `fraexamp`. The analysis will be carried out with a FFT buffer of 512 points and a hanning window. The default conditions of 30% overlap and normalized input data will be met. A graphical display hardcopy file (`*.plt`) will be produced from /OPT=2, and it will be the .pha file from /DISP=5, with the name `fred.plt`.

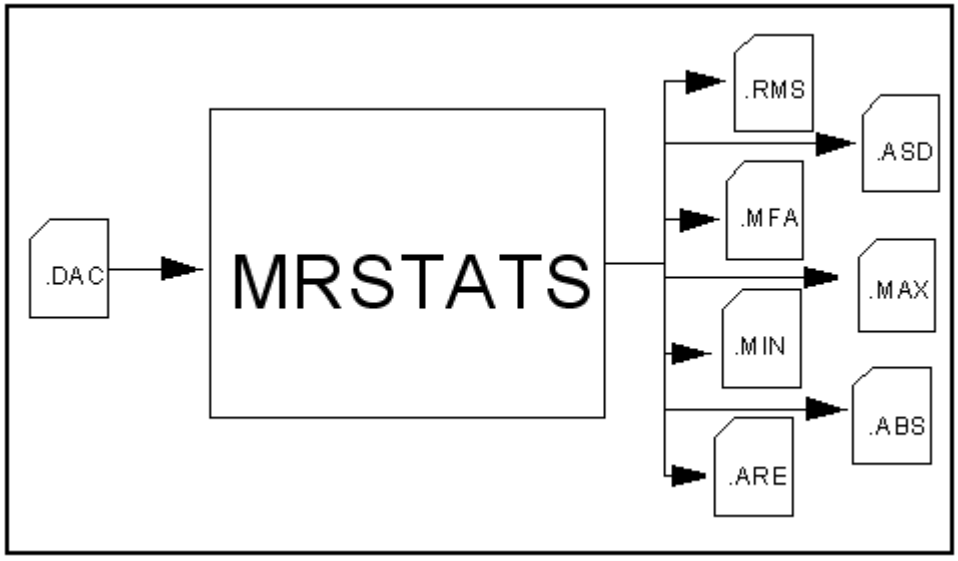
A list of MFRA's batch keywords:

/OPTion	This keyword is used for the first menu where the user chooses either analysis (option 1) or display (option 2)
/INP1	The input filename, /INP1=FILE1
/INP2	The response filename, /INP2=FILE2
/GENeric	The output filename required for the results data file, /GEN=RESULT
/OTYPE	Output Type (PSD,AMP,etc.) P, A, E
/SCALE	RMS or Peak R, P
/AVERage	Linear averaging or Peak Hold L, P
/STArt	Start Time for Analysis
/END	End Time for Analysis
/OV	Whether to overwrite an existing results file, /OV=Y
/NORM	Option to normalize the input time series data (Y/N), //NORM=F,B,N
/OVERLap	The amount of overlap of the FFT buffers, /OVERL=45
/FFT	The permissible FFT buffer size: 32, 64, 128, 256, 512, 1024, 2048, 4096, 8192, /FFT=32
/FLOor	Noise Floor
/WFILE	User Defined Window Filename
/ZERo	Value of 0/0 in gain file Z, O
/WINdow	The type of window function to be used, H, C, R, /WIN=H
/DISPlay	This allows the user to choose the display to be viewed: 1. Power Spectrum of input signal (.SXX), 2. Power Spectrum of response signal (.SYY), 3. Cross Spectrum of input and response signals (.SXY), 4.Gain plotted against frequency (.GAI), 5.Phase angle plotted against frequency (.PHA), 6.Coherance function plotted against frequency (.COH), 7.Display all, 8.Exit display options, /DISP=1
/FMAX	The maximum frequency to be displayed, /FMAX=45
/FMIN	The minimum frequency to be displayed, /FMIN=10
/PLOt	Automatically plot results Y, N
/PLTNAM	Hardcopy filename

It is recommended that, by default, the /OV=Y keyword be included in every batch command line, since if it is omitted and an output file with the specified name already exists, batch operation will cease.

Statistical Analysis - (MRSTAT)

MRSTATS analyses a time signal and produces a number of statistics about that signal. MRSTATS works by breaking the input time signal into segments, and statistically analyzing each segment. Each statistic is fed into an output signal file.



Input files normally have the .dac file extension.

MRSTATS can generate a maximum of seven statistics from an input file. Each set of statistics is fed into its own type of output file, the type of statistic being identifiable by the file extension. For example, the output file that records the variation of the mean signal strength against time has a .mea file extension.

The MRSTATS module accepts input from .dac files and divides the signal recorded by the .dac file into a series of time slices of equal length. Each slice then has up to seven operations carried out upon it. The operations are as follows:

Operation	Output File Extension
Root Mean Square (RMS)	.rms
Standard Deviation	.rsd
Mean	.mea
Maximum Value	.max
Minimum Value	.min

Absolute Maximum	`.abs`
Area Under Data	`.are`

The result of each operation is placed in the appropriate file, and the next slice is then analyzed. Eventually all of the signal will be analyzed, and up to seven output files will have been created that record the variation of the above named statistics against time, for each user specified slice of the input signal.

The Area Under Data option is a record of the area under the curve as derived by integration.

MRSTATS does not have the ability to display the output files. Use data display modules such as MMFD or MQLD to display files.

Module Operation

The RSTATS module can be started and operated in one of three ways:

- From the **Advanced Loading Utilities** pulldown menu under **Tools | MSC Fatigue** (for Patran) or under **Tools | Fatigue Utilities** (for).
- In Stand alone mode by typing **mrstats**
- By incorporating the MRSTATS commands in a batch operation

MRSTATS is composed of three screens. Whether the user has run MRSTATS from the menu, or directly from the system prompt, the first screen that they will see looks like this:

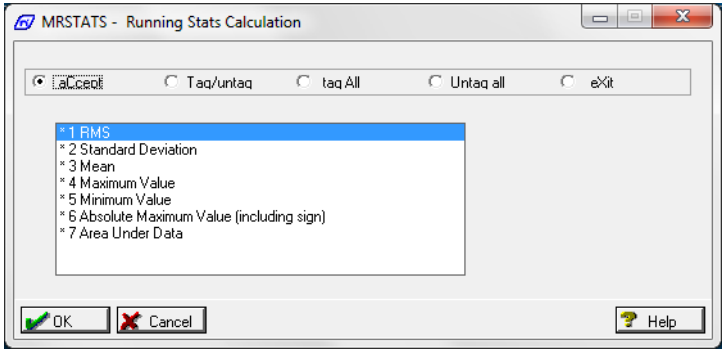


Figure 14-26 Selecting the Statistics

The horizontal bar contains the tag options menu. The items that can be tagged are those statistics named in the statistics list below the tag menu. Tagging a statistic means that MRSTATS will produce an output file containing the tagged type of statistical data.

A statistic is tagged when it has an asterisk (*) next to it.

Note:	Use the left/right arrow keys and the space-bar to select from the tag menu, OR, press the appropriate letter (the capital letter in the option is the 'hot key'. The selected option is the one highlit with the highlight bar or point and click with the mouse.

To move the highlight bar within the list of statistics use the up/down arrow keys. The 5 tagging menu choices do the following:

Option	Description
aCcept (or OK)	When the user has selected those statistics for which they want an output file to be created, they must press OK to accept the screen, OR, press C for aCcept. Pressing OK takes the user to the second (and last) screen of MRSTATS.
Tag/untag	This allows the user to tag individual statistics while leaving others untagged. To tag/untag a statistic move the highlight bar over the statistic and press Enter. Users may select as many, or as few, statistics as they wish.
tag All	As the name suggests, this option allows the user to simply and quickly tag all the statistics. To use tag All, move the highlight bar over it and press Enter. An output file will then be produced for all seven statistics.
Untag all	This has the opposite affect to tag All. The program will not run if there are no statistics tagged (there would be no point because no output would be produced). Therefore, the only use of Untag all is to clear a previous set of selections prior to making new selections.
Exit	This option exits MRSTATS and returns the user to the operating system. Options are not saved.

When the user has selected those statistics for which they want an output file to be created, they must click **OK** to accept the screen. Pressing OK, or C to aCcept, takes the user to the second screen of MRSTATS.

The screen looks like this:

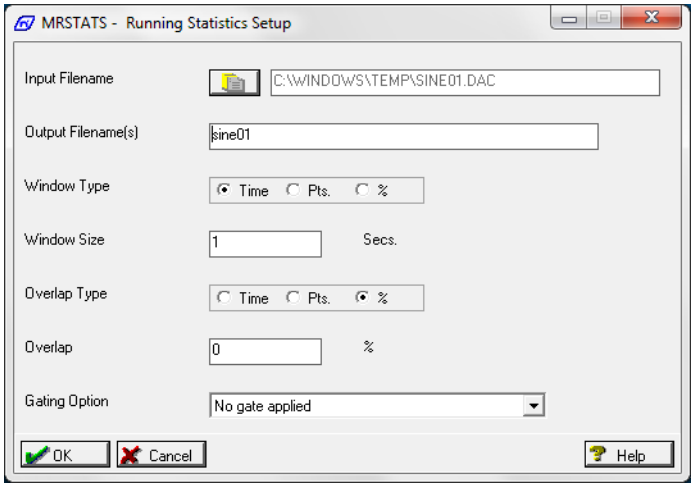


Figure 14-27 Specifying the Analysis

Please note that initially only the first field is displayed (filename of input signal). Only when the input signal filename has been entered will MRSTATS display the other fields.

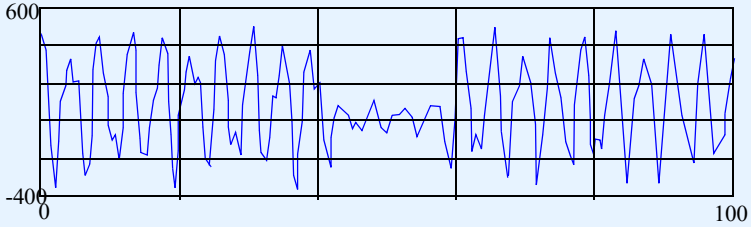
This screen allows the user to name the input (`.dac`) file, the output file names, and specify certain technical parameters.

The fields are as follows:

Option	Description
Input Filename	The name of the file that contain the input signal must be entered here. The input file will normally be a `.dac` file, which is also the default file extension. The full path must be specified only necessary if the file is not in the current directory, or currently defined path.

Output Filename(s)	Here the user can choose the name of the output file, or if more than one statistic has been flagged, the output files. If all seven statistics were tagged, and `data01` was chosen as the output file name, the output files produced will have the following names:
	
	`data01.rms`
	`data01.rsd`
	`datao1.mea`
	`data01.max`
	`data01.min`
	`data01.abs`
	`data01.are`
	
	Both of the above fields can accept up to 32 characters (to name the drive, path, and extension).

Window Type	Enter the method used to specify the window size. The options are: ■ Time - specify a window size in seconds. ■ Points - specify the window as a number of points ■ % (Percent) - specify the window as a percentage of the signal length

Option	Description
Window size	It is in this field that the user specifies the length of the window size units into which the input signal will be divided. The diagram below illustrates how a 100 second long input signal is divided into five 20 second segments by choosing a window size of 20 s. The same rules apply if points or % are selected.  Each of the 5 segments will be analyzed and each of the 7 output files will contain data points each from of the five time segments. If nothing is entered in this field, MRSTATS enters a default value of 10 units (seconds, %, or points). If the signal is less than 10 units long, the default window size is the length of the signal. Users may not specify a window size greater than the length of the signal.
Overlap	This field enables users to overlap time segments. For example, if a window size of 10 seconds was input for a 100 second signal, and an overlap of 50%, the output readings will be at the following times: First reading between 0 and 10 seconds, second between 5 and 15 seconds, third between 10 and 20 seconds, etc. until 100 seconds is reached. The default value is no overlap, i.e., consecutive windows. The maximum overlap is 99% or its equivalent. Negative overlaps are not allowed. The options are: ■ Time - specify the overlap in seconds. ■ Points - specify the overlap as a number of points. ■ Percent - specify the overlap as a percentage of the window length.

Option	Description
Gating Option	A gating signal is a signal that records events which can be used to define which parts of an input signal will be analyzed. In other words it is a kind of filter. For example a user may only be interested in analyzing input signals that correspond to an event such as the brake being applied on a test vehicle. If the use of the vehicle's brakes are recorded, the signal can be used as a gating signal that is mapped to the input signal, so that only those parts of the input signal created by brake application will be analyzed. There are 6 ways of setting the gating signal plus an option for no gate (the default): 1. Window starts when the gating signal reaches a specified value, and ends when the gating signal rises to the specified value. 2. Window starts when the gating signal is NOT at a specified value, and ends when the gating signal reaches the specified. 3. Window starts when the gating signal is greater than the specified value, and ends when it falls to the specified value. 4. Window starts when the gating signal is less than the specified value, and ends when it is outside the specified limits. 5. Window starts when the gating signal is inside the specified limits, and ends when it is outside the specified limits. 6. Window starts when the gating signal is outside specified limits and ends when it is inside the specified limits.

Option	Description
Gating filename	The user enters the name of the gating signal (if any) into this field. Typically it will have a `.dac` file extension. The gating signal will normally be of the same time duration and sample rate as its corresponding input signal. The default file extension for a gating signal is `.dac`.
The Gating value(s)	To complete the screen, move the highlight to the gating values box. The exact layout of the gating values boxes depends upon the Gating Options chosen. The values must lie within the gating signal limits (displayed to the right of the gating value box).
Event Analysis Type	The gate, once triggered, will operate for a certain portion of the signal (an event frame), but it may encounter another trigger before it has reached the end of that event frame. How this is handled is set by the Event Analysis Type field. The event analysis type can be of three forms: 1. Analysis of signal takes place for the whole of the event frame i.e., multiple windows can occur in one event. If an event is shorter than the window time then no analysis will occur for this event. 2. Analysis of signal takes place only for first window in an event frame. If an event is shorter than the window time then the analysis occurs over the event time. 3. The window value will be ignored and the analysis of the signal will take place over the whole of the event time in the gating signal. Note that the x-axis value in the output files is stepped in an arbitrary value of 1 sec.

Statistics are calculated for that part of the signal which lies within the window.

The two drawings below illustrate an input signal (A) and its corresponding gating signal (B). When brakes are applied the gating signal is 0.7, and it drops to 0 when brakes are not applied.

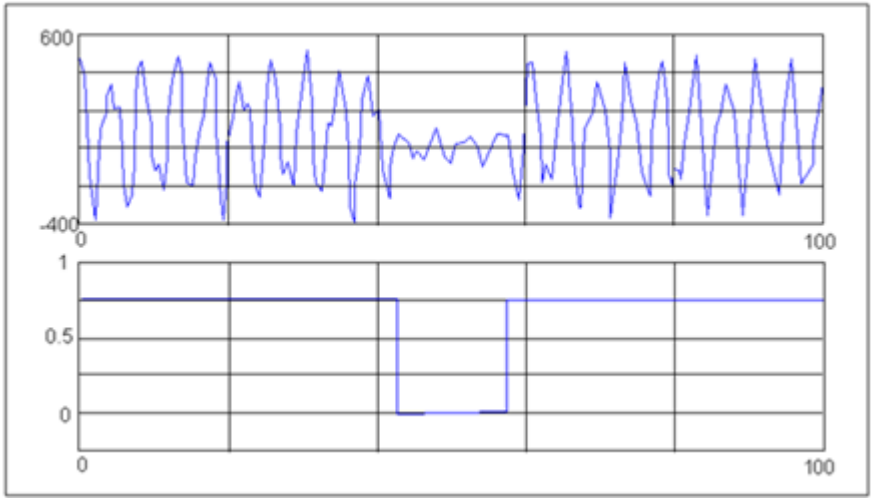


Figure 14-28 A Signal and It's Gate Signal

If the gating value is set to 0.5 then only segments 1, 3, 4 and 5 on graph A will be analyzed. Only those peaks correspond to the 0.5 limit set as the gating signal trigger value. Note that the gating signal has to be greater than the gating value for the entire time window, if it drops below the gating value just once per window then the entire window's data is ignored. If nothing is entered in this field, MRSTATS assumes there is no gating signal and therefore analyzes all of the input signal.

The example given above was produced according to gating option 3.

If a gating option is chosen then the third and final screen is displayed.

If the no-gate option is accepted, the analysis proceeds and a % complete meter appears.

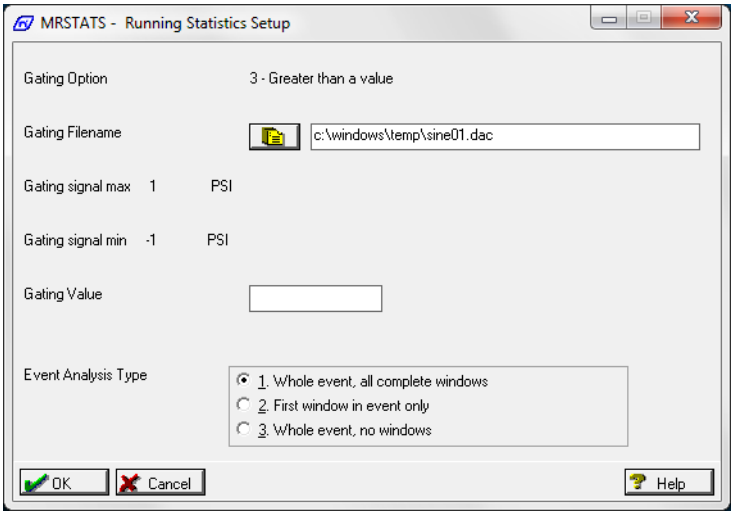


Figure 14-29 Specifying the Gating Parameters (In This Case for Option 5)

To accept the screen and start the analysis, press **OK**.

Once the screen has been accepted a '% processing complete' progress bar appears on the screen.

Batch Operation

MRSTATS can be run in non-interactive batch mode. Typical batch command lines would be:

```
mrstats /opt=all/inp=test/ov=y
```

This would calculate all output statistic signals for the input file `test.dac`.

A list of MRSTATS' batch keywords:

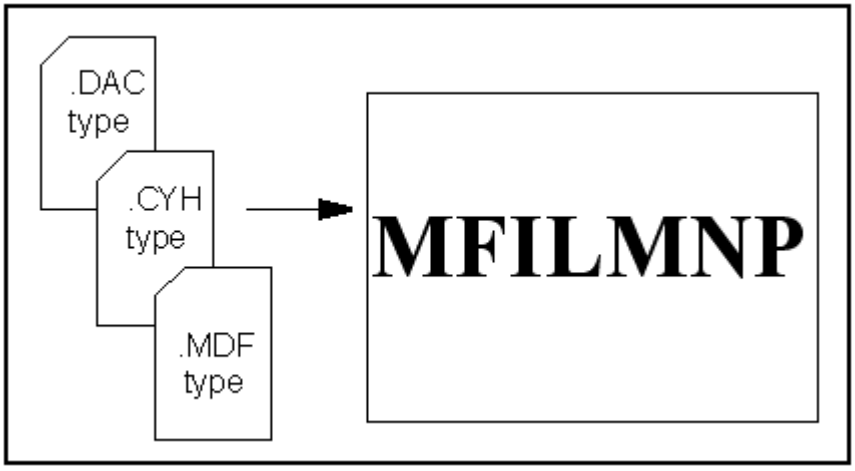
/OPT	The statistics options - 1 - 7 or ALL
/INP	The name of the input .dac file
/OUT	The name(s) of the output .dac files
/OV	Overwrite existing files
/WINTYP	The window type time, points, or percent
/WIN	The window size
/OVLTYP	The overlap type T(ime), P(oints), or %
/WINOV	The window overlap
/GSIG	The name of the gating signal.
/GOPT	The gating option from 1 to 6
/LIM1	The gating value.

/LIM2	Second gating value
/ETYP	The event analysis type

It is recommended that, by default, the /OV=Y keyword be included in every batch command line, since if it is omitted and an output file with the specified name already exists, batch operation will cease.

Header/Footer Manipulation - (MFILMNP)

File manipulation (MFILMNP) allows both header and extra details manipulation. In addition, it can check that a header conforms to MSC Fatigue conventions.



MFILMNP can list and edit data file headers, and control the extra details data that can be written to files. The files types that can be edited include single parameter files e.g. `.dac` files and 2 parameter files e.g. `.mdf`. Three parameter histogram files can also be processed.

Headers tend to contain information relating to the data within the file. Extra Details Areas (also called footers) tend to contain user supplied data, or the results of the analysis of the data.

Header information is stored in at the top of a data file ahead of the data itself. The area is of fixed size and is reserved for a specific set of file details. Footers are appended to the end of files and tend to contain user specified details. The exact contents of the footer cannot be stated without looking at each footer because the extra details keywords, which specify the contents of the footer, vary. This is because the available extra details keywords vary between MSC Fatigue modules, and in any case the user specifies which keywords are to be used.

Because of the difference between header and footer information, they are discussed separately, starting with The Header Area.

The Header Area

Although MSC Fatigue strictly maintains file header accuracy and integrity, it may sometimes become necessary to edit the contents of a header element in order to meet a specific requirement. For example, under

normal circumstances files with differing sampling rates or bases cannot be concatenated. MFILMNP provides a means for "adjusting" any header element so that concatenation, or any other operation which would not normally be allowed, can be undertaken. However, great care must be exercised when using MFILMNP in this way because the fundamental nature of the data is being changed! Because of this danger, these numeric operations can only be carried out, with a precise knowledge of the header format, and by accessing it through its 'dump form'.

In addition to editing specific numeric elements, MFILMNP also provides a means of amending areas of the header that contain textual information such as axis labels and units. This functionality can be particularly useful when data files are being transformed, by some arithmetic or analytical process, from one unit system to another; the two example batch command lines provided at the end of the batch section illustrate this usage. Unlike the numeric operations referred to above, textual manipulation does not require a knowledge of the header format and so is carried out by accessing it through an 'annotated form'.

The Dmp Form

The dump form provides a means of accessing the header in a relatively straightforward way by considering it to be made up of 128 four-byte elements, 512 bytes in all. Data can be stored in each element according to the number of bytes required for each data type; this storage requirement is detailed below:

Data Type	Example	Storage Requirement (Bytes)	Number of values per dump form element
REAL	150.65	4	1
INTEGER	10	2	2
CHARACTER	A	1	4

In this regime, a real floating point number, requiring 4 bytes, will occupy one element of the dump form. An integer, on the other hand, will only half fill a dump form element, and so, two integers can be stored. Similarly, each dump form element will store four characters.

Access to the dump form is through element number i.e. 1 through 128. It is the user's responsibility to load each element with the correct data type and number of values. For example, if dump form element 33 is to be amended, the user will need to be aware that this element contains two integer values and enter them both. Omitting a value will set the second half of the element to zero, see the section on module operation for more details.

In all there are 128 elements numbered. All the elements are fully explained in the chapter called MSC Fatigue DAC File Format Description, 274.

The Annotated Form

Unlike the dump form, the annotated form requires no prior knowledge of the format of the MSC Fatigue file header. It is designed to allow access to specific header fields by name alone. The following fields may be accessed in this way:

- X-axis unit string
- Y-axis unit string

- Z-axis unit string
- X-axis label string
- Y-axis label string
- Z-axis label string
- Original signal name
- Final signal name
- Signal base

Notice that although the signal base is a numeric field, it may nevertheless be accessed through the annotated form. See the section on module operation for more details.

The Footer (Extra Details Area)

Often the header area is not large, or flexible enough, to store non-standard system or even user-defined, information. To overcome this restriction, a system has been devised whereby information may be attached directly to the tail of an existing data file, into a region known as the **Extra Details Area** (**EDA**) or **Footer**.

The extra details area of a specific data file may be examined, modified and extended through the use of the MFILMNP module. However, many MSC Fatigue modules access this region automatically, either to deposit, or recover information, some of these modules are detailed below:

Once extra details have been “attached” to a particular data file, MSC Fatigue will automatically pass them on to any data files “spawned“from the original file and so the information can be carried forward from the beginning to the end of any analysis procedure.

Format of the extra details area.

Information is stored in the extra details area and are in the form of keyword/ value pairs, the sequence of the pairs is irrelevant. The maximum size of each field is illustrated below:

- The Keyword Field - Up to 8 characters in length.
- The Value Field - Up to 120 characters in length.

Since access is always via MFILMNP or other MSC Fatigue modules, a more detailed knowledge of the format of the EDA is not required.

Note:	Since extra details are stored entirely as character text, any detail which is supposed to be numeric will NOT be checked for validity by MFILMNP.

Module Operation

The MFILMNP module can be run in one of the following three modes:

- From the **Advanced Loading Utilities** pulldown menu under **Tools | MSC Fatigue** (for Patran) or under **Tools | Fatigue Utilities** (for).
- In stand alone mode by typing **mfilmnp** at the system prompt
- By incorporating the MFILMNP commands in a batch operation.

The first two modes are interactive. Once initiated in interactive mode, MFILMNP will display the following options:

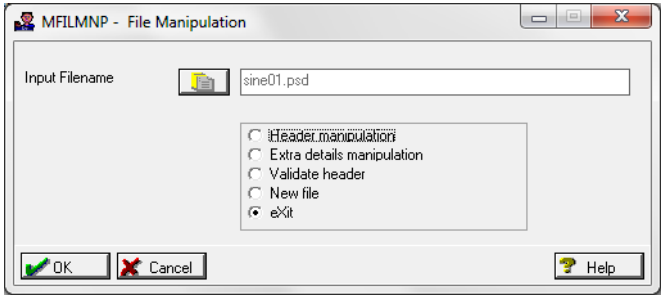


Figure 14-30 The MFILMNP Top-Level Menu

Input File Name

By default MFILMNP expects this file to be a standard MSC Fatigue single parameter file with a file extension of .dac. Therefore, if a file of this type is to be processed, the .dac extension may be omitted. Files with the correct internal format but different file extensions must have their names entered in full, i.e. including the file extension.

Header Manipulation

Option	Description
View Annotated form	The annotated form is an annotated (cut down) version of the full dump form. It only contains those header parameters most often changed, i.e. about 20 of the total of 128 parameters. Output can be sent to either the screen or a file.
View Dump form	Lower limit: The dump form of the file header consists of 128 four byte elements with each element containing either a single floating point number, two integers, or four characters. The response required here is the number of the dump form element at which the listing is to commence. Upper limit: The dump form of the file header consists of 128 four byte elements with each element containing either a single floating point number, two integers, or four characters. The response required here is the number of the dump form element at which the listing is to end. The number entered must be greater than or equal to the lower limit specified above and less than or equal to 128. Operation type: real, integer*2, character (R): The contents of each element of the dump form can be represented and displayed by means of any of the three data types; REAL, INTEGER and CHARACTER. If the user wishes to view the first 3 (of 128) dump screen elements then entering Lower limit = 1 and Upper limit = 3 will display elements 1 to 3.
Update annotated form	Move the highlight bar over the field to update in the screen that is presented.

Option	Description
Update dump form	Dump form number: The dump form of the file header consists of 128 four byte elements with each element containing either a single floating point number, two integers, or four characters. The response required here is the number of the dump form element to edit. Operation type Real, Integer*2, Character: The contents of each element of the dump form can be replaced by any of three data types; REAL, INTEGER and CHARACTER. The type offered as the default is the data type of the element specified for editing. MFILMNP will present the current contents of each dump form element and prompt for an alternative value. Note that in this case the element specified was number 91. Number 91 is a character and characters have up to 4 values to be entered (Integers have 2 values and Real have 1). Special characters such as Escape codes can be entered by surrounding number with <> characters. New Value: Type the new values here. Pressing OK will cause them to be passed to the dump form.
File integrity Check	Selection of this option will cause MFILMNP to recalculate all the statistics which are normally contained within the header area. This procedure is useful since it ensures that the calculated values are a true representation of the data.
Save changes	All edits MUST be explicitly saved before they become permanent. If changes are not saved then MFILMNP will prompt with a Save Changes to Header Yes/No message before leaving the Header manipulation options.
Return	Returns you to the main menu.

Extra Details (Footer) Manipulation

The second major option from the main menu is that which allows manipulation of the footer information.

If this option is selected then you have the following options:

Option	Description

1. Load extra details area from ASCII file	Extra details may be loaded into the specified data file from an external ASCII text file which may, for example, have been created with a text editor or word processor. This feature is very useful for loading additional, user-defined test information, particularly in batch mode; see the second example of typical batch commands for more details of this technique. By default MFILMNP assumes that the text file will have the extension `.asc` and that it resides in the user directory. Note that any extra details with matching keywords already in the file will overwrite. Other extra details are unaffected.
	
	Syntax of the ASCII File:
	
	The ASCII file must have one keyword/value pair per line with the delimiter between the keyword and value strings being the equals sign "=", for example:
	
	`TITLE=This is the test title`
	`ENGNAM=Fred`
	`TEMP=150`
	
	Keywords can be up to 8 characters in length; entry of longer strings will result in an error message. Values fields may by up to 120 characters in length. Any line without an equals sign will be disregarded and so can be used as comment lines. Spaces in the keyword are not allowed.

2. Load extra details area from another data file	It is possible to copy extra details from another file into the extra details area of the current data file. The other file must have version 3.0 (or above) extra details, and it is assumed by MFILMNP to have the default file extension `.dac`. Other file extensions must be entered explicitly. Note that any extra details with matching keywords already in the file will be overwritten. Other extra details are unaffected.
3. List extra details area	This option will simply list the entire contents of the specified data file to the screen or to a specified file. Note that a file must be Loaded from options 1 or 2.
4. Delete all of the extra details area	This option allows the user to delete the entire extra details area of a specified data file. This is a dangerous option since it may destroy important data, as a result MFILMNP will prompt for confirmation to proceed.

Option	Description
5. Add extra details pair.	The additional information stored in the EDA of standard MSC Fatigue files are stored in the form of a keyword identifying the extra detail together with an associated value. Keyword A string of up to 8 characters, which will represent the keyword by which the particular piece of information will be identified, should be entered. If the keyword already exists, an error message will result. Valid keyword strings might be, TIME, Title, or My Name. These keywords will be stored as TIME, TITLE and MY NAME, notice that the SPACE character is maintained as a valid keyword character. Value The additional information stored in the EDA of standard MSC Fatigue files are stored in the form of a keyword identifying the extra detail together with an associated value. The value to be assigned to a particular keyword should be entered, it can be up to 120 characters in length and may be any text string. The 120 characters are split over two lines because of screen restrictions. After entry of an extra details pair, any mistakes can be corrected by selection of option 6.
6. Modify extra details value.	Selection of option 6 will cause EDM to ask for the keyword and its value. The value associated with a particular keyword can be modified by means of this option. If the keyword itself needs to be modified, then it must first be deleted with option 7 and then re-inserted with option 5. Enter the keyword whose value is to be altered, option 3 can be used to list all the keyword/value pairs currently stored. The modified value to be assigned to the specified keyword should be entered, it can be up to **120 characters** in length and may be any text string.
7. Delete extra details pair.	The additional information stored in the EDA of standard MSC Fatigue files are stored in the form of a keyword identifying the extra detail together with an associated value. Entering the name of an unwanted keyword at this point will remove it and it's associated value from the extra details area. Keyword pairs can be displayed using F3/List. This facility also allows the user to tag more than 1 keyword, use a wildcard character (*) to tag families of keywords, and access keywords from other drives/directories.

Option	Description
8. Copy extra detail pair to environment file	MSC Fatigue runs within the context of both a local (from the user's directory) and global system environment. These environments are special data files that contain information to which each MSC Fatigue program module can gain access. Since the structure of the environment file and the extra details are fundamentally the same, data may be copied freely between them. As a result an environment file may then be used as a mailbox to pass information results between data files and program modules. This option allows extra details keyword/value pairs to be copied from the specified data file **TO** the **local environment file**. Enter the name of the keyword which, together with its associated value, is to be copied from the extra details area of the current data file into the local environment file. F3/List will list all available keywords.

Option	Description
9. Copy extra details from environment file	Equivalent to option 8 but instead copies **FROM** the **local environment** file to the specified data file. Enter the name of the keyword which, together with its associated value, is to be copied from the local environment file into the extra details area of the current data file.
A. Extra details calculation	This option can be used to perform arithmetic calculations on keyword values. An example is shown in Figure 14-31 where speed in mph is converted to m/sec. The top line reads the value associated with a header variable called SPEED and multiplies it by 1.8 to get km/hr. The new speed value is automatically fed into line 2 (EDMV2) and multiplied by 1000 and the resultant speed value is fed into field EDMV3 where it is divided by 3600 (the number of seconds in an hour). The results of all three stages are written to the extra details area, in this case as EDMV1, EDMV2 and EDMV3. Available input name/values can come from ■ Extra details ■ Numbers ■ Header Keywords Arithmetic operators can be ■ *(multiply) ■ /(divide) ■ +(Add) ■ -(subtract) ■ ^(raise to the power of) There should be no spaces between words and operators.
B. Compress extra details	Extra details are added to the end of a data file as a string of values. When values/words in the middle of the string are deleted, the blank space is NOT automatically closed up. The deleted areas become 'dead' space because values/words can only be added to the end of the string, not in its middle. This option closes up the spaces, or more accurately, it moves the spaces to the end of the string where they become available again.

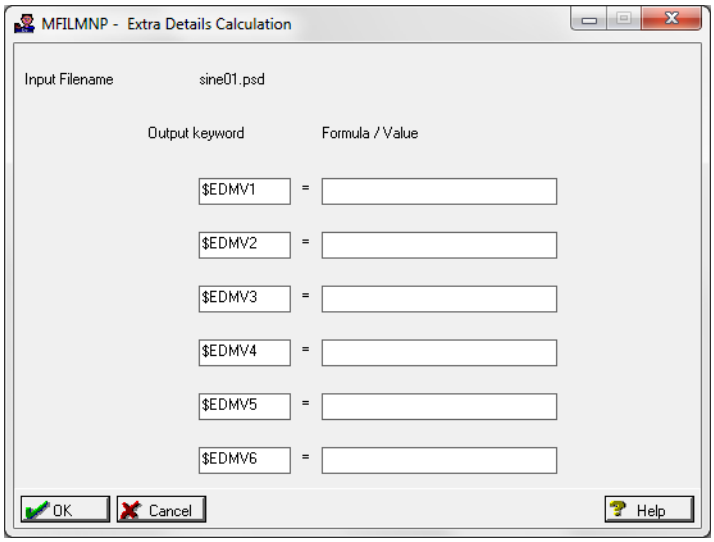


Figure 14-31 The EDA Calculation Screen

Validate Header

This option is a utility that checks the validity of the header. It:

- checks the header values are correct by recalculating the values and comparing the calculated values to those in the header.
- uses double precision for the statistic calculations.
- checks that blocks of extra details are consistent with the header.
- checks that all numbers are credible.

New File

Selection of this option simply removes the menu from the screen and makes the Input File Name field the screen focus once more.

Batch Operation

MFILMNP can be run in non-interactive batch mode. In this mode of operation simple but often tedious alterations to the file header to be made simply and accurately. Typical batch command lines would be:

```
mfilmnp
/inp=test101/opt=h/opt=u/ylabel=MPa/yunits=stress/*=tt
```

This changes the Y-axis label and the Y-axis units of the file `test101.dac`.

```
mfilmnp /inp=test101/opt=e/opt=8/key=*
```

This copies all the extra details pairs from the file `test101.dac` to the environment file. A list of MFILMNP's batch keywords:

/INP=	The name of the input data file. /INP=FILE1
Main Menu options:	
/OPT=	H-Header, E-Extra, V-Validate, N-New file
/OPT=	A submenu option number. 1-Load EDA from an ASCII file, 2-Load EDA from another data file, 3-List EDA, 4-Delete all of the EDA, 5-Add extra details pair, 6-Modify extra details value, 7-Delete extra details pair, 8-Copy EDA to local environment, 9-Copy EDA from local environment. /OPT=7
/KEYn=	The required EDA keyword. /KEY1=TIME, /KEY2=SPEED
/VALn=	The value to associate with the required keyword. /VAL1=60, /VAL2=100
/OV=	Confirmation to allow the deletion of an entire extra details area. /OV=Y
/INPut	The name of the ascii file or the data file from which to copy extra details from. /INP=INDATA
/OPTion	The required option number where: 1-View annotated form, 2-View dump form, 3-Update annotated form, 4-Update dump form, 5-File integrity check. /OPT=3
/DESTination	The destination for the header listing: S-Specifies to the user screen. F-Specifies to a file name. /DEST=F
/OUTput	Output file name, /OUT=`fred.lst`. Note that to direct the listing to a notebook, a notebook name must be defined in the local environment. In batch mode this is accomplished by issuing the following command to MFILMNP... /menm /opt=2/key=notebook/val=myname)
/LOWLIMit	Specifies the element in the header at which the listing of the dump form will begin. Values between 1 to 128 may be entered. /LOWLIM=1
/UPPLIMit	Specifies the element in the header at which the listing of the dump form will end. Values between /LOWLIM and 128 may be entered. /UPPLIM=10
/OPTYPe	Defines the type of the header element to be edited. R-Specifies a REAL (floating point) field, I-Specifies an INTEGER field, C-Specifies a character field. /OPTYP=I
/XUNIT	The text string that will define the units of the x-axis. Up to a maximum of 28 characters may be used. If the value is set to NONE, then this field will be made blank. /XUNIT= Secs.
/YUNIT	The text string that will define the units of the y-axis. Up to a maximum of 28 characters may be used. If the value is set to NONE, then this field will be made blank. /YUNIT= Secs.
/ZUNIT	The text string that will define the units of the z-axis. Up to a maximum of 28 characters may be used. If the value is set to NONE, then this field will be made blank./ ZUNIT= Secs.

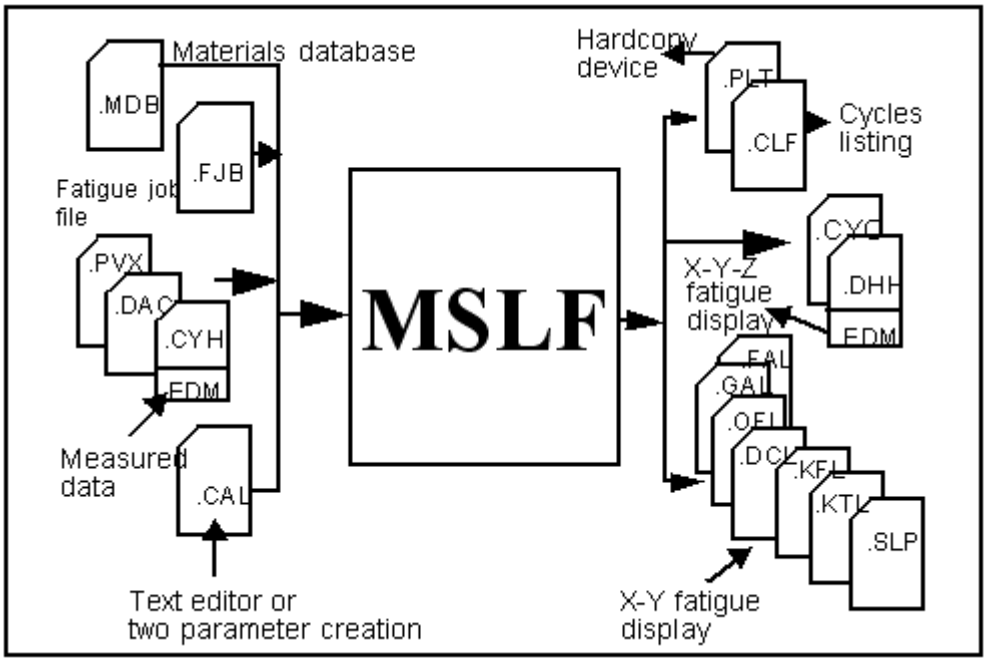
/XLABel	The text string that will define the x-axis label. Up to a maximum of 16 characters may be used. If the value is set to NONE, then this field will be made blank. /XLAB=Elapsed Time.
/YLABel	The text string that will define the y-axis label. Up to a maximum of 16 characters may be used. If the value is set to NONE, then this field will be made. /YLAB=Load.
/ZLABel	The text string that will define the z-axis label. a maximum of 16 characters may be used. If the value is set to NONE, then this field will be made. /ZLAB=Damage.
/BASE	The new base value for the data file. /BASE=10
/REAL	If a REAL (floating point) field is being edited, then this keyword defines the new REAL value required. /REAL=26.5
/DUMPform	Dump form number /DUMP=91
/NUM1	First value for dump form update /NUM1 = A
/NUM2	Second value for dump form update /BASE=10
/NUM3	Third value for dump form update /NUM3 = C
/NUM4	Fourth value for dump form update /NUM4 = D

Advanced Fatigue Utilities

Ten advanced modules are available as single location fatigue analyzers and/or for display of fatigue damage and cycles information. It is assumed that the user has a good working knowledge of MSC Fatigue's analysis capabilities before attempting to use these modules. Please familiarize yourself with Using MSC Fatigue - The Classic Modules (Ch. 3) and Total Life and Crack Initiation (Ch. 6) first, if this is not the case.

Single Location S-N Analysis - (MSLF)

The MSLF program models fatigue life to predict durability based on SN-curves derived from constant amplitude test results for specimens or components. In this way, failure can be predicted in a total life sense for any component subject to any variable amplitude or service measured time history using the Miner Rule of linear damage calculation. In MSLF, a single fatigue life answer is not the end point, only the beginning of potentially very many "what if" and sensitivity studies using multiple input parameters and back calculations with varying scale factor, mean stress offset, % certainty of survival, stress concentration factor, size effect, surface finish and surface treatment.



The reader is referred to Total Life (S-N) Analysis, 1351 for theoretical information on stress-life (S-N) fatigue analysis. This section highlights the functions of the MSLF module.

MSLF is very similar to MCLF and it is assumed that the user has a working knowledge of the MSC Fatigue module FEFAT. The difference between MSLF and MCLF is that MSLF requires stress response information and MCLF requires strain responses. The difference between MSLF and FEFAT is that MSLF begins with a stress response (in the form of a `.dac` file and could be from measurement data) and is a single location analyzer, whereas FEFAT requires load input and stress from FE models. They work together well when a response stress signal is output from FEFAT from a single location of the FE and feed into MSLF.

MSLF can be used when considering the fatigue performance of:

- Welds and other structures which may contain cracks or crack- like defects.
- Machined components
- Forgings
- Castings
- Pressings
- Non-metals; e.g. plastics, composites etc.

MSLF, like the other fatigue analyzers in the MSC Fatigue system, can be used for:

- Durability assessments in the design/test loop to avoid costly and time consuming prototype repeat testing
- Durability Optimization by "what if studies" to achieve better fatigue solutions (loading, material, surface condition, local geometry)

- Failure analysis by mimicking the failure and providing substantiated remedial solutions
- Optimization of inspection and maintenance scheduling by fatigue assessments and updates
- Computer simulation prior to structural testing to optimize test programs from a fatigue content perspective and to optimize time, budget and testing resources
- Optimization of materials selection and manufacturing process routes (casting, forging, machining, shot peening) from a fatigue durability viewpoint.

K_t values can be determined within Time Correlated Damage - (MTCD), 1121 and passed into MSLF as default selections. MSLF also supports the calculation of K_f values from K_t. A full range of postprocessing and results display options are available in MSLF. Other features of MSLF are similar to the other fatigue analyzers in the MSC Fatigue system and only those items unique to MSLF are expounded on here.

Module Operation

The MSLF module can be run in one of the following three modes:

- From the **Advanced Fatigue Utilities** pulldown menu under **Tools | MSC Fatigue** (for Patran) or under **Tools | Fatigue Utilities** (for).
- In stand alone mode by typing **mslf** at the system prompt.
- By incorporating the MSLF commands in a batch operation.

Once running in interactive mode, the MSLF module will display the following screen:

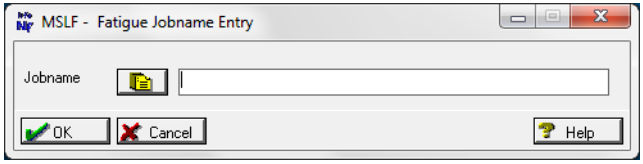


Figure 14-32 The First MSLF Screen

The user must enter the name of an Input Job File, which have `.fjb` extensions. If the name of an existing Job File is entered then MSLF will go to the postprocessing. If a new job is specified then its parameters have to be entered in a series of screens. If no name is entered then the results of processing may NOT be saved.

The flow chart below illustrates the major routes through MSLF.

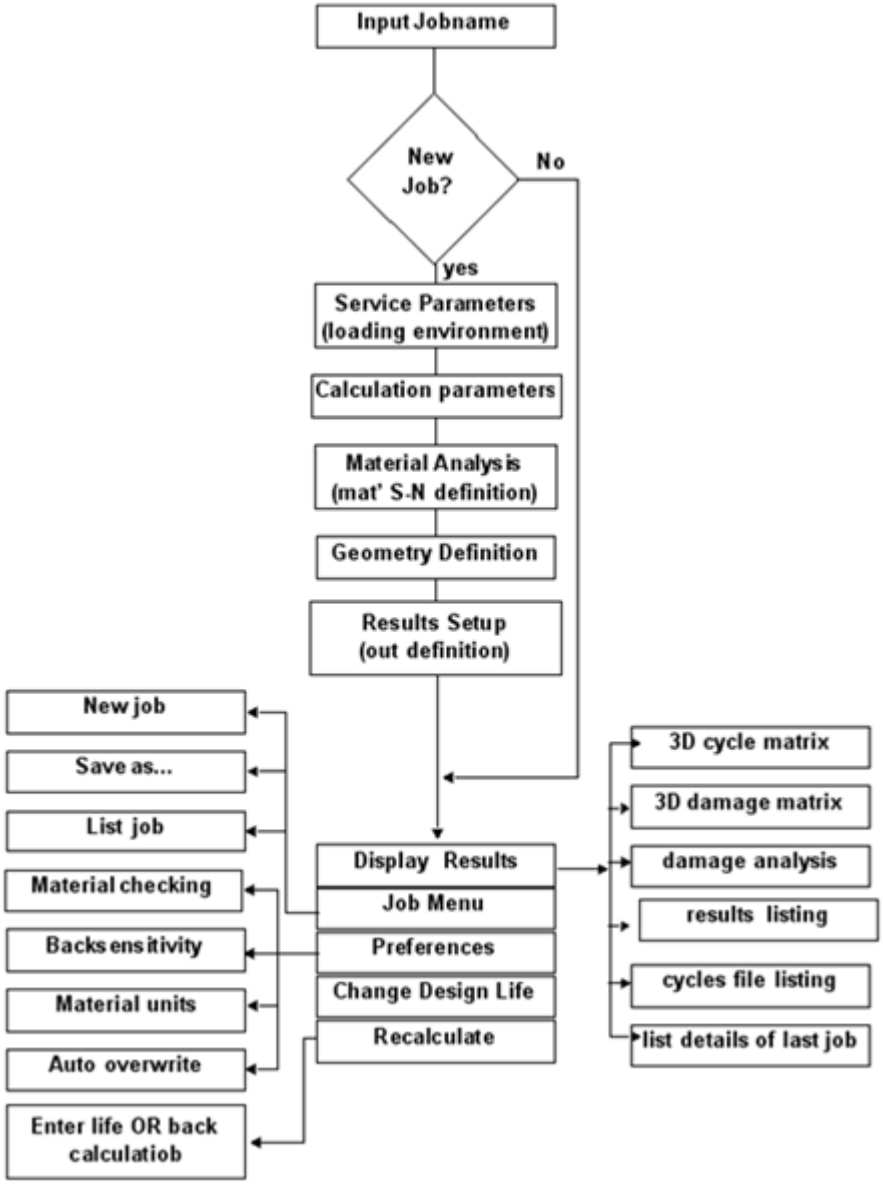


Figure 14-33 The General Module Structure (MSLF)

The Loading Environment

This input for MSLF is identical to that of MCLF with the exception of a few items that MSLF does not need. See Service Loading Environment, 1096.

Three types of loading file can be specified:

- **Time History** which normally means a single parameter .dac time history or .pvx peak valley file. This can be a measured time history or a time history extracted from FEFAT.
- **Range Mean Matrix** which normally means a rainflow matrix file with a .cyh extension.
- **Constant Amplitude** which requires the user to enter individual values of Amplitude and Mean Value.

For all file types the data should preferably be in units of Microstrain (uE). If they are NOT then a calibration file that contains the appropriate conversion factors should be supplied.

Model Parameters

All input on this page is identical to that required for a global FE fatigue analysis. See Model Parameters, 1101.

Nominal S-N curves can be either structure, component, or specimen based. One of the three analysis methods must be chosen. Briefly they are:

- *Material S-N* which are S-N curves generated from the elastic part of the strain controlled fatigue tests or S-N curves for which $K_t = 1.0$
- *Component S-N* which are generated directly from stress based fatigue tests on components and the associated $K_t > 1.0$.
- *BS7608* which are S-N curves designed to be used as part of BS7608 part 10 analysis of welded structures.

It is important to maintain consistency between the analysis methods selected here and the S-N datasets chosen in the next screen.

Material Data Input

All input on this page is identical to that required for a global FE fatigue analysis. See Material Data Input, 1101.

Note that using Material Management (Ch. 4), can approximate S-N damage curves and can be generated purely on the basis of ultimate tensile strength (UTS). The curves are constructed by fixing the stress axis intercept, 1 cycle, at the value of the UTS, fixing the stresses at 1000 cycles and the endurance limit according to the fractions of UTS detailed below:

Alloy Type	Cycles	Stress
Ferrous Alloys	1	1.000 x UTS
	1,000	0.900 x UTS
	1,000,000	0.357 x UTS
Titanium Alloys	1	1.000 x UTS
	1,000	0.800 x UTS
	1,000,000	0.307 x UTS
Aluminum Alloys	1	1.000 x UTS
	1,000	0.700 x UTS
	500,000,000	0.258 x UTS

Alloy Type	Cycles	Stress
Other Alloys	1	1.000 x UTS
	1,000	0.800 x UTS
	100,000,000	0.274 x UTS

Geometry Screen

All input on this page is identical to that required for a MCLF. See Geometry Screen, 1102.

A brief discussion of Notch Correction Factors K_f from K_t:

If K_f is not known then it can be calculated from a user supplied K_t value. MSLF that takes material properties, notch radius, and notch depth into account, and uses hard coded lookup tables, to calculate K_f.

The user-defined K_f value is used to modify the S-N curve at the transition life'NC1' by dividing the stress by K_f. The S-N curve is also modified at N=1E3 by dividing by K_f', which is determined from a hard coded Figure 14-34 below, which is taken from R C Juvenall's paper, 'Engineering Considerations of Stress, Strain, and Strength' McGraw Hill, 1967.

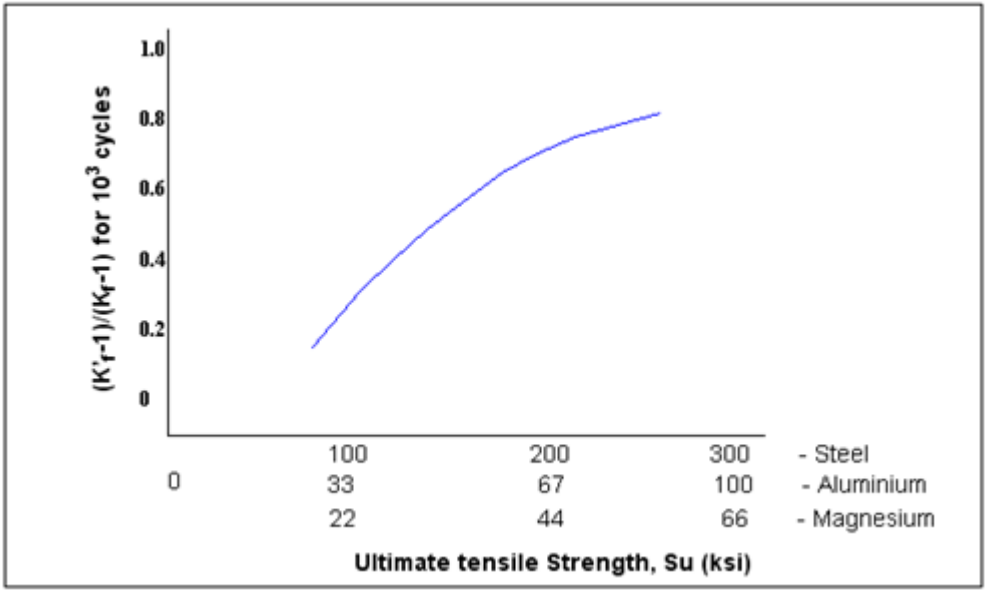


Figure 14-34 Relationship Between K'_f and K_f as a Function of Ultimate Strength

The above figure illustrates the relationship between $(K_f')/(K_f\text{-}1)$ and the UTS of the material at N=1E3. If the user defines a Kt value and the software calculates K_f, the software makes exactly the same adjustment at the transition life. At 1E3 all corrections made to the S-N curve are based on the fatigue notch factor K_f. The second slope is not changed at all - only the intercept is changed to make the transition life the same.

Results Setup Screen

This setup is also virtually identical to that of MCLF. See Output Setup Screen, 1103.

MSLF can generate a number of different output files depending on the analysis path chosen. If a multiple analysis has been specified in any of the input screens, then a single file, with the results name provided, with any one of the following file extensions, depending on the multiple parameter, will be created.

.fal	Scale factor vs. Life
.gal	Hysteresis gate vs. Life
.ofl	Offset vs. Life
.kfl	K_f vs. Life
.ktl	K_t vs. Life
.dcl	Survival probability vs. Life

All these files have the standard MSC Fatigue X-Y paired data format and can be displayed graphically within MSLF itself or by the X-Y parameter display module, MTPD. No other output files will be created.

If, on the other hand, a single shot analysis has been carried out then MSLF can optionally create a group of files which contain different information, have the same results name and the following file extensions.

.cyo	Rainflow matrix suitable for input to MCDA or MP3D
.dhh	Damage matrix suitable for input to MCDA or MP3D
.slf	List of fatigue cycles and associated damage suitable for input to MCDA.

All these files may be displayed graphically from with MSLF.

Like MCLF these files, numerical results are also written to the extra details area of the damage matrix file, `.dhh` extension. See Environment Keywords, 1107.

The Postprocessing Menu

See the The Postprocessing Menu, 1105 postprocessing menu. Operations are identical.

Batch Operation

It is recommended that, by default, the /OV=Y keyword be included in every batch command line, since if it is omitted and an output file with the specified name already exists, batch operation will cease.

When batch processing with a series of different input files, it is necessary to use a new batch line and option definition for each new input. The new line must specify the option from the postprocessing menu into which the new input will go. For example, when using the jobfile `fatjob.fjb`:

```
mslf /job=fatjob.fjb/inp=datafil1.dac/opt=l
mslf /job=fatjob.fjb/inp=datafil2.dac/opt=l/opt=g/kf=2.0
mslf /job=fatjob.fjb/inp=datafil3.dac/opt=l/kf=2.5/opt=g
```

Which will run MSLF three times with a different strain history file each time. Omitting the /OPT keywords will cause the batch file to fail because inputs without the /OPT=L keyword the `.dac` files cannot be loaded

and will therefore be ignored (the values in `fatjob.fjb` will be used instead). Each run of MSLF with a new input for the fatigue job needs a new batch line.

Note that it is permissible to input more than one parameter on a batch line, but they must be different. Note also that the order in which the batch keywords appear is not critical. Also all notes associated with MCLF batch operation also apply to MSLF operation. Batch keyword for MSLF:

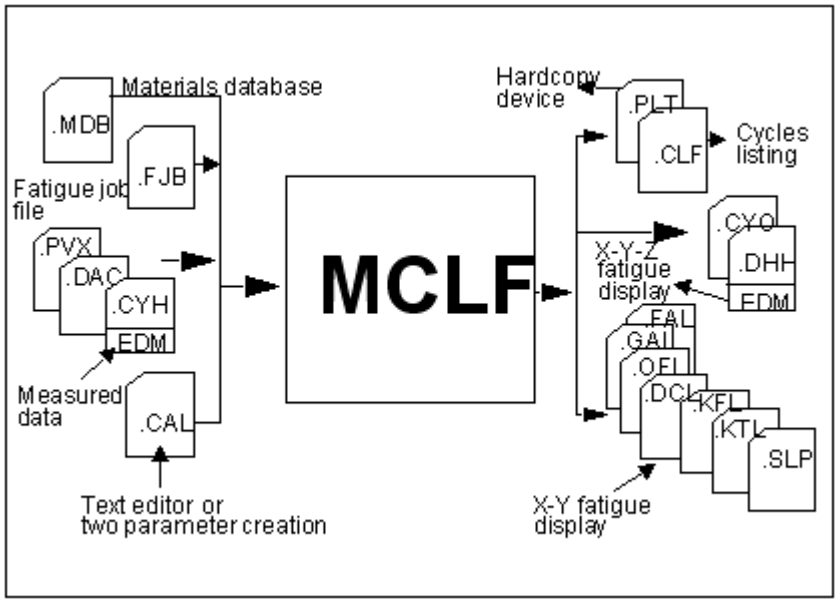
/OPTion	Back page option, e.g. /OPT=M, to input a new Model parameter. Also L,S,G,O,D,J,P,C,R,X
/INPut	The name of the input data file. /INP=INDATA Needs /OPT=L if a new file is to be loaded into a job already created.
/FACT	The required scaling factor(s). /FACT=2.5
/DCrit	The %certainty of survival. /DC=50
/ANAL	The required analytical procedure, M, C, or B.
/MSC	Mean stress correction method, N=None, G=Goodman, B=gerBer, All
/ANAL	The analysis method: Material s-n, Component s-n, or Bs5400. /ANAL=B
/UTS	The Ultimate Tensile Strength of the material. /UTS=675
/GATE	The required hysteresis gate. /GATE=500
/LIFe	/LIF=10000 If BACK has been specified for this keyword then /FA= specifies the required life.
/SNC	The name of the material data set to use. /SNC=SNDATA
/EDIT	Specifier to edit a material parameter prior to analysis. /EDIT=SRI=10075. This keyword can be used any number of times.
/OUT	The name of the output results file. //OUT=RESULT
/OVer	Whether to overwrite an existing results file. /OV=Y
/PLT	Request a hardcopy plot of factors or design criterion vs. life plot. /PLT
/PLTNAM	The name to assign to the required plot file. /PLTNAM=MYPLOT
/TYPE	Loading type is; T=Time, R=Range-mean, or C=Constant temp. /TYPE=R
/CALFIL	Calibration file; N=No, ASCII, or B=Binary. /CALFIL=B
/UNIT	MPa, KSI, uE. /UNIT=MPA
/AMP	Amplitude(s). /AMP=
/MEAN	Mean(s). /MEAN=
/STA	Start time. /STA=10
/END	End time. /END=100
/CALNAM	Calibration file name. /CALNAM=test
/LUNI	Loading units from cal file. /LUNI=MPA
/OFF	Offset. /OFF

/NUMEQU	Number of equivalent units. /NUMEQU=34
/EQUNIT	Equivalent units string. /EQUNIT=
/DESign	The design criteria. /DES=
/MINERs	Miner's value. /MINER=1.5
/JOB	Jobname. /JOB=Newjob
/CREate	Create new job Y/N. /CRE=Y
/MATENTry	Material entry method. /MATENT=
/SNDATa	S-N data set name. /SNDAT=Fred
/SURFace	Surface finish, PO,GR,GO,POO,AM,HO,F,C,WC,SC,UD. /SURF=GR
/TREATments	Surface treatments, NONE,NITRIDED, COLD ROLLED, HOT PEENED, AL. /TREAT=Nitrided
/CLASS	Weld class. /CLASS
/WELDED	Welded yes/no. /WELDED=Y
/THICKness	The thickness of the weld. /THICK=20
/WCORRosion	Correct for corrosion Yes or No. /WCORR=Y
/YModulous	Young,s modulus. /YM=3E3
/N1	The first S-N entry point. /N1=
/S1	The stress amplitude at S1. /S1=
/N2	2nd S-N entry point. /N2=
/S2	Stress amplitude at S2. /S2=
/SLOPE	Slope after N2. /SLOPE=
/SERR	Standard error at log(x). /SERR=
/RATIO	R-ratio at test. /RATIO=
/GENTYP	Type of material generated. /GENTYP=
/SRI1	Intercept on Y-axis of S-N plot for edit option (see fig 7). /SRI1=
/B1	First slope for, edit option. /B1=
/NC1	Transition life, edit option. /NC1=
/B2	Second slope, edit option. /B2=
/KT	Kt value(s). /KT=
/ADDKF	Additional Kf values. /ADDKF=
/HTYPE	Histogram type, None, Input units. /HTYPE=
/CYC	Cycles file, Y/N. /CYC=
/SIZE	Histogram size. /SIZE=
/LIMits	Limits are User or Auto. /LIM=

/DAMUNI	Damage units A, P, N. /DAMUNI=
/XYFIL	Create and X-Y file of multiple results Y=yes or N=no. /XYFIL=
/DAMage	Damage histogram file Y/N. /DAM=
/RMIN	Histogram file limit. /RMIN=
/RMAX	Histogram file limit. /RMAX=
/MMIN	Histogram file limit. /MMIN=
/MMAX	Histogram file limit. /MMAX=
/PRFOPT	Preference option, M,B,U. /PRFOPT=
/UNIOPT	Material units setting, MP,P,K,N,MN. /UNIOPT=
/TOLERance	Back life tolerance. /TOLER=
/MATCHK	Material checking. /MATCHK=
/NEWJOB	Job name to save as. /NEWJOB=
/JOBOPT	Start another job or 'save as' (see NEWJOB)
/AUTOVer	Auto overwrite preference Yes/No. /AUTOV=

Single Location ε-N Analysis - (MCLF)

Local (single location) stress-strain fatigue analysis is used to estimate the number of cycles required to initiate an engineering crack. Similitude between the material at a critical location, such as a notch, and the material in a smooth specimen, subjected to similar loading conditions, is assumed. MCLF uses this methodology to predict the presence of engineering cracks of about 2 mm length.



Each calculation session is associated with a fatigue job file which contains details of all the parameters required for an analysis; in this way a set of default fatigue environments can be defined. For complex jobs, the fatigue job file is particularly useful since it obviates the need to re-enter parameters from one session to the next unnecessarily. Multiple and "back" calculations are performed quickly and easily and provide convenient X-Y type plots of life variation.

Analysis proceeds by tracking local stress and strain, identifying fatigue cycles by means of rainflow cycle counting, correcting for the effects of geometry, surface finish and treatment and estimating partial damage. Finally, total damage is calculated by linear damage summation.

Module Operation

The MCLF module can be run in one of the following three modes:

- From the **Advanced Fatigue Utilities** pulldown menu under **Tools | MSC Fatigue** (for Patran) or under **Tools | Fatigue Utilities** (for).
- In stand alone mode by typing **mclf** at the system prompt
- By incorporating the MCLF commands in a batch operation

The first two modes are interactive. Once running in interactive mode the MCLF module will display the following screen.

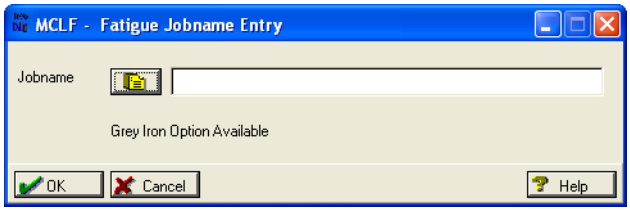


Figure 14-35 The First MCLF Screen

The user must enter the name of an Input Job File, the extension `.fbj` will be appended automatically. If the name of an existing Job File is entered then MCLF will go to the postprocessing menu screen straight away. If a new job is specified then its parameters have to be entered in a series of windows. A name is not required to continue but the results of processing may not be saved.

The flow chart below illustrates the major routes through MCLF.

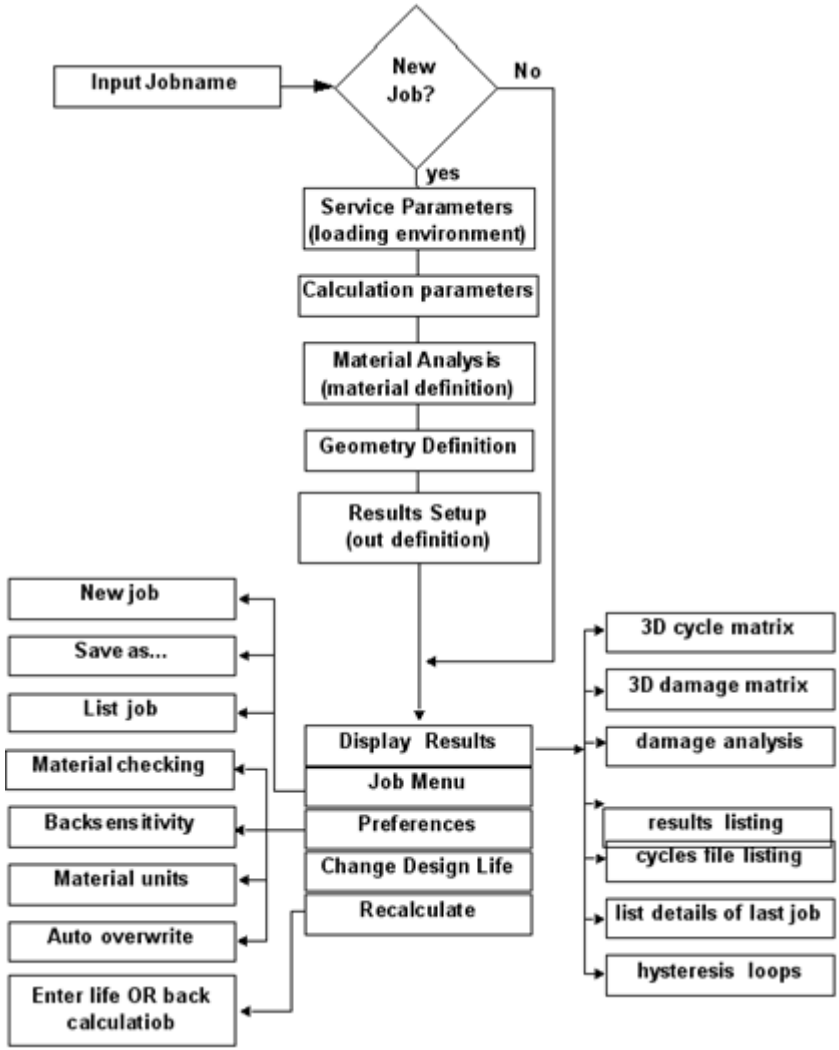


Figure 14-36 The General Module Structure (MCLF)

The MCLF module works in much the same manner as FEFAT. It is assumed that the user has a good understanding of the FEFAT module.

MCLF is essentially an extension of FEFAT to allow more precise fatigue calculations from a strain time history, either measured or calculated from FEA. MCLF is used in conjunction with FEFAT by extracting a strain time history for a particular location to feed into MCLF. Because much of the operation of MCLF is identical to FEFAT, only those features unique to MCLF are explained here. See FE Fatigue Analysis Options (FEFAT), 317 for details of the other options.

The following input is necessary to define a fatigue job:

Service Loading Environment

Option	Description
Loading	One of the main inputs for a fatigue life estimation is the file that describes the loading environment to which the component or assembly is to be subject. Three types of loading file can be specified: **Time History** which normally means a single parameter `.dac` time history or `.pvx` peak valley file. This can be a measured strain time history or a strain time history extracted from FEFAT. **Range Mean Matrix** which normally means a rainflow matrix file with a `.cyh` extension. **Constant Amplitude** which requires the user to enter individual values of Amplitude and Mean Value. For all file types the data should preferably be in units of Microstrain (uE). If they are NOT then a calibration file that contains the appropriate conversion factors should be supplied.

Option	Description

Calibration File	Any other system of units such as those associated with load or force etc., can also be used provided that the appropriate conversion function is supplied by means of a calibration file.
	
	Calibration files have a ASCII format. A calibration file must have the following structure: Input on X and output on Y, where output must be in the default internal units,i.e. microstrain, and input can be any user unit. The data pairs must be entered X (input) followed by Y (output), with one pair per line. A space is a sufficient delimiter between X and Y values.
	
	The calibration should increase monotonically from the smallest values to the largest. If necessary MCLF will linearly extrapolate beyond the ends of the calibration. Internally, MCLF interpolates between specified values.
	
	The units of both the input and output should be specified. In the case of an ASCII file, these units are defined through the use two comment lines (lines preceded by the # character) at the beginning of the file, e.g.
	
	`#input=Newtons`
	`#output=MICROSTRAIN`
	`-1000 -100`
	`0 10`
	`1000 150`
	In this case input in Newtons is related to output units of microstrain. As a guide note that:
	
	Microstrain = absolute strain x 1E6
	Microstrain = per cent strain x 1E4
	Microstrain = per mil strain x 1E3

Option	Description
Strain State	**Axial Loading**: Typically, at the surface of a component, the stress normal to the surface is zero and conditions of plane stress are said to prevail. Under these circumstances (and if the maximum principal strain remains relatively stationary throughout the loading sequence), it is appropriate to use uniaxial properties directly. **Plane Strain Loading**: If the component thickness is large relative to the notch root, and transverse contraction is effectively prevented, then plane strain conditions prevail. Typically, such conditions may be found on the surface of a thick-walled cylinder, or very thick sections. The triaxial stress state, found in plane strain, effectively alters the uniaxial cyclic stress-strain curve in the first principal direction. This modified curve is obtained by noting that the transverse strain and the stress normal to the notch surface are both zero and applying the Hencky plastic flow laws. If no entries are found in the material data base, then in the correction procedure an elastic Poisson's ratio of 0.3 and a plastic ratio of 0.5 are assumed. **Shear Strain**: Conditions of pure shear arise when the biaxiality ratio approaches -1. A typical situation is of a shaft loaded in torsion. If the shear strain state is selected, MCLF will expect the input loading to be in terms of a shear strain history (usually the signed absolute maximum). Such histories are obtained through the use of strain gauge rosettes together with the appropriate outputs from the stress-strain analysis module, MSSA. The uniaxial material parameters will be automatically adjusted to account for this strain state.
Strain type	The strain input to MCLF can be in one of two forms. The first is measured strain, which was the default for this program in previous versions. This strain is assumed to lie on the cyclic stress-strain curve, whereby if yield has occurred then the strain will change in a non-linear way as load increases. The strains will be in this form if a strain gauge has been used. Alternatively, the strain may have been calculated theoretically, from classic elastic theory or from finite element analysis. In this case the strain is usually linear elastic, and MCLF must use a different method to calculate local stress and strain.
File Name	The name of the loading file should be entered in this field. Note that if Loading = Time History then MCLF expects a single parameter `.dac` or `.pvx` file to be entered. If Loading = Range Mean Matrix then a `.cyh` file is expected. File names with other extensions can also be typed in this field.

Option	Description
Amplitude and Mean	Multiple values of amplitude OR mean can be entered. For example entering (1000,5000,500) will calculate values from 1000 to 5000 inclusive at intervals of 500. Entering 1000,2000,3000 (without the brackets) will calculate values at 1000, 2000, and 3000. MCLF will run the calculation for them all (although as only one multiple analysis criteria is allowed per run no other criteria can be multiple). "BACK" can also be entered, in which case the required life must be entered and MCLF will calculate the amplitude and mean required to achieve that life. The Show button on the right of the header bar can be used to display the values that will be processed.
Time Window	Not all of the Loading Time History file has to be used in the calculation. If only a part of the file needs to be used then the start point and end point can be specified in the Time Window. The default is to use all of the time history file, which is why the defaults are START and END. If a part only is to be used then clear the fields and enter the values, i.e., 10 in the Start field and 1000 in the End. Alternatively the syntax START+n and END-n' where n is a number of units (e.g. seconds) from the start of the file and n' is a number of units before the end of the file. START and END are usually recorded in the header area of the input file. Please note that start is not necessarily zero.
Calibration file and units	The use of a calibration file to convert input loading units to microstrain has been specified. In this field the name of the file which contains the calibration, in the appropriate format, must be provided. As a default, calibration files are assumed to have the file name extension `.cal` and so, the required file may be selected by entering its name directly without this extension. In the units field, enter the units of the input, X, half of the calibration file. This can be any text string and is used only for reporting the units back to the user. For **Scale Factor, Offset, and Cycles Gate**: if multiple entry is employed then a Damage Matrix, Rainflow Matrix, or Cycles File will NOT be produced, although MTPD will plot the results and will allow them to be stored as a hardcopy plot file.
Scale Factor	The scale factor is an amount by which the loading history or matrix will be multiplied. Multiple scale factors can be entered separated by commas or as, for example entering (-500,1000,50) in the field will scale from -500 to +1000 inclusive in steps of 50. BACK may also be entered in which case MCLF will calculate the scale factor needed to achieve the required design life.

Option	Description
Offset	The offset is a value added to the results file(s) **after the scale factor has been applied**. For example, if the original loading file had an amplitude of 10 and a scale factor of 30 then the result (300) would then have the Offset applied. If the offset was -5 then it would be 300-5, or 295.
Cycles Gate	The gate is in effect, a filter that speeds processing by filtering any small cycles that are unwanted (for example signal noise). The gate value should not be set too high or it will filter out relevant parts of the signal. Any signal smaller than the gate value specified in this field will be ignored in the life calculation. Multiple gate values can be entered using the same syntax as the example given for scale factors (above). No results file will be saved if a multiple analysis is carried out. A multiple gate analysis can be used to assess the gate value so that the user can choose the gate threshold that just fails to remove genuine damaging events. The gate threshold value is sensitive to changes in material and K_f. Also if a signal is scaled up by 2, the gate is NOT automatically scaled up by 2, the user must manually scale up gate.
This Input File is Equivalent to...	The fatigue analysis to be carried out is through the use of a either a time series, range-mean matrix or a sequence of peak valleys. The results will normally be presented in the form of the number of repeats of these series to failure. An equivalent unit, such as miles, laps, hours, etc. can be attached to the input loading file so that results can be presented in the form of miles, laps or hours to failure.

Model Parameters

All input on this page is identical to that required for a global FE fatigue analysis. See Solution Parameters, 51.

Material Data Input

All input on this page is identical to that required for a global FE fatigue analysis. See Materials Information Form, 61.

Geometry Screen

Option	Description
Method	The elastic stress concentration factor, K_t, is the ratio of the maximum stress at a stress raiser to the nominal stress computed by the ordinary strength- of-material formulae, using the dimensions of the net section. It can be used to account for the presence of a notch within a component or structure. The magnitude of the K_t required depends on the nature of the notch and its geometry. Values of stress concentration factors can be obtained from standard works such as: R.E Peterson's 'Stress Concentration Factors', John Wiley & Sons, Inc. 1974. Alternatively use can be made of the Time Correlated Damage - (MTCD), 1121 module. It is well known that small notches have less effect in fatigue than is indicated by K_t. This has led to the idea of a fatigue concentration factor, K_f, which is normally less than K_t, being introduced and being used to replace K_t within Neuber's rule. K_f is related to K_t according to: $K_f = 1 + (K_t - 1) / \{1 + \sqrt{(p' / r)}\}$ p' is a material constant dependent on grain size and strength and r is the notch root radius. If K_f is not known, then estimate the theoretical stress concentration factor, K_t, and select the calculate option, otherwise select the direct entry option.
Kf vs. Kt	If K_f or K_t are being entered directly other fields, such as notch root radius, do not appear. Single or multiple values of either K_f or K_t may be entered. Alternatively, the word back, or BACK, may be entered in which case MCLF will request the entry of a desired life and then will calculate the K_f required to achieve that life. Note that only a single arithmetic operation between delimeters (comma or space) can be undertaken, so 3+3/2 would cause an error.

Option	Description
Notch Root Radius (r) and Notch index (q)	Enter a value for the notch root radius in mm. A value will enable MCLF to calculate q and display it in the Notch Index q field. The notch sensitivity index, q, has been found to be a function of both material and notch radius. Neuber has defined q as: $q = 1 / \{1 + \sqrt{(p'/r)}\}$ where: p' is a material constant dependent on grain size and strength and r is the notch root radius. The parameter p' has units of distance, mm, and for a medium strength steel with a UTS of 750 MPa has a value of about 0.1 mm. The notch index q can only be calculated for steels that have a UTS of 50 - 250 ksi (about 340 - 1700 MPa).
Additional K_f	The fatigue strength of a component can be reduced further as a result of metallurgical defects such as inclusions or porosity. This field allows such additional K_f effects to be included in the life calculation. The number entered here will be multiplied by the value of K_f above, and the resultant combined value used in Neuber's rule. When the multiplication has been done the calculated K_f is displayed in the Calculated K_f field. When setting up a new Job the next data to be defined is the Output Definition screen (also called the Results Setup screen) which is explained below.

Output Setup Screen

MCLF can generate a number of different output files depending on the analysis path chosen. If a multiple analysis has been specified in any of the input screens, then a single file, with the results name provided with any one of the following file extensions, depending on the multiple parameter, will be created.

.fal	Scale factor vs. Life
.gal	Hysteresis gate vs. Life
.ofl	Offset vs. Life
.kfl	K_f vs. Life
.ktl	K_t vs. Life
.dcl	Survival probability vs. Life

All these files have the standard X-Y paired data format and can be displayed graphically within MCLF itself or by the X-Y parameter display module, MTPD, which is automatically invoked from MCLF.

If, on the other hand, a single shot analysis has been carried out then MCLF can optionally create a group of files which contain different information, have the same results name and the following file extensions.

.cyo	Rainflow matrix suitable for input to MCDA or MP3D
.dhh	Damage matrix suitable for input to MP3D
.clf	List of fatigue cycles and associated damage suitable for input to MCDA.
.slp	Hysteresis loops suitable for input to MTPD.

All the above files may be displayed graphically from within MCLF.

In addition, numerical results are written to the extra details area of the damage matrix file, `.dhh` extension. The following table details the keywords and values stored.

Keyword	Value	Description
ANALYSIS	CLF	Program responsible for analysis
MSC	S-W-T	Mean stress correction method
DCRIT	50.0	Probability of survival used
FACTOR	1	Scale factor
GATE	0.0	Hysteresis gate used
OFFSET	0.0	Offset used
KF	2	Fatigue strength reduction factor used
MINER	1	Miner's constant used
HISTORY	`test101.dac`	Load history used
MATERIAL	MANTEN	Material used
EQUUNITS	Repeats	Equivalent units
NUMEQUNI	1	Number of equivalent units
UTS	552	Ultimate tensile strength of material use
$CYCHEXT	`test101.cyo`	Rainflow matrix id, for system use only
MEANLF	3331	Mean life calculated
MINLF	3780	Life associated with minimum fatigue damage
MAXLF	2899	Life associated with maximum fatigue damage

The type of output requests are:

Option	Description
Histogram type	If none is selected here, then no histogram damage file is generated. If Input Units is selected then a damage histogram file will be generated with the same units as the input file. Selecting Nominal uE or Local uE will produce histograms scaled in strain units, either nominal elastic or local elastic-plastic ones respectively. A histogram with the axes scaled in terms of strain amplitude and maximum stress is produced by selecting the SWT option. The number of bins can also be specified.
Damage Units	The units of the damage displayed in each cell of the damage matrix can be specified. The damage can be output as actual damage values, as percent damage or damage normalized to 1.0. Select the required units.
Cycles file	In the case of a single shot fatigue life estimation, an output results file can be generated. This file contains details every cycle extracted by the rainflow algorithm, in terms of its nominal stresses and strains together with the associated fatigue damage. The cycles file may be viewed and searched using the list cycles option on the display results menu or postprocessed using the MCYL module.
Histogram Limits	The limits of the rainflow and damage matrices can be automatically determined to match the limits of the data, or they can be 'User' specified. If they are user-specified then the range and mean minimum and maximum fields (described below) become active.
Range Minimum/ Maximum	The range limits can be manually specified in these two fields in the units specified.
Mean Minimum/ Maximum	The mean limits can be manually specified in these two fields in the units specified.
Hysteresis Loops and Number of Loops	In the case of a single shot analysis, an output file which contains the 'n' largest hysteresis loops, i.e., stress-strain cycles, extracted by the rainflow algorithm can be generated. The contents of this file may be displayed graphically by selecting the plot loops option on the display results menu or by invoking the X-Y plotting module, MTPD. The file created will have the same generic name as given to the cycles and damage matrices but its extension will be `.slp`. Select Yes or No for this file. The number of loops to be saved in the output file can be specified here. Note that the default is the 5 largest loops. Any positive number can be entered. Use the display loops option to view the loops.

The Postprocessing Menu

The postprocessing menu of MCLF is virtually identical to the Design Optimization, 330 menu of the FEFAT module. The two items that differ are the results display and listing.

This displays the results of the most recently calculated life estimation. Exactly what is available for display depends upon the calculation; if an X-Y file was not created then it cannot be displayed, a multiple calculation will not produce any histogram files, etc.

Option	Description
Cycle matrix 3D	Display a rainflow cycles matrix file as a 3D plot. Such a file may have been calculated or entered into the analyzer. MCLF actually loads the plot 3D module MP3D for this type of display. If a damage matrix file (`.dhh`) was also created during the current analysis it can also be loaded from within MP3D.
Damage matrix 3D	Display a damage matrix file as a 3D plot. MCLF actually loads the Plot 3D module MP3D for this type of display.
Damage analysis	Display both the cycles and damage matrices on a single 2-D plot. MCLF actually loads the Cycles and Damage Postprocessor module Single Location S-N Analysis - (MSLF), 1083.
Hysteresis loops	If the hysteresis loops were saved during a single shot analysis (as a `.slp` file), they may be displayed using the module Two Parameter Display program, MTPD. See (Cycle/Damage Histogram Display (MP3D), 357.

The Results Listing option will list the full results of the current life estimation in tabular form. Such listings are carried out within MCLF. If parameters have been changed but not Recalculated, then the non-recalculated data will NOT be listed. Selecting this will load the Cycles File Listing program Cycle and Damage Analysis - (MCDA), 1111 module.

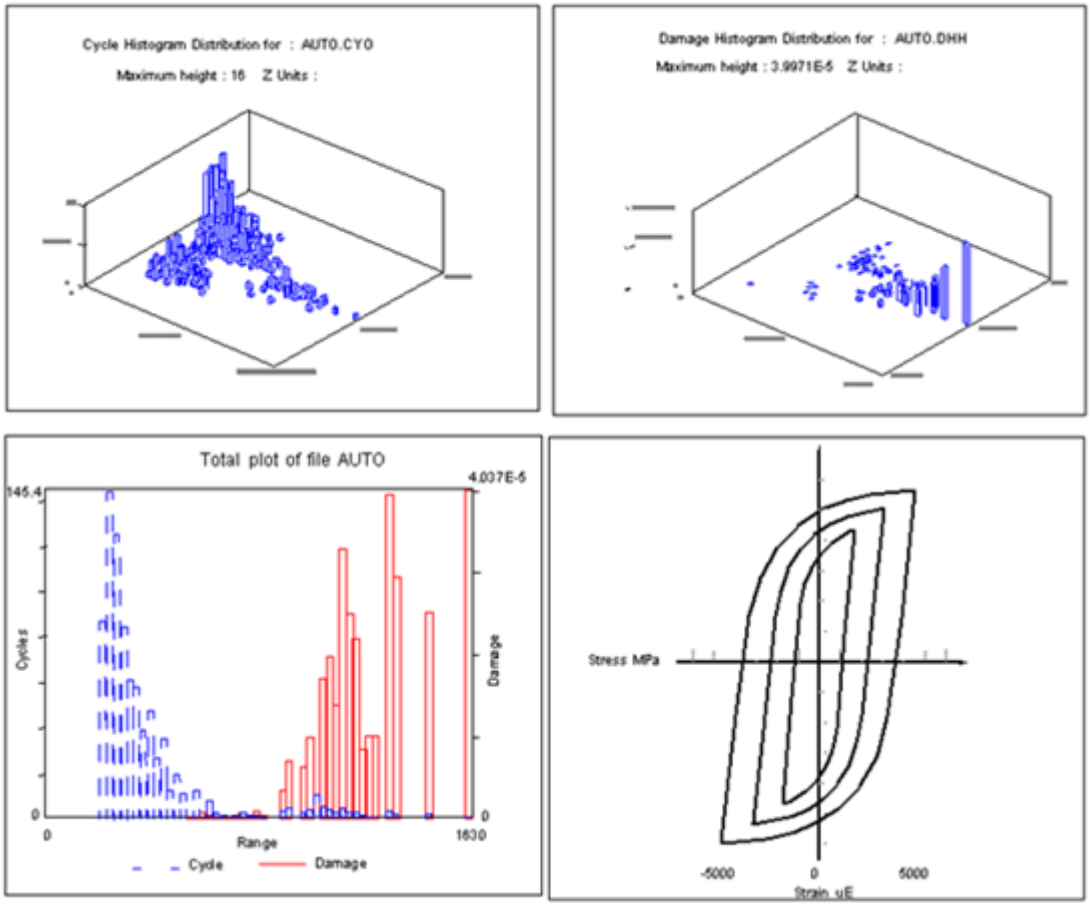


Figure 14-37 Sample MCLF Plots

Environment Keywords

MCLF makes the following entries in the local user environment.

USERNAME	Username
.FBJ	Last job file
PFSTSUNI	Material units
BACKTOLR	Back life calculation tolerance
MATCHECK	Default status of material checking
FATAUTOV	Default status for auto-overwrite
.DAC	Last `.dac` file
$SIGNAL	Last single parameter file

$CYCLES	Last cycles file
$HISTOG	Last histogram file
.CYH	Last cycles histogram file
$PAIRED	Last X-Y file
$ANNOT1	The first annotation string; $ANNOT2 TO $ANNOT6 also available.

Batch Operation

It is recommended that, by default, the /OV=Y keyword be included in every batch command line, since if it is omitted and an output file with the specified name already exists, batch operation will cease. To finish an MCLF batch line use /OPT=X to exit from the postprocessing screen.

When batch processing with a series of different inputs, it is necessary to use a new batch line and option definition for each new input. The new line must specify the option from the postprocessing menu into which the new input will go. For example, when using the jobfile `fatjob.fbj`.

```
mclf /job=fatjob.fbj/inp=datafil1.dac/opt=l
mclf /job=fatjob.fbj/inp=datafil2.dac/opt=l/opt=g/kf=2.0
mclf /job=fatjob.fbj/inp=datafil3.dac/opt=l/kf=2.5/opt=g
```

Omitting the /OPT keywords will cause the batch file to fail because inputs without /OPT keywords cannot be loaded and will therefore be ignored (the values in `fatjob.fbj` will be used instead). Each run of MCLF with one or more new parameters requires a new batch line. It is permissible to input more than one parameter on a batch line, but they must be different. Note also that the order in which the batch keywords appear is not critical.

Note that if a MCLF job file exists as the result of a prior interactive run of MCLF, then its job settings can be overridden in batch by use of the appropriate postprocessing option and the appropriate parameter. For example, suppose a job file called life.fbj exists in which K_f is set to 2. The following batch line will NOT change K_f.

```
mclf /job=life/kf=4/ov=y
```

However, the following batch line WILL change KF and automatically recalculate a new life...

```
mclf /job=life/opt=g/kf=4/ov=y
```

because /OPT=G invokes the Geometry option in which the change is made.

MCLF keywords:

/JOB	Job File Name
/CREate	Confirm Creation of New Job Y,N
/CALFIL	Nature of the Calibration File A,B,N
/CALNAMe	Name of Calibration File
/LUNIts	Input Units of Calibration files
/STATE	Strain State A,P
/STYPE	Strain Type M,E

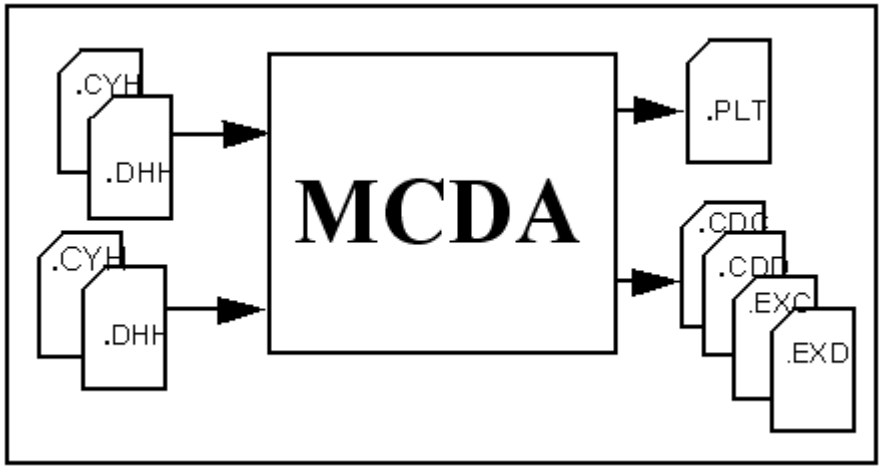
/INPut	Name of the Input Loading File
/TYPE	Type of Loading Input T,C,R
/UNITs	Internal Loading Units M,K,U
/EQUNITs	Equivalent Units
/NUMEQUnits	Number of Equivalent Units
/STArt	Start Time for Analysis
/END	End Time for Analysis
/FACTor	Scale Factor for input Loading
/OFFset	Offset of Input Loading
/GATE	Hysteresis Gate of Analysis
/AMPlitude	Magnitude of Constant Amplitude
/MEAN	Mean of Constant Amplitude
/MATENTry	Mode of Material Parameter Entry L,E,G
/MATSRC	Source of materials data S,U
/DBNAME	Material Database Name
/MATname	Material Parameter Dataset Name
/EDIT	Edit Material Dataset Y,N Parameter Editing & Entry
/UTS	Ultimate Tensile Strength
/YM	Young's Modulus
/SF	Fatigue Strength Coefficient
/BASQ	Fatigue Strength Exponent
/EF	Fatigue Ductility Coefficient
/COFF	Fatigue Ductility Exponent
/NP	Cyclic Hardening Exponent
/KP	Cyclic Hardening Coefficient
/CUTOFF	Endurance Limit Cut-off
/SERRE	Elastic Standard Error
/SERRP	Plastic Standard Error
/SERRC	Cyclic Standard Error
/RATIO	R - Ratio
/SURFace	Surface Finish (=POLished, GRound, GOod Machined, Average Machined, POOrly Machined, HOt Rolled, Forged, Cast, Water Corroded, Sea Corroded, User Defined.)
/TREATment	Surface treatment (=None, Nitrided, Cold rolled, Shot peened, All)

Parameter Generation:	
/GENTYP	Generic Material Type S,A,T,O
/UTS	Ultimate Tensile Strength/
SERR	Standard Error of Log strain
/RAREA	Reduction in Area Calculation Parameters
/DESign	Design Criterion Percent
/MSC	Mean Stress Correction Method N,S,M,A
/MINERs	Value of Miner' Constant
/EPCORR	Elastic-Plastic Correction Method, Neuber or Mertens-Dittmann N,M
/ALPHAP	Shape Factor for Mertens-Dittmann
/GEOTYPe	Notch Geometry E,C
/KF	Fatigue Strength Reduction Factor
/KT	Elastic Stress Concentration
/NOTCH	Notch Root Radius
/ADDKF	Additional K_f
/OUTput	Generic Name for Output Files
/CYCles file	Whether a Cycles File is Required Y,N
/DAMage file	Whether a Damage File is Required Y,N
/DAMUNI	Units of the Damage Matrix A,P,N
/XYFILe	Save X-Y Data in a File Y,N
/HTYPE	Matrix Type I,N
/SIZE	Matrix Size
/LIMit type	Matrix Limits A,U
/RMIN	Minimum Range of Matrix
/RMAX	Maximum Range of Matrix
/MMIN	Minimum Mean of Matrix
/MMAX	Minimum Mean of Matrix
/OPTion	Postprocessing Options L,M,S,G,O,D J,P,C,R
/RESOPTion	Results Display Options R,X
/JOBOPTion	Job File Options N,S,L
/PRFOPTion	Preference Options M,B,U
/UNIOPTion	Units Options MP,P,K,N,MN
/NEWJOB	Start Another Job Y,N
/LIFE	Life required for Back Calculation

/MATCHK	Material Parameter Checking Y,N
/TOLER	Sensitivity of Back Calculation
/OVerwrite	Overwrite Existing Output Files Y,N
/AUTOVerwrite	Overwrite Without Confirmation Y,N
/SWTMTH	S-W-T algorithm method F,I
/PLOt	Whether hardcopy is required Y,N
/PLTNAM	Hardcopy file name

Cycle and Damage Analysis - (MCDA)

MCDA calculates and displays cycles and damage distributions so that different test conditions may be compared and the reasons for variations in fatigue damage may be determined. Displays may be as histograms, continuous curves, or exceedance plots.



The MCDA module provides a means of displaying cycles histograms created from Rainflow analysis, and damage histograms created from Fatigue analysis. The input data is “histogrammed” within a two dimensional array with a maximum of 128 x 128 elements (bins). MCDA takes each array dimension and sums the data for each element across its opposing dimension.

This creates two new 2 dimensional histograms, each with a single array with a maximum of 128 elements. Exceedance data is also calculated for each new array. This is achieved by summing the data values from the current element in the array to its last element. This is repeated for each element until the maximum element is reached for that array.

An example of this calculation for this is shown in the following table.

TOTALS ARRAY					
element	1	2	3	4	5

	34.5	72.45	123.2	-12.0	312.0
EXCEEDANCE ARRAY	530.15	495.65	423.2	300.0	312.0

Module Operation

The MCDA module can be run in one of the following three ways:

- From the **Advanced Fatigue Utilities** pulldown menu under **Tools | MSC Fatigue** (for Patran) or under **Tools | Fatigue Utilities** (for)
- In Stand alone mode by typing **mcda**
- By incorporating the MCDA commands in a batch operation

Once running, MCDA will display the following window.

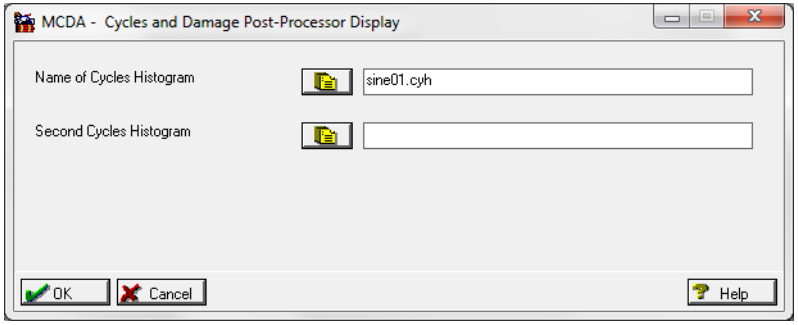


Figure 14-38 Naming the Input Cycle and Damage Files

The user is asked to enter the cycles histogram file name which usually has a `.cyh` extension. If a damage histogram file is found with the same generic name as the cycles histogram file this is also loaded. A default file may already exist in the cycles histogram field.

A second cycles histogram file may be loaded. If the user does not want to load a second histogram file, then this field can be left blank.

Depending on the user's input, one of four menus will be shown. These menus give a list of the available plot options for the histogram files selected.

Option	Description
Plot Cycles Histogram 1	Plots only the cycles of the first file selected.
Plot Cycles Histogram 2	Plots only the cycles of the second file selected.
Plot Both Histograms	Plots the cycles from both files together in the same plot for comparison purposes. This option is only available when two files have been supplied.
Plot Damage File 1	Plots the damage from the corresponding cycles of the first file only. This option is only available when a corresponding damage file exists in conjunction with the cycles plot.
Plot Damage File 2	Plots the damage from the corresponding cycles of the second file only. This option is only available when a corresponding damage file exists in conjunction with the cycles plot.
Plot Both Damage Files	Plots damage from both files. This option is only available when two files have been supplied.
Plot Damage/Cycles - File 1	Plots cycles with the corresponding damage superimposed on top of the cycles plot for the first file which is useful for viewing cumulative damage.
Plot Damage/ Cycles - File 2	Plots cycles with the corresponding damage superimposed on top of the cycles plot for the second file.

When the input cycles and/or damage files have been named, and a menu option chosen, then a plot of the data is displayed. An example plot is shown below in Figure 14-39 (in this case a plot of a cycles file and it's corresponding damage file).

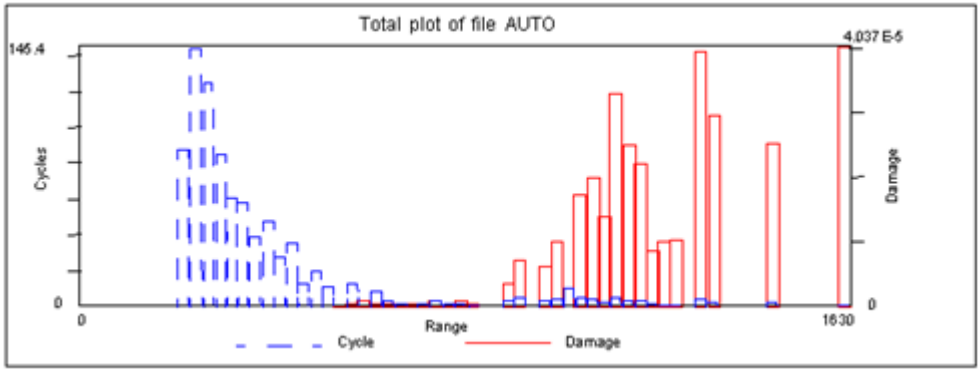


Figure 14-39 A Typical MCDA Plot

Once a plot is on the screen, operation of the interface is from the command line or via the pull down menus. The menu system is in MCDA is very similar to that in other graphical modules in MSC Fatigue, many of which are explained in Plot an Entry Option, 245 in the Loading (PTIME) module. Each menu item can also be invoked form the command line above the graphical display. Some of the command line codes are:

```
SA, NE, HC, RE, EX, PL,JO, TO, FU, WX, RA, ME, EP, TP, CW,
DW, LC, NC, LD, ND, GR, TI, TOUT, TIN, ZON, ZOF, AT, DT,
RON, ROFF, HBON, HBOF, GT, PESA, PEDE, PEPR, PEDA, PETX,
PEAX, PEAN, PEGR, PEBA, PEER, PESU, PEMF, PEMT, PEMO, PEMB,
PEMH, PEMK, XMIN, XMAX, CMIN, CMAX, DMIN, DMAX, CU, OP,
OPDM.
```

Typing OP at the command line will list all the options available within MCDA. Those of particular interest to MCDA are explained below.

Save Files	**SA**	Saves the displayed histograms as a single file.
New File(s)	**NE**	Allows a new input file to be entered.
Range Plot	**RA**	Displays the range of the histogram along the X-axis.
Mean Plot	**ME**	Mean Plot, displays the mean of the histogram along the X-axis.
Log Cycles	**LC**	Displays the cycles on the Y-axis on a logarithmic scale.
Linear Cycles	**NC**	Linear (Normal) Cycles values along the Y-axis (not log scale).
Log Damage	**LD**	Displays the damage plot Y-axis as a log scale.
Linear Damage	**ND**	Linear (Normal) Damage values on the Y-axis.
Exceed. Plot	**EP**	Displays the exceedance plot of the histogram.
Totals Plot	**TP**	Displays the total histogram along the X-axis
X Window	**XW**	Sets a specific X minimum and Y
Cycle Window	**CW**	sets a specific cycle minimum or maximum to plot.
Damage Window	**DW**	Sets a specific damage minimum and maximum to plot.
CMIN/MAX		Sets the minimum and maximum Y-axis cycles file window to plot (functionally similar to CW above).
DMIN/MAX		Sets the minimum and maximum Y-axis damage file window to plot (functionally similar to DW above)
When the cursor is activated the following sub options are available:		
V or Left Hand Mouse Button -		Pick off a data value
W		Window on X-axis
Q		Quit cursor mode
TOUT		Ticks drawn on the outside of the box
TI		Ticks drawn on the inside of the box
CP		Change plot type, returns to the menu
HC =<filename>		(file name optional)

XMIN/XMAX=<Value>		Set range or mean limits on the x-axis
CMIN/CMAX=<Value>		Set cycle limits on the cycles-axis
DMIN/DMAX=<Value>		Set damage limits on the damage-axis

When saved, the following extensions are given to the various signal files.

.cdc	Cycles histogram as totals plot.
.cdd	Damage histogram as totals plot.
.exc	Cycles histogram as exceedance plot.
.exd	Damage histogram as exceedance plot.

Batch operation

MCDA runs in all the standard batch modes supported in MSC Fatigue

A list of MCDA's batch keywords:

INP1	Name of the cycles histogram file	/INP1=SAETRN
INP2	Name of the second cycles histogram file	/NP1=SAETRN2
DOPT	The number of the chosen display option menu	
OPT	The number of the chosen graphics menu option	/OPT=3
XMIN	The minimum X-axis value	/XMIN=100
XMAX	The minimum Y-axis value	/XMAX=1E4
CMIN	The minimum cycles value to be displayed	/CMIN=10
CMAX	The maximum cycles value to be displayed	/CMAX=1000
DMIN	The minimum damage value to be displayed	/DMIN=5
DMAX	The maximum damage value to be displayed	/DMAX=275
OV	Overwrite an existing file Yes/No	/OV=Y
PLTNAM	Plot the file name (used with /OPT=HC)	/PLTNAM=FILENAME
PTIT	Plot the file title (used with /OPT=HC)	/PTIT=TEST RUN 5

A typical batch command line would be:

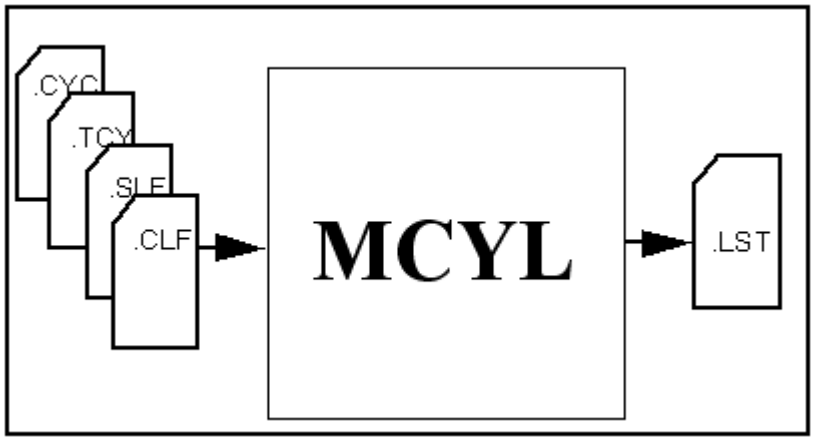
```
mcda /inp1=data1.cyh/inp2=data2.cyh/dopt=2/pltnam=mytest
/ptit=test run 12/ov=y
```

In this case, Histogram files data 1 and data 2 will be loaded. If both files have matching damage histogram files then these will also be loaded. The user has specified plot option 2 on the option menu, the command SA is given, and this saves the plotted histogram as a signal data file. The data file already exists therefore the

OV=Y batch command gives permission to overwrite it. The file name will be `mytest.cyh` with a title of Test Run 12.

Cycles File Lister - (MCYL)

It sometimes becomes necessary to examine the contents of a MSC Fatigue cycles file. In such instances, MCYL can be used to numerically list the contents of such a file, either to the screen or to a list file stored on disk. MCYL can also display a summary of the information resident in the file header region.



MCYL also accesses the cycle results files generated by the fatigue analyzers FEFAT, MCLF and MSLF and create a rainflow matrix of cycles according to user specified engineering units and limits. It can also display the histogram, alphanumerically, and produce an output histogram file which can itself be either plotted, listed or edited.

MCYL can display paired data files and three parameter cycles matrix files. It can also modify such files and save them as a disk file.

The 3-D rainflow matrices can be formed in terms of nominal or local stress or strain, with cycles being classified into a maximum of 128 range classes and 128 mean classes. The size, in terms of physical units, of the range and mean classes can be set independently.

The distribution of fatigue cycles in terms of counts or damage can be displayed alphanumerically or saved for later plotting by the MP3D module.

Although a histogram with 64 columns and 64 rows cannot be displayed on the screen as a unity, windows of 8 columns wide and 8 rows high can be used to scan the entire matrix. The particular use of this module lies in its ability to display quantitatively the relationship between numbers of specific cycles and associated damage content.

The cycle matrices generated can be used directly as inputs to the matrix fatigue analyzers FEFAT, MCLF and MSLF. This feature is particularly useful if the original time series data files are large and it is undesirable to process them directly.

Module Operation

MCYL can be run in one of 3 ways:

- From the **Advanced Fatigue Utilities** pulldown menu under **Tools | MSC Fatigue** (for Patran) or under **Tools | Fatigue Utilities** (for).
- In stand alone mode by typing **mcyl** the system prompt
- By incorporating the MCYL commands in batch mode

The main menu items of MCYL are:

- List a cycles file - List the contents of a cycles file (`.cyl`) to screen or disk.
- Form a cycles matrix - Create a cycles matrix and possibly a damage file (`.cyh` and `.dhh`)
- List a cycles matrix - List to screen or disk a file of the type created in 2 above
- List to EXCEL - Create a text format data file in Lotus Excel spreadsheet format (`.txt`)

List a Cycles file

To list a cycles file the following input is requested:

Option	Description
Input File Name	When this option's mask is first run only the Input File Name field is displayed. The user must tell MCYL the name and location of the cycles file to list. This file must be a standard cycles file from the fatigue analysis section or from the signal classification section. By default, MCYL expects to find any input data files to be resident in the user's directory. Probably the easiest method of entering input file names is to use the pick list facility. This also enables other drives/directories to be accessed. If a file name is entered without an extension then the default extension `.clf` will be given to the file name entered. A default file name may also be given in the question window. When the Input File Name field is filled the other fields become active. They are explained below.
Listing	In a critical location analysis, results are produced for the nominal location, and at the critical (local) location. Because of limited listing space, only one of these sets of results may be listed at one time. Therefore select either 1 or 2.
Gate Value (also applies to Damage Gate)	A gate may have originally been applied during the analysis to limit the calculation to large cycles only (setting a gate value is a method of reducing signal noise). It is possible to increase the original value with a higher gate value to further limit the listing. The gate must be a range value in the units of the file listing. For example in a stressed based file the gate value should be in MPa, whereas for strain based the units should be uE.

Option	Description
Display Type	The cycles may be shown in Max/Min format (where the peak and trough values of the cycles are displayed) or Range/Mean (where the range is the distance between max and min and the mean the average of the max and min.) Enter the value 1 or 2. Alternatively, the strings MAX and RANGE are also recognized.
Output Destination	The cycles file data may be displayed at the user's terminal by selecting S for Screen, or written out to a list file by selecting F for file. If File is selected then an additional Output File Name field must be filled. The default output file name is that of the input file but with a `.lst` extension. The default file name can be edited by the user (to avoid overwriting an existing file for example).
Output File Name	Enter a file name to which the data should be sent. The default extension for the file is `.lst` (see Output Destination above). The output file contains header information and a complete listing of the data in the cycles file.

Form a Cycles Matrix

To form a cycles matrix, the following input is required.

Option	Description
Input File Name	A cycles file from FEFAT, MCLF or MSLF should be input here. A damage histogram input field is also available (MCYL knows the source of the input file by reading it's header). The damage histogram field is optional.
Generic Output Name	Type the name of the file (or files if a damage file is also to be created). The default is to copy the Input File Name although this may require the input file to be overwritten.
Cycles Matrix Type	MCYL is able to form up a range-mean matrix from a cycles results file in terms of either stress or strain. Toggle the appropriate option into the field. MCYL will then display the limits of the largest and smallest cycles present in the cycles results file and prompt for the following:
Minimum Range Limit	For the purpose of scaling the histogram the range of the smallest cycle to be represented in the histogram must be entered in physical units (usually microstrain). If there are any cycles smaller than the minimum range specified, then those cycles will be excluded from the histogram. However, a warning message will appear and a note made in the note book.

Option	Description
Maximum Range Limit	For the purpose of scaling the histogram the range of the largest cycle to be represented in the histogram must be entered in physical units (usually microstrain). If there are any cycles larger than the maximum range specified, then those cycles will be excluded from the histogram. However, a warning message will appear and a note made in the note book.
Number of Columns	For the purpose of scaling the histogram it is possible here to specify the number of classes (bins) into which to classify the cycle means. Any integer number up to a maximum of 128 (the default) may be entered.
Number of Rows	For the purpose of scaling the histogram it is possible to specify the number of classes (bins) into which to classify the cycle ranges. Any integer number up to a maximum of 128 (the default) may be entered.
Minimum Mean Limit	For the purpose of scaling the histogram the smallest mean value to be represented in the histogram must be entered in physical units (usually microstrain). If there are any cycles whose mean values are smaller than the value specified, then those cycles will be excluded from the histogram. However, a warning message will appear and a note made in the note book.
Maximum Mean Limit	For the purpose of scaling the histogram the largest mean value to be represented in the histogram must be entered in physical units (usually microstrain). If there are any cycles whose mean values are greater than the value specified, then those cycles will be excluded from the histogram. However, a warning message will appear and a note made in the note book.

List a Cycles Matrix

A common usage for this option is to list the cycles file created in the above option although an existing file can also be listed. This option can list BOTH Cycles and Damage files. The following input is required:

Option	Description
Cycles File Name	Specify a cycles histogram file in this field, for example one generated in the Form a Cycles Matrix option. The default extension is `.cyh` but other compatible file types can be typed with their extension. This field may be left blank if only a damage histogram file is to be listed.
Damage File Name	Specify a damage histogram file in this field, for example one generated in the Form a Cycles Matrix option. The default extension is `.dhh`. This field may be left blank if no damage histogram file is to be listed.
List Zero Bins	List Zero Bins = No, will filter out from the listing any data entries that contain zero cycles (and therefore contribute nothing to the analysis). If this is set to NO it will ONLY apply to the cycles matrix NOT a damage matrix, should one be included in the processing.

Option	Description
List Type	3D is a normal 3D histogram plot, 2D = Range plots the sum of the X-axis values, 2D = Mean plots the sum of the Y-axis values.
Bin Location	The matrix files hold values in mean or range bins. The maximum of the bins can be used for the listing, or the mean of the bins (mean is (maximum-minimum) /2).
Damage Gate	Setting a gate at any value will filter out any data entries with a value LESS than the gate. Leaving this field blank will filter out nothing.
Destination - Screen or File	If the input file(s) are to be listed to screen select Screen. To save them to disk select File. They will be saved in the current directory.

List to EXCEL

This main menu suboption will save the listings file in a format that can be imported into a Microsoft Excel spreadsheet. The following input is required:

Option	Description
Input File Name	This field should contain the name of the cycles file that is to be converted to Excel text (`.txt`) format and saved on to disk.
Output File Name	By default this is the same as the Input File Name not with a `.txt` extension. However any valid name and extension can be specified here.
Output Format	The program can create two types of file. In the first case, all the data is written to the file, including header and label information. In the Raw Data option, only the matrix data is written.
Label Data	The data is normally output to the tab separated file without labels and totals. To add labels and totals to the data, answer Yes to this question.

Batch Operation

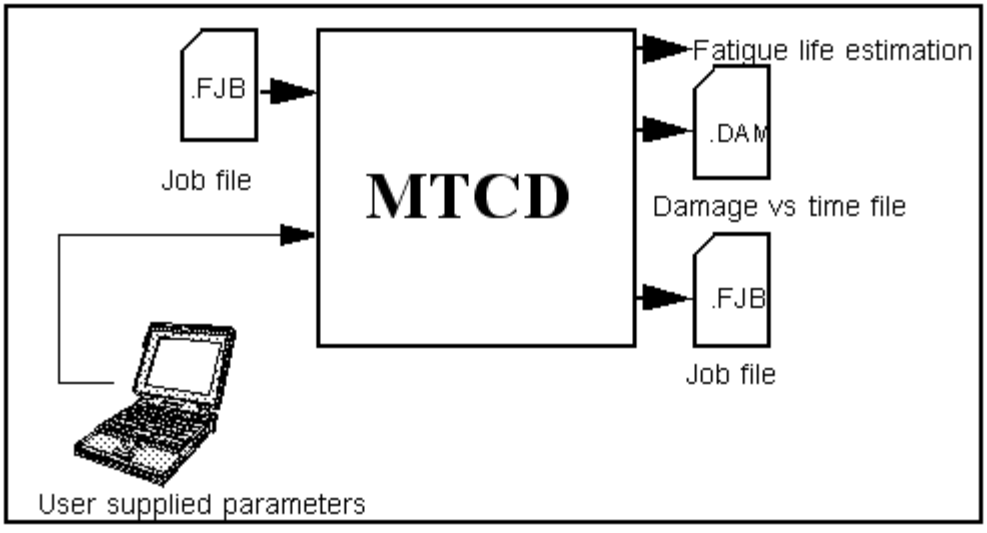
MCYL can be run in macro mode. In this mode of operation simple listings can be automatically generated. Please refer to the 'batch operation' section for more general details of invoking modules in batch.

/INPut	The name of the input cycles file. /INP=SAETRN
/LOCation	Whether the nominal or critical location is to be used. /LOC=2
/GATe=	The gate value to limit the listing. /GAT=50
/DISTYPe	The required display format. /DISTYP=2
/DESTination	The destination of the output. /DEST=F
/OUTput	The name of the output list file. /OUT=CYCLES
/OVerwrite	Overwrite existing files, Yes or No. /OV=Y
/DAMage	Damage gat. /DAM=12
/HSTOPT	Cycles matrix type. /HSTOPT=

/RMIN	Minimum range limit. /RMIN=10
/RMAX	Maximum range limit. /RMAX=8000
/NCOL	Number of columns. /NCOL=32
/NROW	Number of rows. /NROW=64
/MMIN	Minimum mean limit. /MMIN=200
/MMAX	Maximum mean limit. /MMAX=3000
/DMAT	Damage matrix, Y or N. /DMAT=Y
/DAMSC	Damage units Actual, %, or Normalized. /DAMSC=%
/OPT	Main option, 1, 2, 3, 4. /OPT=3
/CYH	Cycles histogram file name. /CYH=TEST1.CYH
/DHH	Damage histogram file name. /DHH=TEST1.DHH
/ZERO	List zero bins. /ZERO=YES
/TYPE	List type - 3 (3D),R (2D range), M (2D mean). /TYPE=2DR
/BINLOC	Bin location, Mean or Range. /BINLOC=R
/DGAT	Damage gate. /DGAT=20

Time Correlated Damage - (MTCD)

MTCD is a fatigue analyzer that can be used to pin-point fatigue damage within a loading history. It uses the local stress-strain approach to track local stresses and strains by means of a single-pass algorithm. An estimate of the total damage accrued by one pass through the load history is made and displayed graphically.



The local stress strain approach can be used to estimate the time taken to generate an engineering crack 2-5 mm in length.

The methodology needs a description of the loading history, the component geometry and material properties.

Typically, the loading history is measured in the field, and is assumed to be representative of the actual loading the component will experience throughout it's life. Life estimates are usually expressed in the form of a number of repeats of that history to failure, i.e. that the history essentially repeats itself. This assumption allows the analysis to be simplified.

Because its local stress-strain coordinates must lie on the cyclic stress strain curve, analysis starts at the largest absolute strain excursion, continues to the end of the sequence, wraps around to the start, and finishes at the largest excursion again. In this way all cycles are closed and repeatable. This procedure is often referred to as the two-pass algorithm, one pass to locate the largest excursion, and a second pass to carry out the fatigue analysis.

The single pass algorithm commences analysis at the beginning of the loading sequence and accumulates damage as each cycle closes and each reversal is replaced by larger relatives at the head of the rainflow stack. At the end of the analysis, the largest strain excursion will be at the head of the stack and accompanied by a number of residual peaks and valleys which cannot be closed.

The local stress and strain are tracked at each strain excursion, by using the cyclic stress strain relationship, together with a rheological material model using segment exhaustion.

Once a cycle or appropriate reversal has been identified, the fatigue damage is estimated and ascribed equally to both of its turning points. The damage is written in the correct sequential position of the output damage file. Loading data values which are not turning points, are written out to the damage file with zero damage. In this way the time-base is maintained and the two files may be compared directly. The number of cycles, unclosed reversals, and the total damage accumulated, are presented for scrutiny.

It is important to note that MCLF and FEFAT both use a two pass algorithm, and when analyzing the same loading sequence it will provide total damage values slightly different to those given by MTCD which commences analysis at the first point in the sequence. Naturally, MTCD will agree with the other analyzers when MTCD is also forced to start at the largest excursion.

Module Operation

The MTCD module can be run in one of the following three ways:

- From the **Advanced Fatigue Utilities** pulldown menu under **Tools | MSC Fatigue** (for Patran) or under **Tools | Fatigue Utilities** (for).
- In stand alone mode by typing **mtcd** at the system prompt
- By incorporating the MTCD commands in a batch operation

When run in interactive mode, MTCD's first screen will ask for the name of the file to process and looks like this:

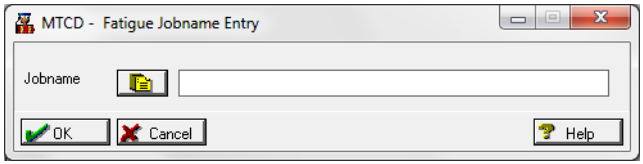


Figure 14-40 The First MTCD Screen

The user must enter the name of an Input Job File, which has a `.fjb` extension. If the name of an existing Job File is entered, it's parameters will be loaded and MTCD will go straight to a postprocessing screen. If a new job is specified then its parameters have to be entered in a series of screens. If no name is entered then the results of processing may NOT be saved.

The operation of this module is very similar and in many cases identical to MCLF, MSLF, and FEFAT. It is assumed that the user has a working knowledge of FEFAT and therefore only those items specific to MTCD is explained here.

Figure 14-41 illustrates the major routes through MTCD.

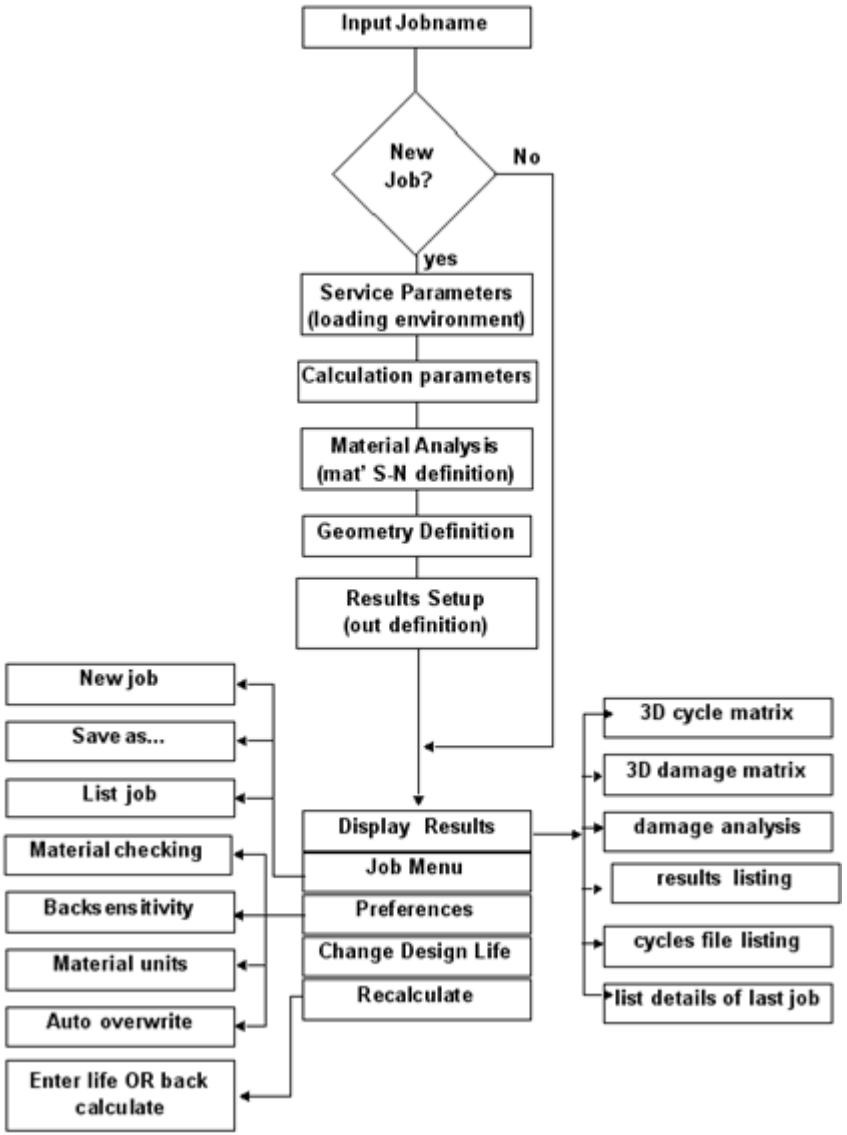


Figure 14-41 The General Module Structure (MTCD)

The above flowchart is mirrored by the options that are on the Postprocessing menu. If a new job is being defined, then the following input is required:

The Loading Environment

This information is identical to that needed for MCLF. See Service Loading Environment, 1096.

Model Parameters

This information is identical to that needed for MCLF. See Model Parameters, 1101.

Material Data Input

This information is identical to that needed for MCLF. See Material Data Input, 1101.

Geometry

This information is identical to that needed for MCLF. See Geometry Screen, 1102.

Results Setup

This is the only input that is somewhat different.

Option	Description
Output File	Enter the name of the output file. The default extension is `.dam`. This file is in `.dac` format.
Output File Type	Output damage files can have either of two formats. In the first, the single value format, the absolute damage associated with each turning point, half for each cycle or reversal, is written to the output file at the same point in time it occupied in the time series file. Non turning points have zero damage ascribed to them. In the second format, the damage is accumulated as it occurs and so provides insight into the rate of damage accumulation.
Start At	MTCD uses a single pass algorithm to accumulate fatigue damage. As a result, analysis can commence at the beginning of the loading sequence, instead of at the largest excursion, and so provide a more accurate distribution of damage within the time series. For compatibility with MCLF and FEFAT, MTCD has been provided with the option to commence analysis at the largest absolute strain excursion. When run in this mode, MTCD will provide values of total damage similar to those calculated by MCLF, if not, the values will be slightly different.

When all of the data for a calculation has been entered MTCD will create the required damage file. Results will also be listed to the screen in the first instance. Plots can be displayed from the Postprocessing menu.

Note that the result is presented as accumulated damage rather than repeats to failure.

The Postprocessing Main Menu

This menu is identical to that needed for MCLF. See The Postprocessing Menu, 1105.

Note: If any parameters are changed the Recalculate option must be run for the changes to take effect.

This display of results is of the most recently calculated life estimation. Exactly what is available for display depends upon the calculation; if an X-Y file was not created then it cannot be displayed, a multiple calculation will not produce any histogram files, etc.

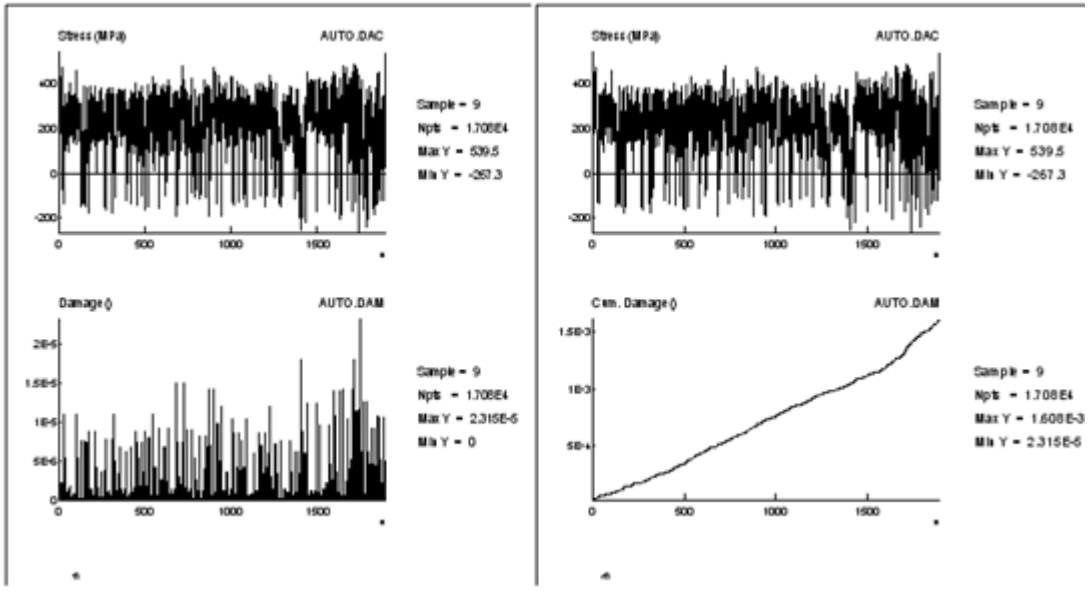


Figure 14-42 Sample MTCD Plots. Top: Job Details Page, Bottom: A Cumulative Damage Plot, an Instantaneous Damage Plot

Batch Keyword Summary

A sample batch line is shown below:

```
mtcd  /job=test1/opt=l/inp=test101/ov=y/opt=r/opt=d/resopt=t
/pltnam=test101
```

In this example, the time series contained in file `test101.dac` will be processed according to the defaults set in job file `test1.fjb` and the resulting damage distribution will be plotted together with the input time series and save in a plot file call `test101.plt`.

Note that if a MTCD job file exists as the result of a prior interactive run of MTCD, then its job settings can be overridden in batch by use of the appropriate postprocessing option and the appropriate parameter. For example, suppose a job file called `life.fbj` exists in which K_f is set to 2. The following batch line will NOT change K_f.

```
mtcd /job=life/kf=4/ov=y
```

However the following batch line WILL change K_f and automatically recalculate a new life...

```
mtcd /job=life/opt=g/kf=4/ov=y
```

...because OPT=G invokes the Geometry option in which the change is made.

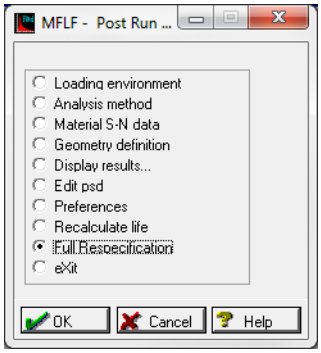
A list of MTCD's batch keywords:

/JOB	Job File Name
/CREate	Confirm Creation of New Job Y,N
/INPut	Name of the Input Loading File
/CALFIL	Nature of the Calibration File A,B,N
/CALNAMe	Name of Calibration File
/LUNIts	Input Units of Calibration files
/STATE	Strain State A,P
/FACTor	Scale Factor for input Loading
/OFFset	Offset of Input Loading
/GATE	Hysteresis Gate of Analysis
/EQUNIts	Equivalent Units
/NUMEQUnits	Number of Equivalent Units
/MATENTry	Mode of Material Parameter Entry L,E,G
/MATname	Material Parameter Dataset Name
/EDIT	Edit Material Dataset Y,N
/PLTNAM.	Request a hardcopy plot of the results file, and give it a file name. /PLTNAM=`MYTEST.PLT`
Parameter Editing & Entry	
/UTS	Ultimate Tensile Strength
/YM	Young's Modulus
/SF	Fatigue Strength Coefficient
/BASQ	Fatigue Strength Exponent
/EF	Fatigue Ductility Coefficient
/COFF	Fatigue Ductility Exponent
/NP	Cyclic Hardening Exponent
/KP	Cyclic Hardening Coefficient
/CUTOFF	Endurance Limit Cut-off
/SERR	Standard Error of Log Reversals
/RATIO	R - Ratio
Parameter Generation	
/GENTYP	Generic Material Type S,A,T,O
/UTS	Ultimate Tensile Strength
/SERR	Standard Error of Log Reversals
/RAREA	Reduction in Area, %

Time Correlated Damage Analysis	
/SURFace	Surface Finish: Polished, Ground, GOod Machined, Average Machined, Poorly Machined, Hot Rolled, Forged, Cast, Water Corroded, Sea Corroded, User Defined. SUR=HOT ROLLED
/TREATment	Surface treatment, None, Nitrided, Cold rolled, Shot peened. /TREAT=NONE,/TREAT=SHOT PEENED
/DESign	Design Criterion Percent
/MSC	Mean Stress Correction Method N,S,M
/MINERs	Value of Miner' Constant
/GEOTYPe	Notch Geometry E,C
/KF	Fatigue Strength Reduction Factor
/KT	Elastic Stress Concentration
/NOTCH	Notch Root Radius
/ADDKF	Additional Kf.
/OUTput	Generic Name for Output Files
/OTYPE	Damage File Type S,C
/ANAStart	Analysis Start Position B,L
/OPTion	Postprocessing options L,M,S,G,O,D,J,P,C,R, X, (i.e. the menu hot key)
/RESOPTion	Results Display options T,X, (T= plot time series)
/JOBOPTion	Job File Options N,S,L
/PRFOPTion	Preference Options M,B,U
/UNIOPTion	Units Options MP,P,K,N,MN
/NEWJOB	Start Another Job Y,N
/MATCHK	Material Parameter Checking Y,N
/OVerwrite	Overwrite Existing Output Files Y,N
/AUTOVerwrite	Overwrite Without Confirmation Y,N

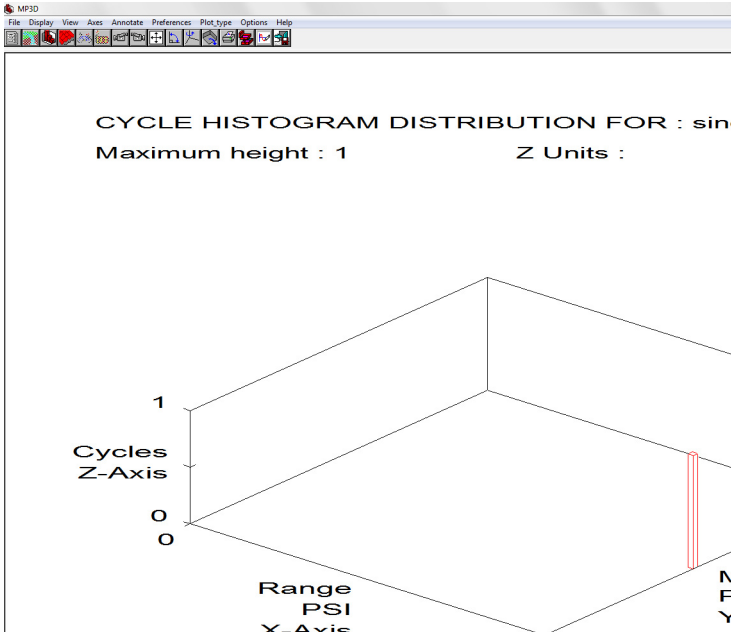
Single Location Vibration Fatigue - (MFLF)

MFLF is a single location, stress-based fatigue analysis module that accepts stress response PSDFs as input. This module has also been mentioned in an earlier chapter. As an example of usage copy over the original SAE history **saetrn.dac** to your working directory. This signal is assumed to contain a stress time response.



Use MASD to convert the time signal into the frequency domain by converting it to a PSDF. See the section on MASD in this chapter for instruction on how to do this. Use all the default settings. The output file name should be **saetrn.psd**.

1. Invoke MFLF from the system prompt by typing **mflf** or choose the Single Location Vibration Fatigue option from the **Advanced Fatigue Utilities** pulldown menu under **Tools | MSC Fatigue** (for Patran) or under **Tools | Fatigue Utilities** (for).
2. Accept all defaults for all setup screens except for these: Input Filename: **saetrn.psd**; Dataset Name: **MANTEN**



The analysis will proceed, the results will be presented and eventually you will be placed in the Post Processing Options. Answer **Yes** to any overwrite permission questions.

Go to **Display results... | Cycles histogram**.

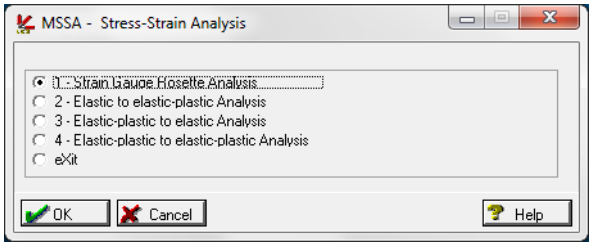
Exit from MFLF when you are finished.

Note: This example is for illustration purposes only. The signal used in this example is not actually an appropriate signal to use in that it is not truly random or gaussian as required by a random vibration fatigue analysis.

See Frequency Fatigue Life Estimation (MFLF), 695 for a more detailed description of this utility.

Stress-Strain Analysis - (MSSA)

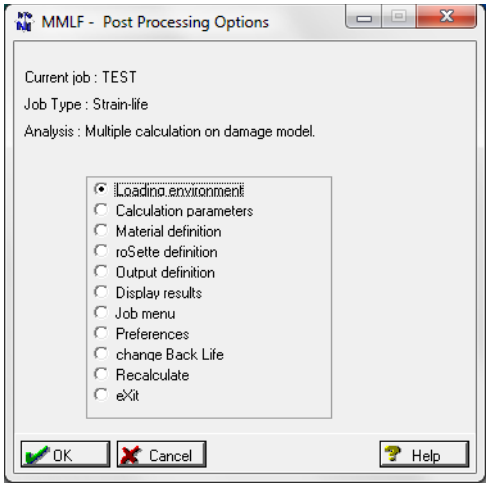
Stress-Strain Analysis processes rosetta data and finite element data from MSC Fatigue, including software strain gauges. It creates outputs suitable for use by either the stress or strain-life fatigue analyzers. It also provides an indication of the state of multiaxiality present, suggests possible processing routines through the fatigue analyzers and has a multiaxial fatigue analyzer that works by using a MSC Fatigue .fes file. In addition to this, the module can be used to convert elastic-plastic strain records, measured on one material, to that of another material. It can also convert elastic-plastic strain records to equivalent fully elastic ones and vice-versa.



Note: See Stress-Strain Analysis (MSSA), 911 for a more detailed description of this utility.

Multi-Axial Life Analysis - (MMLF)

MMLF is a single location multiaxial fatigue analyzer based on Crack Initiation and has been briefly referred to in a previous chapter. It requires three strain input signals which typically come from strain gauge rosettes. For rectangular rosettes the signals are separated by 45 degrees. For delta rosettes the signals are separated by 60 degrees.

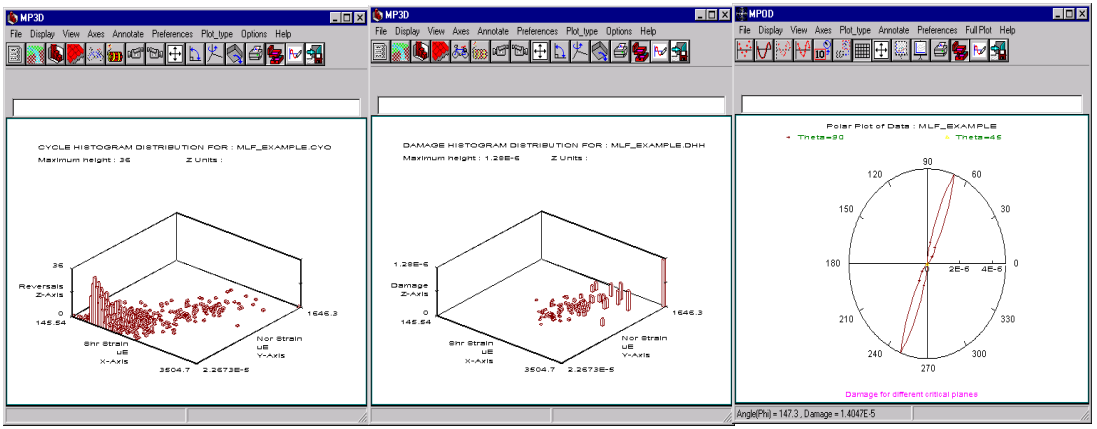


As an example, take the three SAE histories that we have been using thus far (**saetrn.dac**, **saesus.dac**, **saebrakt.dac**), except run them through MLEN and chop them all to **1800** seconds. (See the previous section on MLEN to learn how to do this.) We will assume that these new signals are from a rectangular rosette.

1. Invoke MMLF from the system prompt by typing **mmlf** or choose the Multi-Axial Life Analysis option from the **Advanced Fatigue Utilities** pulldown menu under **Tools | MSC Fatigue** (for Patran) or under **Tools | Fatigue Utilities** (for).
2. Enter a new job name such as "**mlf_example**." It is new, so answer **Yes** to the ensuing question.
3. Accept all defaults for all setup screens except for these: Gauge 1: **saetrn.dac**; Gauge 2: **saesus.dac**; Gauge3:**saebrakt.dac**;Material Name: **MANTEN**
4. The analysis will proceed, the results will be presented and eventually you will be placed in the Post Processing Options. Answer **Yes** to any overwrite permission questions.

   Go to **Display results | Stress and Strain**. Plot this result and any of the others you wish in this menu selection.

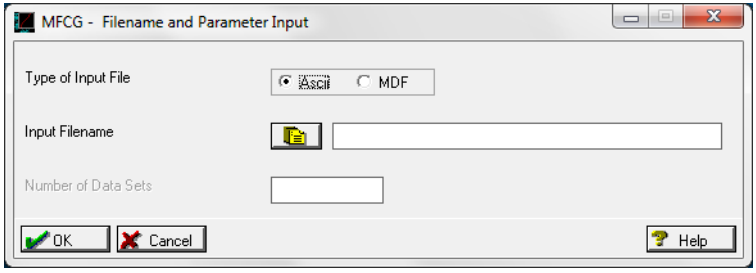
Exit from MMLF when you are finished.



Note: Strain signals input to MMLF are assumed to be elastic-plastic. No elastic-plastic corrections are performed in MMLF. Use MSSA and/or SSG to do this if necessary from FE data.

See Local Multiaxial Stress/Strain Fatigue Analyzer (MMLF), 438 for a more detailed description of this utility.

Crack Growth Data Analysis - (MFCG)

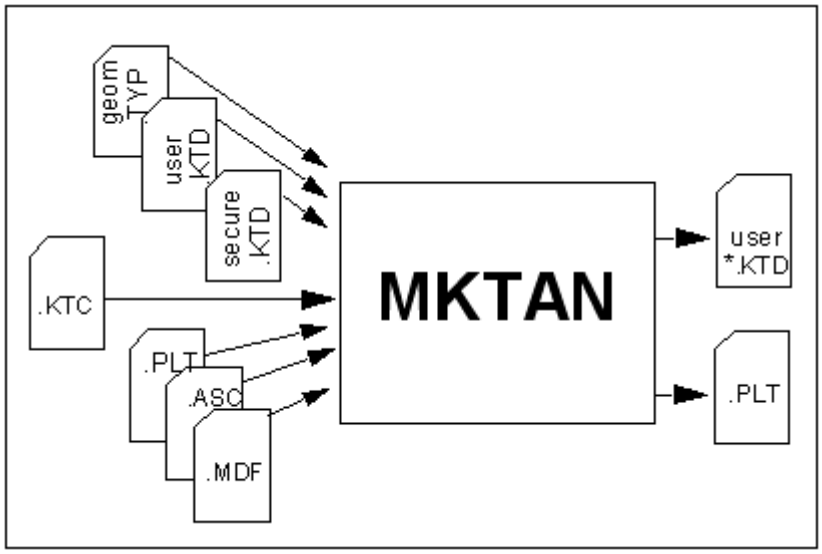


MFCG calculates the Paris Law coefficient, C, and exponent, m, in the expression $da/dN = C(\Delta K)^m$ from actual raw test data obtained under constant amplitude loading conditions.

Note: See Crack Growth Data Analysis (MFCG), 580 for a more detailed description of this utility.

K_t/K_f Evaluation - (MKTAN)

MKTAN stores and retrieves values for stress concentration factor (K_t) solutions for geometric details, and calculates K_t and K_f. It allows users without finite element analysis (FEA) software rapid and convenient access to K_t values for a range of common component geometries. The K_t values can be used in programs such as MTCD to predict the fatigue life of an engineering component which require the user to input a value for stress concentration factor K_t, or fatigue strength reduction factor K_f. Such values have been calculated for a wide range of shapes (geometries) and sizes of engineering components and are available in standard reference works such as Peterson's book on Stress Concentration Factors.



Tables exist for geometries such as elliptical holes with a range of long and short diameters, in plates of various widths, crankshafts, springs, keyways, inclusions, etc. MKTAN stores these values in a library of cases and allows the user easy access to the libraries for calculation purposes.

The library cases cover popular and common geometries such as holes and notches in plates and bars, plus effects of surface condition on crack initiation, and strength reduction K_f values for various environments such as seawater.

Subsequent releases of MKTAN will contain progressively more comprehensive cases for a wider range of geometries and conditions. Library entries supplied with MKTAN are detailed in an index file called `secure.ktd`. Actual entries are held in a secure database. Supplied files cannot be edited. However the user can copy entries into a user database and edit them at will. Details of the user database are held in a file called `user.ktd`.

Module Operation

MKTAN can be run in one of the following 3 ways:

- From the **Advanced Fatigue Utilities** pulldown menu under **Tools | MSC Fatigue** (for Patran) or under **Tools | Fatigue Utilities** (for).

- In stand alone mode by typing, **mktan** at the system prompt
- By incorporating the MKTAN commands in a batch operation.

There are two main operations that MKTAN performs:

- Calculate a K_t value from the Secure or User database.
- Edit the User database or view the Secure or User databases.

There are a variety of steps and options to both of the above operations. However, both types of operation are started from MKTAN's main menu. The main menu screen is the first screen to appear when MKTAN is operated in interactive mode. The main menu screen is shown overleaf.

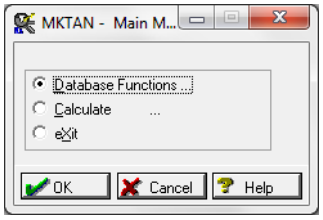


Figure 14-43 The MKTAN Main Menu Screen

The options available from the main menu screen are illustrated in the flowchart below.

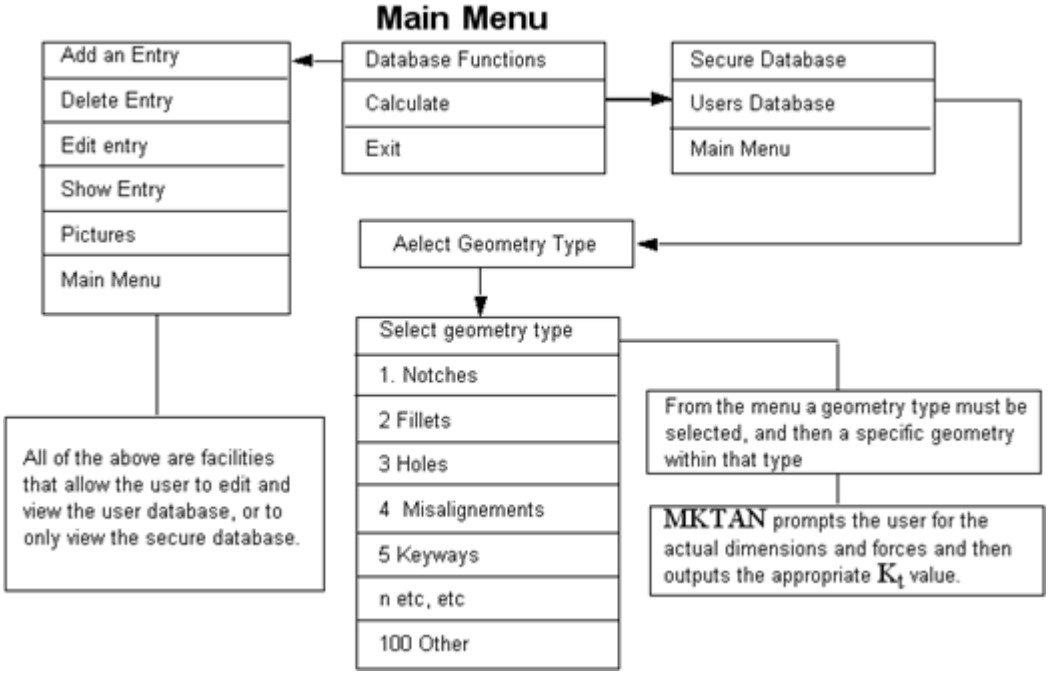


Figure 14-44 Flowchart Summarizing MKTAN Menu System

The options on the left of the flowchart allow for user database functions such as display or edit. Entries from the secure database can also be copied to the user database and then edited although the secure database cannot be changed.

The options on the right of the flowchart enable the component geometry to be selected, and the exact dimensions of that geometry, plus any other information needed to calculate a K_t value to be input.

Database Functions Menu

All database functions are achieved via this menu. The text below details the purpose and function of each of the options from the Database Functions selection on the main menu.

Add an Entry

Choose this option when an entry to the database is being made. The user will be sent to a screen which has empty fields into which a definition of the entry will be made.

Option	Description
Description field	Enter a description for the database entry. The description can NOT be left blank.
Comments field	Enter some comments about the database entry. The comments field can be left blank.
The Reference field	Enter a reference for the database entry (the Reference field). The reference can be left blank.
Name of Storefile	This is the name of the file where the database information will be kept. The name of the store file must be entered, and must be unique.
X type	The X parameter type can be toggled between Single and Ratio by pressing the space bar or clicking the left mouse button. The X parameter type Single means: a single variable (numerator), e.g. A, such as the width of a plate. An example of a X parameter type Ratio: A / B, such as the long and short radii of an elliptical hole.
X Scale	This scale refers to the lookup table X scale. The X scale can be either Linear or Log 10. The X scale can be toggled between Linear and Log 10 by pressing the space bar or clicking the left mouse button.

Option	Description
Family Type	The Family type can be Single, Ratio or None. Each of these values can be chosen by toggling the type, by pressing the space bar, or by clicking the left mouse button. The family type Single refers to a single variable representing the family value (e.g. A). The family type Ratio refers to two variables representing the family value as a ratio (e.g. A / B). The family type None means that there are no variables which represent the family value. This means that there is only one curve (lookup table) in the family.
Variable 1 - 4	The variables entered are used to represent the X parameter and/or Family parameter.
Geometry type	The available geometries can be toggled through and selected. When the fields have been filled the 2nd Add Entry screen is displayed. Some of the fields will NOT be shown if they are not appropriate to the parameter and choices specified on the previous screen.

A second Add an Entry screen will be presented whose input parameters are:

Option	Description
X Parameter numerator	The available X parameter variables can be shown/selected by pressing the space bar or clicking the left mouse button. Ensure that this selection is for the numerator for the X-axis ratio, for example a in a/b.
X Parameter denominator	The available X parameter variables can be shown/selected by pressing the space bar or clicking the left mouse button. Ensure that this selection is for the numerator for the X-axis ratio, for example b in a/b.
Family parameter numerator	The available family parameter variables can be shown/selected by pressing the space bar or clicking the left mouse button. Ensure that this solution is for the numerator family parameter ratio, for example r in r/D.
Family parameter denominator	The available family parameter variables can be shown/selected by pressing the space bar or clicking the left mouse button. Ensure that this solution is for the denominator family parameter ratio, for example D in r/D.
Plot File Name	This field is where the picture/plot entry is named. The default file type is `.plt`. The plot file should contain an image of the Geometry to be used in the calculation of K_t. The image will be shown if the geometry is selected.
Lookup tables	If the type chosen is `.mdf` (MSC Fatigue two parameter file) or ASCII the lookup table file name is specified here. If the lookup table type is polynomial then the polynomial constants and limits are required to be entered. The format of this file is shown after this table.

Option	Description
Lookup File Name	This screen allows the name of the lookup table to be entered.
Select database screen (geometry type)	This screen allows the database to use for drawing the pictures (geometries) to be selected.
Polynomial constant screen	A screen appears if Polynomial was selected. Enter the value for the polynomial constant A0 to A3. n is from 1 to 4 and all 4 constants should be entered even if they are all zero. The Family Parameter values: For each family (sub-set) the value of the family should be entered, for example A=0.1. For each subset the family parameter value must be larger or smaller than the previous one (uniformly increasing or decreasing).

Format of MKTAN ASCII lookup files:

If the lookup table type is ASCII then the file type is defaulted to `.asc` (e.g. entering `fred` is interpreted as `fred.asc`). An ASCII file can be created using any editor or word processor, however the format provided by MKTAN is as follows. The first line of the ASCII file must contain the following words starting in column 1: KTAN ASCII FILE.

The second line must contain the number of families specified in this file, e.g. 3. The format is then:

- The family value for the lookup table
- The number of lookup table pairs
- The lookup table x, y pairs (each pair on separate lines)

An example of an ASCII file is as follows:

```
 KTAN ASCII FILE<ASCII file identifier>
2<number of families (2 shown below)>
0.1 <family value for family 1>
6<number of lookup table x, y pairs>
10.0,20.0<X,Y pairs>
30.0,30.0
50.0,35.0
60.0,40.0
70.0,80.0
80.0,90.0
0.2<family value for family 2>
3<number of lookup table x, y pairs>
8.0,15.0<x,y pairs>
10.0,25.0
20.0,100.0
```

Note: Items in < > are for reference only, and do not appear in the file.

Delete entry

This question relates to the deletion (removal from database) of a user database entry. The Storefile associated with the user database entry is also deleted. After selecting an entry to be deleted, you must confirm the deletion by answering in an affirmative manner (Yes) to the Are you Sure? prompt to delete the entry. Answering Negatively (No) will not delete the user entry. The database entry and the user database will be removed if this question is answered in an affirmative manner (Yes).

Answering No will leave the User Database without making changes. If the user database is deleted (removed) then no user database operations will be allowed until further entries are made to the user database (i.e. by completing an 'Add an entry' operation).

Edit an Entry

This screen allows the editing of a User Database entry. At first this option lists all the database entries. An entry must be selected for editing. Note that a page of information on the currently highlighted entry can be obtained by pressing the Info button. The Info data is there to help users choose. When a selection has been made another menu appears. The following explains what each menu option does.

Option	Description
Edit general details	This allows the description, comments, reference and geometry type to be changed.
Replace Plot File	This allows a new plot file (geometry drawing) to replace the existing geometry drawing.
Extract to MDF file	This allows the extraction of the lookup table data to a `.mdf` format (paired) file. Polynomial lookup table cannot be extracted.
Extract to ASCII file	This allows the extraction of the lookup table data to an ASCII format file. The ASCII file produced is the same format as required to be input to MKTAN to represent a lookup table. Polynomial lookup table cannot be extracted.
Replace Lookup table with MDF	This replaces the existing lookup table with an `.mdf` format file.
Replace Lookup table with ASCII -	This replaces the existing lookup table with an ASCII format file.
Edit Polynomial constants	This allows the editing of the polynomial constants lookup table

Show Entries Screen

This screen allows the user to view the database entries. It contains these suboptions:

Option	Description
Show all entries	The complete database is listed. Details of individual entries are stored as pages of general information. To view a page of information about an entry move the highlight bar over the entry and click on Info or press F3. Alternatively a semi-sorted list of database entries can be viewed by using the sub-option below.
List by Geometry Type	This screen shows a list of the database entries with a given geometry type. Entries are displayed in pages with general information displayed about each entry. Note that a page of information on the currently highlighted entry can be obtained by pressing or by clicking on the Info button. The Info data is there to aid users in their choice.

Pictures

The final option on the Database Functions menu is Pictures. All database entries should have a corresponding picture associated with them (specified on the Add Entry form). The picture shows the user what the dimensions are and what shape the component is. The pictures will be produced on a graphics package and will be a plot file with a `.plt` extension. When 'Pictures' is selected, the user is given a list of available geometry types. Selecting a geometry type, which may have several cases, leads to a page of pictures.

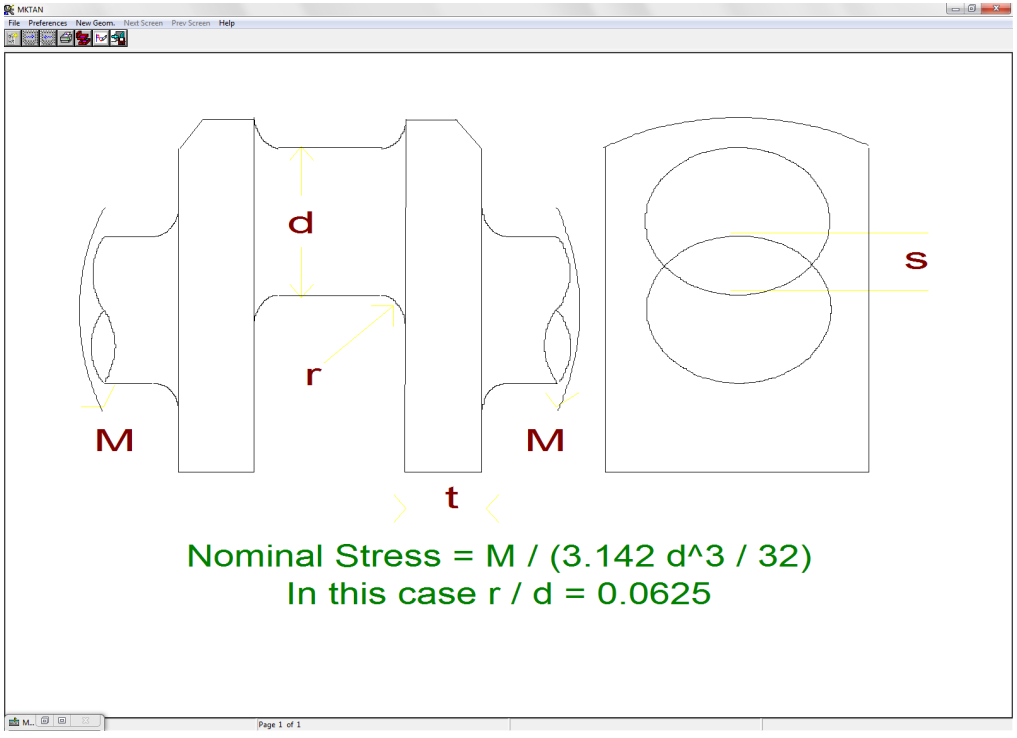


Figure 14-45 A K_t Picture (This Example is a Crankshaft)

The Calculate Menu

The other option from the Main Menu is Calculate. This allows K_t values to be calculated for any geometry contained within the database.

When Calculate is selected, the user must choose to use either the Secure database or the User Database. This choice is made on a pop up menu that appears next to the main menu. There then follows an established series of steps that lead to a K_t value being calculated. When the database is chosen a form appears.

It is here that the geometry type of the component being analyzed is chosen from the database of geometry types. After the geometry type is selected a list of entries of that type is displayed. An entry must be selected. Any single geometry type will probably contain more than 1 entry or case. For example holes in a plate of finite width, and holes in a plate of infinite width. One of them must be selected before K_t can be calculated.

Selection is by one of two methods, either:

- From the list shown on a screen such as the screen overleaf where the user can move the highlight bar over the desired choice and select it by pressing ENTER or by pointing and clicking the mouse pointer over the selection.

  or

- The user can click on the Picture button to produce a picture of the case. In the above example it will cause the picture over the page to be displayed. One picture is of a plate with a finite width of value w. The other picture is of a plate with an infinite width. A selection can be made by pointing and clicking on the appropriate picture.

When selected, a screen like this will appear:

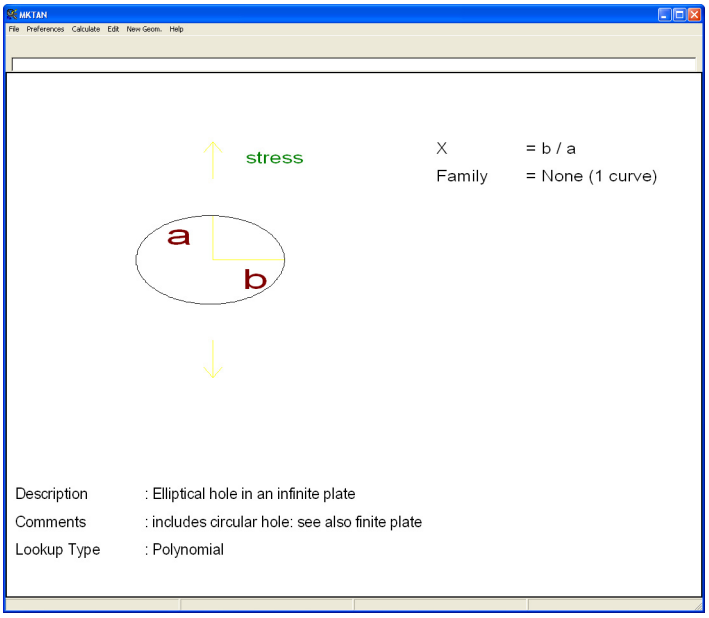


Figure 14-46 K_t Hole Dimension Definition

The picture defines the relevant dimensions of the geometry and gives guidance to the user as to what numbers should be entered where. All the dimensions and variable names etc. were defined when the entry into the database was made using the Database Functions/add Entry facility.

The menu to the right of the picture contains the Calculate command. Clicking on Calculate causes MKTAN to ask the user to input the actual values for the dimensions. Note the value for K_t (in this case 4.333).

The graphics menu (shown overleaf) also contains the hardcopy command so users can record the case, with its dimensions, and the calculated K_t value for reporting purposes.

Graphics Mode Operation

Some of the graphics commands of MKTAN are shown here. These commands can be entered in the command line of the graphics window or accessed directly from one of the pulldown menus:

```
CA, ED, NG, HC, PL, EX, RON, ROFF, PESA, PEDA, PETX, PEAX, PEAN,
PEGR, PEBA, PEER, PESU, PEMF, PEMT, PEMO, PEMB, PEMH, PEMK, OP,
HBON, HBOF, CU.
```

The graphics screens are menu or command line operated. Most of the options are standard MSC Fatigue operations and are explained in Graphical Display, 254. Options of particular interest to MKTAN are explained below.

Edit	ED	Allows a second calculation to be performed with new (edited) values. Note that Calculate must be clicked upon before the new values are acted upon.
Calculate	CA	Triggers the process that calculates new K values. The user must supply values for whatever variables have been defined for the current geometry.
New Geom		This option sends the user back to the Select Geometry Type screen.
Main Menu		Sends the user back to the MKTAN Main Menu screen.

Batch Operation

A list of MKTAN's batch keywords:

/MEN	Main menu option. (E = Exit, D = Database Menu, C = Calculate)
/DB	Database menu option. (A = Add, E = Edit, etc.)
/CAL	Calculate menu option. (U = User database, S = Secure database)
/SE	Show entry menu option. (L = List by type, S = Show all entries)
/EDT	Edit menu option. (number of option, e.g. 1 is Edit general)
Database menu options	
Add an entry batch keywords (also Edit entry keywords where applicable)	
/DES	Description of entry
/COM	Comments about the entry
/REF	Reference

/STF	Storefile name (unique)
/XTYP	The type of X parameter required. (S - Single, R - Ratio)
/XSC	The type of X lookup table scaling. (Li - Linear, Lo - Log 10)
/FAM	The type of Family parameter. (S - Single, R - Ratio, N - None)
/VAR1	Variable 1 name (e.g. A)
/VAR2	Variable 2 name (e.g. B)
/VAR3	Variable 3 name (e.g. C)
/VAR4	Variable 4 name (e.g. D)
/GEOM	Geometry name (e.g. Holes)
/XP1	X parameter numerator
/XP2	X parameter denominator
/FP1	Family parameter numerator
/FP2	Family parameter denominator
/PLT	Plot File name
/LUT	Lookup table type (M - MDF, A - ASCII, P - Polynomial)
/LUF	Lookup file name
MDF lookup file add entry	
/FVn	Family value, where n represents the set number, and can be 1 to 10. (e.g. /FV1 = 0.5 for family value 1)
Polynomial lookup file add entry	
/FVn	Family value, where n represents the set number, and can be 1 to 10. (e.g./FV1 = 0.5 for family value 1)
/XMINn	The minimum X value for the lookup table (n is set number)
/XMAXn	The maximum X value for the lookup table (n is set number) |
/CnA0	The first polynomial constant (n is set number)
/CnA1	The second polynomial constant (n is set number)
/CnA2	The third polynomial constant (n is set number)
/CnA3	The fourth polynomial constant (n is set number)
Database menu options	
Delete an entry	
/ENT	The entry to delete (the first occurrence found of the entry will be used, when the full entry specification is not given)
/DEL	Confirmation required to delete entry (Y - Yes, N - No)

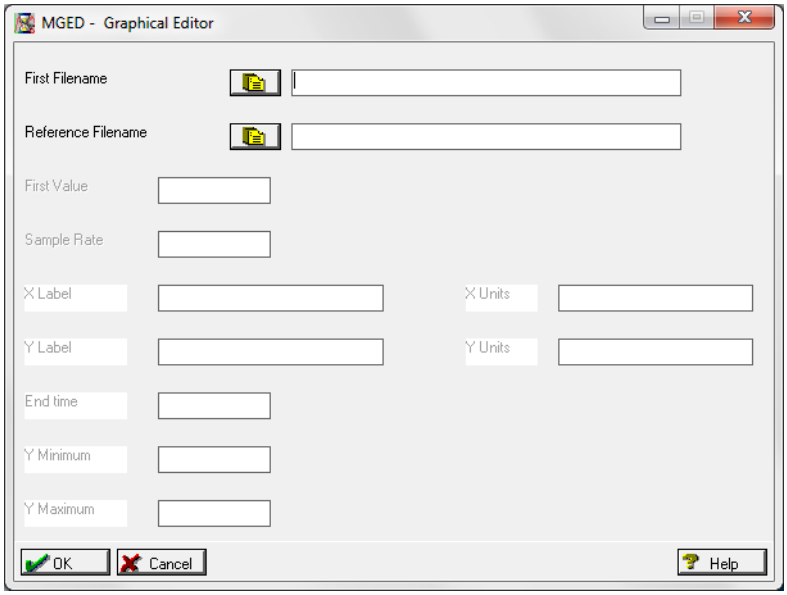
/KIL	Confirmation required to remove the user database (Y - Yes, N - No). This situation occurs when deleting the last entry in the user database, this will cause the user database to be removed as well as the entry. Selecting No will not delete the entry or database.
Edit entry menu options	
Edit general details - keywords same as add entry	
/NPLT	Replace Plot File. The new plot file name.
/NMDF	Extract to MDF file. The file name of the output MDF file.
/NMDF	Extract to ASCII file. The file name of the output ASCII file.
Replace Lookup tables with MDF	
/LUF	The file name of the MDF format lookup file.
/FVn	Family values for each set (n is set number)
Replace Lookup tables with ASCII	
/LUF	The file name of the ASCII format lookup file Edit
Polynomial constants - same as Add entry Polynomial lookup table	
Database menu option	
Pictures - not applicable in batch mode as the option draws geometry to screen.	
Calculate Menu	
/GTYP	The geometry type to list (use geometry type number)
/ENT	The geometry entry to use in the calculation of Kt

Graphics Display Utilities

Several display utilities and three hardcopy plotting/printing utilities exist for MSC Fatigue graphics.

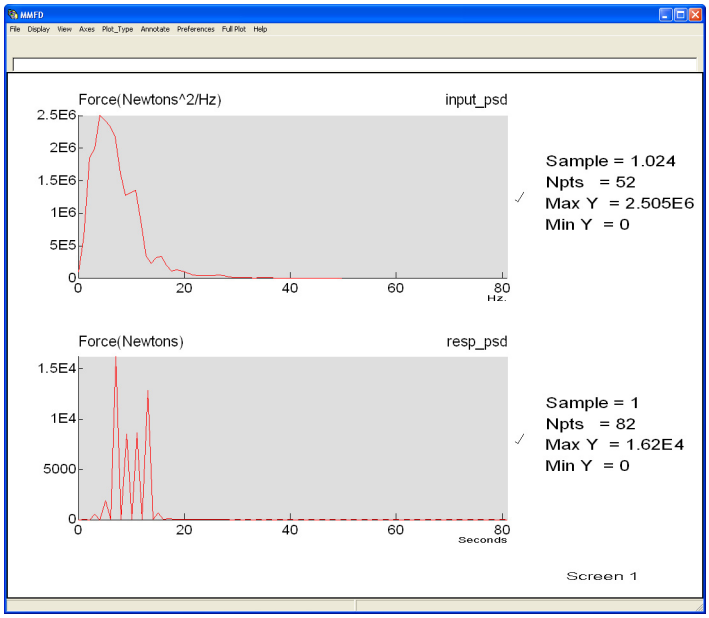
Graphical Editing - (MGED)

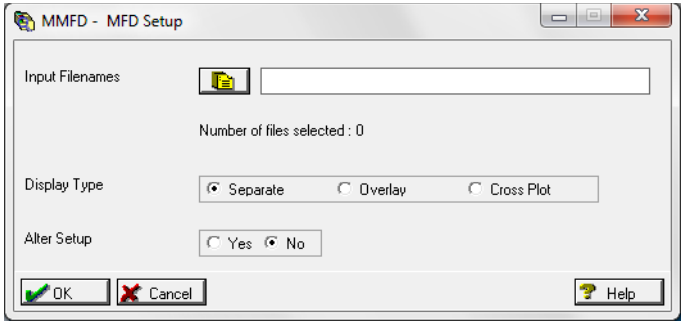
This module is the multi-channel interactive graphical editor for time series data allowing online manipulation of a signal. Tasks such as cleaning up bad data, creating data, extending a signal, spike removal, etc., are all easy and quick to carry out. This module can also operate in batch. For multi-channel edits it creates it's own NCL macro so that operations defined for one channel can be applied to all others, without the need to do them interactively. The assumption is that the other signals are from the same test or at least exhibit the same sample rate, etc.



Multi-File Display - (MMFD)

This module displays single parameter data files. The files may contain any type of sequential data including time series, power spectra, time at level distributions, etc. Files may be displayed across four screen pages, with a maximum of eight files per page. Thus, allowing up to 32 files to be presented. Three modes are offered for displaying the files on each page. They are: separate plots, overlaid plots, and cross-plots. Separate plots are those where each file is plotted independently of the others. Overlaid plots are where all of the files are plotted using common axes. Cross-plots are where one file nominally forms the X-values against which the other files are plotted on common axes.

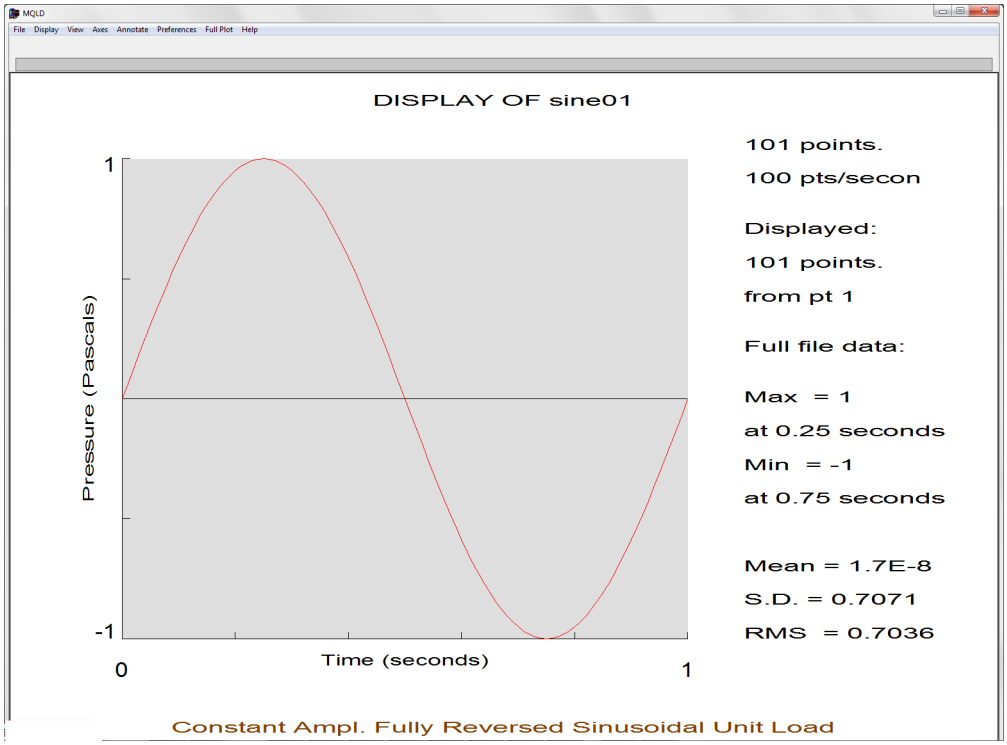




Note: See Multi-File Display (MMFD), 251 for a more detailed description of this utility.

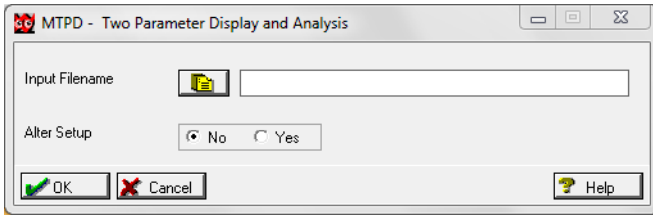
Quick Look Display - (MQLD)

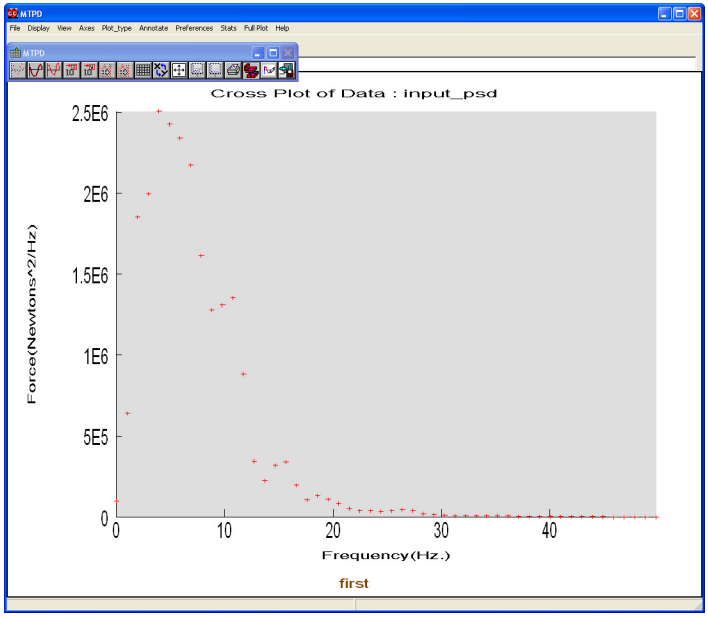
This module displays single channel data file. The file must be in the **.dac** format, which includes time histories, ASD results, ADA results, and any other results that have a constant X-axis increment. Use mTPD for pared (X-Y) data and mP3D for histogram and waterfall data.



Two Parameter Display - (MTPD)

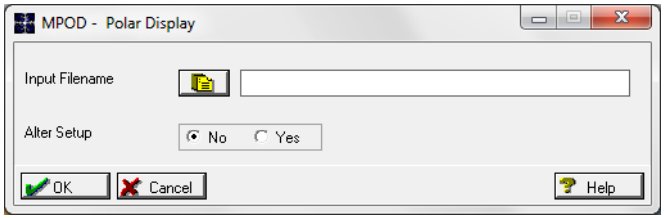
The two parameter display module displays pared (X-Y) data files. Displays may be scaled in various ways. Functions for windowing specific fields and picking off coordinate pairs are also available. After the data has been displayed, a menu will appear. Select your options and give the name of a data file to display. The file is assumed to be in the local directory and have an extension **.mdf**. If you wish to access another directory or use a file with a different extension, you will need to type in a fuller file specification.





Polar Display - (MPOD)

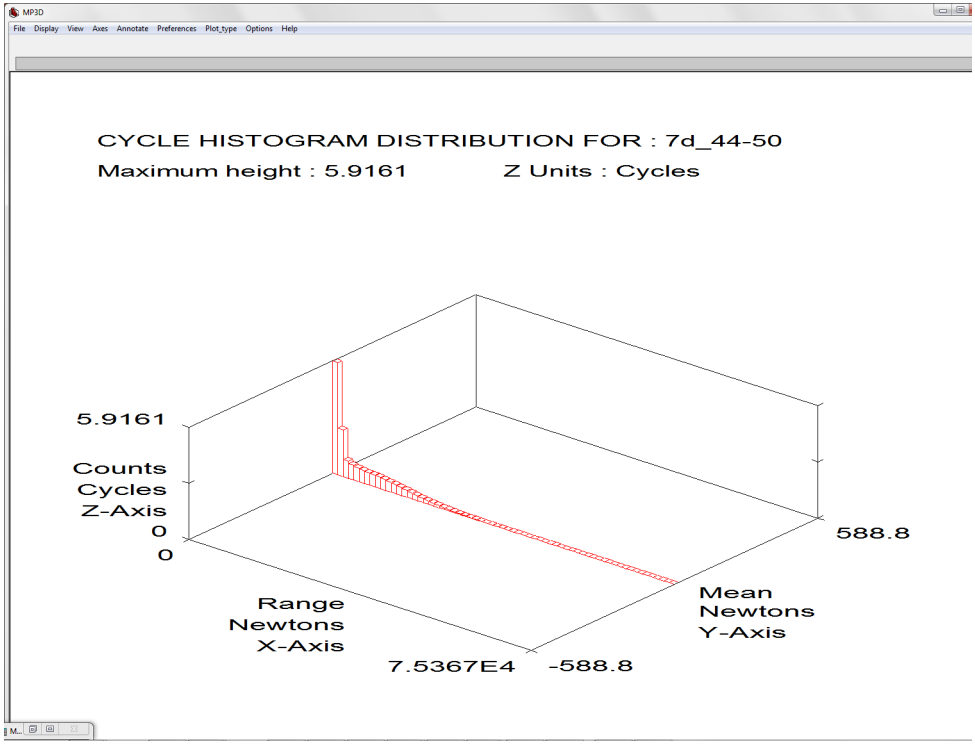
The polar display module displays pared (X-Y) data files. Displays may be scaled in various ways. Functions for windowing specific fields and picking off coordinate pairs are also available. After the data has been displayed, a menu will appear. Select your options and give the name of a data file to display. The file is assumed to be in the local directory and have an extension **.pod**. If you wish to access another directory or use a file with a different extension you will need to type in a fuller file specification.



Note: See Polar Display (MPOD), 820 for a more detailed description of this utility.

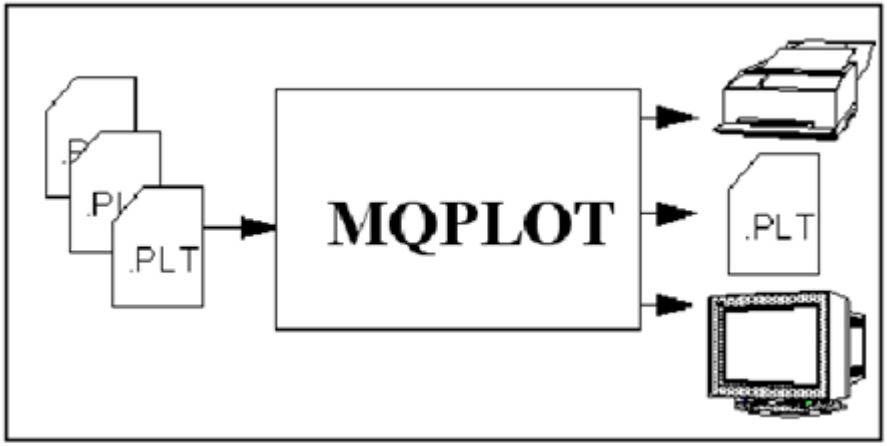
Three Dimensional Display - (MP3D)

This module is the histogram and waterfall display module. It accesses a standard nSoft histogram or waterfall file and provides a 3D graphical representation in the form of a tower, surface, or waterfall plot. The display can then be zoomed into, and positioning using rotation, tilt, and quadrant operations may expose hidden areas. For histogram files originating from fatigue analysis damage/cycles files may be plotted directly. In addition, mP3D will display the sum total occurrences of values along the X or Y-axis and display the result as a 2D plot. For waterfall files, 2D plots of X-slice and Y-slice may be produced.



Linux Based Plotting Utility - (MQPLOT)

MQPLOT is used to view or print plot (`.plt`) files. Up to 2000 plot files can be loaded onto MQPLOT and viewed on screen and/or printed or plotted as hard copy. A large number of output devices are supported. MQPLOT can also be used to convert plot files into different formats, for example `.plt` to `.eps` (encapsulated postscript).



MQPLOT is designed to be simple and very easy to use. MQPLOT can also set up the number of plots per page, landscape or portrait orientation, number of copies and whether the plot title should be on or off. When viewing, MQPLOT can scroll through plots by mouse or key press, alter plot titles, deselect plots, change print device and print. Plot files are produced by using the hardcopy option in any MSC Fatigue graphics module. Unless the plot is given a specific name, then they take the form `plnnnnnn.plt` where `nnnnnn` is a number starting at `PL000001.PLT`. The next value of `n` is stored in the local environment against the keyword $NEXTPLT. A plot file has a 'plotted' status, yes or no, so that when MQPLOT selects files to view or plot the user has the choice of 'All plots' regardless of whether they have been plotted or not OR 'All unplotted' which plots all unplotted files in the local directory. The F3/pick list shows the plotted status, and has additional options for sorting alphabetically or by creation date, for displaying the plot title or creation date and for deleting plotted plots.

In viewing mode, MQPLOT has a number of options including next/previous, scrolling by time display or key press, deselection and print. In plotting mode, a plot device can be selected from the index (`plotters.ind`) stored locally or centrally and the setup can be customized to set the number of plots per page (maximum 8), plot title inclusion or not, landscape/portrait orientation and number of copies. When plotting (using the MGRAPHIC program), a status display can be given.

Module Operation

MQPLOT can be run in one of three ways:

- From the **Graphical Display Utilities** pulldown menu under **Tools | MSC Fatigue** (for Patran) or under **Tools | Fatigue Utilities** (for).
- In stand alone mode by typing **mqplot** at the operating system prompt
- In batch mode

When running in interactive mode the first screen to appear is where the plot files are named. If they are to be output to a print or plotting device, the device is also specified on this screen. The fields are used as follows.

Option	Description
Select All Plots, all Unplotted, Select Plots	Select the input type for plots required: All Plots - All plot files in the local directory, all Unplotted - All unplotted files in the local directory, Select Plots - Select names from a list or enter via keyboard. Up to 2000 plot files may be selected.
File Names	The plot file names can be typed directly into the field. Multiple files can be specified, for example FILE(1-4) would cause MQPLOT to load FILE1, FILE2, FILE3 and FILE4. However, probably the easiest way to enter file names is by using the List button). When using the List facility, files can be tagged/untag (selected) in the usual way prior to acceptance but MQPLOT's top bar menu also has an options button for sorting and display. The Options are: ■ **Delete plotted** - Delete all the files which have been plotted and regenerate the list. ■ **Sort alphabetically** - Sort the list of file names alphabetically. ■ **Sort on creation date** - Sort the list of file names into creation date order. ■ **Creation date display** - Display the creation dates of all the files and which program created them. ■ **Plot Title display** - Display the plot titles of all the files. The default is to sort on data with date of creation display; when displaying title, the environment keyword QPDTIT is set to Y. The F3/List facility will not function if there are no plots in the current directory.

Option	Description
Output	This is a simple toggle and only one option can be selected, not both. If Screen is selected the files chosen in File Names will be displayed on the user's screen. For Device, select a device by double clicking on the name of that device. A list of devices must first be created using the Windows-Based Plotting Utility - (MWNPLOT), 1152 program. Use MPLTSYS to edit the list.
Device	If Device is selected then the Device field becomes active. It contains the user's choice, from a pick list of devices, that are available from F3 or the List button. The exact contents of the pick list is dependent upon what the user has installed using MPLTSYS.
Alter Setup	Selection of Yes will allow the following plotter parameters to be set. ■ **Plots per page** - from 1 to 8 (dependent upon the number of plots selected). ■ **Plot Title** - Whether the plot title should be output or not. ■ **Orientation** - Whether the plot orientation should be landscape or portrait. ■ **Status Display** - Whether the status display should be shown or not. ■ **Number of copies** - The number of copies of the plot requested When the plot system has been setup according to the user's wishes, pressing F1 or OK will cause plotting or display to occur. If plotting is to disk file or printer then the user takes no further part in the process. In the event of problems MQPLOT will attempt to print to the selected device for a period of time equivalent to the TIMEOUT variable as set in the environment, e.g. TIMEOUT=15 for a time-out of 15 seconds. The time-out default is 30 seconds. Use module MENM to set the TIMEOUT. See Modifying the MSC Fatigue Environment (MENM), 1463.

If Display is set to Screen, some additional menu items are available for graphical manipulation. Instructions for using the graphics menu and command line interface are typical to other MSC Fatigue graphical programs and not detailed here. The full menu system and list of command line codes are shown below.

Most of the codes are standard and therefore explained elsewhere, but two of particular interest to MQPLOT are NX and PR. Pressing NX (Next) and PR (Previous) allows the user to scroll forwards and backwards through up to 50 plots, which is just about all that MQPLOT is intended to do in view mode.

Menu Option	Prompt Code	Meaning
MAIN		The Main (and only) menu
Next Scrn	NS	Show the next plot in the queue
Prev. Scrn	PS	Show the previous plot in the queue

Device	DV	Go back to the figure 2 screen to set the device
Scroll	SCR	Causes MQPLOT to automatically page through selected plots without the user having to press Previous or Next
Alter Title	AT	Change the plot title
Deselect	DES	Remove the currently selected plot from the list
Print Setup	PSU	Go back to the screen to set the device
Print	PRI	Print a plot `.plt` file to disk
Return	RT	Return to the file name input screen of MQPLOT
Exit	EX	Exit out of MQPLOT
	CU	Go into cursor mode
	OP	List the options codes

Batch Mode

A list of MQPLOT batch keywords:

/INPut	Plot file name(s) to print, display, or 'print' to disk (convert). /INP=FILE(1-9)
/TYPe	Type of output, Screen or Display (S or D). /TYP=S
/NAMe	Name of the plotting device. /NAM=HPGL
/SELect	Select the input type for plots A (all), U (unplotted), S (select). /SEL=A
/ALTer	Alter Setup Y,N. /ALT=Y
/NUMber	Plots per page, 1 to 8. /NUM=4
/TITle	Plot Title Y,N. /TIT=My plot
/LAYout	Orientation P,L. /LAY=L
/COPies	Number of copies. /COP=2

Windows-Based Plotting Utility - (MWNPLOT)

This application allows MSC Fatigue plot files to be selected, previewed and plotted to any device supported by Windows. The software is unprotected and may be loaded onto many computers simultaneously. A typical application might be on a PC networked to a Linux installation, in which case plot files, after conversion by MCONFIL to PC format, may be sent from the work station to the PC for inclusion within a document being created by a word processor.

Initially screen displays are rendered in the same colors as set within each MSC Fatigue graphic module. Prior to printing it is advisable to select a white background from the preferences menu.

Graphic diagrams can be imported, via the Windows clipboard directly into other applications such as word processors. Better resolution can be achieved by copying to the clipboard via the meta-file format rather than a bitmap.

Text may be reproduced by means of vectors (characters drawn by means of lines), or the fonts supported by specific printers.

This is an interactive only program and can be invoked by typing WNPLOT at a system prompt of by access from the **Graphical Display Utilities** pulldown menu under **Tools | MSC Fatigue** (for Patran) or under **Tools | Fatigue Utilities** (for).

The File Pulldown Menu

Option	Description
Open	This option allows one or more plot files to be selected and loaded ready for viewing. A standard Windows File Open dialogue box is used to select the required path and files. Dragging the mouse whilst pressing the left hand button will mark several files for selection (although only one at a time can be viewed). WNPLOT remembers the name of the last directory from which files were loaded. Therefore storing your plot files in the same plot file directory will reduce the need to change directory. Having selected one or more files (up to 1000 can be selected), click the OK button to load and display file(s). Pressing the print button will cause the currently displayed plot to be sent to the current plotting device as specified in Windows. Select the multiple print option in order to print a set of selected files.
Print...	Uses the standard Windows Print folder
Print Setup...	Uses the standard Windows Print Setup folder
Page Setup...	Allows the size and paper source to be selected (printer and paper must be available), and the orientation (Portrait/Landscape), and the paper margins.
Load Settings	Just about anything that can be set from the other menu and button options, e.g. font, font size, color, metafile vector/bitmap clipboard saves, etc. can be saved in a settings file (using the Save Settings As option). Such files can be retrieved and applied to the current plot(s). See also Pen Colors Setup folder. Settings files have `.set` extensions. See also the `nplot.ini` file.
Save Settings As	The current settings, e.g. font, font size and color, metafile vector/bitmap clipboard saves, etc. can be saved to a settings file for future retrieval and application by the user. In this way a library of settings can be created that are applicable to specific plots via the Load Settings option. When saving settings files, save them with their chosen name and the `.set` file extension, e.g. `myprefs.set`. Also, take care to save the files in a directory in which you can find them in the future.

Options Menu

Option	Description
Save Settings on Exit	This option allows preferences such as line thickness, pen colors and background colors to be saved on exit so that when the application is restarted, they are immediately available.
Auto Redraw	Some plot files may be quite large and, therefore, require some time to plot on screen. Under these circumstances it may be desirable to select several preferences between each repainting of the screen. Set Auto-redraw to off to disable automatic re-draw after each option. The display will be updated only when the View button is pressed.

Clipboard Command

This option copies the currently displayed plot file to the clipboard from which it can be pasted into any other suitable Windows application such as a word processor.

> **Note:** The Preferences-Settings-Clipboard Settings folder must not have the Save to File check button ticked (on).

Or...

Save as .WMF

This option copies the currently displayed plot file to either the clipboard, or to a file, as a Windows metafile (`.wmf`). See More details on Bitmaps and Metafiles for details of the metafile types available.

> **Note:** The Preferences-Settings-Clipboard Settings folder must have the Meta File radio button set to on. See More details on Bitmaps and Metafiles for details of the bitmap types available.

Or...

Save as Bitmap (.bmp)

This option copies the currently displayed plot file to either the clipboard, or to a file, as a bitmap. See More details on Bitmaps and Metafiles for details of the bitmap types available.

> **Note:** The Preferences-Settings-Clipboard Settings folder must have the Bitmap radio button set to on.

The Preferences Menu

Background Color...

By default, this application reproduces graphic images in the same color scheme as the original graphic modules; which means that the background color is generally black. See 'Pen Colors Setup' folder also.

This option allows an alternative background color to be chosen. If the save settings option has been selected, then the color setting will be saved.

Settings... This reveals a form with four different folder options:

Font Table Setup folder

Having elected to display textual information through the use of the Windows system of fonts, this option allows specific fonts to be assigned to specific textual areas of the display. The font uses are as follows:

1. Default Font
2. Title Font
3. Annotation Font
4. Axes Labels Font
5. Axes Numbers
6. Text Font
7. to 16. User Defined

The Edit... button pops up the font folder that allows the font, font size, color, and style, to be set for any Font Description. The settings can be saved to a `.set` file (see Save Settings As).

If a `.set` file is loaded (see Load Settings) it can be used in future runs of WNPLOT until overridden by the default settings (see below) or by the loading of another `.set` file.

The Default button loads the defaults, regardless of the current settings.

The Copy all button imposes the defaults onto the current plots.

Note that if text has been rotated by the original graphic application then a True Type font must be selected if the text is to be rotated on the hardcopy also.

Pen Colors Setup folder

The palette from which pens can be assigned colors is set here. There are 16 color slots available from which pens can be assigned colors and the total number of colors from which to choose can be from 256 to 16 million (depending upon the PC's hardware and settings).

The colors specified will affect the screen display, and most color plotters. Use the following table as a guide to pen number and function:

- pen 1 Default
- pen 2 Data plots
- pen 3 Text (only applicable to vector text)
- pen 5 Axes including all labels
- pen 6 Annotation text (only applicable to vector text)
- pen 8 Grids within graphs

Clicking the edit button pops up the color palette from which colors can be selected.

Clicking the define custom colors button on the color palette pops up the windows color picker. Clicking anywhere on the spectrum of colors causes that hue to be assigned to the 16 available. Adjacent to the spectrum is a slider that controls the lightness of the hue

The Default button of the pen colors folder makes the current settings (including changes) become the new default.

Monochrome devices will not respond to color definitions although up to four colors are automatically simulated in mono by different line styles. Line thickness settings are effective in color and mono.

Line Thickness

The thickness of the line to be drawn can be set up from this dialogue box. Note that when text is set to fonts the line thickness setting of fonts has no effect (Windows controls the font thickness), however, lines and axes are still affected.

Clipboard Settings...

The Clipboard Settings folder - Creating and defining graphics files.

Images can be copied as `.bmp` bit-maps using the current screen resolution, or in the form of Windows Meta File vector files with `.wmf` extension. With vector files the resolution matches itself to that of the plot device.

Bitmaps will have `.bmp` file extensions, and windows meta files `.wmf` extensions.

Bitmap

The default dimensions of the image to be copied to the clipboard have been set to provide a graphic which occupies about a third of a page of A4 paper, with an aspect ratio similar to the original screen image, i.e. 400 x 400 pixels. If a different size or aspect ratio are required then alternative settings can be set within this dialogue box.

Bitmap images can easily be edited in graphics editors such as Windows Paintbrush. Vector formats such as `.wmf` tend to be less easy to edit but their variable and usually superior resolution makes them very popular for CAD and presentation applications.

Meta File - Scalable

The clipboard stores the current graphic to RAM memory, or a disk file, depending upon whether Save to File is ticked. Selecting Settings leads to a Settings window for the clipboard type (bitmap or metafile).

More details on Bitmaps and Metafiles.

Save to file - When Save to File is On, clicking Copy to Clipboard causes the Save Bitmap, or metafiles save folder, to appear. Their File Type buttons allow any of the above graphics types to be the save format.

Miscellaneous folder

16 Colors/Monochrome

For monochrome displays, such as those found on certain lap-top computers, it is often preferable to view images in monochrome. Selecting the monochrome option will reproduce the image in monochrome on both the display and any color plotting device.

When printing to a black and white printer, e.g. most laser printers, it is advisable to set Preferences to monochrome; results tend to be clearer (black on white has the highest contrast of all) and monochrome tends to use the least toner.

If a plot was originally created with several lines differentiated in different colors, and mono is chosen then up to 4 line colors are represented by different line styles.

Text Preferences - Fonts/Vectors

Text may be displayed on screen and on a hard copy by representing each letter as a series of vectors (lines) which may limit their resolution, or through the use of the Windows font system. The latter invariably provides a more satisfactory result which is generally much more compatible with the results obtained from word processors and graphics packages etc.

One possible problem area in is text rotation whereby certain Windows print drivers may rotate text in an unforeseen way. In such circumstances it will be necessary to use vector-based text.

Zoom Preferences

This option sets the mode of operation of the zoom buttons.

When set to Magnify, text or plot lines remain the same absolute thickness, and thus appear progressively thicker as you zoom in.

When set to More Detail, the text and lines remain the same relative thickness, thus dense plots with many points etc. can be easier to resolve.

The Button Bar

The function of most of the buttons is, given by their titles, self explanatory. In cases where the function is not so obvious it is explained.



 File Open - pops up the select file pick list and allows one or more plots to be loaded, i.e. it has the same effect as the Open command on the File Menu. WNPLOT remembers the name of the last directory opened.

 The navigation and control buttons are analogous to those found on a video recorder or audio tape player. They allow the user to page through a list of plots, jump quickly to either end or beginning of the list, and start/pause the slide show. Some navigation keys have keyboard equivalents as described in the topic Active Keyboard Keys.

 Slide show Time Delay. The time delay in seconds for which plots are displayed during a slide show.

 Redraw screen. Redraws the current plot, i.e. to remove any drawing marks or to put into effect a series of commands that are pending because redraw is set to off (on the options menu).

 Copy to clipboard. Copies the current plot to the clipboard (see also clipboard settings on the clipboard menu).

 Print. Pops up the Windows Print dialogue box.

 Full Screen. Make the plot occupy the full screen. Navigation keys is still possible by using certain keyboard keys as described in the topic Active Keyboard Keys.

 Zoom in by a factor of 2. Up to 4 zooms are possible. See also the Zoom settings on the preferences menu. Zoom out by a factor of 2. Up to 4 zoom-outs are possible. See also the Zoom settings on the preferences menu.

Drawing on the plot

The mouse cursor becomes a drawing tool pen if it is pressed and moved when it is over the plot area.

The pen is intended only for use as a marker or presentation tool, e.g. to temporarily highlight an area of interest on the plot. None of the marks or writing drawn on the plot can be saved.

The pen is red when preferences are set to monochrome, and when set to color the pen is the reverse of the background color (which is also set in preferences).

Active Keyboard Keys

Several control buttons have keyboard equivalents:

- Home - Jump to the first plot in the list.
- Page Up - Go back to the previous plot.
- Page Down, Enter or Return - Go forward to the next plot in the list.
- End - Jump to the last plot in the list.

More details on Bitmaps and Metafiles

The minimum of information that must be set is the height and width of the bitmap. However, bitmaps can be specified in much greater detail by toggling 'Save to file' on and then clicking the File Options button.

The bitmap options are as follows:

- RGB - Standard bitmap format that stores the palette
- Bits per pixel - 1 = mono, usually black or white, 4 = 16 colors (4 bits=16 in base ten). If the graphic has >16 colors then Windows picks the nearest color of the 16 and substitutes it - this can produce unusual results), 8 = 256 colors, 24 = True color (16million colors)

- RLE4 - Run Length Embedded bitmap (4 bits 16 color)
- RLE8 - Run Length Embedded bitmap (8 bits 256 color)

The RLE formats are readable by many graphics applications.

Metafiles types can be specified when the Save to folder appears after Clipboard-Copy to Clipboard has been clicked. The types are:

- Scalable - Aldus placable - a very common format that includes the header (which makes it scalable). This format is readable by 16 bit applications such as Windows 3.1programs.
- Enhance - New to Windows 95 and NT - 32 bit only and non 3.1 compatible.

The nplot .ini File

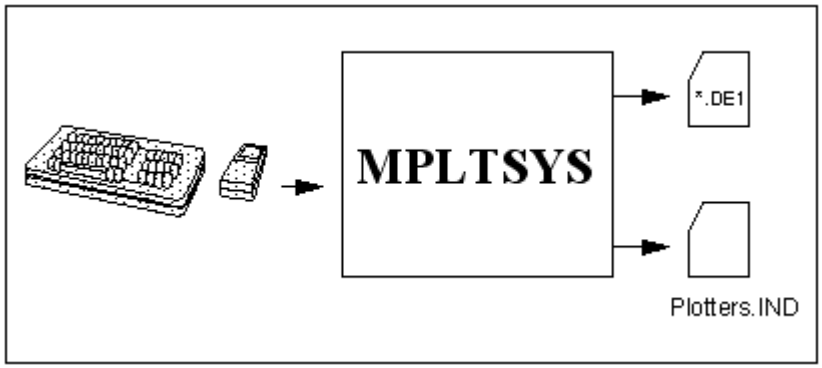
Settings for fonts, pens, the clipboard, and general settings, are all stored in the `nplot.ini` file. This is a windows file that is loaded by windows immediately before WNPLOT, and sets many of WNPLOT's characteristics.

It is an ASCII text file that resides in the `winnt` directory, and it can be edited by users. Make a backup copy of `nplot.ini` before editing it. An example of a `nplot.ini` file is given below.

```
[GENERAL]
SaveSettingsOnExit=Yes
AutoRedraw=Yes
Colour=Yes
Fonts=Yes
Slidetime=5,
Zoom=2
workdir=c:
[CLIPBOARD_SETTINGS]
Bitmap=1
Limits=400,400
BitmapFileType=1
BitmapBits=8
[FONTS]
Font1=20,10,0,0,400,0,0,0,0,1,2,1,26,Arial
Font2=20,10,0,0,400,0,0,0,0,1,2,1,26,Arial
Font3=20,10,0,0,400,0,0,0,0,1,2,1,26,Arial
Font4=20,10,0,0,400,0,0,0,0,1,2,1,26,Arial
etc...
[PENS]
Pen1=64,128,128
Pen2=64,0,64
Pen3=0,0,0
Pen4=0,0,0
etc...
[PEN_WIDTH]
Pen1=5
Pen2=1
Pen3=1
etc....
```

Printer/Plotter Definition Module - (MPLTSYS)

The MSC Fatigue system can produce graphical hardcopy on a wide range of output devices. These include pen plotters such as the wide range of HP devices, laser printer-plotters, and dot matrix printers.



Before any hardcopy can be produced, the plotters that are available on the system must first be defined.

The plotter definition system controls all aspects of defining the available plotters, and their setup (plot size, position etc.). MPLTSYS allows definitions to be created, modified deleted and listed.

The definition of a plotter includes two parts, description and setup.

The description includes the name that the plotter will be known by, a general description of the plotter its output port or file extension, and whether it is currently available for use. It is possible to have a defined plotter that is currently unavailable for some reason. If the plotter is marked as being unavailable, users will not be able to send hardcopies to it.

The setup includes details of the required plot size, position on the page, number of copies, physical connection between plotter and computer and other things.

Two types of setup are possible: central and local. The central setup for each plotter is held in a central location and is used by all users. When a definition is first added, the setup data are stored centrally automatically. The modification option allows a local setup for a plotter to be created or modified as required. The local setup is held (permanently) in the local directory. There can therefore be as many local setups for each plotter as there are local directories.

When a hardcopy output is produced, any settings found in a local setup override the corresponding settings in the central setup. Individual users may therefore have plotter setups which are modified over the standard central setup without having to modify the central setup.

Up to 16 definitions may be stored. This can be 16 different setups for a single device, a single setup for each of 16 different devices, or any mixture between these extremes. The index of defined plotters is held in the file `<install_dir>\ mscfatigue_files\nssys\plotters.ind`. The central setups for the defined plotters are held in the same directory in files of type `.de1`. Local copies may be held in the local directory under the same name as the central copy.

A schematic overview of the commands and options within the plotter definition system is shown in Figure 14-47.

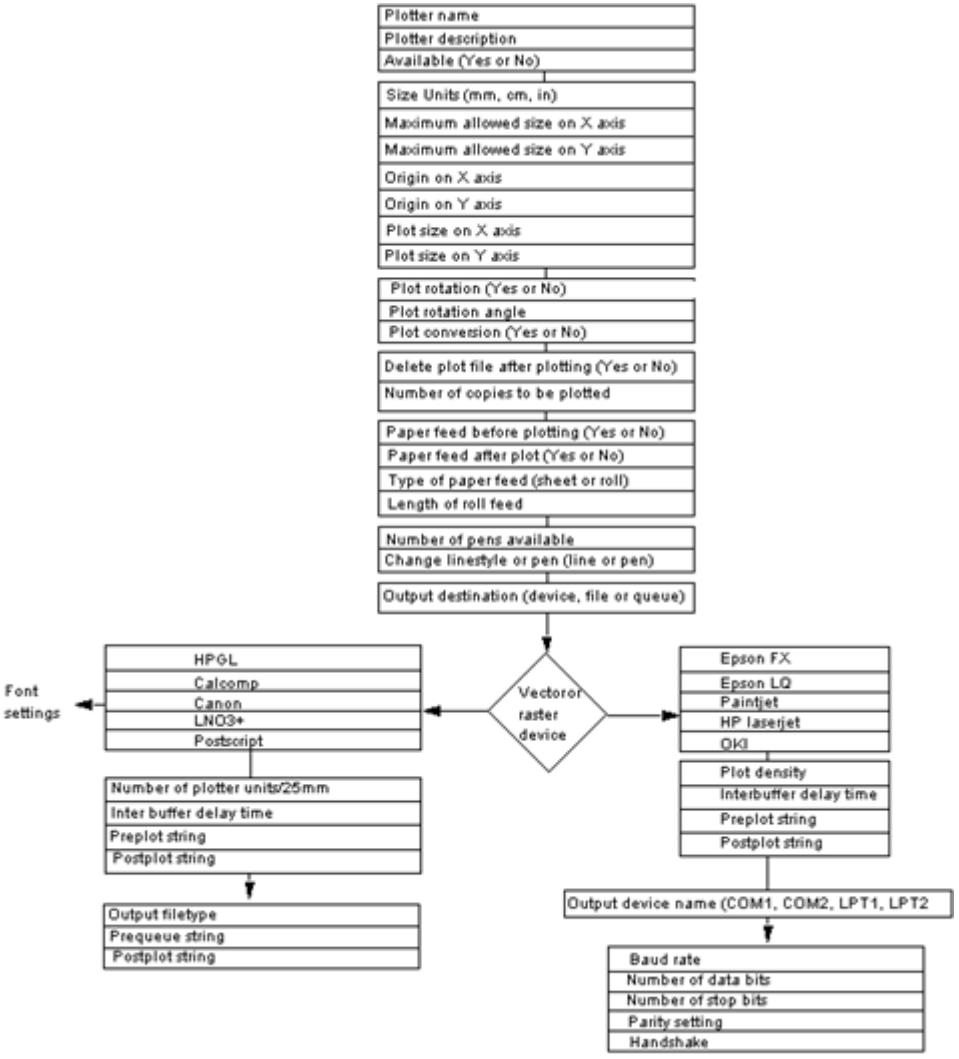


Figure 14-47 Possible Plotter Definitions

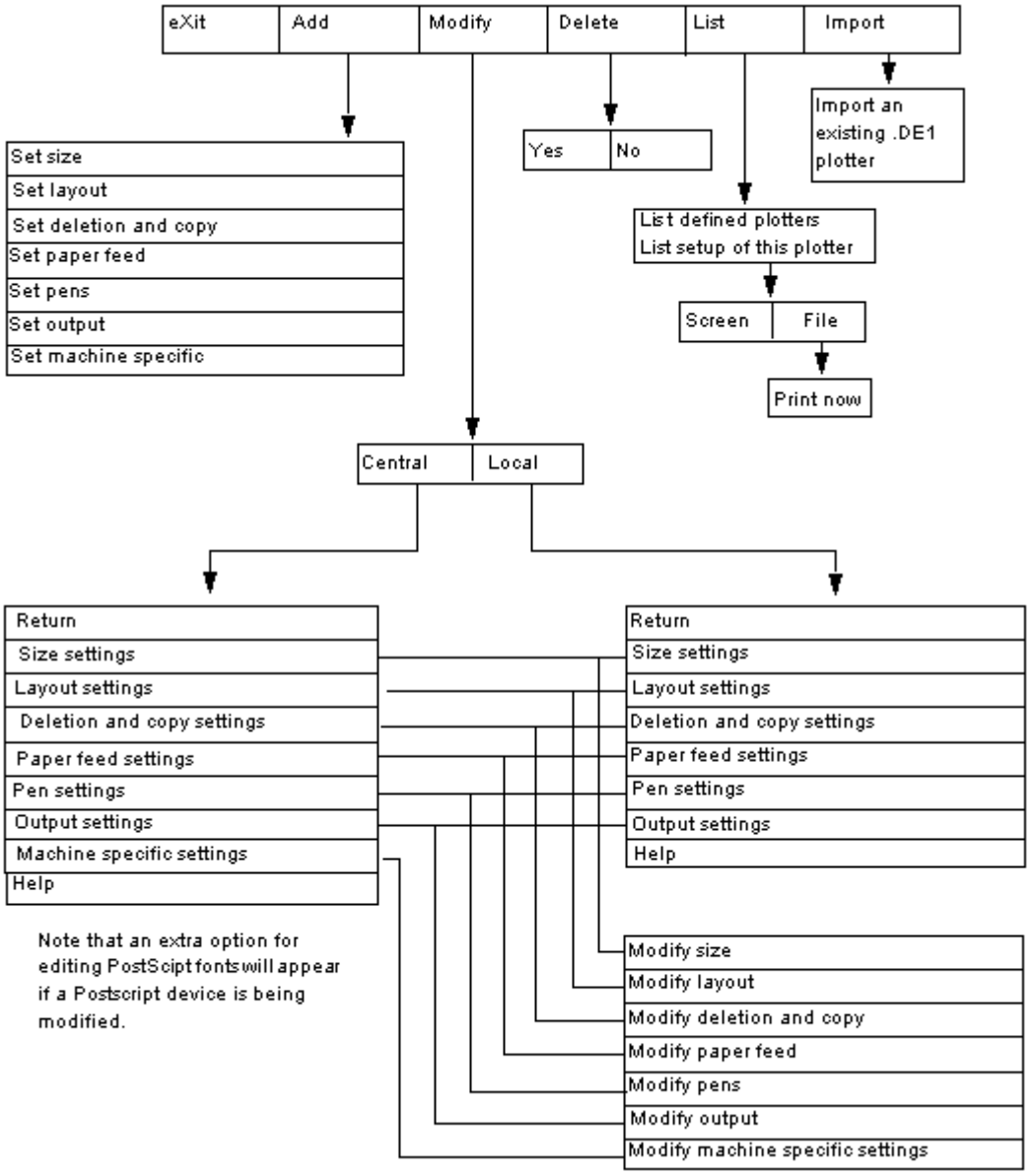


Figure 14-48 MSYSPLT Module Overview

Module Operation

The MPLTSYS module can be started in one of the following two ways:

- From the **Graphical Display Utilities** pulldown menu under **Tools | MSC Fatigue** (for Patran) or under **Tools | Fatigue Utilities** (for).

- In stand alone mode by typing **mpltsys** at the system prompt

The main options screen displays the defined plotters in a vertical list. For each plotter its name and description are displayed. The following options are displayed:

- eXit
- Add
- Modify
- Delete
- List
- Import

The Modify, Delete, and L options occur on the currently highlight plotter definition.

These individual options are be described below except those that are self-explanatory.

Add a Printer/Plotter

Select the main option Add to add a plotter definition to the existing list. You must enter these three pieces of information:

1. **Plotter name** - The plotter name must be unique in the list of defined plotters, and is only a name. It does not need to be the physical device name. For example the device could be a CANON laser printer-plotter. In this case you do not need to specify the name as CANON. MPLTSYS does not know that it is a CANON type device by its name. You could call this plotter PLOTTER1, for example. Note that the listing page will list only the first 6 characters of the name.
2. **Description** - The description field can be used, for example, to identify the setup of this plotter. You could enter CANON plotter - A3 paper. Users would then know that A3 size paper is loaded into this device.
3. **Available** - Select available or not as appropriate. If you select that the plotter is not available, it will not be displayed on the list of plotters that graphical output can be sent to by the plot control system.

Once input is complete, press the **OK** button to save the data. At this point the name will be checked for uniqueness. If a matching name is found an error will be displayed and the data will not be saved.

The program will then proceed through the following sections in turn:

- Plot size selection
- Plot orientation selection
- Plot file disposal and number of copies selection
- Paper feed selection
- Pen selection
- Plotter type selection
- Plotter control parameters
- Plot output destination selection

Plot Size Selection

Option	Description
Units of size	The first field allows you to select the units of size used in the setup to be millimeters, centimeters or inches.
Maximum Sizes on X and Y axes	The maximum allowed sizes are for informational use only. No check is made against the values entered when the hardcopy is produced.
Origins	The X and Y origins are expressed relative to the mechanical origin of the plotter. The relationship between the mechanical origin of the plotter and the paper is not always clear. You can examine this relationship by setting the origin values to 0, making a plot and seeing where the plot begins on the paper. You may enter negative values here if required.
Plot Sizes on X and Y axes	The X and Y-axis plot sizes are measured from the origin points specified above.

Plot Orientation Selection

This section allows you to rotate or invert the plots. This setup section is included only for compatibility with previous releases of the software. It is better to use the Landscape/Portrait options available in the plot control system.

Option	Description
Plot Rotation	The plot can be caused to be rotated about the origin by answering yes to this question.
Plot Rotation Angle	If you specify Yes as the response to the plot rotation question, you will be allowed to enter a rotation angle. The rotation angle is expressed in degrees and must be positive. All rotations are taken from the origin in an anti-clockwise direction.
Plot Inversion	If you select inversion, the plots produced will be rotated by 180 degrees about their centre.

Plot File Disposal and Number of Copies Selection

Option	Description
Delete Plot File After Plotting	When hardcopy is selected in the display program, a plot file is produced containing the device-independent codes required to produce a plotted hardcopy. By selecting Yes as the response to the "delete plot file after plotting question" these plot files will be deleted after the plot has completed successfully.
Number of Copies to be Plotted	The number of copies produced for each plot file may be selected on this screen by entering the required number here.

Paper Feed Selection

You may select to have a paper feed before and/or after each plot, or no paper feed at all. If you specify no paper feed, the plot control system will prompt you to enter paper before each plot is produced.

Option	Description
Paper Feed Before Plot	Select Yes here to cause a paper feed to occur before the plot is produced on the plotter.
Paper Feed After Plot	Select Yes here to cause a paper feed to occur after the plot is produced on the plotter.
Paper Feed Type	You may select either sheet or roll paper feed. On dot matrix printers where the device is fed by continuous stationery, you should specify this as sheet, not roll. If you select roll paper, you must enter the length of paper to be rolled out.

Option	Description
Length of Roll Feed	If you have selected that the paper feed type is roll, you must specify the length of the feed required.
Pen Selection **Number of Pens Available (maximum 8)**	The pen selection screen allows you to specify the number of pens available, and whether pen or line style change is to be used on an overlaid data plot. In the graphical display software, it is possible to produce overlaid and cross-plotted data graphs. It is important to be able to differentiate the different data sources on the plotted hardcopy. The graphical hardcopy device drivers use the information on this screen to decide how to display this type of data. The driver has two options, use different pens (colors or thickness) or different line styles. Different line styles are achieved by drawing lines made up of different length dashes and spaces. The procedure that the hard copy driver uses for determining what it will do is as follows. If you specify that the device has multiple pens and that pen change is to be used, the device driver will use the different pens to display the different data lines. If, for example, you specified that the device had 8 pens, and you plotted an eight data line overlaid plot, each line would be drawn with a solid line using a different pen, on the assumption that each pen is a different color. If you specified 4 pens with the same data plot, the first four lines would be drawn as solid lines using the four pens, the second four lines would be drawn as dashed lines (all the same pattern) using the four pens. If you specified 3 pens with this plot, the first three lines would be drawn solid with the three pens, the second three would be drawn dashed (pattern 1) with the three pens, and the last two would be drawn as dashed (pattern 2, different to 1) with the first 2 pens.
Change Line style/Pen	If set to Pen with multiple pens available, then each pen will be used. If you specify one pen, or multiple pens with line style change instead of pen change, each data line would be drawn primarily in a different line style rather than a different color. The facility to force multi-pen plotters to use line style change can be useful in the case where the plot will need to be photocopied, when different colors may not show up on the copies.

Plotter Type Selection

Vector devices are those devices which can directly understand a vector command. That is, they can directly process a command to draw a line from A to B, and can process these commands in any order. Vector devices include pen plotters, some laser printer plotters and postscript type devices.

Raster devices are devices for which the vector command must be translated before it can be understood. These devices are impact printers and some laser printers. Consider a simple impact printer. In order to draw a line from the top to the bottom of a piece of paper, the device must print one dot per line moving down the sheet.

If the next command were to draw a line back to the top of the paper, the printer could not do this because it could not move the paper backwards. Instead, the graphical hardcopy driver must split the plot into a series of strips arranged vertically down the paper, and ensure that all the dots required in the plot in each strip are produced whilst the print head is on this line. This means that the plot file must be processed several times to actually produce the plot.

Pen color settings in the hardcopy environment:

This document gives some help and advice on the use of pen settings within the hardcopy environment. The standard colors are: 1 - Black, 2 - Red, 3 - Green, 4 - Blue, 5 - Cyan, 6 - Yellow, 7 - Magenta, 8 - Brown

To change the pen colors, you need to create a file in the `<install_dir>/mscfatigue_files/nssys` directory which has the same name as your plotter name, but with an extension of .pen. For example, if you are using a plotter called `deskjet`, then create a file called `deskjet.pen`. This file, which can be created with any editor and should contain lines which specify a new pen number and the associated pen color required. Each line in the file should be in the form:

```
<pen number>=<color>
```

where the pen number is the number of the pen to change, and <color> is the new color, entered as a name. So, for example, if you wish to replace yellow with black, enter a line as follows

```
6=black
```

The color name is NOT case sensitive. Comment lines can be included in the file by having a ! as the first character in the line.

Plotter Type Selection = Vector

If you select vector, the toggle field shows the list of supported vector plotters (one of which must be selected).

Option	Description
HPGL	This is a generic entry which includes any device that implements the Hewlett-Packard Graphing Language (HPGL). This includes HP devices themselves, but since HPGL has become a *de facto* standard, devices from many other manufacturers (other than HP) support this protocol.
CALCOMP 81/84	These are A3/A4 size plotters driven by Calcomp graphing commands. As with HPGL, devices from other manufacturers (other than Calcomp) may support this protocol.
CANON (Vector)	This entry refers to the CANON LBP8 series of laser printer/plotters that include the VDM (Vector Draw Mode) facility fitted either as standard or as an option.

Option	Description
DEC LN03+	This is the DEC LN03 laser printer/plotter fitted with the Tektronix graphing option.
GRAPHTEC 4731	For driving the Graphteck 4371 continuous paper plotter.
PostScript	This is a generic description including any device that implements the PostScript language. The implementation of PostScript enables different fonts to be printed on a plot, and also has two additional font selection screens, and for this reason merits further explanation.

For PostScript printers:

Option	Description
Text Output - Fonts | Vectors	If set to *Fonts,* the plotter definition being edited can potentially make use of all the PS fonts in the currently loaded cartridge. If set to *Vectors* then the PS fonts are ignored and the default font is used in vector format - this can be useful if problems are being experienced with Postscript settings.
Edit Font Table	If set to Yes then the font table can be edited. For example, the setting 2 (the Title font) can be set to Palatino-Bold, which is a more stylish font than Courier, and the height scale factor Y is 1.5 which makes the title text 50% taller than the other text. Scale factor can also be used to adjust inter-letter spacing.

What do the numbers designate?

The usage of each numerical designation is shown below.

1. Default Font
2. Title Font (For example File name)
3. Annotation (User added text on particular plots)
4. Axes Labels Font (i.e. X and Y-axis titles)
5. Axes Numbers Font (i.e. along X and Y-axis on plots)
6. Text (i.e. stats)
7. To 16 User Defined (Not Used)

What fonts are available in my printers PostScript Cartridge?

A file called `fntlst.pos` should be located in the `<install_dir>/mscfatigue_fiels/nssys` subdirectory. If it is sent to a PS printer then it will cause it to print a list of all the fonts available in the currently loaded cartridge.

Note that when entering a font name in the User Defined column that PostScript cartridges are *very* case sensitive; type the name *exactly* as it is shown in the font list, i.e. ZapfDingbats is not the same as Zapfdingbats

because the incorrect lower case 'd' would cause the whole name to be unrecognized (which would cause the default Courier font to be used instead).

Standard PostScript fonts which can be used on any PostScript device are: Courier, Courier-Oblique, Courier-Bold, Courier-BoldOblique, Times-Roman, Times-Italic, Times-Bold, Times-BoldItalic, Helvetica, Helvetica-Oblique, Helvetica-Bold, Helvetica-BoldOblique

These are available by toggling on any cell in column 1 with the space bar. Only Courier has a constant letter width, the other fonts have variable letter widths, i.e. whereby an 'm' is wider than an 'i', etc.

Note:	Scientific notation, e.g. Greek symbols, is available with the Symbol or ZapfDingbats fonts on most PostScript cartridges.

Plotter Type Selection = Raster

If you select raster, the list of the supported raster plotter types will be displayed in the toggle list, one of which must be selected.

Option	Description
EPSON FX	The graphical hardcopy driver is capable of driving EPSON FX printers at three different print densities.
EPSON LQ	The graphical hardcopy driver is capable of driving EPSON LQ printers at four different print densities.
HP PAINTJET	The hardcopy driver is capable of driving the Hewlett Packard paintjet plotter at two different plot densities.
HP LASERJET II	The hardcopy driver is capable of driving the Hewlett Packard Laserjet II plotter at four different plot densities.
OKI 290 Series	The hardcopy driver is capable of driving the OKI 290 series color printer at three different plot densities.
HP LASERJET III	For driving the HP Laserjet III. These printers are faster than HP II s' because compressed files are transmitted.
Deskjet	Use HP laserjet III driver and choose 7 pens for color deskjet or 1 pen for a monochrome deskjet.
Epson Escape P2	Use for Epson stylus printers. Choose 7 pens for color printer or 1 pen for a monochrome printer.

Note:	Raster devices can only be output to a device, not to a file or a queue.

Plotter Control Parameters

The final set of data required in the plotter setup could be referred to as device specific data. The first is for vector devices, the second for raster. The only difference between them is the first field on the screen. For the vector device it is number of plotter units per 25.4 mm (one inch), for the raster it is a selection of plotter density.

Option	Description
Internal Scaling (Plot Units per 254mm and Plotter Density)	For a vector device, the plotter units per 25.4 mm field will always appear with a default value which varies according to the plotter type selected. It should not be necessary to change this value. The ability to change it is included to cope with the situation where a device may emulate another device type, but with a different internal scaling. For a raster device, a selection of plotter densities will appear, also varying with the plotter type selected. The density figures shown will not necessarily relate directly to any printer documentation in respect of their absolute values. For example, the three densities available for the EPSON FX are 60 120 and 180, and these are not the dots per inch figures published in the EPSON manual. Instead, the figures should be used in the mode of low, medium. and high. The higher the value the more dense (and better-looking) the plot will be. Also, the higher this value, the longer the plot will take to produce.

Option	Description
Plotter Inter-Buffer Delay Time	The next field allows the specification of an inter-buffer delay time for the device. This field will only have any significance for plotters whose destination has been set to be a serial device (via a COM port). The output device and the computer should 'handshake' using the method selected on the serial device screen to ensure that the computer does not send data that the output device is not ready to receive and process. On some early devices, at high data transmission rates this process can break down. The result is a corrupted plot, with lines appearing to be drawn at random on the paper. The response to this field defines a time (in seconds) that the computer will wait between each buffer of data (approximately 78 characters). If this problem is experienced, judicious use of this parameter will allow it to be overcome.
Pre and Post Plot Command Strings	The final two fields on the screen allow the specification of pre and post plot command strings. These allow you to define commands that will be sent to the device before and after the plot is produced. The pre-plot command could, for example, be the command string required to change the device from one mode of operation to another to allow a plot to be produced. The post-plot command could then be used to switch the device back again. The values entered here must be in ADE (ASCII Decimal Equivalent) format. That is, if a part of the string is the letter A, this must be entered as 65 (characters 6 and 5), since this is the ADE code for A. The specification as ADE values is necessary since the typical types of commands to be sent include non-printable characters.

Output Destination

Option	Description
Output Destination Selection	Various screen displays are possible from this point, depending on the response to the first screen in this set. There are three possibilities for the output destination, Device, File and Queue. If Device is selected, graphical hardcopy output using this plotter definition will go directly to the plotter. If you select File, the device-independent plot file will be converted into a device-specific intermediate file that can be sent to the plotter by, for example, a print command. If Queue is selected, first a file will be produced as in the previous option, then this file will be submitted to a print queue using user-defined commands. A print queue is a facility whereby printing can occur on a device at the same time that other programs are running.
Output Destination - Device	If you select device as the output destination, a screen allowing the device to be selected will appear. This screen allows you to select the actual computer port to be used. This can be COM1 or COM2 (serial (RS232) ports) or LPT1 or LPT2 (parallel ports). Baud rate, data and stop bits, parity setting and handshake all have to be specified if the output device is COM1 or COM2.

Option	Description
Output Destination Device - Serial	This screen is used to set the communications parameters for the serial port. The responses to each of these fields must be set to match the settings on the plotter. If they do not match exactly, the hardcopy will either not appear or will be corrupted. If there is any doubt about the setup of your plotter, consult the appropriate plotter manual. In order for the computer to communicate with a printer or plotter over a serial cable, the two devices must be capable of handshaking. This means that the printer or plotter can tell the computer when to stop sending data, and when it can re-start. If this does not happen successfully, data will be lost. There are two handshake methods. The first is software controlled and is known as Xon/Xoff handshaking. With this, the printer or plotter sends the Xoff character (normally the DC3 ASCII character) to the computer when it wants the computer to stop sending data. It sends an Xon character (normally the DC1 ASCII character) when communication is to be re-started. The second handshake method uses connections on the serial interface and is known as hardware handshake. The status (high or low) of a connection is changed and monitored. One of two strategies is normally employed. The two strategies are to interconnect the DTR and DSR lines (pins 6 and 20), or the CTS and RTS lines (pins 4 and 5). For serial devices, in the plotter system, the handshaking can be selected to be either Xon/Xoff or hardware. The hardware handshake method supported is the RTS-CTS link. If your device uses DTR-DSR, then a cable must be made connecting the DTR signal to RTS, and the DSR signal to CTS. This information is not relevant for the parallel ports and is not asked for. Only the port number is asked if you select the LPT ports.

Option	Description
Output Destination - File	On selecting this option a screen will be displayed prompting for the file type of the temporary device specific file that will be produced. The file type without the period (.) should be entered. If you had, for example, two plotters connected, one CANON and one HP, they could be setup such that the file type for the files to be sent to the CANON is CAN, and HP for the HP. In this way they would be distinguishable from each other, and it would be easy to see which should go to which device. When these files are created, they are created with the name `n.ype`, where `n` is a number and `ype` is the answer to this question. The plot control system controls the creation of these files. Please note that the plot file deletion setting does not affect whether these intermediate files are deleted. This setting cannot be chosen for dot matrix printer type devices (Raster) devices. The intermediate file that would be produced for each plot file would occupy several megabytes of disk storage. This is because of the need to convert the graphical data to a bitmap that can be printed on the device. These devices must be driven direct through a serial (often called COM1 or 2) or a parallel port (often called LPT1 or LPT2).
Output Destination - Queue	Choosing queue as the output destination causes an additional input screen to appear. The first field on this screen prompts for a file type. The comments in the previous section - output destination file - apply to the response to be given here. The device specific information is stored in this file.
File Name Method Incremental | Plot File Name	Two other fields for input appear on this screen: queue text string to precede file name and queue text string to follow file name. These allow the definition of the queue command and any of its qualifiers. The graphical hardcopy output system will build a command line using these two responses and the file name, and then issue this command to the computer operating system. For example, let us assume that one temporary file has been created whose name is `7.spl`. The program that will queue this file to a device is called PLOT, and the file can be deleted after it has been sent to the device by appending /DELETE to the file name. You would enter PLOT as the queue text string to precede the file name, and /DELETE as the queue text string to follow the file name. The software would build the command line `PLOT 7.spl /DELETE` and then send this command to the operating system.

Once this final screen has been accepted, the program will return to the main options screen.

Modifying a Plotter Definition

Once a plotter definition has been created, the modification option can be used to change any of the stored details. The modification option also allows a local setup for the plotter to be created and edited. On selecting modification on the main options page with the currently highlight plotter selected, a screen is presented.

Any of the four fields on this screen can be modified. Please note that in order to modify the setup data, the fourth (Modify Setup) field must be set to yes. Once this screen has been accepted and the choice to modify the setup data has been made, if you have chosen to modify the setup data, another screen appears to allow the selection of either the central or local setup. If local is selected, and a local setup file does not exist, it will be created automatically, and the contents of the central setup file will be copied to it.

Hint:	If you are trying to modify settings that appear to be unwilling to change, then please check that the settings of a Local copy of the plot definition file of the same name are not overruling the Global definition file that is undergoing to modification (Local definitions always take priority over Global definitions).

There is a difference in operation between modifying a central and a local setup. With the central setup all the setup data can be edited. With the local setup, the plotter type and the plotter control parameters are not able to be edited.

Having selected central or local, the main modification control screen appears. Any of the individual groups of data can be selected. Once a group has been selected, the modification screen that appears is similar to the addition screen and all the comments noted in the addition section apply.

Note that the Postscript font settings option only appears when a PostScript device is being modified.

Listing

On selecting the listing option, a further menu screen appears. This allows the selection to list either the defined plotters (their name and description only) or the setup for the selected plotter.

One of two destinations can be selected for the listing: screen or file. Screen displays are sent directly to the screen, with the screen pausing between each successive screen of data. If the listing is sent to a file, the name of the file to which the data will be sent is needed. A default name will be displayed. By convention the file type associated with list files is `.lst` If the default name is not accepted it is possible to omit the file type or specify it as `.lst`.

If an existing file of the name specified is found, you will be asked if you want to overwrite it. If Overwrite=No, the initial selection screen will appear, otherwise the file will be overwritten and the list file will be created.

Import

Import allows an existing plotter definition file - they always have `.de1` extensions - to be imported into the current plotter table without having to define the new files characteristics, etc.

- Imports must be in the `<install_dir/mscfatigue_files/nssys` sub-directory - MPLTSYS will not import plotters from other directories.

- If the definition file originated from a different machine then it must first be converted using the module MCONFIL.

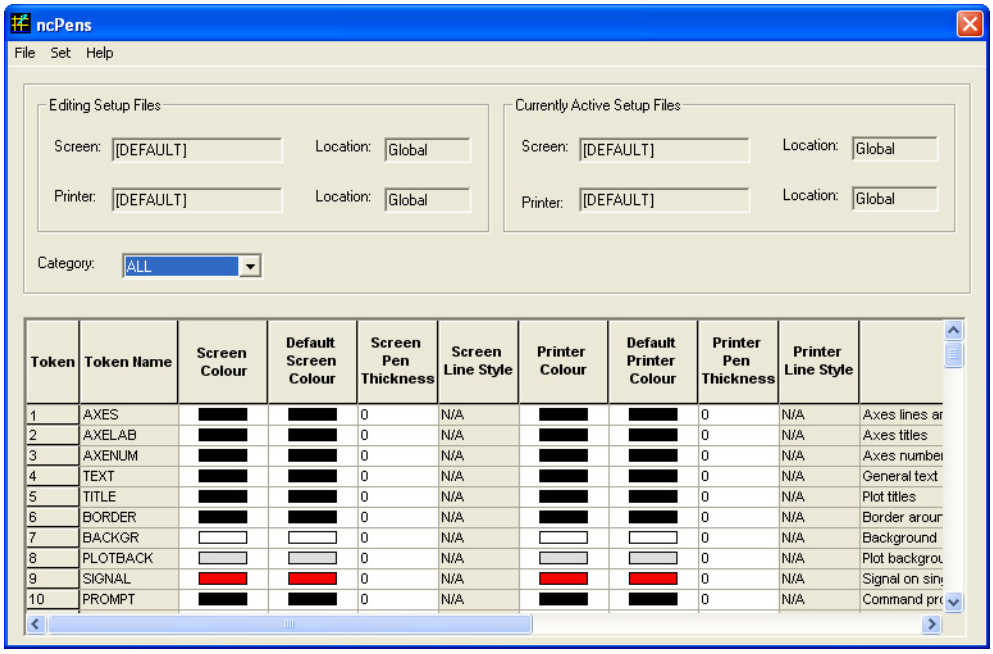
Import has only one screen whose fields are:

Option	Description
Plotter File Name	Enter the file name of the `.de1` to import. The name must be unique, it is not allowed to have multiple plotters with the same name. The file MUST reside in the MSC Fatigue system (`nssys`) directory.
Plotter Name	This is non-editable text taken from the plotter file.
Description	Type a description of the imported plotter. It can be used to differentiate the plotters for the users of the system by describing the physical plotter that has this name.
Available	If this plotter is available for use, e.g. connected to your computer, select Yes. If not, select No.

Plot/Pen Colors Utility - (MNCPENS)

MNCPENS is a simple utility to allow you to change graphic colors on any plot. It is accessible through most all graphic utilities such as MQLD, MMFD, MGED, MPOD, etc. from the Preference pulldown menu. Select the Pen Setup option from this pulldown menu or invoke the program from a Windows or Linux command prompt with the symbol **mncpens**.

You can customize colors of any plot attribute such as background, curve data, grid lines, titles, text annotation, etc. You save the settings to a file in either the local working directory, your home directory, or the global installation area (if you have privileges). Once this has been set, you can select whether the local, home, or global color environment is used, or some other custom color file is selected which remains in effect until you change it. The best way to learn this program is to experiment and see what it does.



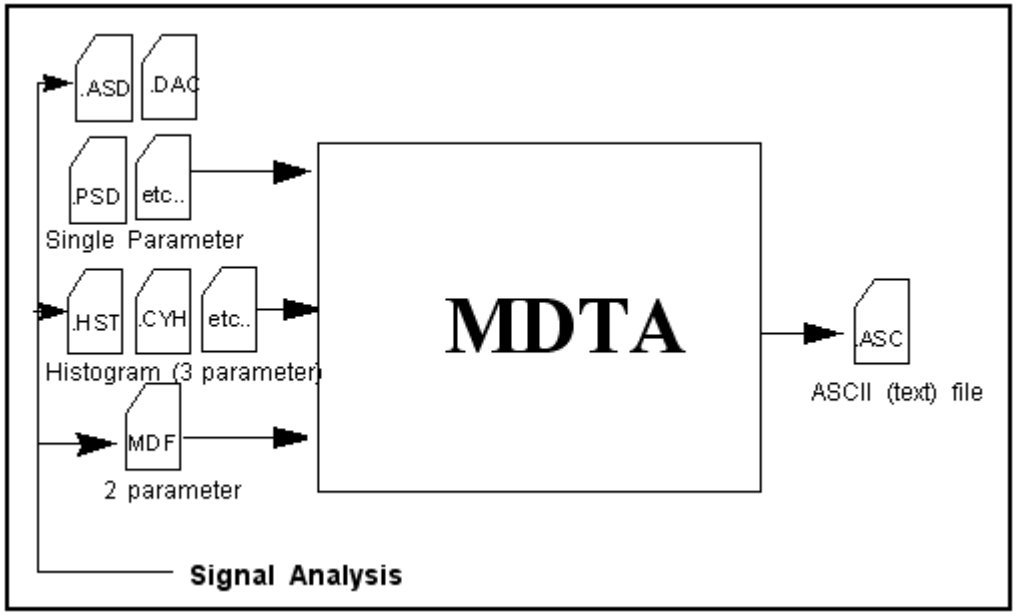
File Conversion Utilities

Various additional file manipulation and translation modules are also available.

Binary/ASCII Convertor - (MDTA/MATD)

MDTA - Binary to ASCII

The MSC Fatigue signal to ASCII module, MDTA, converts a single parameter, X-Y, or histogram binary file into ASCII format.



It can also write or omit header details, and write an ASCII file as single or multiple column.

MDTA is complimentary to module PTIME which can convert ASCII files to binary files.

Signal files cannot be easily examined on a terminal or printing device because they are in binary format. For the same reason they are difficult to manually edit (in a word processor for example). However, ASCII files can be viewed or printed easily because they are letters and numbers. MDTA provides the means to quickly and easily produce an ASCII version of many MSC Fatigue signal files.

The resultant ASCII files are meant to be read by PTIME and are not a generic file type.

The header of the input signal file can also be converted (or omitted) and MDTA can write multi or single column.

Note that any extra details area data within the `.dac` file will be lost in this process. In order to retain such keywords, write them out to a separate ASCII file, then import them into the data file using MFILMNP.

If multi-column is chosen the program will attempt to fit as many values onto a line as possible. This will minimize the ASCII file size. The alternative is single column whereby each value (or pair of values for X-Y files) will be written to a new line.

If two parameter files are to be created from a single parameter file then MDTA can read the header information of the input `.dac` and create a second data column of X values e.g. time.

Numbers in an ASCII file are delimited (separated) by 1 of 3 characters from the ASCII character set. These are the space, comma, or tab.

The file types supported are:

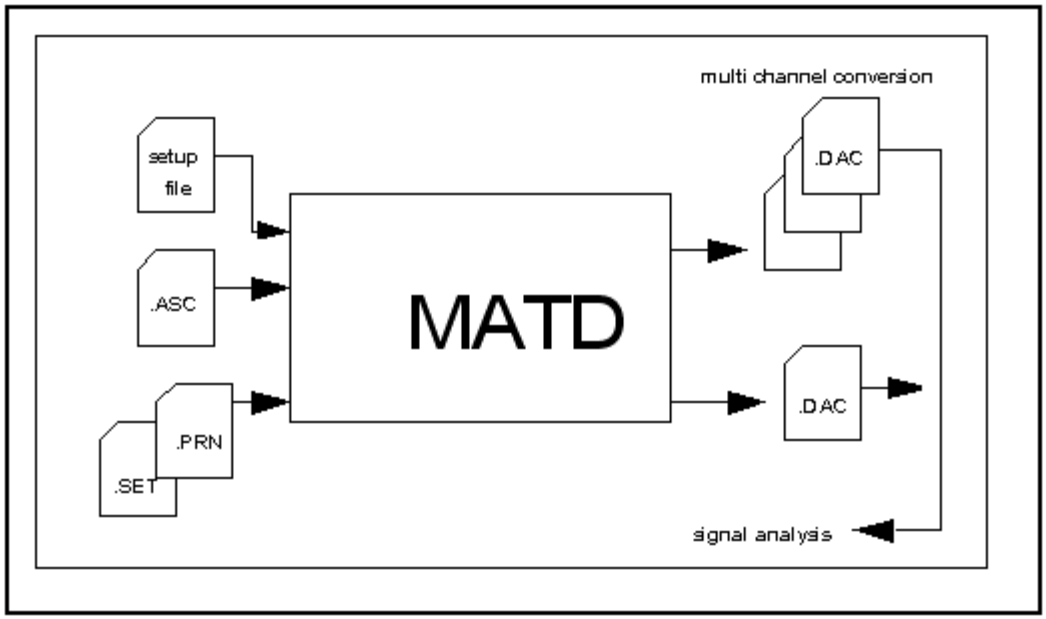
- **1 Parameter:** `.dac, .psd, .acf, .ada, .asd,, .pvx, .sxx, .sxy, .syy`
- **Histogram:** `.cyh, .dhh, .hst, .san`

- **X-Y (2 Parameter):** `.mdf, .css, .sl, .fal`

The above table is not necessarily exhaustive.

MATD - ASCII to Binary

The ASCII to signal module, MATD, converts an ASCII text file into a format binary data file, for example a 1 parameter .DAC file.



MATD processes files of data in ASCII format and converts them to binary format files that can, for example, be plotted in MQLD or MMFD.

MATD's complementary module is MDTA which converts binary to ASCII.

There are three options/modes of operation from ATD's main menu:

1. Single channel file conversion

   MATD can convert single parameter time history files, paired data files, and histograms. If the data came from MDTA then header details are automatically used.

   MATD can convert non-equally spaced data, and can selectively choose or ignore data from a file as required. This selectivity is made possible by the user supplied parameters, explained in the Module Operation section.

2. Multi-channel file conversion

   This option extracts many channels of data from an input file simultaneously. The data must be sequential (i.e., time history format).

   The program will need the following information to be supplied:

   - The number of channels in the input file

- A list of channels to be demultiplexed
- The sample rate of the data
- The base offset of the data
- For each channel, its X and Y axes' units and labels

The information may be input using the keyboard or from an ascii file (hereafter referred to as a general setup file). The format of the general setup file is shown at the end of this technical overview.

3. Chart logger (dAtagate) file conversion

   This option extracts many channels of data from an input file created by the dAtagate chart logger (from now on referred to as a chart logger).

   The program will normally take the default setup from the chart logger's setup file, however, a general setup file may also be used.

Spreadsheet

For multi-channel demultiplex operations a spreadsheet facility is offered for specifying and editing the default X and Y axes labels and units.

MDTA Module Operation

The MDTA module can be run in one of the following 3 modes:

- From the **File Conversion Utilities** pulldown menu under **Tools | MSC Fatigue** (for Patran) or under **Tools | Fatigue Utilities** (for).
- In stand-alone mode by typing **mdta** at the system prompt
- By incorporating the MDTA commands in a batch operation

Once running in interactive mode the MDTA module prompts for the following input. The purpose and usage of each field is explained below. Note that until Input File Name is completed and confirmed (by clicking **OK**) then none of the other fields are active.

Option	Description
Input File Name	In this field the user should type the name of an input binary file (often a single parameter .dac file). By default MDTA assumes a .dac file extension but if a file with a different file extension is to be processed then enter the File Name plus extension in full. Two parameter files and histogram (3 parameter) files can also be processed. MDTA will expect to find the input data files resident in the user's directory. However, other directories can also be accessed if the complete file specification path name and file name are entered. Probably the easiest way to name an input file is to use the pick List facility.
Output File Name	In this field the output ASCII file must be named. The file extension will by default be .asc although if a different name and extension is entered here then the alternative extension will be used. By default MDTA uses the input file name as the output file name (the file extensions will be different). If a file with the same name as he proposed output file name already exists on the target drive and directory then MDTA will prompt the user for permission to overwrite the existing file.
Write Header Yes/No	MDTA can optionally write the header details to the ASCII file. If the ASCII output file is subsequently converted back to binary (by module PTIME) then the header information will also be converted. This MDTA/PTIME pairing is useful, for example, if a binary header needs to be edited and then re-converted back to binary. The header details will occupy their usual fixed 20 first lines of the file.
Multiple Column Yes/No	If multiple column is selected, the program will attempt to fit as many values into a line as possible. If no value is selected then every data value X-Y pair will be written on a separate line (although the ASCII file will be larger).
Separator	This field enables the user to select which of three ASCII characters will be used to delimit or act as a boundary marker, between the data values.
Include X-Values?	If two parameter files are to be created from a single parameter file then MDTA can read the header information of the input .dac and create a second data column of X values e.g. time

MDTA Batch Example

MDTA can be invoked in all the standard batch modes. A typical batch command line would be:

```
mdta /inp=saetrn/out=saetrn/ov=y/mul=y/del=t
```

In this case, MDTA would convert a data file called `saetrn.dac`. The results would be written to a file `saetrn.asc` and this file would be overwritten if it already existed. The ASCII file will be multi- column with tabs as delimeters.

A list of MDTA's batch keywords:

/INPut	The input file name. /INP=FILE
/OUTput	The output file name required for the results data file. /OUT=RESULT
/OVerwrite	Whether to overwrite an existing results file. /OV=Y
/HEADer	Whether to write header, Yes or No. /HEAD=Y
/MULtiple	Whether to write multiple columns (or single) Yes/No. /MUL=Y
/DELimiter	The delimiter character Space, Comma, Tab /DEL=C

MATD Module Operation

The MATD module can be run in one of the following 3 modes:

- From the MSC Fatigue Tools pulldown menu system
- In stand alone mode by typing **matd** at the system prompt
- By incorporating the MATD commands in a batch operation

Once running in interactive mode the MATD module appears allowing access to the three modes of operation.

Mode 1 - Single Channel

The following input is available for Single Channel operation:

Option	Description
ASCII Filename	This field asks for the name of an ASCII text file data to be converted to a time series file format .DAC file. The assumed file extension is .ASC though other file extensions and pathnames are accepted if they are entered explicitly. Use the pick list facility to select an input file. Note that until this field has been completed, none of the other fields will appear.
Output Filename	This requests the name of a time series file (e.g. *.DAC) as the output of the ASCII to DAC conversion process. The generic name of the input file is offered as default, and if the output file already exists in the specified directory, the user will be asked to confirm that it may be overwritten.
Sample rate	Enter the sample rate on points per second (Hz). The default is 1 Hz, or the value in the macro of the ASCII if it was made by DTA, or if the keyword ATDSAMP is set in the environment then any values associated with that keyword are used. If the time increment is known then the sample rate is the time rate's reciprocal.
Equally Spaced Data ? Yes/No	MATD can convert two types of ASCII data file: ■ Where the data consists of values only, where the values are assumed to be equally spaced apart in time. If the time values are present in this case, they may be omitted provided the correct sample rate is entered. (a single parameter file) ■ Where both time and values are present, and the time values are not equally spaced. The data is assumed to be formatted with a time followed by a value sampled at that time. This is the two parameter file type. The user must answer Yes if the data format is like that of 1) above, or No if the data format is like that of 2) above. If Yes is entered and Take All Numbers = No, then the following fields appear. Start Position for Accepting, Number of Values to Skip, and Number of Values to Take.
Header Lines to Skip	The ASCII text file being used as input to MATD may have some header lines containing information about the data rather than data itself. The user is requested to enter the number of lines of header the program is to ignore as valid data lines. The default is determined by the environment keyword ATDSKIP, which MATD both sets and reads in the local environment. It may be necessary to view the ASCII file to count the header lines. If the ASCII file was produced by the module MDTA then the correct number of header lines to skip to recreate the original signal file is 20.

Option	Description
Take All the Numbers [Y/N]	The conversion program is only interested in data points and not in any other numbers which may be in the file, such as time base values. This question is asking whether all the numbers in the ASCII file are to be used or whether some are to be ignored. When the file is a file of time-data value pairs where the time values are NOT equally spaced then answering Yes will cause MATD to treat all data as time-value pairs. If No then ATD will ask for the time column number and the data column number.
Start Position for Accepting	This field appears if Equally Spaced Data=Yes and Take All Numbers=No. If the user wishes to skip some data values in the file, for example in the case of a multiplexed ASCII file such as: 1 2 3 4 5 6 7 8 1 2 3 4 5 6 7 8 1 2 3 4 5 6 7 8 etc. (8 channels) Each channel may be taken out one at a time by specifying the start location of the channel. In this case, for channel 4, selecting a start value of 4 and the skip value (next question) of 7 (skip 7 values before the next data value for channel 4), will convert only the ASCII data for channel 4. The default is to begin taking data at the first value.
Number of Values to Skip	This field appears if Equally Spaced Data=Yes and Take All Numbers=No If the user wishes to skip some of the data values in the file, then the number of values to skip at any one time must be given in response to this prompt. For example, if the file is like this: 1 2.5 2 3.45 3 etc. and the values 1,2,3 are not required, then the number to skip is 1. The default value is 0, meaning do not skip.

Option	Description
Number of Values to Take	This field appears if Equally Spaced Data=Yes and Take All Numbers=No. If the user wishes to skip some of the data values in the file, then the number of values to accept at any one time must be entered at this prompt. For example, if the file is like this: 1 2.5 3.5 4.5 2 3.45 4.2 3.1 3 etc. and the values 1,2,3 are not required, then the number to skip is 1 and the number to take is 3. The default number to take is 1. If Equally Spaced Data = No, then Start Position for accepting, etc., does not appear. They are replaced by Time Column and Data Column.
Time Column	This field appears if Equally Spaced Data=No and Take All Numbers=No. If non equally spaced data is in the input file then the program needs time value pairs. If the file has multiple data columns, the user must specify which column is time. Therefore, in this field the time column number is entered. MATD will assume that each line has the same number of columns.
Data Column	This field appears if Equally Spaced Data=No and Take All Numbers=No. In addition to knowing which is the Time Column, MATD must also know which is the Data Column.
Y-Axis Label/Units	All screen displays contain these two fields. The label for the Y-Axis is Magnitude and Volts by default, however the user can enter any 28 characters in these fields should they wish to change the Label/Units. Having completed all inputs, MATD begins the data conversion and gives the following on screen message.

Mode 2 - Multi Channel

The following input is available for the multi-channel operation:

Option	Description
General Setup Filename	The name of a general setup file should be entered into this field. General setup file usually take an .ATD extension and this will be assumed if none is given. If the field is left blank it will be assumed that the program is to create a temporary default setup for the demultiplex operation. A list of .ATD files is offered by pressing the pick list button or by clicking the mouse on the LIST button. A file may be selected from this list.
ASCII Data Filename	The name of the ASCII data file should be entered into this field. ASCII data files usually contain a number of header lines followed by the data to be demultiplexed. The standard extension for this type of data file is .ASC and this will be assumed if none is entered. A list of .ASC files is given by clicking the pick list button); A file may then be selected from this list.
Generic Output Filename	A generic file name for the output filenames should be entered here. The output filenames are created by appending the channel numbers to the filename and adding a .DAC extension, e.g., Demultiplexing channels 1, 5 and 6 using a file stem FRED will produce the following output filenames: FRED01.DAC - Contains data from channel 1 FRED05.DAC - Contains data from channel 5 FRED06.DAC - Contains data from channel 6
Header Lines to Skip	The standard ASCII data file may optionally contain a header area proceeding the data to be demultiplexed. This question asks for the number of lines of text comprising the header area. The first non-blank line after this area is then assumed to be the start of the data to be demultiplexed. The default is determined by the environment keyword ATDSKIP, which MATD both sets and reads in the local environment. If the ASCII data file does not contain any header information then the answer to this question will be 0.
Number of Chans in File	This question is asking for the number of channels, NCHANS, defined in the data file. The program can then synchronize the data knowing that a new sample begins after every NCHANS data values in the input file. The channels are numerically tagged, incrementing sequentially from 1 to NCHANS (i.e. if 4 channels are defined these are tagged 1,2,3,4). Channels are subsequently referred to by their numerical tag. If a general setup file is being used the default number of defined channels offered in this field is taken from this file.

Option	Description
Sample Rate of Data	This question determines the sample rate of the data. It must be the same for all channels. The sample rate is defined to be the number of samples per channel per second. It is also the reciprocal of the time increment between each sample of a given channel. If a general setup file is being used the default sample rate offered in this field is taken from this file. If no setup has been nominated the default value offered is 1 unless the keyword ATDSAMP is in the environment in which case its associated value is used.
Base Offset of Data	This question determines the X-Axis base offset of the data. Most commonly this means the time at which the first data in the input file was sampled and it usually has a value of zero. It must be the same for all channels. If a general setup file is being used the default value for the base offset is taken from this file. If no setup is being used a default value of 0.0 is offered.
Demultiplex Channel List	The list of channels to demultiplex is entered in this field. The input may be entered using the keyboard as follows: 1,5,7-10- selects channels 1,5,7,8,9 and 10 ALL - selects all channels in data file DEFAULT - uses the default channel list from the setup file or ALL channels if a setup file is not being used Alternatively, the channels may be selected from a list of all active channels offered by pressing the pick list button or by clicking the mouse over the CHAN button. The word <SELECTED> is written to the field to denote a list of channels selected this way. The number of channels selected is displayed below this field. When all the above fields have been filled, ATD will use the spreadsheet function to display the units and labels for each channel. The spreadsheet function is explained four pages ahead.

Mode 3 - Chart Logger

The following input is available for the chart logger operation:

Option	Description
Logger Setup Filename	A chart logger setup filename should be entered into this field. Logger files take a .SET extension (this is implied if none is given). Files with other extensions are assumed to be general setup files. If the field is left blank it will be assumed that the program is to create a temporary default setup for the demultiplex operation. A list of .SET files is offered by clicking the mouse on the LIST button. A file may be selected from this list. When the logger file has been selected the following items are requested:
Logger Data Filename	The chart logger data filename should be entered into this field. Chart logger data files take the extension .PRN and this will be assumed if none is entered. A warning is issued if the specified file does not exists. A list of .PRN files is given by pressing the pick list button. A file may then be selected from this list.
Generic Output Filename	A generic stem for the output filenames should be entered into this field. The output filenames are created by appending the channel numbers to the filestem and adding a .DAC extension, e.g., Demultiplexing channels 1,5 and 6 using a file stem FRED will produce the following output filenames: FRED01.DAC - Contains data from channel 1 FRED05.DAC - Contains data from channel 5 FRED06.DAC - Contains data from channel 6
Header Lines to Skip	MATD would fail if it attempted to start extracting data in the header region of the input file. So, because the header region consists of a discrete number of lines at the top of the file, it is possible to instruct MATD to skip that number of lines before data extraction will begin. The default is determined by the environment keyword ATDSKIP, which MATD both sets and reads in the local environment. By default MATD will skip whatever number of header lines it encounters in the input file. Also, because ASCII files can be loaded into a text editor, it is possible to manually count the number of lines that need skipping in order to answer this question.
Number of Chans in File	This question is asking for the number of channels, NCHANS, defined in the data file. The program can then synchronize the data knowing that a new sample begins after every NCHANS data values in the input file. The channels are numerically tagged, incrementing sequentially from 1 to NCHANS (i.e. if 4 channels are defined these are tagged 1,2,3,4). Channels are subsequently referred to by their numerical tag. If a general setup file is being used the default number of defined channels offered in this field is taken from this file.

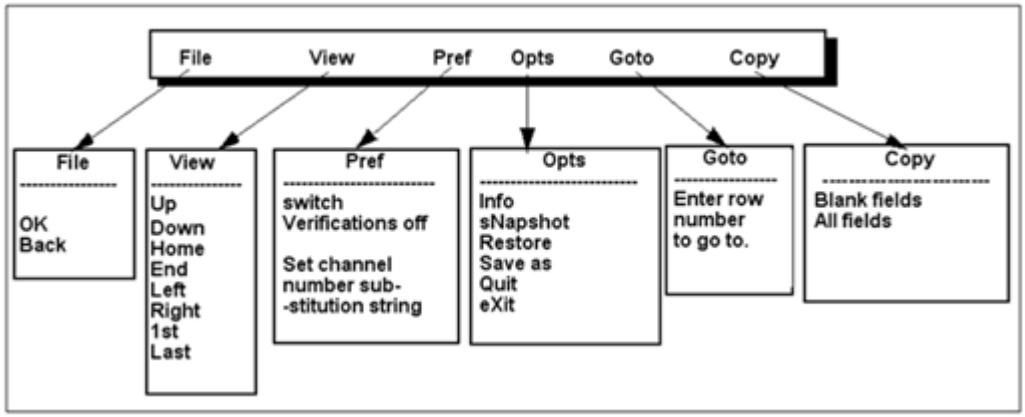
Option	Description
Sample Rate of Data	This question determines the sample rate of the data. The sample rate is defined to be the number of samples per channel per second. It is also the reciprocal of the time increment between each sample of a given channel. If a general setup file is being used the default sample rate offered in this field is taken from this file. If no setup has been nominated, the default value offered is 1 unless the keyword ATDSAMP is in the environment in which case its associated value is used.
Base Offset of Data	This question determines the X-Axis base offset of the data. Most commonly this means the time at which the first data in the input file was sampled and it usually takes the value 0. If a general setup file is being used the default value for the base offset is taken from this file. If no setup is being used a default value of 0.0 is offered.
Demultiplex Channel List	The list of channels to demultiplex is entered in this field. The input may be entered using the keyboard as follows: 1,5,7-10- selects channels 1,5,7,8,9 and 10 ALL - selects all channels in data file DEFAULT- uses the default channel list of setup file or ALL channels if a setup file is not being used Alternatively, the channels may be selected from a list of all active channels offered by pressing the pick list button. The word <SELECTED> is written to the field to denote a list of channels selected this way. The number of channels selected is displayed below this field. When all the above fields have been filled MATD will use the spreadsheet function to display the channel units and labels for each channel (see next section). Exiting from the spreadsheet editor will initiate the demultiplexing operation (after the user has confirmed this action). During logger file demultiplexing, lines containing errors will cause a temporary halt with a message to select one action from Remove, remove All, or Exit. Any lines removed in this way will be written or appended to a test file, ATDERR.LOG, for subsequent viewing.

The MATD Spreadsheet

There are three main areas on a spreadsheet screen:

1. The **data area** itself which consists of rows and columns of data cells.
   - **Column A** contains the channel number It can contain numerical designations for up to 256 rows. These labels can NOT be edited.
   - **Column B** contains the X-axis labels. Values can be edited. See the Copy function for details of quick global edits.
   - **Column C** is the X-axis units and **columns** D and E contain the corresponding values for the Y-axis.

2. The **edit window** at the bottom of the screen. It contains temporary or volatile data, such as the data value currently highlighted or being edited.
3. The **header bar** across the top of the screen which contains the buttons for manipulating the data and navigating around the columns and rows. All of the buttons have keys mapped to a function, as described in the footer.



Most of the operations in the header bar are self evident. Those that are not are listed here:

Option	Description
OK	When the edits have been made this option quits the spreadsheet environment and starts the demultiplexing operation.
Back	This option quits the spreadsheet environment and returns to the previously viewed screen.
Set Verifications Off	Suppresses the less crucial verification requests. For example the verification for restoring a snapshot will not be asked.

Option	Description
Set Channel Substitution String	This option allows a string to be defined such that wherever it is encountered as a substring in any channel label or unit it represents the channel number. For example, assume the channel substitution string is set to "nn" and the Y-labels for channels 3,57 and 92 are set as follows: 3:Strain Gauge nn 57:Serial No PK00nn 92:Channel-nn The actual labels written to the output files will be Strain Gauge 3 Serial No PK0057 Cha92el-92 Note that the substring, ***nn***, is obviously inappropriate since it produces an unrequired substitution in the word Cha***nn***el.
SNAPSHOT	Causes the current spreadsheet values to be remembered. The values are reinstated by using the restore option. A snapshot is automatically taken when the spreadsheet is first entered.
RESTORE	Spreadsheet values are re-instated to those stored when the last snapshot was taken. Verification is only required if the verify preference is switched on.

MATD Batch Operation and Setup Files

MATD can run in batch mode because the spreadsheet operations are inherently interactive and are not available in batch. In the absence of interactive instructions, *Setup files* in which the interactive instructions are preprepared are used. Setup files are described here.

Option	Description
Chart Logger Header Files	The default setup for chart logger data is taken from the corresponding .SET file. However, it is permitted to use an .ATD setup file if required.
No Setup File	If no setup file is nominated then all input fields corresponding to demultiplex parameters are initialized to NULL thus forcing values to be entered.
General, .ATD, Setup files	The general setup file will be structured as follows. The first line of the file will distinguish an MATD general setup file from other files: MATD General Setup File: Created 5/2/95 for testing with MYDATA The characters up to and including the colon form the identification string. All characters after this colon are optional and are ignored. All remaining lines of importance will contain recognizable tokens followed by their associated parameters. TOKENS / Example <NHEAD>12The number of header lines preceding the data <SRATE>0.01The sample rate <XBASE>0.00The x-base offset <XLAB>3, TimeThe channel number (3) followed by the label <YLAB>3, StrainThe channel number (3) followed by the label <XUN>3, SecondsThe channel number (3) followed by the units <YUN>3, uEThe channel number (3) followed by the units <NCHANS>4The number of channels in the data file <DMXLST>2,3The list of channels to be demultiplexed <SUBSTR>nnnThe channel number substitution string <LOGMAP>1-9,17,18The logical channel mapping information used when saving information from a logger setup file to a general setup file.

A minimal specification in a general setup file must contain values for the sample rate, base offset, number of channels and the list of channels to be demultiplexed. The file format of general setup files is rigid and the following rules must be observed when they are created:

1. Tokens must be delimited by triangular brackets < >
2. The only tokens recognized are those listed above. Any unrecognized token will be assumed to be the result of misspelling and will cause an error.
3. Any line not beginning with a token is ignored and treated as a comment.

4. Token lines must contain their corresponding parameters. Labels and units may seem not to be specified in some files where their values comprise solely of white-space (TAB characters and spaces).
5. Lines containing tokens should not contain any other information other than the token's associated parameters.
6. In the example above channels 2 and 3 are to be demultiplexed from a 4 channel file, however, the units and labels for channel 2 will be initialized to NULL because they have not been defined.
7. The following tokens may appear more than once to allow the specification of information to be spread over several lines of the setup file which otherwise would result in a line too long to fit within screen viewing limits:
   - <DMXLST>
   - <LOGMAP>
   - An error is invoked if the setup file contains any of the tokens in the following list more than once:
   - <SRATE>
   - <XBASE>
   - <NHEAD>
   - <NCHANS>
   - <SUBSTR>

A list of MATD's batch keywords:

/INP	File name of the ASCII file to convert (.ASC) /INP = AERO
/OUT	File name of the output signal file (.DAC) /OUT=AERO
/OV	Whether it overwrites the existing output filename /OV = Y
/SPA	Whether the data is equally spaced in the frame /SPA = N
/HEAD	Number of header lines to skip /HEAD =
/ALL	Whether to take all the numbers in the file /ALL = N
/STA	The start position for accepting /STA = 10
/SKIP	How many values to skip /SKIP = 4
/TAKE	How many values to take /TAKE = 1
/SAM	Sample rate for the output file /SAM = 2
/UNI	Units for the output file /UNI= mV
/XBASE	Base offset of X-axis
/LAB	Output file label /LAB = Title
/TCOL	Time column /TCOL = 2
/DCOL	Data column /DCOL = 2
/SETup	Setup filename.

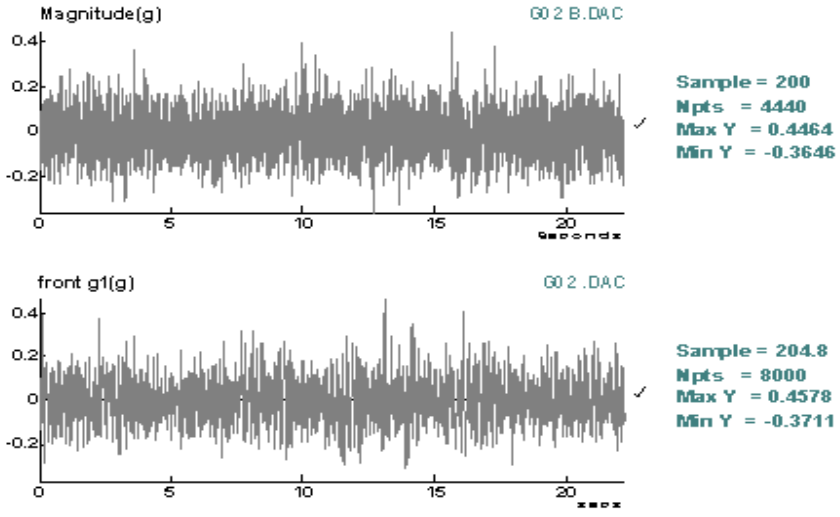
/INPut	Input filename.
/OUTput	Output filename.
/NCHANS	Number of active channels in data file /NCHANS=8
/CHAnnels	List of channels to be demultiplexed.
/ERRACT	Action to take if multichanel datafile has errors (R, A, X)
/MISDAT	Action to take if multichannel samples not complete (Z, R, X)

Signal Regeneration - (MREGEN)

MREGEN can:

- regenerate a single parameter signal file (.dac extension) from a three parameter range-mean cycles histogram file (.cyh type).
- regenerate a single parameter signal file (.dac extension) from a three parameter maximum-minimum cycles histogram file (.cyh type).
- regenerate a single parameter signal file (.dac extension) from a three parameter Markov Matrix (.mkh type).
- generate a Gaussian series from a user supplied irregularity factor and save it as a .dac file.
- generate a random time history (.dac) from input psd.

The cycles histogram files are, in the first instance, created from single parameter files and the creation process loses time information (sample rate and frequency). Because of this loss of original information, MREGEN produces an approximation of the input file, i.e. with the same number of cycles at each specific range and mean.



The only output file is the signal (.dac) file regenerated from the input cycles histogram.

The plots in Figure 14-49 compares a regenerated file with its original.

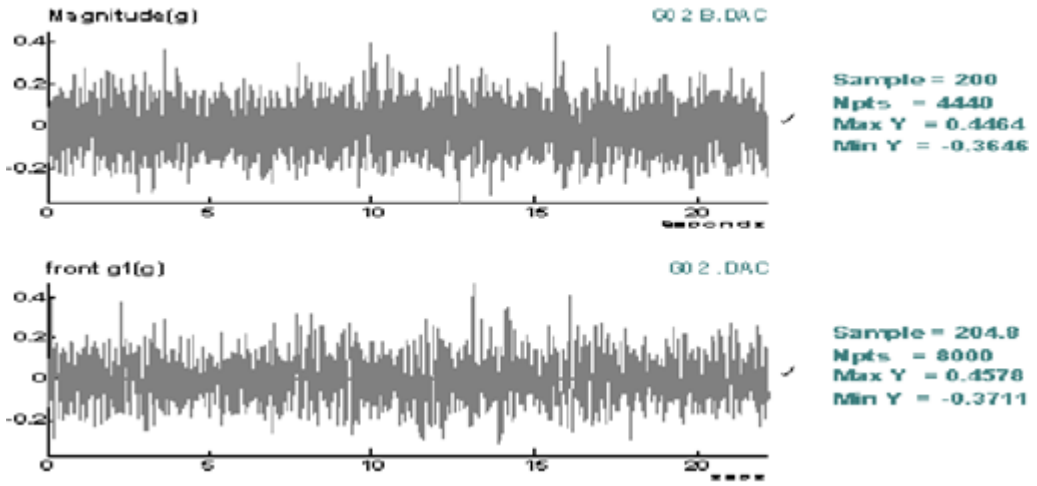


Figure 14-49 The Original .dac File, (Bottom) Compared to the Regenerated File (Top)

The regenerated file is similar to the original (compare the statistics to the right of each plot) except for the fact that the *sample rate* is wrong; the original was 204.8 samples/sec and the new file was assigned the estimate of 200 sample/sec. MREGEN always allows the user to enter a sample rate because it cannot be calculated from a .cyc or .mkh file because time information is lost. The default time allocated is 1 sample/sec.

Rainflow Reconstruction:

The MREGEN module constructs a peak-valley sequence from two types of rainflow matrix, one where the cycles are stored in the matrix according to their range and mean, and another where they are stored by their maximum and minimum (peak and trough). In the case of the range-mean type of matrix, the data is converted from range-mean to max-min format before the reconstruction occurs. This results in a slight inaccuracy when the resulting regenerated sequence is cycle counted and compared with the input matrix. This inaccuracy is due to the overlap of range-mean and max-min matrix bins, i.e. a cycle from a particular range-mean bin may lie in one of four max-min bins. Care must therefore be taken when comparing results from a regeneration from a range-mean matrix.

Once a max-min matrix has been obtained, either directly or by conversion from range-mean, the regeneration process may begin.

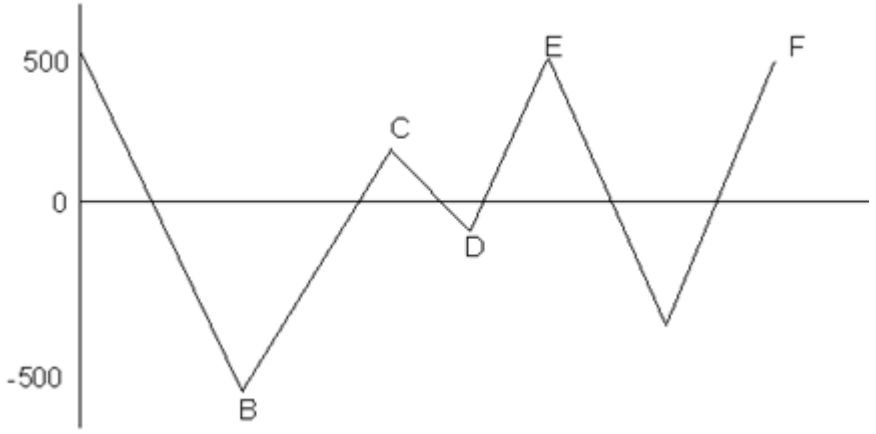
The procedure for regeneration is as follows:

1. Find the largest cycle, that is the one inside which all other cycles will hang or stand. Generate three points to define this cycle. Use a random number to determine whether the cycle is min-max-min or max-min-max. If there is more than one entry in the bin, generate 2n+1 points for n entries in the bin.

   Use the user-supplied method to generate the data value from the bin location. The extreme option chooses the largest possible maximum and smallest possible minimum. The mean option uses the value from the centre of the bin and the random option selects a random location within the bin.

2. For all other bins with non-zero entries, obtain the maximum and minimum value of the cycle classification. Find all the possible locations for this cycle to be inserted within the current regenerated sequence. A possible location is defined by the ability to fit the new cycle inside two consecutive existing samples. For example, in the following sequence we are trying to fit a cycle whose maximum is 100 and minimum is -100.

Existing sequence: 500 (A),-500 (B),100 (C),-50 (D),250 (E),-300 (F), 300 (G).



In this case there are 4 possible locations, between A and B, B and C, E and F, F and G. It is not possible to insert the cycle between C and D or between D and E and create the correct sequence. Note that 'equal to' is allowed, as in the case of B to C.

A random number is then used to determine which of these four locations to use. The data to the right of the pair of points, including the right hand point, is then shuffled right to leave a two point gap into which the cycle can be placed. The points are then inserted in the correct sequence to retain a peak-valley order. In the above example if the second pair (B-C) were selected then points C-G would be moved right and the values inserted to give the following sequence:

New sequence:

200 (A),-500 (B), 100(new), -100 (new), 100 (C),-50 (D),250 (E),-300 (F), 200 (G).

This procedure is repeated for the number of cycles in the bin.

The process then continues until all non-zero bins have been handled.

To obtain satisfactory speed performance, memory management and extensive software optimization is performed. This leads to a possible limitation on some machines of a maximum of 800,000 cycles in the regenerated sequence. This requires 10Mb of actual or paged memory. On Linux systems the number of cycles which can be regenerated in one go is limited to the amount of memory available to the user.

When the sequence is written from memory to disk, the first point written to the file is selected randomly from within the complete sequence. This method is based on the paper in reference (Ref. 39).

Markov Transition Matrix Reconstruction:

When regenerating a data stream from a Markov Transition Matrix, there are two approaches, firstly to reconstruct segments which result in an exact match when a Markov count is performed on the regenerated

samples. Secondly, it is possible to regenerate a sequence of any length whose transition matrix has the same characteristic as the original.

To understand this last approach, consider the Markov matrix as a joint probability matrix. In this matrix, each bin represents the probability that the next data point will be selected from that bin, given that the current point is known. The additional restriction is that from a minimum (valley) point, the next sample must be from a bin greater than the current bin, and similarly for a maximum (peak) value the next data point must be from a bin whose value is less than the current bin.

The algorithm normalizes the transition matrix to be a joint probability matrix and then selects a random start point. The start point is selected randomly, but in such a way that the algorithm will proceed correctly. Subsequent samples are generated by a random value shaped by the probability distribution. This will continue to an indefinite number of samples provided that the Markov Transition matrix was valid. If the matrix was invalid, it may be possible to arrive at a bin with no valid bin to go to. In this case an error will be generated. As the sequence becomes longer, the distribution of the regenerated samples will more closely resemble the original distribution.

Irregularity Factor Reconstruction:

The irregularity factor of a data stream is defined as

$$\gamma = \frac{\text{The number of positive mean crossings}}{\text{The number of peaks}} \tag{14-1}$$

Note that the band width, E, is defined as

$$E = E = (1 - (\gamma^2))^{0.5} \tag{14-2}$$

So when $\gamma = 1$ and E= 0 the data is pure narrow band. When gamma=0 and E=1 the data is pure broad band and when $0 < \gamma < 1$ the data is wide band.

Kowalewski (Ref.40.) proposes a method by which a joint p.d.f. may be generated from a single value of gamma. The algorithm uses this method to generate the p.d.f. and then regenerates a sequence of peak valleys by the method defined in the section on Markov Transition Matrix Reconstruction. The Kowalewski p.d.f. is normalized in the X, Y and Z axes so the output data is scaled to the values entered by the user.

Module Operation

The MREGEN module can be run in one of the following three modes:

- From the **File Conversion Utilities** pulldown menu under **Tools | MSC Fatigue** (for Patran) or under **Tools | Fatigue Utilities** (for).
- In stand alone mode by typing **mregen** at the system prompt
- By incorporating the MREGEN commands in a batch operation

Once running in interactive mode, the MREGEN module will display the four options listed below.

Option	Description
Range-Mean Rainflow Matrix, 1199	Regenerates a .dac files from a range-mean rainflow matrix (.cyh). As pointed out in the technical overview, this type of regeneration has to go via an intermediate maximum-minimum stage and is as a consequence 3% less accurate than the maximum-minimum option.
Max-Min Rainflow Matrix, 1200	Regenerates a .dac file from a maximum-minimum rainflow matrix (.cyh) file. Both the above methods accept .cyh files as their input and MREGEN cannot guarantee that it is the correct type unless it was created from another MSC Fatigue module (Range-mean or max-min is identified in the header). The wrong input type will probably cause an error message to be output at some stage but this can NOT be guaranteed.
Transition Markov Matrix, 1200	Regenerates a .dac files from a from-to Markov transition matrix. It takes a Markov Transition matrix and converts it to a joint peak-trough probability density function (p.d.f.) from which a variable length time series may be reconstructed. The reconstructed history will have the same, or similar, peak trough p.d.f. to the original data. It does not reconstruct exact transition segments.
Irregularity Factor, 1201	Generates a Gaussian distribution from a user supplied irregularity factor.

Range-Mean Rainflow Matrix

This option requires the following input.

Option	Description
Input File Name	The name of the file from which a .dac file is to be regenerated must be entered here. The List button will automatically look for files with the correct extension depending upon what was chosen in the Input Type field above.
Output File Name	The name of the .dac file to be regenerated must be entered here. By default it will have the same name as the input file, but a .dac extension. MREGEN will ask for permission before overwriting any existing files.
Bin value Position Extreme\ Mean\Random	When a matrix element is selected to reconstruct a time history value, the value itself may be determined from any part of the bin. Three cases are offered here: Select the extreme edge of the bin (maximum for peaks, minimum for troughs), the mean value (centre of the bin), or a random location.

Option	Description
Sample Rate	This field specifies the sample rate of the output .dac file. Because the rainflow process loses all time information it is necessary to supply it in this field, which will allow a new time series (.dac) file to be created. Note that the new time series is really a peak valley sequence with a single cyclic frequency equal to half the supplied sample rate.
Seed	The number seed can be one of two types: Random or User defined. If it is random then the program will generate a pseudo-random number as the seed (it will in fact repeat after a very large number of cycles). For repeatable results it should be the same seed value for each run of the program, hence the option of the user entering a seed number.
Seed Value	This field only appears if Seed is set to User generated. It has been proved that the best seed is an odd number.

Max-Min Rainflow Matrix

The input fields for this option are all the same as for Range-mean Rainflow Matrix, so please refer to the previous text for details of how to complete them.

Transition Markov Matrix

This option requires the following input.

Option	Description
Input File Name	Enter the name of the matrix from which a time history is to be regenerated. The default file extension is .mkh.
Output File Name	Enter the name of the time series file to be created. If this file exists, the program will ask whether the file may be overwritten. The default extension for this file is .dac.
Sequence length Points	Enter the number of points to be recreated. For reconstruction from a Markov matrix, the default value will reconstruct a sequence containing the same number of peaks and troughs as the input matrix.
Bin Value Position Extreme\ Mean\Random	When a matrix element is selected to reconstruct a time history value, the value itself may be determined from any part of the bin. Three cases are offered here, to select the extreme edge of the bin (maximum for peaks, minimum for troughs), the mean value (centre of the bin) and a random location.

Option	Description
Sample Rate	This field specifies the sample rate of the output .dac file. Because the rainflow process loses all time information it is necessary to supply it in this field, which will allow a new time series (.dac) file to be created. Note that the new time series is really a peak valley sequence with a single cyclic frequency equal to half the supplied sample rate.
Seed	The number seed can be one of two types: Random or User defined. If it is random then the program will generate a pseudo-random number as the seed (it will in fact repeat after a very large number of cycles). For repeatable results it should be the same seed value for each run of the program, hence the option of the user entering a seed number.
Seed Value	This field only appears if Seed is set to User generated. It has been proved that the best seed is an odd number.

Irregularity Factor

This option requires the following input.

Option	Description
Output File Name	Type the name of the output .dac file that will be created.
Irregularity Factor	Enter the irregularity factor from which a transition matrix will be created. The irregularity factor is defined as the number of mean crossings divided by the number of peaks. The value lies in the range, 0.0 to 1.0 where 1 implies pure narrow band.
Y-Axis Label / Units	Type the Y-axis label and it's units (if any) of the output .dac file.
Maximum	Reconstruction from an irregularity factor generates a normalized probability matrix. To calibrate the time history, specify the maximum value which the time history may attain.
Sequence length	Enter the number of points to be recreated.
Sample Rate	Type the output file's sample rate (sample rate information is not part of the 3 parameter .cyh file). Note that the new time series is really a peak valley sequence with a single cyclic frequency equal to half the supplied sample rate.
Seed	The number seed can be one of two types; Random or User defined. If it is random then the program will generate a pseudo-random number as the seed (it will in fact repeat after a very large number of cycles). For repeatable results it should be the same seed value for each run of the program, hence the option of the user entering a seed number.
Seed Value	This field only appears if Seed is set to User generated. It has been proven that the best seed is an odd number.

Batch Example

A sample batch line is given below.

```
mregen /typ=r/inp=aero/out=reg_aero/sam=0.5/stype=r
```

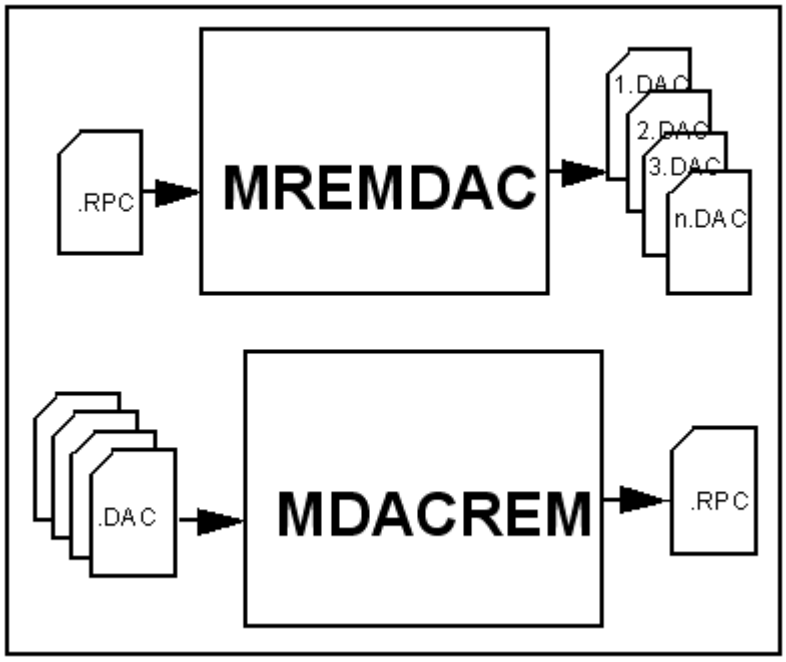
Which will input `aero.cyh` and output a `.dac` file.

A list of MREGEN's batch keywords:

/OPT	Input Type R,M,T,I,X
/INP	Input File Name. /INP=`results.cyh`
/OUT	Output File Name. /OUT=`newfile.dac`
/BINPOS	Bin Position - E,M,R
/SAM	Sample Rate in samples per second. /SAM=50
/STYPe	The type of random number seed; Random R or User defined U. /STYP=R
/SEED	The random number seed when STYP=U. /SEED=3
/SEQ	Sequence length
/BINPOS	Bin Position - E,M,R
/IRR	Irregularity Fact
/LAB	Y-Axis Label
/UNI	Y-Axis Units
/MAX	Maximum value
/MIN	Minimum value
/OV	Overwrite, Y/N

RPC to DAC - DAC to RPC - (MREMDAC/MDACREM)

The MREMDAC program extracts channels of data from MTS RPC^{tm} remote parameter (RPC) files, and creates a single `.dac` file for each channel of the RPC file. It is possible to selectively extract one or more channel numbers. A time window within the RPC file may be selected which will apply to all channels extracted. Both RPC II and RPC III files may be processed. Full details of the extraction and creation process can be saved to a report file.



The MDACREM program combines a set of related .dac files into a single RPC file ('remote parameter') suitable for driving a test rig. The .dac files are often individual channels of a multi-channel data acquisition test.

Note:	Damage may occur to a test rig if it is driven by an unsuitable *RPC* file. When creating an *RPC* file it is strongly recommended that the file is tested for validity within the RPC software suite before the file is used to drive a test rig.

There are several different forms of RPC files, particularly for RPC II. The subsections below briefly describe the types of file that can be processed by this program (see the MTS documentation for full details).

RPC II File Types

The files that can be processed are demultiplexed drive or response time histories. The data type must be 16 bit integer. This corresponds to MTS file types 00 or 100 with data type 00.

RPC III File Types

Any drive or response time history file can be processed.

Module Operation

The MREMDAC and MDACREM modules can be run in one of the following three modes:

- From the **File Conversion Utilities** pulldown menu under **Tools | MSC Fatigue** (for Patran) or under **Tools | Fatigue Utilities** (for).

- In stand alone mode by typing **mremdac** or **mdacrem** at the system prompt
- By incorporating commands in a batch operation

RPC to DAC Conversion (MREMDAC)

When first invoked, MREMDAC, only the Remote Parameter File field will appear. When a name has been supplied the rest of the fields appear.

Option	Description
Remote Parameter File	The first field is the name of the RPC file to convert. The default extension for the RPC file name is `.rsp`, and the file is assumed to reside in the current working directory. A file name may be picked from a list by clicking on the pick list button. Once the RPC file name has been entered, the remainder of the input fields are displayed on screen. These fields are described below.
Data Channels	This field allows the channels required for extraction to be specified. Channel numbers may be entered directly into the field by entering them in the form `1,3-5,10-15 etc.` where the hyphen indicates an inclusive range of channel numbers. The word 'ALL' may be entered in the field which indicates that all the channels in the RPC file should be processed. Click on LIST to pick one or more channel numbers from a list of those which exist in the RPC file. In this case a screen will be presented. The vertical list shows all the channels which exist in the RPC file. One or more channels may be individually tagged (selected with the mouse). Pressing the T key over an already tagged channel will untag it. Use the A key to tag all channels, or U to untag all channels. Select the Accept option when all the required channels have been tagged.
Output File Root Name	One output file is created for each channel extracted from the input file. The output file names are constructed as `<root name> <channel number>.dac` where the `<root name>` is the name entered in this field. This name will be the same for all the output files. The channel number occupies two or three digits, padded with a preceding zero if required.
Output File Extension	This field allows the extension of the output files to be specified. The default offered is `.dac`, which is the standard extension of MSC Fatigue time series data files. Any other three letter extension may be entered if required.
Report Selection	If this field is set to Yes, then a report file will be generated during processing, which contains a list of the channels which have been extracted, with their channel descriptions and units. Also a table of the statistics of each of the extracted channels is written to the file. The report file will have the same name as the root name specified above, but a file extension of `.lst`. It is a text file which may be printed if required.

Option	Description
Window Selection Method	It is not necessary to extract the complete signal for each channel from the RPC file. A time window for extraction can be specified by entering the start and end position. The start and end position may be specified either as times in seconds measured from the start of the data file, as point numbers counted from the start of the data file, or as frame numbers, also counted from the start of the data file. At the bottom of the screen the length of the data file in terms of time, points and frames is displayed. This field requests the method by which the time window is to be selected.
Start Position	This field will actually request Start Time, Start Point Number or Start Frame Number, depending on the setting of the previous field. Either enter the required start position in the appropriate units, or enter the word START which indicates the start of the data file.
End Position	This field will actually request End Time, End Point Number or End Frame Number, depending on the setting of the Window Selection Method field. Either enter the required end position in the appropriate units, or enter the word END which indicates the end of the data file. It is not possible to enter offsets from the end of the file (using END-5 etc.). When this screen is accepted, the processing will begin. As each channel is processed its channel number and description is displayed at the bottom of the screen. Also, when each channel is completed, the statistics of the created .`dac` file will be displayed. If an output file is already found to exist during processing, then a box will be popped on screen asking whether to overwrite the existing file or not.

Processing begins when all input is entered, then channel-by-channel. If an existing file exists or a channel will be overwritten you will be prompted for overwrite permission. A summary screen is presented after processing is done showing details of the operation.

DAC to RPC Conversion (MDACREM)

This following input is required for this conversion.

Option	Description
Input Test Name	The .dac files which are to be combined should all be different channels of the same test. The name of each file should follow the convention below. <test name> <channel number>.dac where the <test name> is the same for all the .dac files. The <channel number> should be at least two characters and is padded with a leading zero if required, i.e. test01.dac would be channel 1 for test name test. The test name for the required set of input .dac files should be entered in this field.
Data Channels	As described in the previous section, the names of the input .dac files should be related by a common test name. Enter in this field the channel numbers of this test that are to be processed. The channel numbers should be entered in the form 1,3-15,21-30 etc., where the hyphen indicates an inclusive range of channel numbers. Alternatively, the word 'ALL' may be entered in the field, and the program will locate and process all the .dac files which match the specified test name. Please note that all the .dac files specified for input must have the same sample rate and number of points.
Titles/Units Source	Select the source of the channel titles and units. Header will get them from the input .dac file headers. Extra Details will get any extended titles and units from their extra details areas.
Output File Name	This field specifies the name of the output RPC file to create. This file will be created in the current working directory unless the name is preceded with a directory specification. The name should not include a file extension. If the name of an existing file is specified, then a question box will appear on screen asking whether the existing file should be overwritten or not.
Output File Extension	This field specifies the extension which will be attached to the output file. The default offered is .rsp.
RPC Type	The program can create either RPC II or RPC III files. Select the required file type here. If RPC III is selected, then the last 4 fields on this screen are not offered, since they do not apply to RPC III files.
Test Title	The test title is a textual description of the data in the file. It is stored in the header of the output RPC file.
Report Selection	If this field is set to Yes, then a report file will be generated during processing which contains a list of the channels processed, their descriptions, units and statistics. The report file is a text file which may be printed if required. The program will inform the user of the name of the report file when processing is complete.
Points/Frame	RPC files are organized in 'groups'. Each group contains 1, 2 or 4 frames, and each frame contains a number of points which is a power of 2 between 256 and 8192. The file must always contain a whole number of groups. Select the number of points per frame from the list offered.

Option	Description
Frames/Group	As described in the previous paragraph, each group within the file must contain either 1, 2 or 4 frames. Select the required number of frames per group from the list.
Window Selection Method	It is not necessary to process the entire length of the input .dac files. A time window can be selected for processing by entering a start and end position.This field controls the method by which the start and end positions are set, which may be Time, measured in seconds from the start of the .dac files, or Point numbers, counted from the start of the .dac files.
Start Position	This field will either request Start Time or Start Point Number, depending on the setting of the previous field. Time is measured in seconds from the start of the data file. Point numbers are counted from the start of the data files. Alternatively, the word 'START' may be specified indicating the start of the data file. Please note that the 'START+' construct used elsewhere in MSC Fatigue does not work here.
End Position	This field will either request End Time or End Point Number, depending on the setting of the Window Selection Method field. Times or point numbers are measured from the start of the data files as described above. Alternatively, the word 'END' may be specified indicating the end of the data files. Please note that the 'END+' construct used elsewhere in MSC Fatigue does not work here.
RPC II only fields, The fields below appear if RPC II files are being created	
Maximum Number of Characters in Channel Description	This field and the following three are only used for RPC II files. These files allow the maximum length of the channel description storage area to be set at the time of file creation. A number between 1 and 80 may be specified.
Maximum Number of Characters in Units Description	RPC II files allow the maximum length of the channel units description storage area to be set at the time of file creation. A number between 1 and 80 may be specified.
Clock Pulses per Second	This field is the clock pulses per second, which may have a value between 1 and 30 000. The sample rate per point of RPC II files is held in the form of a clock speed and a divider. The clock speed, in pulses per second should be entered in this field and the divider (pulses per data point) in the following field. The clock speed divided by the divider value should be equal to the sample rate in points per second. The defaults offered by the program are calculated from the sample rate of the input files, but may be changed if required.
Clock Pulses per Point	As described in the previous paragraph, this field and the clock pulses per second define the sample rate of the RPC file. Enter a number between 1 and 999 in this field. If the values entered in these two fields are not equivalent to the sample rate of the input files, then a message box will be popped on screen. A choice will be offered of whether to continue or re-select the clock settings.

Incomplete Data Groups

Data in RPC files are stored in groups, each group containing 1 or more frames; only complete groups can be stored. However, sometimes the combination of points and groups, divided by the total number of points in the .dac files, mean that the last group in the RPC file will not be completely filled. In such cases the last group must be padded out with a series of values, or discarded.

If the last group is incomplete the Incomplete Data Group window is shown.

The padding options are:

- Zero - All extra values are set to zero
- Last - All extra values are set to whatever is the last value in that channel
- Mean - All extra values are set to the mean value of that channel
- Truncate to last full group, i.e. discard the unfilled group
- Linear Ramp from last to start value
- Haversine join (a cosine wave form) from last to start value

When all the parameters have been set and accepted, processing will begin.

As each channel is processed, the program will display its description, units, and the statistics of that channel. If the report file option was set to Yes, then a report of the processing will be generated. If the MSC Fatigue notebook system is active, then this report is written to the notebook file, otherwise the report is written to a file which has the same name as the output RPC file but an extension of .lst. A message will be displayed when processing is complete describing where the report has been written.

Extra Details

The .dac files only allow 24 characters for channel descriptions and 28 characters for the channel units as stored in the header. RPC files may have more characters than this for both description and units. The description and units written into the .dac file header is a truncated form of the RPC file description and units. However, the full description and units are written into the extra details area of the .dac files under the keywords $YTITLE and $YUNIT.

In addition, the test title as stored in the header of the RPC file is written into the extra details area under the keyword $RPTITLE.

Batch Mode Operation

A sample batch line is given below:

```
mdacrem /inp=test/out=result/win=p/fill=m/ov=y/cho=c/fra=256
```

This will use test as the generic input (test101, test102, etc.) with ALL channels being processed by default. The file name results is the output file name (results.rsp and results.lst). The window selection is points (/win=p) and the frame size 256 points.

If the combination of points per group and the total number of points to be processed result in the last group being incomplete then a fill value must be supplied (/fill=m in this example). In the event of anomalies such as incomplete groups, MDACREM must be told to continue despite the incomplete group by the /cho=c command; otherwise it would abort the batch run.

A list of MREMDAC's batch keywords:

INPut	Input RPC file name. /INP=test
CHAnnels	Channel Numbers. /CHA=all
OUTput	Output File root name. /OUT=test
EXTension	Output file extension. /EXT=DAC
REPort	Report Selection, yes (Y) or no (N). /REP=Y
WINdow	Window Selection Method T=time, P=Points, F=Frame. /WIN=P
STArt	Start Position. /STA=10
END	End Position. /END=30
OVerwrite	Overwrite Output Files Y=yes, N=no, A=all, E=end. /OV=A

A list of MDACREM's batch keywords:

INPut	Input (DAC) Root File Name, /INP=test
CHAnnels	Channel Numbers, /CHA=1-10
OUTput	Output RPC File Name, /OUT=test
CHOice	Whether to proceed if channels not contiguous, /CHO=Y
EXTension	Output File Extension, /EXT=RSP
REPort	Report Selection, /REP=Y
FRAme	Frame Size, /FRA=512
GRP	Frames per Group, /GRP=2
TITle	Test Title, /TIT=Test title
WINdow	Window Selection Method, /WIN=P
STArt	Start Position, /STA=10
END	End Position, /END=20
OVerwrite	Overwrite Output File, /OV=Y
TYPe	RPC File Type, /TYP=3
CD	Maximum Length of Channel Description, /CD=40
EU	Maximum Length of Channel Units, /EU=40
PULSEC	Pulses per Second, /PULSEC=10000
PULPNT	Pulses per Point, /PULPNT=16
SCAL	Scale Factors, /SCAL=O
FILL	Fill value for missing points, Z, L, or M, /FILL=M

Regenerate Time Histories from PSDs

To regenerate a time signal from the frequency domain ideally we would perform an 'Inverse Fourier Transformation' on the complex vector of frequency domain results. This would yield exactly the same time history as we started with. However when starting with a PSD this method is inappropriate because the PSD does not contain any of the original phase information. For certain time histories, however, we are able to make assumptions about the original phase content and therefore regenerate a statistically equivalent time history. The assumption we make is that the time history is taken from an ergodic stationary Gaussian random process.

The method used by MREGEN to generate a time signal is as follows:

1. Generate a white noise signal of a sample rate and length proposed by the user. White noise is a random signal with equal amplitudes at all frequencies up to the Nyquist frequency (half the sample rate). The minimum sample rate allowed is twice the maximum frequency in the PSD file. There is also a minimum signal length, which is related to the FFT buffer size.
2. Create a "gain" file based on the PSD input. If the PSD input is in XY format, or if the specified sample rate or buffer size is different from the suggested default, some interpolation will be required between the points of the PSD, and the square root of the interpolated file is taken to produce a gain file. Interpolation can be linear or logarithmic.
3. The gain file is then used in a Fast Fourier Filtering operation applied to the white noise signal.
4. Finally, the amplitude of the filtered signal is adjusted to ensure its RMS matches the area under the PSD, and any required mean offset is added.

The output of this process will be a signal whose PSD approximately matches that of the input PSD. Since the signal is based on a sample of white noise, and since the PSD represents an average over time, longer time signals will provide a closer match to the original signal.

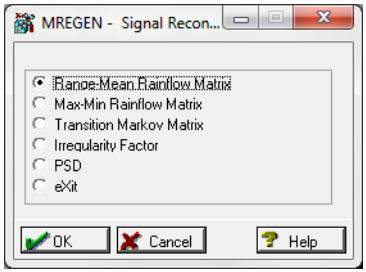
Caution:	Since an "inverse fourier transform" is not being performed, a limitation exists in algorithm defined above. If your input PSD file (and hence the gain file) includes a sharp edge, slight discontinuities will be produced at the end of each buffer. This is due to the sharp "edge" in the time domain translating to a "ringing" in the time domain. If the input PSD is a very sharp spike, defined by only one or two points, the effect can be more serious.

Module Operation

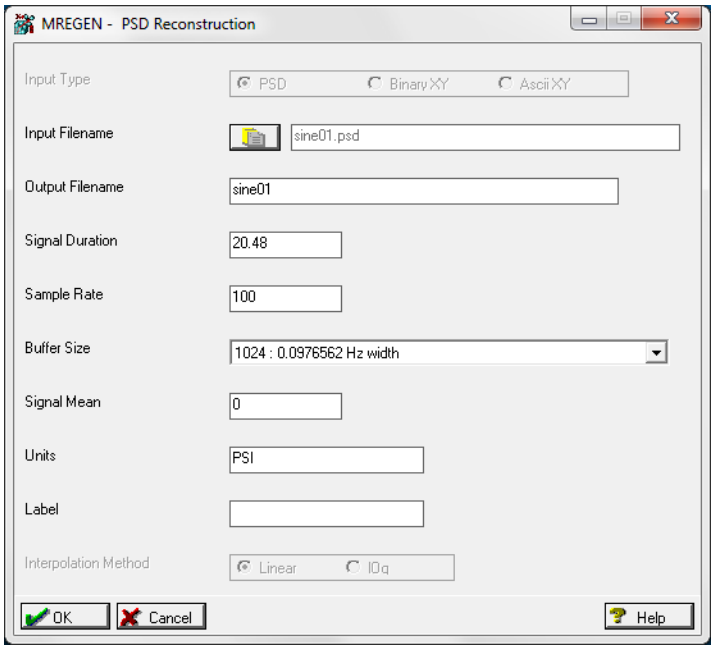
The module can be operated by typing, MREGEN, at the system prompt or by invoking it from the **File Conversion Utilities** pulldown menu under **Tools | MSC Fatigue** (for Patran) or under **Tools | Fatigue Utilities** (for).

There are 2 main steps to operating **MREGEN - Signal Regeneration** Menu.

1. Select the **PSD** Option from the **MREGEN - Signal Regeneration** Menu.



2. Tell the the type and name of the file that is to be converted, and input values for other options in the **MREGEN - Signal Reconstruction** form shown below:



MREGEN_ PSD Reconstruction Form Entries

This following input is required for the regeneration.

Option	Description

Input Type	Inputs can be an nCode format PSD file (e.g. generated by MASD) or binary or ASCII xy file. The ASCII file needs two columns of data, representing frequency (in Hz) and PSD value, and the units should be defined. The first point MUST be at zero frequency. An example file follows (lines preceeded by # are optional):
	
	# ASCII X-Y PSD file
	# Lines preceded by # are comments
	# Data is XY points - Frequency, PSD
	UNITS=MPa^2/Hz# for example
	0, 0
	5, 1
	6, 10
	10, 10
	12, 2
	14, 0
	16, 4
	17, 2
	20, 0

Input Filename	Input or select the filename to be processed. Default filename extensions are .PSD, .MDF, and .ASC for the three file types.
Output Filename	Enter the name of the time series file to be created. If this file exists, the program will ask whether the file may be overwritten. The default extension for this file is .dac
Signal Duration	Enter here the signal duration (in seconds). The default signal length is the minimum recommended, based on an assumed FFT buffer size. Bear in mind that what is generated by this module is a random sample representing a stationary random process. The longer the sample, the better the representation of the process.
Sample Rate	The sample rate of the generated signal must be at least twice the maximum frequency in the PSD (the default). This is necessary to avoid aliasing. If the signal is to be used for fatigue analysis, a frequency of at least 10 times the maximum frequency of interest in the PSD is recommended.
Buffer Size	This is the size of the FFT buffer used in the filtering operation, and controls the resolution of the filtering operation.
Signal Mean	Enter the desired mean of the output signal.
Units	Enter the units of the output signal.
Label	Enter the Y-axis label for the output signal.
Interpolation Method	Select the interpolation method where appropriate.

Cross-Platform Conversion - (MCONFIL)

MCONFIL is a MSC Fatigue utility module that converts a wide range of data files between a variety of hardware platforms and operating systems. MCONFIL converts files so that they can be processed on hardware platforms and operating systems other than that on which the file was originally created. MCONFIL is capable of changing the format of binary files so that such files can be used on any of the range of machines supported by MSC Fatigue.

A typical usage situation might be one whereby data is collected in the field with a PC running MS-DOS, but is to be analyzed on a workstation running UNIX. MCONFIL can convert the MS-DOS file to a format usable by the UNIX machine (or the other way around if necessary).

Another scenario is where the conversion may need to be carried out on a VAX so that the data files can be written to standard 6250 bpi computer tapes. MCONFIL supports binary conversions of this kind where the computer on which the conversion is carried out is different from either the source or target computer. MCONFIL can process the following file types:

File Type	Typical File Extension
Demultiplexed Data	DAC,KSN, PSD
Time History Database	TDB
Materials Database	MDB

The above file types could have been produced on any of the following hardware platforms/operating systems:

RSX11M	(for the DEC PDP range of minicomputers)
MS-DOS	(for DEC VAX computers)
UNIX	(for the IBM PC and compatibles)
UNIX	(Masscomp workstations)
UNIX	(Apollo workstations)
UNIX	(Hewlett Packard)
UNIX	(Sun)
UNIX	(SGI)
UNIX	(IBM 6000)
UNIX	(DECstation)

The files may be converted to be usable on any of the same hardware platforms/operating systems.

Module Operation

The module can be operated by typing, MCONFIL, at the system prompt or by invoking it from the **File Conversion Utilities** pulldown menu under **Tools | MSC Fatigue** (for Patran) or under **Tools | Fatigue Utilities** (for).

There are 3 basic steps to operating MCONFIL:

1. Tell MCONFIL the name and the type of file that is to be converted (i.e., name the input file).
2. Tell MCONFIL the hardware platform/operating system the input file was created on.
3. Tell MCONFIL on which hardware platform/operating system that the output file will be configured to run under.

There are other details that MCONFIL needs to know, such as the name of the output file and whether or not existing files can be overwritten, etc.

Important:	Converting (and overwriting) an already converted `.dac` file is possible. For example, a user may convert from `afile.dac` for DOS to `afile.dac` for UNIX, and then, if they forget that the conversion has already been done, inadvertently attempt to convert the UNIX version to UNIX. This release of CONFIL will not prevent the user from corrupting files in such fashion. It is up to the user to keep track of file names and file types.

These are the steps that must be followed when operating MCONFIL in interactive (nonbatch) mode.

Naming the Input File (Screen 1)

When MCONFIL is first invoked a screen appears. It is here where the input file to be converted is identified. It is necessary to type the full file name and its file extension. Only those file types named previously can be processed. Input file names may be specified with wildcards (an asterisk "*").

When the file name has been input, the user should confirm the choice by pressing **OK, F1** or **ENTER**. This will cause the next screen to be displayed.

When the input file has been named, the user is prompted to supply the output file name. By default, the output file name is the same as the input file. If the same name and directory are specified then MCONFIL will prompt for permission to overwrite the input file. If the input file is to be preserved then the user should select another name or directory for the output file.

From which File Format to what File Format (Screen 2)

The second is screen where the user tells MCONFIL the hardware platform/operating system the input file was created on, and which hardware platform/operating system that the output file will be configured to run under.

The numbers at the side of each option are the MSC Fatigue designation codes for that option. These numbers are used when operating MCONFIL in batch mode.

To specify format from and format to, a selection must be made from at least two of the three menus. The first menu's (left of the screen) choice was effectively made when the file extension was specified on the previous screen, although it can be changed here. The second menu (middle) is the menu upon which the user must specify the operating system that the input file was created under. This is the file source operating system.

The third menu (right of the screen) is the menu upon which the user must specify the operating system that the output file will be created to run under. This is the file destination operating system.

The easiest way of selecting the source and destination operating systems is to point and click the mouse pointer. So if the user wishes to select MSDOS as the source and VMS as the destination, they need only to click on MSDOS and VMS in the appropriate menu.

When using the keyboard, the left/right arrow keys move the highlight bar between the menu boxes.

The up/down arrow keys move the highlight bar up and down within menus.

Pressing **ENTER** selects an option within a menu.

When all selections have been made, clicking **OK** or **F1** causes MCONFIL to start the conversion. MCONFIL tells the user about the progress of the conversion with an on-screen % converted meter.

If no selection has been made from a menu then MCONFIL will take the current or last position of the highlight cursor as the selection.

When conversion is complete, MCONFIL offers the user the chance to perform another conversion.

Batch Operation

Batch operation has the following batch keywords:

/INPut	The name of the file to be converted (/INP=FILENAME.EXT)
/OUTput	The name of the post conversion output file (/OUT=NEWNAME.EXT)
/TYPe	Select the type of file to be converted. Files can be one of the following: `.plt, .dif, .mux`. Standard single channel data files: `.dac, .acf, ada, .asd, ccf, coh, csd, .gai, .pha, .pvx,.sxx, .syy, .sxy, .ksn` (all type 11 files) (/TYPE=11 (the numbers correspond to those on the menu)) Note that this is actually a character string and not a number, so if the pure number does not appear to be working, try typing in the entire string as it appears in the interactive window of MCONFIL, e.g. 7-TDB.)
/SOUrce	Select the source operating system: DEC RSX11M, DEC VAX, MS-DOS, UNIX (APOLLO, SUN, HP, MASSCOMP, SGI, IBM, DEC).(/SOU=3 (the numbers correspond to those on the menu))
/DEStination	Select the source operating system: DEC RSX11M, DEC VAX, MS-DOS, UNIX (APOLLO, SUN, HP, MASSCOMP, SGI, IBM, DEC).(/DES=3 (the numbers correspond to those on the menu))
/OVerwrite	Is used in both generic and individual cases. If the output file(s) exist, then you can choose to overwrite them with this keyword. (/OV=Y)
/AGN	Convert another file - Yes or No, the default is No to prevent MCONFIL looping. (/AGN=Y or N)

Example:

```
mconfil /inp=saetrn/out=saetrn/type=11/sou=2 /des=7/ov=yes/agn=n
```

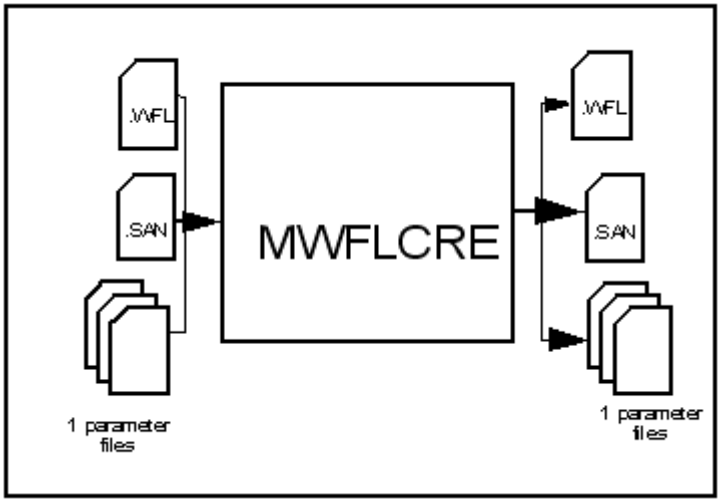
This batch line would convert `saetrn.dac` which is a type 11 file (demultiplexed data) from a VAX (/SOU=2) compatible operating system to a Sun (/DES=7) operating system and give it the same name, overwriting the existing file. The program would then quit and not perform another conversion (/AGN=N).

Important:	The choices that appear in the /TYPe keyword and subsequently on the CONFIL interactive window are dependent on a configuration file called `confil.sys` found in the `nssys` directory of the MSC Fatigue system. It can be altered to include or exclude different supported file types. All file types that are used by MSC Fatigue are included so this should not be necessary.

Waterfall File Create - (MWFLCRE)

Module MWFLCRE has four functions. These are:

- Combine single parameter files into three parameter waterfall (.WFL) files
- Split .WFL files into multiple single parameter files
- Convert a .WFL file into a .SAN histogram file
- Convert a .SAN histogram file into a .WFL file



The above functions can, for example, enable the data contained in .WFL files to be operated by analysis modules that cannot normally operate on waterfall data, but which can operate on single parameter files.

1. Combining single parameter files into 3 parameter waterfall (.WFL) files - Waterfall file creation.

   This part of the program combines 2 or more spectral density files in a .WFL file. The spectral density files are single parameter and typically have an extension of .PSD, .ASD, .DAC, or .AMP. They can be made by programs such as ASD.

2. Splitting .WFL files into multiple single parameter files.

This function generates output files from an input waterfall file. The spectra at a single speed can be stripped out of a waterfall file but more probably a series of spectra can be stripped out into a series of 1 parameter files. The generic name of the output files is specified by the user but they will normally take their name from the input file with numbers appended to them.

The number of .ASD files created will depend upon the Z-axis range specified by the user multiplied by the incremental spacing within that range. For example if the Z-axis speed specified is from 0 to 50 , in increments of 1, then 50 ASD files will be created. If the increment is 0.1 then 500 ASD files will be created (50 / 0.1). If the increment is 10 then only 5 ASD files will be created.

It is up to the user to decide how many 1 parameter files are to be created, although even if the entire input .WFL file were split, the sum total of the output files will usually only be about 25% larger then the original waterfall file size.

3. Converting a .WFL file into a .SAN histogram file.

   This option converts the input file into a single .SAN histogram file. The .SAN file can be manipulated as a standard histogram file, but not plotted using P3D.

4. Converting a .SAN file into a .WFL file.

   This option converts the input file into a single .WFL file.

Option 1: Create .WFL from individual files

This option merges a range of single parameter files into a waterfall file. The filename definition screen contains the following fields.

Option	Description
Input Filename Extension	As mentioned, the single parameter input files typically have an .ASD extension. However if the files to be merged have a different extension, such as .DAC, then the extension should be entered in this field.
Output Filename	This is the name of the output .WFL file. It will be the sum of all the input files.
Input Filename(s)	Here is where the input files are named. Their names can be entered in full, or a generic name can be entered. For example ORD1P3(1-20) will input ORD1P301, ORD1P302, ORD1P303...to ORD1P320, i.e. 20 files. A maximum of 128 input files can be specified. A minimum of 2 files must be specified.

The final screen of waterfall construction enables the user to label the axes of the output waterfall file. The defaults for the frequency and Z axes are read from the header area of the 1st input file, although they can be edited. When this screen has been accepted MWFLCRE will process the input files, create the output file, and return to the Main Menu.

Option 2: Split .WFL to .ASD files

This option is the opposite to create .WFL form .ASD. When selected the following screen will appear. The field must be completed as follows.

Option	Description
Input filename	This is where the input file is named. It has to be a waterfall file format, although if its file extension is not .WFL, but its format is that of a waterfall file then it can be processed. The full file name (including extension) only has to be entered if the extension is not .WFL.
Output filename	This is where the generic name for all the output files is entered. Note that the final filename cannot be more than 8 characters long, so entering TESTFIL as a generic name and generating more than 9 files will produce an error message because TESTFIL10 contains too many characters.
Output extension	Any extension can be specified here, although .PSD is the default extension.
Split from/to	Whatever is the range of values and increments along the Z-axis, a subset of those values can be entered in these fields. These 2 fields allow the input file to be split selectively, i.e. not all the Z-axis values have to be processed (every Z-axis value will produce and output file). Only numbers are allowed (not START, or END etc.). The exact number of output files created is dependent upon the range and its incremental spacing. See Technical Overview above (split WFL files into multiple 1 parameter files) for a full explanation. When the above fields have been filled and accepted MWFLCRE returns to the Main Menu.

Option 3: Convert .WFL file to .SAN file

The only input that the user has to make is to name the input waterfall file and its corresponding output file.

Option 4: Convert .SAN to .WFL file

The only input that the user has to make is to name the input waterfall file and its corresponding output file.

Batch Operation

A list of MWFLCRE's batch keywords:

/EXTension	The file extension
/OUTput	The output file name

/OVerwrite	Whether to overwrite an existing output file
/INPut	The input file names
/SPEed	The speed values
/SMINimum	Minimum speed to fitS
/MAXimum	Maximum speed to fit
/FLABel	The frequency labe
/lFUNIts	The frequency units
/SLABel	The speed label
/SUNIts	The speed units
/ZLABel	The Z data labe
/lZUNIits	The Z data units

15 Validation Problems

- Introduction 1222
- Problem 1: Analysis of a Keyhole Specimen 1223
- Problem 2: Analysis of an Underwater Pressure Vessel 1262
- Problem 3: Comparison to Another Code 1276
- Problem 4: Ten Simple Notched Geometries 1286
- Problem 5: Transient Results 1296
- Problem 6: Spot Weld Analysis 1305
- Problem 7: Vibration Fatigue Analysis 1311

Introduction

This chapter includes several problems used to validate MSC Fatigue. Although not specifically structured as tutorial example problems, the results from each can be reproduced and the files necessary to do so are included with the MSC Fatigue installation media.

For a complete tutorial and a quick start to learning and using MSC Fatigue, please refer to the MSC Fatigue Quick Start Guide.

There are validation seven problems with the first being the most significant by validating the methodologies behind the three basic analysis types: S-N, ε -N (crack initiation), and LEFM (linear elastic fracture mechanics - crack growth) with comparisons to measured data. Comparisons to hand calculations and another fatigue code are included in subsequent problems. The later problems validate different mechanisms of solving the different analysis types such as the difference between using linear static results versus a direct transient analysis or using a frequency approach versus the time domain as is done with vibration fatigue problems.

The problems are laid out as follows:

1. Problem 1: Analysis of a Keyhole Specimen, 1223
   - S-N Analysis of Keyhole, 1225
   - Crack Initiation of Keyhole, 1235
   - Keyhole Results, 1250
2. Problem 2: Analysis of an Underwater Pressure Vessel, 1262
   - Crack Initiation of Weld, 1263
   - Crack Growth of Weld, 1270
3. Problem 3: Comparison to Another Code, 1276
4. Problem 4: Ten Simple Notched Geometries, 1286
5. Problem 5: Transient Results, 1296
6. Problem 6: Spot Weld Analysis, 1305
7. Problem 7: Vibration Fatigue Analysis, 1311

Problem 1: Analysis of a Keyhole Specimen

Problem Description

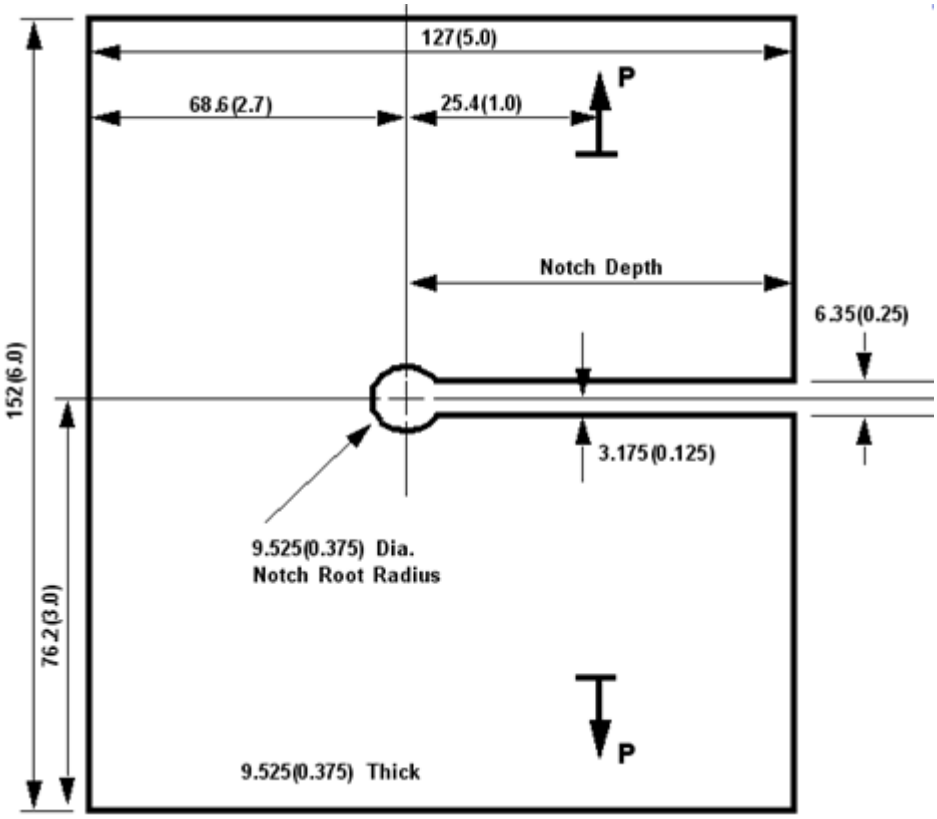


Figure 15-1 SAE Keyhole Specimen in Millimeters (Inches)

This first problem is concerned with the fatigue analysis of a simple "keyhole" specimen. The geometry is shown in Figure 15-1. All three standard fatigue analyzers (Stress-Life {S-N}, Strain-Life {ε -N} and Crack Growth {LEFM}) are demonstrated with three different service load histories and two different materials. The results are then compared to test data accumulated during The SAE Cumulative Fatigue Damage Test Program conducted from 1970-1974. See References (Ch. 17).

In a series of experiments, different constant amplitude cyclic loads were applied to the specimen until it broke into two pieces. To accumulate S-N data, the stress was monitored at the root of the notch (1/2 radius from the hole along the line of direction of the notch) and a stress-life line determined by regression analysis of the scattered points. Two different materials were used for this specimen, MANTEN and RQC100, leading to two S-N data sets as listed in Table 15-1.

These materials were also tested using a special test specimen under strain control to obtain cyclic stress-strain and strain-life scatter plots. A regression analysis was performed to calculate the local strain parameters listed also in Table 15-1 as well as the crack growth properties. All of these values are stored in the MSC Fatigue materials database.

Table 15-1 Fatigue Material Properties: MANTEN, RQC100

Material Properties	MANTEN	RQC100
S-N Properties		
Stress Range Intercept, SRI1 (MPa)	3162	4680
Slope, b1	-0.2	-0.216
Transition life, NC1 (cycles)	2E8	1E8
Slope, b2	0	0
Fatigue limit, FL (MPa)	10	10
Standard error, SE	0.137	0.433
Strain Properties		
Fatigue strength coefficient, Sf'	917	1158
Fatigue strength exponent, b	-0.095	-0.075
Fatigue ductility coefficient, Ef'	0.26	1.06
Fatigue ductility exponent, c	-0.47	-0.75
Cyclic strain hardening exponent, n'	0.19	0.1
Cyclic strength coefficient, k' (Mpa)	1103	1151
LEFM Properties		
Unnotched fatigue strength, FL (MPa)	226	410
Paris law coefficient, C (m/cycle)	3E-12	10E-11
Delta K threshold at R=0, D0 (MPa m1/2)	8	5.35
Delta K threshold at R->1, D1 (MPa m1/2)	2	2
Stress ratio at threshold knee, Rc	0.75	0.5
Stress corrosion th'hld, K1SCC (MPa m1/2)	121	112.8
Monotonic Properties		
Young's Modulus, E (Mpa)	2.034E5	2.034E5
UTS (MPa)	552	863

S-N Analysis of Keyhole

Objectives

1. To calculate the fatigue life for a component made from MANTEN under a "transmission" loading whose maximum is 3.5 Kips using the Total Life method.
2. To investigate the effect of scatter on the S-N curve via the design criterion.
3. To investigate the effect of mean stress on fatigue life.
4. To make a comparison between the two materials MANTEN and RQC100 under the above conditions.
5. To compare predicted lives to measured lives.

Step 1: Geometry and FEA Results

The original FE stresses used in the SAE program were determined using MSC.Nastran. In our example, we use Patran FEA and validate the results. Therefore, it is advantageous to plot the results of the FE analysis. Figure 15-2 show maximum principal stress as a function of the distance from the notch. The original MSC.Nastran finite element analysis and actual measured notch root strains were used to describe the relationship between applied load and local notch root strain in the SAE program. The following shows that Patran FEA can reproduce these same results. The relationship is expressed as

$$e_r = \frac{P}{C_1} + \left(\frac{P}{C_2}\right)^{1/d} \tag{15-1}$$

e_r		is the notch root strain amplitude
P		is the applied load
C_1		is a material constant
C_2		is a material constant
d		is a material constant

Elastic analysis showed that the relationship between nominal elastic notch root stress, S_n, and applied load, P, can be described by

$$S_n = mP \tag{15-2}$$

m		11.24 MPa/kN (7.25 ksi/kip)

From the above stress concentration, K_t, for the geometry may be estimated by considering the elastic part of the local notch root strain calibration

$$e_r = e_n K_t = \frac{P}{C_1} \tag{15-3}$$

and from S = mP

$$e = m\frac{P}{E} \tag{15-4}$$

and so

$$K_t = \frac{E}{(C_1 m)} \tag{15-5}$$

K_t = 203403/(6271.7 x 11.24) = 2.88

With the applied load of 30kN the expected local elastic stress as predicted by the original MSC.Nastran analysis

= EP/C_1

=203403 x 30 / 6271.7

= 973 MPa

The equivalent stress predicted by Patran FEA is 1001 MPa at node 1 and so it is reasonable to assume that the current Patran FEA model is sufficiently accurate to use in the MSC Fatigue validations. The reference node (the node which defines the nominal stress) has been chosen to be node 61 since this gives rise to a K_t of 1001/347 = 2.88, which is located at just about one notch root radius away from the notch itself. This reference stress will be called the nominal notch root stress. The MSC.Nastran analysis showed that nominal elastic notch root stress, S_n, divided by the applied load is equal to 11.24 MPa/kN (7.25 ksi/kip). The Patran FEA analysis gives 347/30 = 11.57 MPa/kN (7.45 ksi/kip). Note that a MSC Fatigue design optimization analysis carried out at node 181 using a K_t of 1001/87 = 11.5 gives the same answer as an analysis at node 1 with K_t = 1. This illustrates the importance of carrying out a FE analysis to help in estimation of K_t and strain gauge placement. From the above analysis, sufficient confidence is generated by the Patran FEA model to enable it to be used as part of the MSC Fatigue global validation procedures.

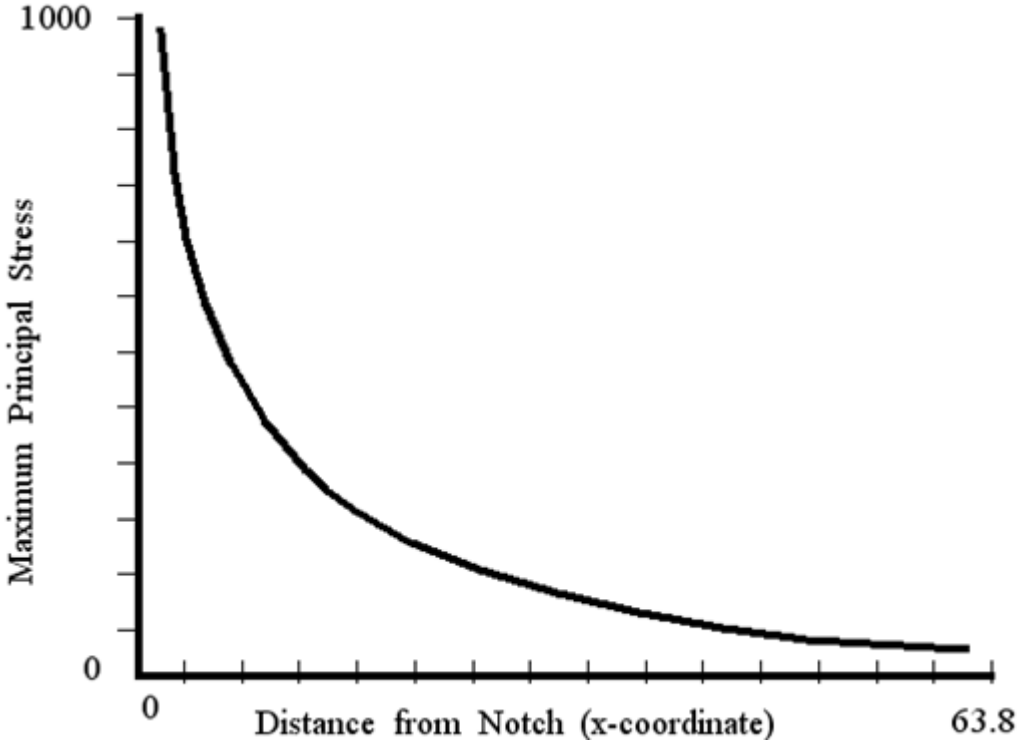


Figure 15-2 Maximum Principal Stresses as a Function of Distance from Notch

Step 2: Material Characterization

The materials selected by the SAE Committee were both steels. One was U.S. Steel's Man-Ten alloy and the other was Belthlehem's high alloy steel, RQC-100. A summary of the basic smooth (polished) specimen fatigue properties have already been discussed.

The stress-life properties are stored in MSC Fatigue's materials database manager, PFMAT, as MANTEN_SN and RQC100_SN. PFMAT can be invoked from within Patran under the MSC Fatigue Materials Info form by pressing the Database Management button in the top right corner or directly from the command level by simply typing the symbol **pfmat**. It is not necessary to actually run PFMAT for to reproduce the results of this problem unless you wish to view the S-N curves.

When invoking PFMAT the main menu allows you to list, search, create, edit, and graphically display existing data sets as well as help to classify welded details. In this case, the datasets for MANTEN and RQC100 are already stored in the standard database, which is held in a read-only location for security reasons. Since you do not need to edit the data, you can use the data stored in the secure database.

Now you can look at the S-N curves graphically. First use the Load menu to load MANTEN_SN and RQC100_SN datasets into the database manager memory locations for material 1 and material 2. Then choose the graph option from the main menu. Note that RQC100_SN gives longer lives at all stress ranges. The key to determining this is to pick a certain stress level and go across horizontally until you hit the S-N curves. The more spread apart two S-N curves are in the horizontal direction the longer lives the one further to the right will give relative to the other.

Use the following keystrokes to perform the above task (note that most of these commands can also be accomplished with a mouse):

Option	Comments
pfmat	Invoke PFMAT either from the system prompt or from the MSC Fatigue Materials Info form by clicking on the Materials Database button.
Load / data set 1	Click on the Load option and select data set 1.
Type Name / MANTEN_SN	Click on Type Name option and type the name of the material. Click on OK when done. A message will appear confirming the loading of the material.
Load / data set 2	Click on the Load option and select data set 2.
Type Name / RQC100_SN	Repeat operation for RQC100_SN.
Graphical display	Select the Graphical display option. The S-N curves will be plotted.
File / Exit/ eXit	Close the graphical display by selecting Exit under the File menu and then select the eXit option to leave PFMAT.

Step 3: Loading Histories

Three load histories were selected by the SAE Committee for use in their evaluation. It is important to emphasize that these sequences are not intended to represent standard loading spectra in the same way that Carlos or Falstaf have done. However, they do contain many features which are typical of ground vehicle applications and, therefore, are useful in the evaluation of life estimation methods.

The first load history has a predominantly tensile mean which reflects sudden changes in mean such as occur from transmission torque measured on a tractor engaged in front-end load work and is referred to as the TRANSMISSION history. The second load history has a predominantly negative mean obtained from the bending moment on a vehicle suspension driven over a proving ground and is referred to as the SUSPENSION history. The last is a load history representing a vibration with nearly zero mean load obtained from a mounting bracket, referred to as the BRACKET history. All of these histories contain uncalibrated values scaled to +/- 999 and are stored in files `saetrn.dac`, `saesus.dac`, and `saebrackt.dac`, respectively.

For the purposes of the global validation the three time histories have been scaled by MSC Fatigue's time history database manager, PTIME to provide a set of data which correspond to the local load levels used in the actual laboratory tests. The loading levels are given in Table 15-2.

Table 15-2 Load History Scale Factors

Time History	Load kN	Level kip	Scaling Factor
SAETRN:	71.2	16	71.271271
	35.6	8	35.635635
	15.6	3.5	15.615615
SAESUS:	71.2	16	71.271271
	40.0	9	40.04004
	31.1	7	31.131131
	26.7	6	26.726726
	20.0	4.5	20.02002
	13.3	3	13.313313
SAEBRAKT:	71.2	16	71.271271
	35.6	8	35.635635
	15.6	3.5	15.615615
	13.3	3	13.313313
	11.1	2.5	11.111111

An example of how to use PTIME to scale the above mentioned time histories is given here. Once PTIME has been invoked, (either from the command level by typing the symbol **ptime** or from within Patran from the MSC Fatigue Loading Info form by pressing the Database Management button), use the following operations. The instructions shown below need to be repeated for each of the scaled time histories in Table 15-2, if you desire to investigate all of them. Only the first is shown.

After this PTIME session a file called `saetrn15.dac` will exist in your directory as well as the `ptime.adb` and `ptime.tdb` files. This session assumes you are invoking PTIME for the very first time and you are working in a clean directory. Most of these commands can also be performed with the mouse.

Operation	Comments
ptime	Invoke PTIME either from the system prompt (or do it from the MSC Fatigue Loading Info form by clicking on the Database Manager button).
Copy from central	Select the “Copy from central” option so as to copy the time histories from the central holding area.
List	Click on the List button to show the available time histories stored in the central area.
SAETRN / SAESUS / SAEBRACKT	Highlight these three time histories by selecting them with the mouse then click on OK. They will be copied into your local directory.

Operation	Comments
Add an entry	Select the Add an entry option.
Duplicate file	Select the Duplicate file option from the Add a time history menu.
List / SAETRN	Press the List button and then select SAETRN from the listbox. Press OK.
SAETRN15	Give SAETRN the new name, *SAETRN15*, and also new description. Press OK. The time history SAETRN will be duplicated and will be called SAETRN15.
Change a time history	Select the Change a time history option.
Polynomial transform	Select the Polynomial transform option from the Change menu. Press OK to accept the default file which should be SAETRN15. Confirm Yes to overwrite.
15.6156	Enter *15.6156* into the second field of the polynomial transform form that appears for applying scale factors to the time history. Click OK. This will scale the time history.
Force / Newtons	The next form that appears is for changing details specific to the time history. Change the load type from Uncalibrated to *Force* and the Units from none to *Newtons*. Press OK.
Plot an entry	Select the Plot option and accept the default. Note the maximum and minimum values. They should be 1.56E4 and -7730, respectively.
File / Exit / eXit	Close the graphical display by selecting Exit under the File menu and then select the eXit option to leave PTIME.

Step 4: Setup MSC Fatigue Job

To set up the MSC Fatigue job for an S-N component fatigue analysis of the keyhole specimen, invoke Patran or MSC Fatigue and fill out the MSC Fatigue forms as shown below. In order for the fatigue analysis to be performed correctly you will need the files `key.out` and `key.txt` which can be found in the `examples` directory delivered with your MSC Fatigue system.

```
<install_dir>/mscfatigue_files/examples
```

The file `key.out` contains the geometry of the specimen and `key.txt` contains the FEA results which must be translated into binary format before proceeding. This is done with a special utility program called RESTXT delivered with your MSC Fatigue system. (It can be invoked by simply typing the symbol **restxt** at the system prompt.) After translation the new file is called `key.res`. (In most instances FE results will be stored in and accessed from the database.)

Unless stated otherwise, parameters not specifically specified should retain their default values.

Option	Comments
patran	Invoke Patran (or MSC Fatigue) if you have not already done so.
File / New...	Open a new database from the File pull-down menu. Call it "key." Set the analysis preference to Patran FEA if asked. Ignore any warning messages.
File / Import...	Import the neutral file `key.out` into the database. At this time you may wish to manipulate the model, turn labels off, or other Patran operations. When ready, go on to the next step.
Tools / FATIGUE... **(Analysis)**	Invoke Patran's FATIGUE interface by selecting it from under the Tools pull-down menu (or the Analysis application switch in MSC Fatigue).
General Setup:	
Analysis: S-N	Set the analysis type to S-N on the main form.
Jobname: keysn	Give the job a name. Use *keysn*.
Title: S-N analysis of keyhole	Give the job a title.
Solution Parameters Form:	
Mean Stress Correction: None	Set the mean stress correction method to None.
Materials Information Form:	
Material: MANTEN_SN	Place the cursor in the cell under the word Material and click on the mouse. A listbox will appear. Select the material MANTEN_SN from this listbox.
Surface Finish/Treatment	Select no surface finish and no treatment.
Region: default_group	Select the default group as the region. It contains all the nodes of the entire model for which the fatigue analysis will be applied to. The Materials Information form can be closed down now by clicking the OK button. All other information in the spreadsheet can be left blank.
Loading Information Form:	
Results From: Patran FEA	The results are from an external Patran FEA results file.
Select a Patran FEA Job: key	Press the Select File button and select the `key.res` file from the listbox.
Load Case ID: 1	Place the cursor in the Load Case ID cell and click the mouse button. A databox appears in which the load case ID from the FE analysis is to be entered. Press RETURN to accept the default (1).
Time History: SAETRN15	Select a time history from the list that appears. Our time history is the one that was previously created and scaled, SAETRN15.

Option	Comments
Load Magnitude: 30000	Enter 30,000 Newtons as the load magnitude. This is used to normalize the stresses from the FE analysis. Press RETURN to enter this value into the cell. This form can now be closed down by pressing the OK button. Everything else in the spreadsheet can be left blank.
Job Control Form:	
Full Analysis	Set the action to Full Analysis and click the Apply button. The database will close down and translation will begin. The database will open when translation is done and the job is submitted.
Monitor Job	At this point, the job has been submitted and can be monitored as to its progress if desired. Set the Action to Monitor and click Apply each time you wish to see the progress state of the job. Once the message *Fatigue analysis completed successfully* appears as the status message, the results can be examined.

Step 5: Evaluate Results

Results for all load history and material combinations are tabulated in Keyhole Results, 1250. For our specific example here, the results can be seen by opening the Results form. From this form, you can read results into the database for making contour plots or enter a separate program (PFPOST) for listing the results in tabular form. This latter method is convenient for quickly identifying the life at our reference node 61. To quickly assess the damage:

Operation	Comments
Results Form:	
List Results	Set the Action to List Results. The separate MSC Fatigue module PFPOST will be spawned. Click the Apply button.
OK / OK	Click OK twice when the PFPOST form comes up to accept the default jobname.
Filtered nodes	Select this option. Note however that in this specific problem we are actually only interested in the result at node 61 since this is the location 1/2 a radius distance from the notch. This is where the strain gage was placed to measure nominal stress from which the S-N curve was created. Stress at this location determines actual failure, not at the notch root itself.
End / exit	Press End and then exit PFPOST.

The analysis at node 61 predicted life of approximately 20,000 repeats of the time history. When using component S-N curves, it is only necessary or valid to evaluate the fatigue life at the reference location. The stress plotted on the S-N curve corresponds to the stress at node 61, however, we know that failure will actually occur at node 1 (at the notch). Therefore the actual fatigue life at node 1 is what is reported at node 61 in this example. Life or damage contour plots of S-N component results will be meaningless and therefore the results are not necessary to be read into the Patran database.

Note: The above explanation may be confusing if you do not understand the difference between a component and a material S-N curve. This example could also be done by using the material S-N curves for MANTEN and RQC100 as opposed to the component S-N curves used in this example. The advantage to this is that they have been converted from being component based to being material based and allow specification of surface finish and surface treatment. The material S-N curves are the component S-N curves except they have been scaled by the K_t value of 2.88 determined earlier in this example and are stored in the database as MANTEN_MSN and RQC100_MSN. See Component vs. Material S-N Curves, 169 for a more in depth explanation of the reference location and the differences in these to curve types. In short, material S-N curves are geometry independent whereas component S-N curves are dependent on and only valid for the specific component geometry for which they were created. Color contour plots of life for material S-N curves do have valid meaning. They can be treated exactly like a crack initiation analysis as explained in the next example in this section.

Step 6: Design Optimization

Now to answer some of the objectives of this problem, you must set the action to Optimize from the MSC Fatigue Results form. This will invoke a separate MSC Fatigue module called FEFAT from which we can answer these questions. Remember you want to investigate the effect of the S-N scatter via the design criterion, the effect of mean stress, and the difference between the two materials under investigation:

Operation	Comments
Results Form:	
Optimize	Set the Action to Optimize. The separate MSC Fatigue module FEFAT will be spawned. You can select a node from the graphics screen or type one in if you wish. Leave it blank for now. Click the Apply button.
User Select node	Select the User Select option and put in 61 as the node in the Node/Element databox in place of `@pfatigue.ents`. You must also supply a Design Life. Use 20,000. Click OK when node 61 and the design life have been specified.

Note at this point that the same exact result is presented for Node61 (i.e., ~20,000 repeats). Now change the design criterion.

Operation	Comments
End	Press the End button to clear the summary form.
Change Parameters	Select the Change Parameters option.
Design criterion	Change the design criterion to 96% certainty of survival. Click OK.
Recalculate	Select the Recalculate option.

For a 96 percent certainty of survival you can expect the life to decrease to approximately 12,000 repeats. Now investigate mean stress.

Operation	Comments
End	Press the End button to clear the summary form.
Change Parameters	Select the Change Parameters option.
Mean Stress Correction/ Goodman	Change mean stress correction method to Goodman. Press OK.
Recalculate	Note the new life of approximately 8000 Repeats. Press the End button.
Change Parameters	Select the Change Parameters option again.
Mean Stress Correction/ Gerber	Change the mean stress correction method to Gerber. Press OK.
Recalculate	Note the new life of approximately 11,000 repeats.

For a most conservative answer, you would want to select Goodman mean stress correction answer of ~8000 repeats at 96% certainty of survival. Now investigate the other material.

Operation	Comments
End	Press the End button to clear the summary form.
Original parameters	Select Original Parameters option to reset all values back to the original values set up in the fatigue analysis.
Material optimization	Select the Material Optimization option.
S-N Dataset/ RQC100_SN	Select the S-N Dataset option and either type in RQC100_SN or use the List button to select it from a listbox. Click OK when done.
Recalculate	Note the life. Press the End button.
Change Parameters / Design criterion	Change the design criterion to 96% certainty of survival.
Recalculate	Note the new life. Press the End button and Exit from FEFAT.

Note that at 50% certainty of survival, RQC100 does better with a life of ~47,000 repeats (~20,000 repeats for MANTEN) than it does at 96% (~7400 repeats, ~12,000 repeats for MANTEN) even though earlier we determined that RQC100 was a better material at all lives based on the S-N curves. This fact is due to the statistical nature of S-N curves where the scatter in the S-N data for RQC100 is much more variable than for MANTEN.

Crack Initiation of Keyhole

Objectives

1. To calculate the fatigue life for a component made from MANTEN under a “transmission” loading whose maximum is 3.5 Kips using the “local strain” method for crack initiation.
2. To investigate the effect of surface finish on the fatigue life by analyzing the effect of machining marks around the notch.
3. To investigate the effect of mean stress on fatigue life.
4. To make a comparison between MANTEN and RQC100 under the above conditions.
5. To compare predicted lives to measured lives.

Step 1: Geometry and FEA Results

The exact same finite element analysis results are used in this example as were used in the previous S-N example of the keyhole specimen. The finite element analysis was performed with a static loading of 30kN and the results stored under the jobname KEY.

In case you skipped the previous example, the results are stored in the file `key.txt` and must be translated from a text format to a binary format using the Patran’s utility program called RESTXT delivered with your Patran system. After translation the new file is called `key.res`. (It can be invoked by simply typing the

symbol **restxt** at the system prompt.) In order for the fatigue analysis to be performed correctly, you will also need the file `key.out`. Both can be found in the `examples` directory delivered with your MSC Fatigue system. The file `key.out` contains the geometry of the specimen.

```
<install_dir>/mscfatigue_files/examples
```

Step 2: Material Characterization

The same materials as in the previous S-N analysis example are again used in this example. One is U.S. Steel's Man-Ten alloy and the other was Belthlehem's high alloy steel, RQC-100. The specimen was polished, and the surface was untreated. A summary of the basic smooth (polished) specimen fatigue properties have already been discussed.

The strain-life and cyclic stress-strain properties are stored in MSC Fatigue's materials database manager, PFMAT, as MANTEN and RQC100. PFMAT can be invoked from within Patran under the MSC Fatigue Materials Information form by pressing the Database Manager button or directly from the command level by simply typing the symbol **pfmat**. It is not necessary to actually run PFMAT for this problem unless you want to view the strain-life, or cyclic stress-strain curves.

In this case, the datasets for MANTEN and RQC100 are already stored in the standard database, which is held in a read-only location for security reasons. Since you do not wish to edit the data, you can use the data stored in the secure database.

You can look at the strain-life curves graphically. First use the Load menu to load MANTEN and RQC100 datasets into the database manager memory locations for material 1 and material 2. Then choose the Graphical display option from the main menu. Then choose to graph the strain-life curves. Note that RQC100 gives longer lives at all strain levels except in the transition zone. The transition zone is where the elastic and plastic lines that make up the strain-life curve cross each other. To the right of the transition zone is known as the high-cycle regime where elastic effects dominate. To the left is the low-cycle regime where plastic effects dominate. Plot also the cyclic and monotonic stress-strain curves for the two materials. Can you tell which one exhibits cyclic hardening and which exhibits cyclic softening? (MANTEN is hardening and RQC100 is softening.)

Use the following keystrokes to perform the above task (note that most of these commands can also be accomplished with a mouse).

Operation	Comments
pfmat	Invoke PFMAT either from the system prompt or from the MSC Fatigue Materials form.
Load / data set 1	Click on the Load option and select data set 1.
Type Name / MANTEN	Click on Type Name option and type the name of the material. Click on OK when done. A message will appear confirming the loading of the material.
Load / data set 2	Click on the Load option and select data set 2.
Type Name /RQC100	Repeat operation for RQC100.
Graphical display	Enter the graphical display options.
Strain life plot	Plot the strain-life plot for both materials.
Plot Type / E-P Lines	Plot the Elastic and Plastic lines that make up the strain-life curves.
File / New Plot	Close the graphical display. Select New Plot from the File menu.
cYclic & Monotonic stress-strain curves plot	Enter the Graphical Display option again and plot the cyclic and monotonic stress-strain curve for data set 1 (MANTEN). Press the OK button to plot.
File / New Plot	Close the graphical display.
cYclic & Monotonic stress-strain curves plot	Plot the cyclic and monotonic stress-strain curve for data set 2 (RQC100). Click the OK button to plot.
File / Exit / eXit	Close the Graphical Display and return to the main form and use the eXit option to leave PFMAT.

Step 3: Loading Histories

The specimen was loaded with three random time histories corresponding to typical histories for transmission, suspension, and bracket components at different load levels.

The three load histories were selected by the SAE Committee for use in their evaluation and are the same as those used in the previous S-N analysis of the keyhole. The same scaling was used as already discussed and shown in Table 15-2. Use step three from the previous example to obtain the appropriately scaled time history. It is assumed that you are starting from a fresh, empty directory.

Step 4: Setup MSC Fatigue Job

To set up the MSC Fatigue job for a crack initiation fatigue analysis of the keyhole specimen, enter Patran or MSC Fatigue and use the following keystrokes shown below. It is assumed that you are starting from a fresh, empty directory. If a parameter is not specified, accept its default.

Operation	Comments
patran	Invoke Patran (or MSC Fatigue) if you have not already done so.
File / New...	Open a new database from the File pull-down menu. Call it "key." Set the analysis preference to Patran FEA if asked. Ignore any warning messages.
File / Import...	Import the neutral file `key.out` into the database. At this time you may wish to manipulate the model, turn labels off, or other Patran operations. When ready, go on to the next step.
General Setup:	
Analysis:Initiation	Set the analysis type to Crack Initiation on the main form.
Jobname: keyci	Give the job a name. Use *keyci*.
Title: Crack Initiation analysis of keyhole	Give the job a title.
Tools / FATIGUE... **(Analysis)**	Invoke Patran's FATIGUE interface by selecting it from under the Tools pull-down menu (or the Analysis application switch in MSC Fatigue ()).
Solution Parameters Form: Accept all defaults	
Materials Information Form:	
Material: MANTEN	Place the cursor in the cell under the word Material and click on the mouse. A listbox will appear. Select the material MANTEN from this listbox.
Finish: Polished	Select Polished from the option menu that appears. The word polished appears in the Finish cell. The SAE specimen was a polished specimen with no surface treatment.
Treatment: No Treatment	Select No Treatment from the option menu that appears.
Region: default_group	Select the default group as the region. It contains all the nodes of the entire model for which the fatigue analysis will be applied to. The Materials Information form can be closed down now by clicking the OK button.
Loading Information Form:	
Results From: Patran FEA	The results are from an external Patran FEA results file.
Select a Patran FEA Job: key	Press the Select File button and select the `key.res` file from the listbox.
Load Case ID: 1	Place the cursor in the Load Case ID cell and click the mouse button. A databox appears in which the load case ID from the FE analysis is to be entered. Press RETURN to accept the default (1).
Time History: SAETRN15	Select a time history from the list that appears. Our time history is the one that was previously created and scaled, SAETRN15.
Load Magnitude: 30000	Enter 30,000 Newtons as the load magnitude. This is used to normalize the stresses from the FE analysis. Press RETURN to enter this value into the cell. This form can now be closed down by clicking the OK button.

Operation	Comments
Job Control...	
Full Analysis	Set the action to Full Analysis and click the Apply button. The database will close down and translation will begin. The database will open when translation is done and the job is submitted.
Monitor Job	At this point the job has been submitted and can be monitored as to its progress if desired. Set the Action to Monitor in the Job Control form and click Apply each time you wish to see the progress state of the job. Once the message *Fatigue analysis completed successfully* appears as the status message, the results can be examined.

Step 5: Evaluate Results

Results for all load history and material combinations are tabulated in Keyhole Results, 1250. For our specific problem, the results can be seen by opening the Results form from the MSC Fatigue main form. You can either read results into the database for making contour plots or you can enter a separate program (PFPOST) for listing the results in tabular form. This latter method is convenient for quickly identifying which node has the shortest life. To quickly assess the most damaged nodes:

Operation	Comments
Results Form:	
List Results	Set the Action to List Results. The separate MSC Fatigue module PFPOST will be spawned. Click the Apply button.
OK	Press OK twice when the PFPOST form comes up to accept the default jobname.
Most damaged nodes	Select the first option to view the most damaged nodes. We expect Node 1 to have the most damage.
End / eXt	Press End and then exit PFPOST.

From this, we can see that Node 1 has the most damage associated with it and a fatigue life of approximately 6200 repeats.

For a quick contour plot of the log of life in repeats of the time history, do the following:

Operation	Comments
Results Form:	
Read Results...	Set the action to Read Results on the Results form in MSC Fatigue. Click the Apply button. The results will be read into the database.
Results	Click on Results Application on the main form. This will bring up the Results application. This is not the MSC Fatigue Results Form.
Select Result Case	Select the result case from the listbox that was just read. It should be called *Crack Initiation* with the name of the job attached to the end.
Select Result	Select *Log of Life, in Repeats* from the Fringe Result listbox.
Apply	Click the Apply button to make the contour plot of log of life in repeats of the time history. Note that a special spectrum has been created for better viewing of life results. For damage results you will want to change the spectrum back to the Standard Spectrum. This can be done under the Display / Spectrums option from the top menu bar.
Results	Click on the Results switch again on the main form. This will close down the Results application.

You may want to zoom-in on the critical area to see it better.

Step 6: Design Optimization

Now to answer some of the objectives of this problem you must change the action to Optimize on the Results form in MSC Fatigue. This will invoke a separate MSC Fatigue module called FEFAT from which we can answer the questions. Remember we wish to analyze the effect of different surface finishes and treatments and also investigate the effect of mean stress as well as evaluate the two different materials.

Opearation	Comments
Results Form:	
Optimize	Set the Action to Optimize. The separate MSC Fatigue module FEFAT will be spawned. You can select a node from the graphics screen or type one in if you wish. Leave it blank for now. Click the Apply button.
Worst case node	Select the Worst Case Node option from the listbox. The program will automatically scan the results file and pick the node with the shortest life (node 1). Click OK when node 1 has been selected and you have supplied a design life, say 5,000.

Note at this point that the same exact result is presented for Node 1 (i.e., ~6200 repeats). Before answering the above questions, look at the damage distribution. Press the End button to continue.

Operation	Comments
End	Press the End button to clear the summary form.
Results Display... / Plot Cycles Histogram	Select the Results Display option to plot the Cycles histogram.
Plot Type / Damage	Change the plot type from a Cycles histogram to a Damage histogram. This is done from the Plot Type pull down menu.
File / Exit	Close the graphics and return to the main FEFAT design optimization form.

This Cycles/Damage histogram capability gives you an idea of where the majority of the damage is coming from. In our example, we can see that there are a lot of cycles with a low stress range and fewer with a high range. We would expect the high stress ranges (this means a broader hysteresis loop on the stress-strain plane) to give us most of the damage. When you toggle the cycles histogram to a damage histogram you see that this is indeed the case. There is, however, a fairly wide damage distribution at the higher ranges which means we cannot point to a single event causing damage. Now to see the effect of surface treatment and finish.

Operation	Comments
Sensitivity analysis...	Select the Sensitivity analysis option.
Surface Finishes (all)	Select all surface finish conditions to be calculated.
Recalculate	Recalculate the results. Press the End button when done.

Each time you do a recalculation, you are presented with a table listing the various surface finishes or treatments and the corresponding lives. From these tables, you can see that there is at least a factor of 2 difference better or worse when using various surface finishes and treatments. This means the effect is worth considering as a way to improve or reduce the life depending on your design life requirements. Now for the effect of mean stress:

Operation	Comments
Sensitivity analysis...	Select the Sensitivity analysis option.
Mean Stress Correction (all)	Select all mean stress methods to be calculated.
Recalculate	Recalculate the results. Press the End button when done.

You can see that the two mean stress options, Smith-Watson-Topper (SWT) and Morrow give lives less than that achieved using no mean stress correction (~6200, ~9100, ~10500 repeats, respectively) with the SWT method being the most conservative. This is to be expected since the time history we are using (SAETRN) has a predominantly tensile mean. If the time history had a predominantly compressive mean (SAESUS), then we would expect to see the two mean stress correction methods giving longer lives than no correction. If the time history had a roughly zero mean (SAEBRAKT), then all three methods would give approximately the same answers. This is indeed what the results show in Keyhole Results, 1250. Now change the material to RQC100.

Operation	Comments
Material optimization...	Select the Material optimization option.
Material change	Chose to change materials.
RQC100	Type in the new material name, RQC100.
Recalculate	Recalculate the results. Press the End button and exit when done.

It is clear that RQC100 is a superior material using all mean stress methods.

Crack Growth of Keyhole

Objectives

1. To calculate the crack propagation life for a component made from MANTEN under a "transmission" loading whose maximum is 3.5 Kips using the "local strain" method for crack initiation.
2. To investigate the effect of a residual stress on crack growth life.
3. To make a comparison between materials MANTEN and RQC100.
4. To compare predicted lives to measured lives.

Step 1: Geometry and FEA Results

The exact same finite element analysis results are used in this example as were used in the previous two examples of the keyhole specimen. The finite element analysis was performed with a static loading of 30kN and the results stored under the jobname KEY.

In case you skipped the previous examples, the results are stored in the file `key.txt` and must be translated from a text format to a binary format using the Patran utility program called RESTXT delivered with your Patran system. After translation the new file is called `key.res`. (It can be invoked by simply typing the symbol **restxt** at the system prompt.) In order for the fatigue analysis to be performed correctly you will also need the `file key.out`. Both can be found in the `examples` directory delivered with your MSC Fatigue system. The file `key.out` contains the geometry of the specimen.

```
<install_dir>/mscfatigue_files/examples
```

In addition, you will need to define a shape factor, K solution, sometimes known as a Y-compliance or beta function. MSC Fatigue contains a library of compliance functions which contain the means to calculate the fracture mechanics stress intensity factor, K. Please see Crack Growth (Ch. 8) for a discussion of the K Solution Library.

The geometry of our keyhole specimen is similar to the compact tension specimen for which there is an entry already in the K Solution Library. To define the particular dimensions of the keyhole you must invoke PKSOL by typing the symbol **pksol** at the system prompt or you may access is during the MSC Fatigue setup in the Solution Parameter form when the Analysis Type is set to Crack Growth. Once in the PKSOL module, enter the following keystrokes to define the proper K solution. A file called `keyhole.ksn` will exist in your directory at the conclusion of the PKSOL execution.

Operation	Comments
pksol	Invoke PKSOL either from the system prompt or from within the Solution Parameter form in MSC Fatigue when the Analysis Type is set to Crack Growth by pressing the Compliance Generator button in the top right corner.
Inches	Define units in inches.
Generate a Y function table	Choose the Generate a compliance function option.
keyhole	Give it a name. A file called `keyhole.ksn` will be generated.

Operation	Comments
Standard specimens	Pick standard specimens.
Compact tension specimen (CTS)	Pick compact tension specimen.
Define	A graphical display appears to allow dimension definitions. Click on Define from the main form.
B = 0.375	Enter the dimension of B (0.375 inches) and press RETURN.
W = 3.7	Enter the dimension of W (3.7 inches).
None	Just press RETURN. No changes are necessary.
Calculate	Pick the Calculate option from the main form.
Plot Y function	You may now plot the compliance if desired.
File / Exit / exit	Exit PKSOL.

The dimensions can be set up in any units as long as the units used when defining the initial and final crack lengths and any notch dimensions are the same as defined in the compliance function. Initial and final crack lengths are defined in the MSC Fatigue setup mode. If you wish you can plot the Y function vs. crack ratio using Patran's XY plot application from the Solution Parameters form.

Step 2: Material Characterization

The same materials as in the previous S-N and ε -N analyses examples are again used in this example. One is U.S. Steel's Man-Ten alloy and the other was Belthlehem-s high alloy steel, RQC-100. The specimen was polished, and the surface was untreated. A summary of the basic smooth (polished) specimen fatigue properties have already been discussed.

The da/dn curves are stored in MSC Fatigue's materials database manager, PFMAT, as MANTEN and RQC100. PFMAT can be invoked from within Patran under the MSC Fatigue material form by pressing the Database Manager button or directly from the command level by simply typing the symbol **pfmat**. It is not necessary to actually run PFMAT for this problem unless you want to view these da/dn curves.

In this case, the datasets for MANTEN and RQC100 are already stored in the standard database, which is held in a read-only location for security reasons. Since you do not need to edit the data, you can use the data stored in the secure database.

You can look at the da/dn curves graphically. First, use the Load menu to load MANTEN and RQC100 datasets into the database manager memory locations for material 1 and material 2. Then choose the graph option from the main menu. Then choose the Graphical display option to plot the effective and/or apparent ΔK plots. Note that the apparent ΔK plot is non-linear whereas the effective ΔK plot is linearized. The effective ΔK plot has taken into account and correctly modeled all correction effects such as environment, history, closure, etc., and linearized the apparent ΔK plot.

Operation	Comments
pfmat	Invoke PFMAT either from the system prompt or from the MSC Fatigue Materials form.
Load / data set 1	Click on the Load option and select data set 1.
Type Name / MANTEN	Click on Type Name option and type the name of the material. Click on OK when done. A message will appear confirming the loading of the material.
Load / data set 2	Click on the Load option and select data set 2.
Type Name /RQC100	Repeat operation for RQC100.
Graphical display	Enter the graphical display options.
Effective delta k plot	Plot the effective delta K plot for both materials.
File / Exit	Close the graphical display.
Graphical display / Apparent delta k plot	Plot the apparent delta K curve for data set 1 (MANTEN).
0.1 and 0.9	Enter two stress ratios between 0 and 1. Say 0.2 and 0.9. Press the OK button to plot.
File / Exit	Close the graphical display.
eXit	Use the eXit option to leave PFMAT.

Step 3: Loading Histories

The specimen was loaded with three random time histories corresponding to typical histories for transmission, suspension and bracket components at different load levels.

The three load histories were selected by the SAE Committee for use in their evaluation and are the same as those used in the previous two examples of the keyhole. The same scaling was used as already discussed and shown in Table 15-2. See step 3 from the first example for an explanation of the loading histories and their proper scaling. It is assumed that you are starting in a clean, empty directory.

Step 4: Setup MSC Fatigue Job

To set up the MSC Fatigue job for a crack growth analysis of the keyhole specimen, enter Patran and enter the following keystrokes shown below. Unless specified, accept all defaults.

Operation	Comments
patran	Invoke Patran (or MSC Fatigue) if you have not already done so.
File / New...	Open a new database from the File pull -down menu. Call it "key." Set the analysis preference to Patran FEA if asked. Ignore any warning messages.
File / Import...	Import the neutral file `key.out` into the database. At this time you may wish to manipulate the model, turn labels off, or other Patran operations. When ready, go on to the next step.
Tools / FATIGUE... **(Analysis)**	Invoke Patran's FATIGUE interface by selecting it from under the Tools pull-down menu (or the Analysis application switch in MSC Fatigue).
General Setup:	
Analysis: Growth	Set the analysis type to Crack Growth on the main form.
Jobname: keycg	Give the job a name. Use *keycg*.
Title: Crack Growth analysis of keyhole	Give the job a title.
Solution Parameters Form:	
Select a Compliance Function	Select the `keyhole.ksn` file that you previously created using PKSOL from the listbox. Make sure that it is highlighted.
Plane Stress Correction: Off	Set this parameter OFF.
Stress Combination: X Normal	Select the X Normal component as the stress combination parameter from which fatigue life will be determined.
Crack Length Units: Inches	Define the crack length in inches.
Initial Crack Length: 0	Initial crack length can be left at zero.
Final Crack Length: 2	Set the final crack length to 2 inches.
Notch Depth: 2.3	The notch depth is 2.3 inches.
Notch Radius: 0.1875	The notch radius is 0.1875 inches.
Sharp Crack Radius: 3.937E-5	A small sharp crack. Press OK to close the form. See note below.
Materials Information Form:	
Material: MANTEN	Place the cursor in the cell under the word Material and click on the mouse. A listbox will appear. Select the material MANTEN from this listbox.
Environment: AIR	Only one environment is available for MANTEN. Select it with the cursor.

Before completely filling out the Materials form, we need to digress a bit to create a special Patran group which contains the node or nodes of the specimen which define where the nominal or far field stress occurs

since this is the stress that a crack growth analysis expects. (The stress that would be there if no crack were present including the notch.) To do this follow these instructions without closing down the MSC Fatigue forms.

Operation	Comments
Group / Create	From the top main form, select the Group pull-down menu and choose the Create option.
New Group Name	In the form that appears enter *Node349* as the New Group Name. Make sure you do not include any spaces in the group name.
Entity Selection / Node 349	In the Entity Selection databox either type in **Node 349** or select node 349 from the graphics screen with the cursor. Then click the Apply button to create the group and then the Cancel button to close the form down.

Now that the new group is created you can continue to fill out the MSC Fatigue forms.

Operation	Comments
Material Information Form:	
Region: Node349	Select the group that was just created. It contains the node defining where the far field stress is (the stress that would be there if there were no crack). The Materials Information form can be closed down now by clicking the OK button.
Loading Information Form:	
Results From: Patran FEA	The results are from an external Patran FEA results file.
Select a Patran FEA Job: key	Press the Select File button and select the `key.res` file from the listbox.
Load Case ID: 1	Place the cursor in the Load Case ID cell and click the mouse button. A databox appears in which the load case ID from the FE analysis is to be entered. Press RETURN to accept the default (1).
Time History: SAETRN15	Select a time history from the list that appears. Our time history is the one that was previously created and scaled, SAETRN15.
Load Magnitude: 30000	Enter 30,000 Newtons as the load magnitude. This is used to normalize the stresses from the FE analysis. Press RETURN to enter this value into the cell. This form can now be closed down by pressing the OK button.
Job Control Form:	
Full Analysis	Set the action to Full Analysis and click the Apply button. The database will close down and translation will begin. The database will open when translation is done and the job is submitted.
Monitor Job	At this point the job has been submitted and can be monitored as to its progress if desired. Set the Action to Monitor in the Job Control form and click Apply each time you wish to see the progress state of the job. Once the message *Crack growth analysis completed successfully* appears as the status message, the results can be examined.

The following notes are made about the job setup:

1. When defining properties for the different fatigue analyzers, it is important to remember that for the total life and local strain methods you are defining nodes or elements for which the resulting lives will be determined. For the LEFM method the nodes or elements specified during the setup are used only to obtain an average far-field stress (the stress that would be there if no crack or notch existed) as well as to define the appropriate material and environment. In this example, Node 349 exhibits a nominal stress of 24MPa which is roughly P/A=30000N/(125mm*9.525mm) = 28MPa. This value is used in the Paris equation.

2. A small sharp crack has been introduced. This was necessary to cause crack growth in our case. (A default value of zero does not cause significant growth and the program terminates.) This is deemed acceptable for validation purposes since during the SAE study of the keyhole initial crack sizes were not consistently or properly monitored.

Step 5: Evaluate Results

Results for all load history and material combination are tabulated in Keyhole Results, 1250. For our specific example here, the results can be seen by opening the Results from the MSC Fatigue main form.

Operation	Comments
Results Form:	
List Results	Set the Action to List Results. The separate MSC Fatigue module PCPOST will be spawned. Click the Apply button.
Results summary page	Post the summary page. Note the failure method and life.
End / eXit	Press End and then exit PCPOST.

From the results summary page, we can see that the crack grew to final length of a little over 0.2 inches in approximately 1400 repeats of the time history. The mode of failure was by exceeding the fracture toughness of the material.

Step 6: Design Optimization

Now to answer some of the objectives of this example you must change the Action on the Results form to Optimize. This will invoke a separate MSC Fatigue module called PCRACK from which we can answer these questions.

Operation	Comments
Results Form:	
Optimize	Set the Action to Optimize and click the Apply button. The module PCRACK will be spawned.
OK	Click the OK button to accept the first page.
OK	Click OK twice to accept the Output parameter page and answer Yes to overwrite the output file.
OK	Click OK twice to accept the Geometry information.
OK	Click OK to accept the Material and Environment.

This essentially reruns the analysis and puts up the summary report page again except that this time we are allowed to see the crack grow graphically. The same exact results are had as before. Now introduce a residual stress.

Operation	Comments
End	Close the summary page.
Edit analysis parameters.../ Loading definitions	Select the Edit analysis parameters option.
Time History Offset: -5	Enter -5 MPa into the Time History Offset databox.
Recalculate	Recalculate

The job stops after only approximately 58 repeats with no significant growth. The residual stress has retarded/stopped the crack growth. Now change the material to RQC100. Since we know from the previous example that RQC100 is a superior material, we might expect that the life will be extremely enhanced.

Operation	Comments
End	Close the summary page.
Edit analysis parameters.../ Loading definitions	Select the Edit analysis parameters option to reset the residual stress back to zero.
Time History Offset: 0	Enter 0 MPa into the Time History Offset databox. Press OK.
Edit analysis parameters...	Select the Edit analysis parameters option.
Select Material and Environment	Choose to Select a new material and environment.
Material: RQC100	Change the material to RQC100. Press the OK button when the material parameters form is presented.
Recalculate	Recalculate. Give overwrite permission when requested.
End / eXit	Exit from PCRACK.

Surprisingly, RQC100 has a longer crack initiation life but a shorter crack propagation life than MANTEN. Apparently RQC100 is more ductile that MANTEN by the fact that the crack grew to a length of about 0.6 inches as opposed to MANTEN, which only grew to about 0.2 inches, even though it lasted longer.

Keyhole Results

The results tabulated, starting with Table 15-3, indicate the comparison between the actual measured lives and those calculated by MSC Fatigue. For the purposes of validation, the MSC Fatigue lives listed below will be taken to be the definitive answers. Any significant deviations from these results must be considered to indicate regression of the analysis code.

Plots of measured vs. predicted lives for the Crack Initiation and Crack Growth comparisons of the tabulated data are also presented, starting at Figure 15-3. For the Crack Initiation comparisons only, the Smith-Topper-Watson (STW) results are plotted. The solid straight line on each plot represents the one to one

correspondence if predicted life exactly equaled measured life. The two straight dotted lines represent a three times factor indicating a goodness band. A point lying above the solid line represents a conservative life estimate in comparison to test results. Generally, the correlation is well within the expected scatter band.

The results for the crack growth problem can not be consistently reproduced using this model without experimenting with the initial crack size and mean stress. The crack growth analysis is very sensitive to the initial crack size, the indicated mean stress and also the far-field stress chosen to do the prediction. The results table shows the mean stress and initial crack size chosen for each case. This is deemed acceptable since during the SAE study of the keyhole specimen, the mean stresses and initial crack sizes were not consistently or properly monitored.

The following conclusions can be drawn from this example:

1. Correlation coefficients calculated for local strain life (crack initiation) predictions are consistently higher than those obtained from the S-N approach.
2. For all the results, life estimates from the local strain approach are consistently more conservative than those calculated by the S-N approach.
3. For all results, Gerber appears to handle compressive mean stress more accurately than Goodman. Goodman calculates very nonconservative results.
4. For the MANTEN alloy, local stress strain results for the transmission and suspension histories are consistently better than for the S-N approach.
5. For the RQC100 alloy, no significant difference can be found between the use of the local strain and S-N approaches. It is not that the S-N approach has improved, in fact the degree of error is about the same. What seems to be happening is that the local strain predictions appear to be less accurate than for MANTEN. This seems to cast some doubt as to the validity of the material properties provided for RQC100.

Table 15-3 Measured and Calculated Lives using MSC Fatigue (Total Life Result)

Total Life Results for SAETRN with MANTEN					
Maximum Load		**Measured Life**	**Predicted Life**		
(kN)	**(kip)**		**Gerber**	**Goodman**	**None (Material. S-N)**
		8.4			
71.2	16	12.8	Broken	Broken	Broken (Broken)
		12.5			
		420			
35.6	8	154	~280	~120	~330 (Broken)
		74			
		5,800			
15.6	3.5	4,270	~20,300	~14,000	~21,200 (~19,000)
		3,755			

Total Life Results for SAETRN with RQC100					
Maximum Load		Measured Life	Predicted Life		
(kN)	(kip)		Gerber	Goodman	None (Material. S-N)
		8.4			
71.2	16	12.8	~10	Broken	~30 (Broken)
		12.5			
		420			
35.6	8	154	~900	~500	~1,000 (Broken)
		74			
15.6	3.5	3,755	~46,100	~35,200	~46,800 (~42,700)

Total Life Results for SAESUS with MANTEN					
Maximum Load		Measured Life	Predicted Life		
(kN)	(kip)		Gerber	Goodman	None (Material. S-N)
		7.7			
71.2	16	28	~40	~200	~3 (Broken)
		430			
40.0	9	208	~900	~1,900	~670 (Broken)
		162			
		1,750			
26.7	6	2,240	~5,800	~10,500	~5,000 (~4,600)
		1,410			
20.0	4.5	4,700	~24,500	~39,900	~22,600 (~20,100)
13.3	3	8.5E4	~1.9E5	~2.75E5	~1.84E5 (~1.67E5)

Total Life Results for SAESUS with RQC100					
Maximum Load		Measured Life	Predicted Life		
(kN)	(kip)		Gerber	Goodman	None (Material. S-N)
		19.9			
71.2	16	24.4	~200	~400	~100 (Broken)
		64			

Total Life Results for SAESUS with RQC100					
40.0	9	1,710	~2,200	~3,900	~1,900 (Broken)
31.1	7	11,200	~6,700	~10,900	~6,145 (~5,600)
26.7	6	48,000	~14,100	~22,000	~13,100 (~11,700)

Total Life Results for SAEBRAKT with MANTEN					
Maximum Load		**Measured Life**	**Predicted Life**		
(kN)	**(kip)**		**Gerber**	**Goodman**	**None (Material. S-N)**
		1.5			
71.2	16	2.6	Broken	Broken	Broken (Broken)
		2			
		20.8			
35.6	8	11.5	~50	~60	~50 (Broken)
		23			
		1,588			
15.6	3.5	270	~3,000	~3,300	~2,900 (~2,700)
		510			
		>1E4			
13.3	3	2,666	~6,600	~7,300	~6,600 (~6,100)
11.1	2.5	2E4	~17,000	~18,500	~17,000 (~15,200)

Total Life Results for SAEBRAKT with RQC100					
Maximum Load		**Measured Life**	**Predicted Life**		
(kN)	**(kip)**		**Gerber**	**Goodman**	**None (Material. S-N)**
		3.3			
71.2	16	5.1	~3	~7	~3 (Broken)
		4.2			
		87.5			
35.6	8	47	~100	~150	~100 (Broken)
		113			
		2,673			
15.6	3.5	5,020	~6,300	~6,800	~6,300 (~5,800)

Table 15-4 Measured and Calculated Lives using MSC Fatigue (Crack Initiation Result)

Crack Initiation Results for SAETRN with MANTEN					
Maximum Load		Measured Life	Predicted Life		
(kN)	(kip)		S-T-W	Morrow	None
		8.4			
71.2	16	12.8	~4	~5	~5
		12.5			
		420			
35.6	8	154	~75	~90	~100
		74			
		5,800			
15.6	3.5	4,270	~6,200	~9,000	~10,800
		3,755			

Crack Initiation Results for SAETRN with RQC100					
Maximum Load		Measured Life	Predicted Life		
(kN)	(kip)		S-T-W	Morrow	None
		8.4			
71.2	16	12.8	~6	~6	~7
		12.5			
		420			
35.6	8	154	~120	~140	~160
		74			
15.6	3.5	3,755	~1.04E5	~1.87E5	~6.52E5

Crack Initiation Results for SAESUS with MANTEN					
Maximum Load		Measured Life	Predicted Life		
(kN)	(kip)		S-T-W	Morrow	None
		7.7			
71.2	16	28	~20	~15	~15

Crack Initiation Results for SAESUS with MANTEN					
		430			
40.0	9	208	~340	~230	~210
		162			
		1,750			
26.7	6	2,240	~3,100	~1,900	~1,650
		1,410			
20.0	4.5	4,700	~22,800	~11,850	~9,600
13.3	3	8.5E4	~6,49E5	~2.43E5	~1.69E5

Crack Initiation Results for SAESUS with RQC100					
Maximum Load		**Measured Life**	**Predicted Life**		
(kN)	**(kip)**		**S-T-W**	**Morrow**	**None**
		19.9			
71.2	16	24.4	~25	~23	~22
		64			
40.0	9	1,710	~550	~430	~350
31.1	7	11,200	~4,100	~2,800	~1,900
26.7	6	48,000	~21,200	~12,600	~6,900

Crack Initiation Results for SAEBRAKT with MANTEN					
Maximum Load		**Measured Life**	**Predicted Life**		
(kN)	**(kip)**		**S-T-W**	**Morrow**	**None**
		1.5			
71.2	16	2.6	Broken	Broken	Broken
		2			
		20.8			
35.6	8	11.5	~16	~14	~14
		23			
		1,588			
15.6	3.5	270	~2,400	~2,000	~1,900
		510			

Crack Initiation Results for SAEBRAKT with MANTEN					
		>1E4			
13.3	3	2,666	~7,500	~6,300	~5,800
11.1	2.5	2E4	~37,800	~29,100	~25,750

Crack Initiation Results for SAEBRAKT with RQC100					
Maximum Load		**Measured Life**	**Predicted Life**		
(kN)	**(kip)**		**S-T-W**	**Morrow**	**None**
		3.3			
71.2	16	5.1	Broken	Broken	Broken
		4.2			
		87.5			
35.6	8	47	~32	~30	~28
		113			
		2,673			
15.6	3.5	5,020	~2.28E5	~1.54E5	~8.97E4

Table 15-5 Measured and Calculated Lives Using MSC Fatigue (Crack Growth Result)

Crack Growth Results for SAETRN with MANTEN					
Maximum Load		**Propagation Life**	**Predicted Life**		
(kN)	**(kip)**		**Life**	**Mean Stress (MPa)**	**Initial Crack Length (mm)**
		0.5			
71.2	16	3.2	~1	0	0
		1.5			
		117			
35.6	8	39	~40	0	0
		12			
		1,157			
15.6	3.5	1,510	~1,400	0	0
		3,755			

Crack Growth Results for SAETRN with RQC100					
Maximum Load		Propagation Life	Predicted Life		
(kN)	(kip)		Life	Mean Stress (MPa)	Initial Crack Length (mm)
		5.7			
71.2	16	2.5	~2	0	0
		1.8			
		28			
35.6	8	60	~30	0	0
		62			

Crack Growth Results for SAESUS with MANTEN					
Maximum Load		Propagation Life	Predicted Life		
(kN)	(kip)		Life	Mean Stress (MPa)	Initial Crack Length (mm)
		2.8			
71.2	16	20	~20	12.5	2
		1,790			
40.0	9	357	~900	5	2
		605			
		2.3E4			
26.7	6	3.0E4	~37,000	5	1

Crack Growth Results for SAESUS with RQC100					
Maximum Load		Propagation Life	Predicted Life		
(kN)	(kip)		Life	Mean Stress (MPa)	Initial Crack Length (mm)
		7.6			
71.2	16	75.6	~70	5	2
		154			

Crack Growth Results for SAEBRAKT with MANTEN					
Maximum Load		Propagation Life	Predicted Life		
(kN)	(kip)		Life	Mean Stress (MPa)	Initial Crack Length (mm)
		0.7			
71.2	16	0.3	<1	12.5	1
		1			
		11			
35.6	8	8	~6.5	12.5	0
		21			
		3,356			
15.6	3.5	785	~1,300	5	0
		2,116			
13.3	3	1,410	~1,564	5	0.5

Crack Growth Results for SAEBRAKT with MANTEN					
Maximum Load		Propagation Life	Predicted Life		
(kN)	(kip)		Life	Mean Stress (MPa)	Initial Crack Length (mm)
		2			
71.2	16	2.3	~4.5	12.5	0
		2.4			
		98.5			
35.6	8	61	~50	5	0
		99			
		5,000			
15.6	3.5	7,499	~4,700	0	0.1

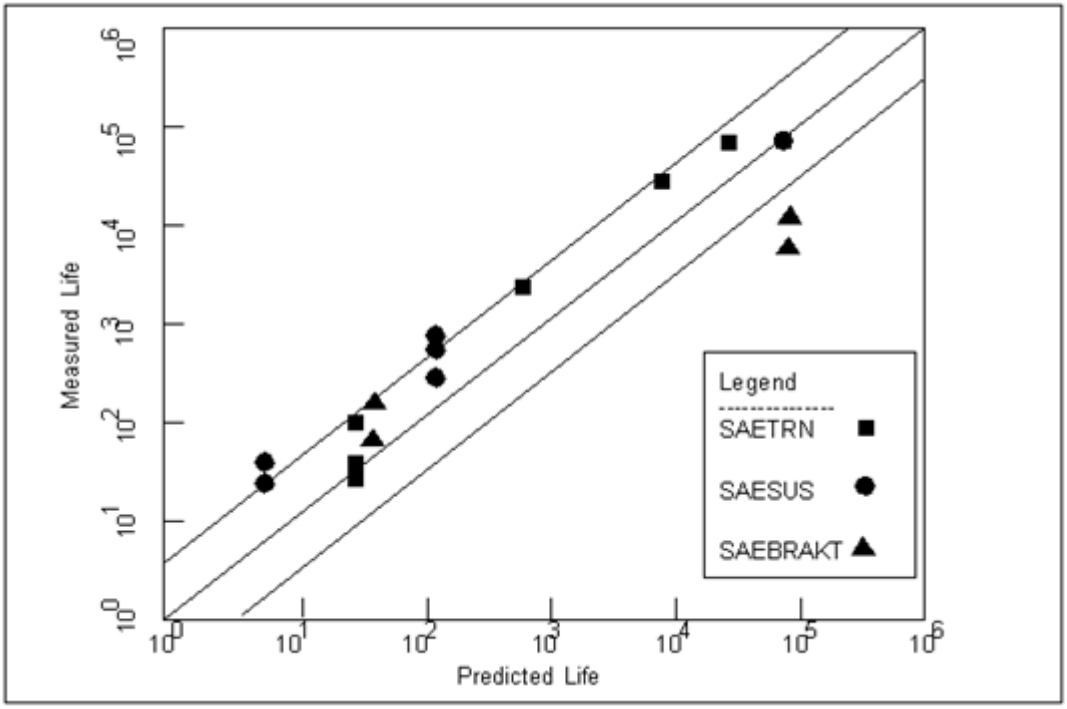


Figure 15-3 MSC Fatigue Measured vs. Predicted Lives, SAE Keyhole*
Round Robin Program (1971-74), Crack Initiation RQC100

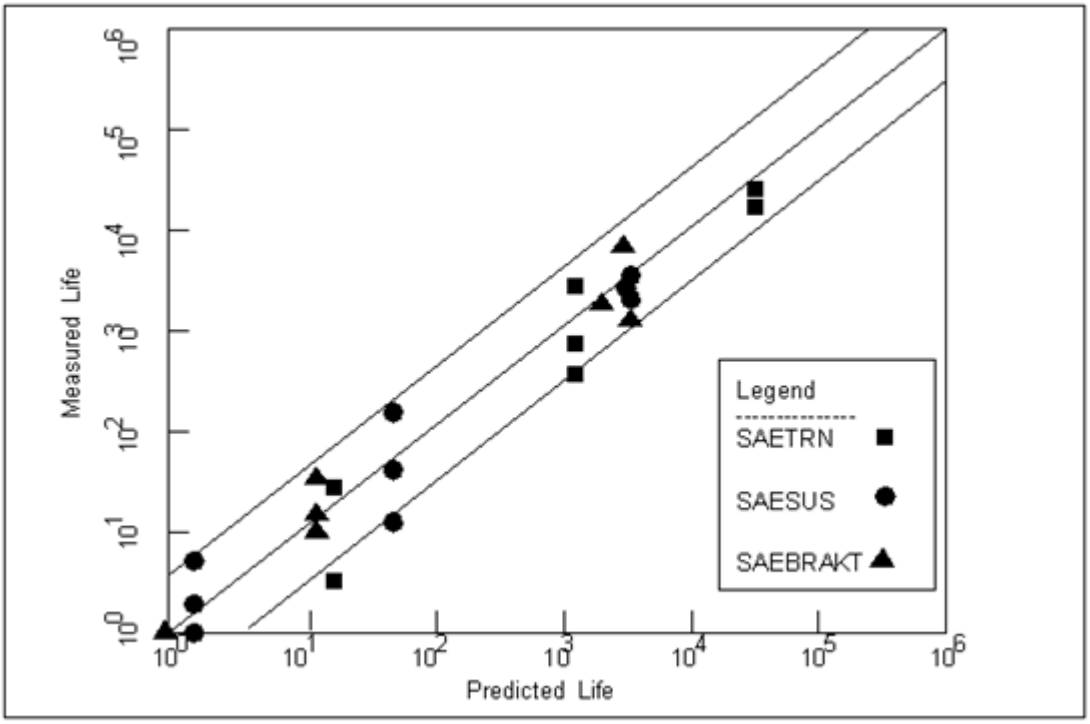


Figure 15-4 MSC Fatigue Measured vs. Predicted Lives, SAE Keyhole Round Robin Program (1971-74), Crack Initiation, MANTEN

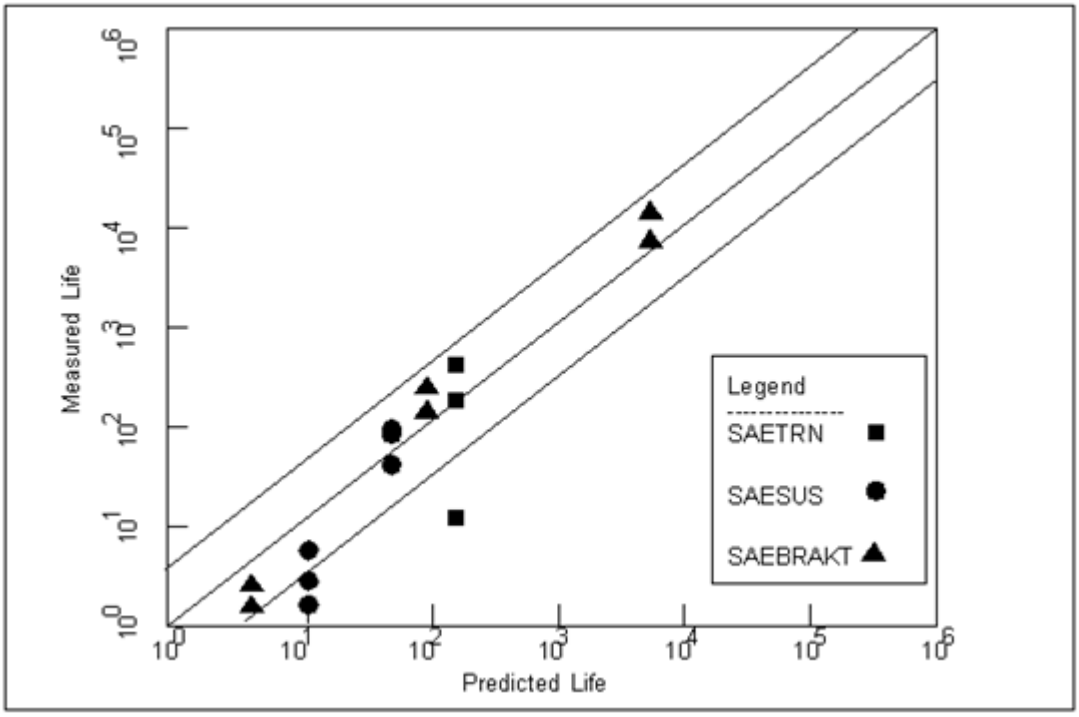


Figure 15-5 MSC Fatigue Measured vs. Predicted Lives, SAE Keyhole Round Robin Program (1971-74), Crack Growth, RQC100

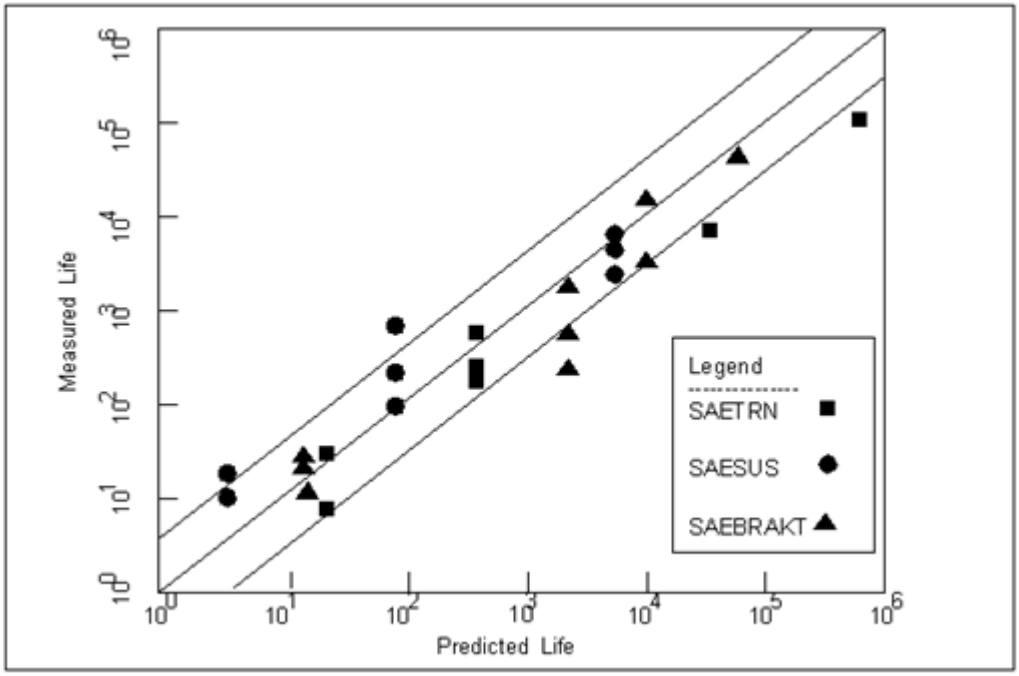


Figure 15-6 MSC Fatigue Measured vs. Predicted Lives, SAE Keyhole Round Robin Program (1971-74), Crack Growth, MANTEN

Problem 2: Analysis of an Underwater Pressure Vessel

Problem Description

This next problem is concerned with the crack initiation of a welded joint as shown in Figure 15-7. The joint is loaded in 4-point bending and is built in Patran using plane strain elements for analysis using Patran FEA. The purpose of this example is to show that MSC Fatigue can predict where in the weld the crack will develop and to compare this with the crack initiation site on the actual structure.

The strain-life method is generally not recommended for welded structures because of the poor quality of most welds. However, this is a high-quality weld and differs very little from the material around it. In most cases, a crack growth analysis would be more appropriate by assuming that the weld already has existing cracks or flaws.

A crack growth analysis is also carried out in this example to show that it is possible to design a leak-before-burst structure. This specimen happens to be a weld from a pressure vessel (submarine). It is important to be able to design with a specification of this sort since a catastrophic rupture could be fatal.

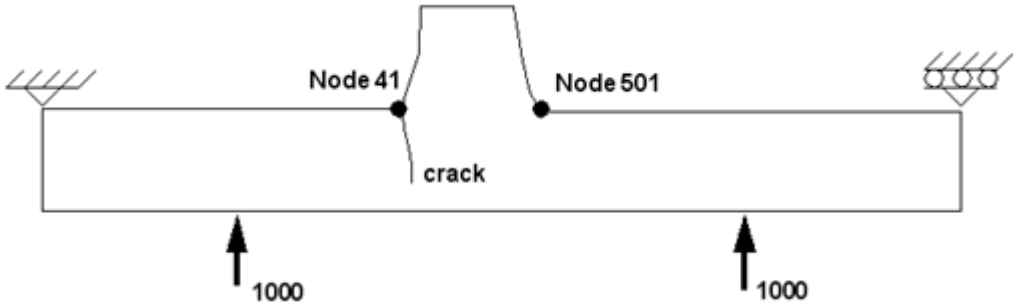


Figure 15-7 Weld Specimen Geometry

Crack Initiation of Weld

Objectives

1. To illustrate the use of the Time History Database Manager to create a simple time history.
2. How to set up your own fatigue equivalent or user fatigue units.
3. To determine the crack initiation life and compare to a simple hand calculation.
4. To determine where the crack will initiate and compare that to the actual structure.

Step 1: Geometry and FEA Results

The specimen was modeled in Patran using plane strain elements. Two 1000 Newton loads were symmetrically placed about the bottom to simulate a 4-point bending (such as pressurizing the vessel). Again this load is arbitrary since a FE analysis is linear and the stresses will later be normalized to a unit load during the MSC Fatigue run. The units are in millimeters and Newtons which gives stresses in MPa.

Stress contour maps of the results indicate that the high stresses are in the butt of the weld on the right and left sides. The maximum stress predicted by Patran FEA is approximately 960 MPa at node 501 (this is a maximum principal stress). The other node that will be of interest is node 41, the left butt of the weld directly opposite node 501. Naturally we expect the crack to initiate at the point of highest stress, all things being equal. However, this is not always the case when surface finish and treatment, residual compressive stresses, or other effects not modeled in the FE analysis are taken into account.

To properly execute this example, you will need the files `weld.out` and `weld.txt` which can be found in the `examples` directory delivered with MSC Fatigue.

```
<install_dir>/mscfatigue_files/examples
```

The `weld.out` file contains the weld geometry and the `weld.txt` file contains the stress results which need to be converted to binary format using the RESTXT program delivered with the MSC Fatigue system. Invoke this utility by typing **restxt** at the system prompt.

Step 2: Material Characterization

The material selected for this example is BS4360-50D. The strain-life properties are shown in Table 15-1. For later discussion, you may want to enter PFMAT and load material BS4360-50D and plot the strain-life curve. Again, since this material exists in the main MSC Fatigue materials database, it is not necessary to actually run PFMAT unless you wish to look at the materials curves graphically.

Table 15-6 Fatigue Material Properties: BS4360-50D

Material Properties	BS4360-50D
Strain Properties	
Fatigue strength coefficient, SF	1036
Fatigue strength exponent, b	-0.123
Fatigue ductility coefficient, Ef	0.622
Fatigue ductility exponent, c	-0.618
Cyclic strain hardening exponent, n	0.163
Cyclic strength coefficient, k' (Mpa)	912
LEFM Properties	
Unnotched Fatigue strength, FL (MPa)	152
Paris law coefficient, C (m/cycle)	2.22E-12
Paris law exponent, m	3.28
Delta K threshold at R=0, D0 (MPa m1/2)	6.17

Table 15-6 Fatigue Material Properties: BS4360-50D

Material Properties	BS4360-50D
Delta K threshold at R->1, D1 (MPa m1/2)	3
Stress ratio at threshold knee, Rc	0.594
Stress corrosion th'hld, K1SCC (MPa m1/2)	75.7
Monotonic Properties	
Young's Modulus, E (Mpa)	1.914E5
UTS (MPa)	480

Step 3: Loading Histories

For this example, a constant amplitude triangular wave is used to simulate the pressurization and depressurization of the pressure vessel. In our case, it is a force since we are simulating the bending of the joint with 4-point bending. PTIME is used to create this load history.

To demonstrate the use of user-defined life units, one block of this triangular wave is defined to be 1/10 of a mission. Or in other words, one mission consists of 10 blocks of the wave history signifying that the vessel is pressurized and depressurized approximately 10 x 10 = 100 times each mission, since there are 10 cycles in each block of the time history. Of course, the time units here are arbitrary and will not affect the life calculations.

The following PTIME keystrokes were used to create the time history. After this PTIME session, a file called `wave.dac` will exist in your directory as well as the `ptime.adb` and `ptime.tdb` files. This session assumes you are starting in a clean, empty directory.

Operation	Comments
ptime	Invoke PTIME either from the system prompt or from the MSC Fatigue Loading Information form.
Wave creation	Select the Wave creations option.
Filename: WAVE Description1: Constant Amplitude WAVE	Enter the name WAVE and also enter a description (Constant Amplitude Wave) in the Description 1 databox.
Number of fatigue equivalent units: 0.1	In this databox, enter the equivalent fatigue units.
Fatigue Equivalent units: Missions	In this databox, enter the name of the fatigue unit. Press the OK button when done.
Triangular	The next form is for specifying the wave properties. Select a Waveform Type of Triangular.
Frequency: 10	Set the frequency to 10.
Amplitude: 500	Set the amplitude to 500. Press the OK button accepting the default for everything else.

Operation	Comments
Finish	Press the Finish option on the Next Action form that appears.
Plot an entry	Now plot the time history. Accept the default name.
File / Exit / eXit	Close the graphical display by selecting Exit under the File menu and then select the eXit option to leave PTIME.

Note that the sample rate is 10 times the maximum frequency. This is a general rule in fatigue analysis as opposed to frequency based analyses that use the 3 times rule. This is because in fatigue, peak and valley information are extremely critical. If you use a sample rate that is to small, you will lose some of the information at these peaks and valleys and therefore predict unconservative lives.

Step 4: Setup MSC Fatigue Job

To set up the MSC Fatigue job for a crack initiation fatigue analysis of this weld, enter Patran or MSC Fatigue and enter the following keystrokes below. The necessary files `weld.out` and `weld.txt` can be found in the examples directory delivered with your MSC Fatigue system. As mentioned before, the `weld.txt` file needs to be converted by the Patran utility RESTXT to a binary format. Unless otherwise specified, accept defaults for any parameters not mentioned in the process below.

Operation	Comments
patran	Invoke Patran (or MSC Fatigue) if you have not already done so.
File / New...	Open a new database from the File pull -down menu. Call it "weld." Set the analysis preference to Patran FEA if asked. Ignore any warning messages.
File / Import...	Import the neutral file `weld.out` into the database. At this time you may wish to manipulate the model, turn labels off, or other Patran operations. When ready, go on to the next step.
Tools / FATIGUE... **(Analysis)**	Invoke Patran's FATIGUE interface by selecting it from under the Tools pull-down menu (or the Analysis application switch in MSC Fatigue).
General Setup Parameters:	
Analysis: Initiation	Set the analysis type to Crack Initiation on the main form.
Jobname: weld_ci	Give the job a name. Use *"weld_ci."*
Title: Crack Initiation analysis of a pressure vessel	Give the job a title.
Solution Parameters Form: Accept all defaults.	
Materials Information Form:	
Material: BS4360-50D	Place the cursor in the cell under the word Material and click on the mouse. A listbox will appear. Select the material BS4360-50D from this listbox.
Finish: Cast	Select Cast from the option menu that appears. The word Cast appears in the Finish cell.

Operation	Comments
Treatment: No Treatment	Select No Treatment from the option menu that appears.
Region: default_group	Select the default group as the region. It contains all the nodes of the entire model for which the fatigue analysis will be applied to. The Materials Information form can be closed down now by pressing the OK button.
Loading Information Form:	
Results From: Patran FEA	The results are from an external Patran FEA results file.
Select a Patran FEA Job: weld	Select the `weld.res` file from the listbox. Appear only as "weld". Make sure you actually select it so that it is highlighted.
Load Case ID: 1	Place the cursor in the Load Case ID cell and click the mouse button. A databox appears in which the load case ID from the FE analysis is to be entered. Press RETURN to accept the default (1).
Time History: WAVE	Select a time history from the list that appears. Our time history is the one that was previously created and scaled, WAVE.
Load Magnitude: 1000	Enter 1000 Newtons as the load magnitude. This is used to normalize the stresses from the FE analysis. Press RETURN to enter this value into the cell. This form can now be closed down by pressing the OK button.
Job Control Form:	
Full Analysis	Set the action to Full Analysis and click the Apply button. The database will close, translation will occur, and then the database will open again and the job will be submitted.
Monitor Job	Set the Action to Monitor on the Job Control form and click Apply each time you wish to see the progress state of the job.

Step 5: Evaluate Results

Since this is a simple example with a constant amplitude load we can easily come up with a rough approximation of the answer. To approach the problem we will follow these steps:

1. Roughly calculate the maximum stress.

   The FE analysis gives 960 MPa maximum principal stress at node 501 due to a 1000 Newton load. Since the stresses are linear with respect to the applied load, and since our load history amplitude is ±500 Newtons, the equivalent stress seen at node 501 should be approximately:

$$\frac{\sigma_{fea}}{P_{fea}} P_{max}(t) = S_{max} = \frac{960}{1000} 500 = 480 MPa$$

2. Convert the maximum stress to maximum strain.

We can divide this number by E=1.914E5 MPa to obtain a strain of approximately 0.0025. (Note: This can be dangerous if multi-axial stresses are present which would invalidate dividing by E to obtain strain. The stress we are considering is maximum principal stress but since our model is simple we assume that we have fairly uniaxial stress. Otherwise you would have to break the stress down to its components, convert the components to strains via the more complicated triaxial formulas (i.e., von Mises), and then convert back to maximum principal strain).

3. Calculate a Neuber product to account for plasticity.

   At this point, we could determine the life by going directly to the strain-life curve of BS4360-50D (use PFMAT). A strain of 0.0025 corresponds to about 3.35E4 reversals, which is 2N cycles or 16750 cycles. Remember that each block of our time history contains 10 cycles, so this corresponds to 1675 blocks or 167.5 missions since there are 10 blocks per mission.

   This calculation did not, however, take into account the elastic-plastic correction which MSC Fatigue uses. To account for this, we need to create a Neuber product which is simply:

   $S_{max} * \varepsilon_{max}$ = Neuber Product (NP) = 480 * 0.0025 = 1.2

   To correct for plasticity using this Neuber Product, we need to look up on the cyclic stress-strain curve (for BS4360-50D) an equivalent stress-strain product of 1.2 that lies on the cyclic stress-strain curve. From this we will determine the corrected strain and look up this value and its corresponding life on the strain-life plot. MSC Fatigue's material database manager PFMAT is quite easy to use in this exercise for this purpose. Follow Step 2 of Crack Initiation of Keyhole, 1235, except load material BS4360-50D instead. Once it is loaded, plot the cyclic stress-strain curve. Use the mouse to position the cursor over the curve and click on the left mouse button. The coordinates of the curve will be displayed in the upper left corner of the screen every time you depress the left mouse button. From these coordinate points, you can determine the NP and find the location and new strain where NP=1.2. To get better resolution, you can zoom in on the curve by using the right mouse button twice, one to specify the lower left corner of the new window and once to specify the upper right corner of the new window. The new corrected strain from this exercise gives 0.00365, with a corresponding stress of 329 MPa.

   $\sigma * \varepsilon$ = NP = 1.2 = 329*0.00365

4. Look up the approximate life on the strain-life plot.

   Now by looking at the strain-life curve of BS4360-50D (use PFMAT), a strain of 0.00365 corresponds to about 1.418E4 reversals, which is 2N cycles or 7090 cycles. Remember that each block of our time history contains 10 cycles, so this corresponds to 709 blocks or 71 missions since there are 10 blocks per mission.

To see the results, open the Results form from the MSC Fatigue main form. To quickly assess the most damaged nodes:

Operation	Comments
Results Form:	
List Results	Set the Action to List Results. The separate MSC Fatigue module PFPOST will be spawned. Click the Apply button.
OK	Press OK twice when the PFPOST form comes up to accept the default jobname.
Most damaged nodes	Select the first option to view the most damaged nodes.
End / eXit	Press End and then exit PFPOST.

The results of the crack initiation analysis (Table 15-7) indicate that a crack will initiate approximately after about 400 repeats of the load history or 40 missions at node 41 which is the left side in the butt of the weld. This is well within the same order of magnitude that we predicted by the hand calculation in the previous paragraph. This shows that MSC Fatigue can correctly predict where a crack will develop since this is where the crack initiated in the actual structure, at the location of node 41.

Table 15-7 Results of Crack Initiation Analysis of Weld

Method:	Life (Missions)
Hand Calculation (no plastic correction)	67.5
Hand Calculation (plastic correction)	71
MSC Fatigue (STW mean stress correction)	~40
MSC Fatigue (Morrow mean stress correction)	~75
MSC Fatigue (No mean stress correction)	~75

Note that during the job setup phase we indicated that MSC Fatigue was to use a Maximum Absolute Principal stress in its fatigue calculation. Since we only were concerned with the Maximum Principal stress in the hand calculation, we predicted the shortest life at node 501 but this turned out not to be the case in the actual analysis.

For a quick contour plot of log of life in Missions:

Operation	Comments
Results Form:	
Read Results...	Set the action to Read Results on the Results form in MSC Fatigue. Click the Apply button. The results will be read into the Patran database.
Results	Click on the Results application switch on the main form. This will bring up the Results application.
Select Results Cases	Select the results case from the listbox that was just read. It should be called *Crack Initiation* with the name of the job attached to the end.

Operation	Comments
Select Fringe Result	Select *Log of Life, in Repeats* from the Select Result listbox.
Apply	Click the Apply button to make the contour plot of log of life in repeats of the time history. Note that a special spectrum has been created for better viewing of life results. For damage results you will want to change the spectrum back to the Standard Spectrum. This can be done under the Display / Spectrums option from the top menu bar.
Results	Click on Results again in the main form. This will close down the Results application.

Step 6: Design Optimization

There are no design optimization objectives in this exercise. However, if you wish to see the results of the other mean stress correction methods set the Action to Optimize from the MSC Fatigue Results form. This will invoke the separate MSC Fatigue module called FEFAT.

Operation	Comments
Results Form:	
Optimize	Set the Action to Optimize. The separate MSC Fatigue module FEFAT will be spawned. You can select a node from the graphics screen or type one in if you wish. Leave it blank for now. Click the Apply button.
Worst case node	Select the Worst Case Node option from the listbox. The program will automatically scan the results file and pick the node with the shortest life (node 41).
Design Life	A design life must be entered. Put in any value, say 100.
End	Clear the summary page.
Sensitivity analysis...	Select the Sensitivity analysis option.
Mean stress correction (all)	Choose to analyze all mean stress correction methods.
Recalculate	Recalculate
End / eXit	Quit from FEFAT.

Perhaps the results suggest that no mean stress correction was necessary. This would make sense since our time history has zero mean. See Table 15-7.

Crack Growth of Weld

Objectives

1. To determine the propagation life of the weld.
2. To ensure that a leak before burst condition exists.

3. To compare the amount of time involved in initiating the crack as opposed to that of propagating the crack.
4. To investigate a different environment (seawater).

Step 1: Geometry and FEA Results

The finite element analysis is identical to that of the crack initiation example.

The compliance function for the LEFM method must exist prior to setting up the job. The compliance function defines the particular crack geometry and how it will grow or how the structure's stiffness changes as the crack grows. You can create it while setting up the MSC Fatigue job within Patran but in this case it is suggested that you create it by running PKSOL. You can create the compliance function by entering the units in millimeters and using "weld" for the file name. It is a crack in a welded plate joint and is a weld toe surface crack in bending. The plate thickness, T is 37.338 mm(1.47 in) and the breadth can be large since this is a plain strain problem, so say 500 mm(>18 in). The length is 457.2 mm (18 in) and the crack aspect ratio is 1. This means that the crack will grow outwardly in a circular pattern as opposed to an elliptical manner. The parametric angle is zero degrees which means the crack will grow straight down from the tee weld. PKSOL can be invoked from the command level by typing PKSOL. Use the following keystrokes:

Operation	Comments
pksol	Invoke PKSOL either from the system prompt or from within the Solution Parameter form in MSC Fatigue when the Analysis Type is set to Crack Growth by pressing the Comliance Generator button in the top right corner.
Millimeters	Define units in millimeters.
Generate a Y function table	Select the Generate a compliance function option.
weld	Type in a name. A file called `weld.ksn` will be generated.
Cracks in welded plate joints	Pick option number 7.
Weld toe surface cracks in bending	Pick option number 2.
Define	A graphical display appears to allow dimension definitions. Click on Define from the main form.
T= 37.338	Enter the dimension of T. Press RETURN.
B = 500	Enter the dimension of B.
L = 457.2	Enter the dimension of L.
a/c = 1	Accept the default.
Parametric angle: 0	Parametric angle of crack growth — straight down.
None	Just press RETURN. No changes are necessary.

Operation	Comments
Calculate	Pick the Calculate option from the main form.
Plot Y function	You may now plot the compliance if desired.
File / Exit / eXit	Exit PKSOL.

The dimensions can be set up in any units as long as you are consistent. The Y-function is a nondimensional table of numbers. The initial and final crack lengths and any notch dimensions can also be set up in any units as long as you are consistent and specify the units consistently. Initial and final crack lengths are defined in the MSC Fatigue setup mode within Patran or MSC Fatigue .

Step 2: Material Characterization

The material selected for this example is again BS4360-50D. The LEFM properties are shown back in Table 15-1 for a seawater environment. Again since this material exists in the main MSC Fatigue materials database, it is not necessary to actually run PFMAT unless you wish to look at the materials curves graphically.

Step 3: Loading Histories

The load history is identical to that of the previous crack initiation model and will require the same files. You must do Step 3: Loading Histories, 1265 of the crack initiation run if you skipped it.

Step 4: Setup MSC Fatigue Job

To set up the MSC Fatigue job for a crack growth analysis of this vessel weld, enter Patran and enter the following keystrokes below. This example assumes you are starting in a clean, empty directory. Accept all defaults unless otherwise directed.

Operation	Comments
patran	Invoke Patran (or MSC Fatigue) if you have not already done so.
File / New...	Open a new database from the File pull -down menu. Call it "weld." Set the analysis preference to Patran FEA if asked. Ignore any warning messages.
File / Import...	Import the neutral file `weld.out` into the database. At this time you may wish to manipulate the model, turn labels off, or other Patran operations. When ready, go on to the next step.
Tools / FATIGUE... **(Analysis)**	Invoke Patran's FATIGUE interface by selecting it from under the Tools pull-down menu (or the Analysis application switch in MSC Fatigue).
General Setup Parameters:	
Analysis: Growth	Set the analysis type to Crack Growth on the main form.
Jobname: weld_cg	Give the job a name. Use *weld_cg*.
Title: Crack Growth analysis of weld	Give the job a title.

Operation	Comments
Solution Parameters Form:	
Select a Compliance Function	Select the `weld.ksn` file that you previously created using PKSOL from the listbox. Make sure that it is highlighted.
Crack Length Units: Millimeters	Define the crack length in millimeters.
Initial Crack Length: 2	Set the initial crack length to 2 mm.
Final Crack Length: 37.3	Set the final crack length to 37.3 mm.
Materials Information Form:	
Material: BS4360-50D	Place the cursor in the cell under the word Material and click on the mouse. A listbox will appear. Select the material BS4360-50D from this listbox.
Environment: AIR	Choose air as the environment.

Before completely filling out the Materials form we need to digress a bit to create a special Patran group which contains the node or nodes of the specimen which define where the nominal or far field stress occurs since this is the stress that a crack growth analysis expects. This generally takes a bit of engineering judgment combined with an understanding of fracture mechanics theory. To do this, follow these instructions without closing down the MSC Fatigue forms.

Operation	Comments
Group / Create	From the top Patran form, select the Group pull down menu and choose the Create option.
New Group Name	In the form that appears, enter Node386 as the New Group Name. Make sure no spaces are in the name.
Entity Selection / Node 386	In the Entity Selection databox either type in **Node 386** or select node 386 from the graphics screen with the cursor. Then click the Apply button to create the group and then the Cancel button to close the form down.

Now that the new group is created you can continue to fill out the MSC Fatigue forms.

Operation	Comments
Materials Information Form:	
Region: Node386	Select the group that was just created. It contains the node defining where the far field stress is (an area away from the crack). The Materials Information form can be closed down now by clicking the OK button.
Loading Information Form:	
Results From: Patran FEA	The results are from an external Patran FEA results file.

Operation	Comments
Select a Patran FEA Job: weld	Press the Select File button and select the `weld.res` file from the listbox.
Load Case ID: 1	Place the cursor in the Load Case ID cell and click the mouse button. A databox appears in which the load case ID from the FE analysis is to be entered. Press RETURN to accept the default (1).
Time History: WAVE	Select a time history from the list that appears. Our time history is the one that was previously created and scaled, WAVE.
Load Magnitude: 1000	Enter 1000 Newtons as the load magnitude. This is used to normalize the stresses from the FE analysis. Press RETURN to enter this value into the cell. This form can now be closed down by clicking the OK button.
Job Control Form:	
Full Analysis	Set the action to Full Analysis and click the Apply button. The database will close, translation will occur, and then the database will open again and the job will be submitted.
Monitor Job	At this point, the job has been submitted and can be monitored as to its progress if desired. Set the Action to Monitor and click Apply each time you wish to see the progress state of the job.

At this point, the job has been submitted and can be monitored as to its progress if desired. You will notice the messages *After xxx Missions the crack length is y mm* appear as the status of the job. You will also notice after a time that the crack is not growing significantly, so abort the job.

Operation	Comments
Job Control Form:	
Abort Job	Set the Action to Abort on the Job Control form and click the Apply button.
Monitor Job	If you set the Action back to monitor you should see the message *Crack growth analysis completed successfully*, meaning, in this case, that the job has been stopped by the user.

Step 5: Evaluate Results

It is interesting to note, that when doing a crack growth analysis at the same load level as the crack initiation and letting MSC Fatigue calculate the initial crack size by specifying a zero initial crack length, there is no significant growth. The crack actually stops growing once it has been initiated at this load level. (This could have been determined also by plotting the compliance function.) Check the results by opening the Results from the MSC Fatigue form.

Operation	Comments
Results Form:	
List Results	Set the action to List Results and click the Apply button. The program PCPOST will be spawned.
Results summary page	When PCPOST has appeared, choose to view the Results summary page. Note that the job has been aborted by the user and the there was no significant growth of the crack.
End / eXit	Exit from PCPOST.

Step 6: Design Optimization

Run PCRACK now by setting the Action to Optimize from the MSC Fatigue Results form. Enter a scale factor of 2 on the load history and set the initial crack size to 10 mm to see if the structure will leak before bursting by growing the crack through the plate to the other side without exceeding the fracture toughness of the material. It will grow 37.3mm (which is the thickness of the plate) in approximately 250 missions. So, due to the type of material and the loading, most of the life is spent propagating a crack and very little initiating a crack in this example.

Operation	Comments
Results Form:	
Optimize	Set the Action to Optimize and click the Apply button. The module PCRACK will be spawned.
Scaling Factor: 2	Change the scaling factor to 2 on this first setup form.
OK	Press the OK button to accept the first page.
OK	Press OK twice to accept the Output parameter page and answer Yes to overwrite the output file.
OK	Press OK to accept the K solution information.
Initial Crack Size: 10	Change the initial crack size to 10mm. Press OK to continue.
OK	Press OK to accept the Material and Environment . PCRACK will initiate the analysis.

Most likely this weld will be subject to a more harsh environment than air when in service. Change the environment to seawater and rerun the analysis.

Operation	Comments
End	Close the summary page (final situation of the previous analysis).
Edit analysis parameters	Select to edit analysis parameters.
Select material and environment	Choose to change material and environment.
List / seawater, free corr.	Press the List button next to Environment and select the first seawater choice in the listbox. Press the Accept button.
OK	Press OK to accept the material and environment.
Recalculate	Recalculate

At a life of under 150 missions, it is likely to last only half as long as it would in air. Based on this information you could then determine some sort of safe inspection period for crack development in the pressure vessel weld joints.

It might be beneficial to point out that with these types of structures (welds), crack initiation analyses are generally not appropriate in that welds can be viewed as already cracked. Therefore, it would be more beneficial to either use a total life (S-N) approach or go directly to a fracture mechanics (crack growth) approach and disregard the initiation life assuming that a crack already exists. Also in this example very little time is used in initiating a crack as opposed to propagating the crack, therefore the crack initiation could have been skipped entirely.

Problem 3: Comparison to Another Code

Problem Description

This example is concerned with validating MSC Fatigue against a widely used, in-house code from a large aerospace company. Only the local strain (crack initiation) method is considered here. The MSC Fatigue results are compared not only against the in-house code but also to actual test data.

Objectives

1. To validate MSC Fatigue against a widely used, in-house fatigue prediction code from a large aerospace company and actual test data.
2. To demonstrate a calibration to test data for accurate fatigue predictions.
3. To demonstrate a method to use stress time histories directly without respect to a specific geometry.
4. To demonstrate the input of your own material property parameters via Materials Database Manager.

Step 1: Geometry and FEA Results

For this series of benchmark problems, the geometry used is arbitrary because stress time histories are used as the loading inputs. Since MSC Fatigue is highly coupled to the FE model and needs to retrieve stresses (or strains) from the FE analysis, an arbitrary geometry and loading were created so as to reproduce a unit stress

due to a unit load. This ensures that the actual stress time history is used exactly as input without any scaling necessary.

If you look carefully at the geometry, you will notice that the units are inches and that a 1 PSI pressure has been applied at one end, while the other end has been constrained. At Node 1 a stress of 1000 PSI results. See the definition of the time history in Step 3: Loading Histories, 1279 of this example.

The geometry is in a file called `code.out` with the stress results in a file called `code.txt`, which needs to be converted via RESTXT to `code.res` similar to all previous example problems. Files can be found in

```
<install_dir>/mscfatigue_files/examples
```

Note: It is possible to set up a job without a finite element mesh using an MSC Fatigue module called FEFTRN or by using the Utilities option in FEFAT or PCRACK.

Step 2: Material Characterization

The material used is the same as that used by the in-house code (i.e., Aluminum 7075-T6). In order to get the exact same strain data curves into MSC Fatigue, the strain-life and stress-strain curves used by the in-house code were digitized and curve fit to produce the material properties as shown in Table 15-1. The MSC Fatigue calculations assume a good machined surface finish and no surface treatment. The following keystrokes can be used to enter the materials information into the MSC Fatigue materials database manager. Once the materials data is entered, you can inspect the corresponding strain-life and stress-strain plots, if you wish. Use these operations to enter the data into the MSC Fatigue materials database:

Operation	Comments
pfmat	Invoke PFMAT either from the system prompt or from the MSC Fatigue Materials form.
Create / data set 1	Click on the Create option and select data set 1.
OK	No password is necessary. A local copy of the database will be created in your directory.
Primary name: AL7075-T6	Give the material a name. You may fill out the other fields in this form if you desire.
OK	Accept this first page of descriptive data.
Generic Data	Enter the Monotonic properties from Table 15-8. Properties not listed may be left blank.
E-N Data	Enter the Strain properties from Table 15-8. Properties not listed may be left blank. The defaults will be used.
S-N Data	Do not enter data in this form. Simply click the OK button.
Fracture Mechanics Data	Do not enter data in this form. Simply click the OK button. After this a confirmation that the material has been loaded will be given.
Graphical Display	Select the Graphical display option.
Strain Life Plot	Choose to plot the Strain Life plot.
File / Exit / eXit	Exit from PFMAT.

Table 15-8 Fatigue Material Properties: 7075-T6

Material Properties	7075-T6
Strain Properties	
Fatigue strength coefficient, Sf'	821
Fatigue strength exponent, b	-0.0953
Fatigue ductility coefficient, Ef'	0.398
Fatigue ductility exponent, c	-0.9251

Table 15-8 Fatigue Material Properties: 7075-T6

Material Properties	7075-T6
Cyclic strain hardening exponent, n'	0.0685
Cyclic strength coefficient, K' (Mpa)	799
Monotonic Properties	
Young's Modulus, E (Mpa)	70200
UTS (MPa)	558

Step 3: Loading Histories

The loading histories, as mentioned before, are stress histories. Because the FE model was created such as to give a unit stress due to a unit load, the stress time histories are taken as is, with no scaling factors applied. Table 15-9 shows the stress time history spectra for each problem. Basically, each block consists of an initial overload (and underload sometimes) followed by 999 (or 998) constant amplitude cycles for a total of 1000 cycles per block.

We will be creating these time histories by reading in ASCII versions of them which can be found in the examples directory delivered with your MSC Fatigue system. The names of the files are `coden.asc` where the *n* refers to the time history number from Table 15-9. Note that the values in these files are PSI and not KSI as reported in Table 15-9. This is because we are using stress at Node 1 which was reported to be 1000 PSI which scales the values in the files to KSI.

Table 15-9 Loading Stress Spectra

Problem No.	Step No.	Min. Stress	Max. Stress	No. of Cycles
1	1	0.0ksi	26.4ksi	1
	2	5.2ksi	26.4ksi	999
2	1	0.0ksi	47.3ksi	1
	2	5.2ksi	26.4ksi	999
3	1	0.0ksi	47.3ksi	1
	2	-7.9ksi	26.4ksi	1
	3	5.2ksi	26.4ksi	998
4	1	0.0ksi	47.3ksi	1
	2	-15.8ksi	26.4ksi	1
	3	5.2ksi	26.4ksi	998

Table 15-9 Loading Stress Spectra

Problem No.	Step No.	Min. Stress	Max. Stress	No. of Cycles
5	1	0.0ksi	47.3ksi	1
	2	-25.0ksi	26.4ksi	1
	3	5.2ksi	26.4ksi	998
6	1	0.0ksi	47.3ksi	1
	2	-32.5ksi	26.4ksi	1
	3	5.2ksi	26.4ksi	998
7	1	-50.0ksi	26.4ksi	1
	2	5.2ksi	26.4ksi	999

The following keystrokes are used to set up the load spectra into PTIME. At the end of the PTIME execution, there will be files called `coden.dac` in your directory. The ASCII read utility of PTIME is used to read in the load spectra and convert it into a form that PTIME can understand. This session assumes you are starting in a clean, empty directory.

Operation	Comments
ptime	Invoke PTIME either from the system prompt or from the MSC Fatigue Loading form.
ASCII convert & load	Select the ASCII load option.
ASCII Filename: code1	The form that appears asks for the filename to be read. This file ending with the suffix `.asc` will be read.
Time History: CODE1	Give the new time history a name. Use CODE1
OK	Accept this form after placing the asterisk in the filename databox.
Description 1	You must enter at least one descriptive title. Since you are loading all files at once, this description will be given to all load histories created.
Load type: Pressure	Set the Load type to Pressure.
Units: PSI	Set the Units to PSI. Press OK when ready. The time histories will be read from the ASCII files. Press End when done.
Plot an entry	Select the Plot an entry option to plot the new time histories. Use the List button to select one for viewing. Repeat this operation as many times as you wish to view the time histories.
File / Exit / eXit	Close the graphical display by selecting Exit under the File menu and then select the Exit option to leave PTIME.

Step 4: Setup MSC Fatigue Job

The files needed to set up the MSC Fatigue job can be found in the examples directory delivered with your MSC Fatigue system. The are called `code.out` and `code.txt`, where `code.out` contains the arbitrary geometry and `code.txt` contains the finite element results. The finite element results come from Patran FEA and therefore need to be translated into the binary results file `code.res` by way of the Patran utility RESTXT. The following operations setup the MSC Fatigue job:

Operation	Comments
patran	Invoke Patran (or MSC Fatigue) if you have not already done so.
File / New...	Open a new database from the File pull -down menu. Call it "code." Set the analysis preference to Patran FEA if asked. Ignore any warning messages.
File / Import...	Import the neutral file `code.out` into the database. At this time you may wish to manipulate the model, turn labels off, or other Patran operations. When ready, go on to the next step.
Tools / FATIGUE... **(Analysis)**	Invoke Patran's FATIGUE interface by selecting it from under the Tools pull-down menu (or the Analysis application switch in MSC Fatigue .
General Setup Parameters:	
Analysis: Initiation	Set the analysis type to Crack Initiation on the main form.
Units: PSI	Set the Stress Units to PSI.
Jobname: code	Give the job a name. Use *code*.
Title: Crack Initiation analysis comparison to another code	Give the job a title.
Solution Parameters Form: Accept all defaults.	
Materials Information Form:	
Material: AL7075-T6	Place the cursor in the cell under the word Material and click on the mouse. A listbox will appear. Select the material AL7075-T6 from this listbox.
Finish: Good Machined	Select Good Machined from the option menu that appears. The word Good Machined appears in the Finish cell.
Treatment: No Treatment	Select No Treatment from the option menu that appears.

Before completely filling out the Materials form we need to digress a bit and create a special Patran group which contains only the node with the unit stress associated with it. We only care to do a fatigue analysis at this node since the entire geometry is actually arbitrary.

Operation	Comments
Group / Create	From the top Patran form, select the Group pull down menu and choose the Create option.
New Group Name	In the form that appears, enter *Node1* as the New Group Name. Make sure there is no space in the group name.
Entity Selection / Node 1	In the Entity Selection databox either type in **Node 1** or select node 1 from the graphics screen with the cursor. Then click the Apply button to create the group and then the Cancel button to close the form down.

Operation	Comments
Materials Information Form:	
Region: Node1	Select the Node1 group as the region of the fatigue analysis now.
Loading Information Form:	
Results From: Patran FEA	The results are from an external Patran FEA results file.
Select a Patran FEA Job: code	Select the `code.res` file from the listbox. Appear only as "code." Make sure you actually select it so that it is highlighted.
Load Case ID: 1	Place the cursor in the Load Case ID cell and click the mouse button. A databox appears in which the load case ID from the FE analysis is to be entered. Press RETURN to accept the default (1).
Time History: CODE1	Select a time history from the list that appears. Our time history is the one that was previously created and scaled, CODE1.
Load Magnitude: 1.0	Enter 1.0 as the load magnitude. There is no need to normalize the stresses from the FE analysis. Press RETURN to enter this value into the cell. This form can now be closed down by clicking the OK button.
Job Control Form:	
Full Analysis	Set the action to Full Analysis and click the Apply button.
Monitor Job	At this point, the job has been submitted and can be monitored as to its progress if desired. Set the Action to Monitor and click Apply each time you wish to see the progress state of the job. Once the message *Fatigue analysis completed successfully* appears as the status message, the results can be examined.

Step 5: Evaluate Results

Quickly view the results.

Operation	Comments
Results Form:	
List Results	Set the Action to List Results. The separate MSC Fatigue module PFPOST will be spawned. Press the Apply button.
OK	Press OK when the PFPOST form comes up to accept the default jobname.
Most damaged nodes	Select the first option to view the most damaged nodes.
End / eXit	Press End and then exit PFPOST.

You will notice that the answer from MSC Fatigue is a very large number of Repeats. This number is actually meaningless without calibration. Our goal now is to calibrate MSC Fatigue to come in line with the actual test data results of 47 Repeats.

Step 6: Design Optimization

To illustrate this, invoke the MSC Fatigue module called FEFAT by typing the symbol **fefat** at the system prompt or pick Optimize from the Results form in MSC Fatigue. First reanalyze the original problem to see that the same answer is achieved and then set the parameter optimization to do a scale factor back calculation. You will be asked for a back calculation life. This life should be the actual test value of 47.657 Repeats. Reanalyze and note the scale factor calculated. Now you can run the rest of the problems by changing the time history and using this new calibration scale factor.

Operation	Comments
Results Form:	Or type fefat at the system prompt.
Optimize	Set the Action to Optimize. The separate MSC Fatigue module FEFAT will be spawned. You can select a node from the graphics screen or type one in if you wish. Type in "Node 1" since it is node 1 we wish to look at.
OK	Press the OK button to reanalyze using node 1.
Design Life [Repeats]: 47.647	Enter the targeted design life. Use a design life of 47.
End	Close down the summary page.
Parameter optimization... \ Scaling Factor	Select the Parameter optimization option to do a back calculation of life. Choose the scaling factor option.
Recalculate	Recalculate. Note the back calculated scale factor.
End / exit	Quit from FEFAT.

A back calculated scaling factor of about 2.2 is reported. Note the exact number on your computer and use it in the rest of the exercise. Instead of re-entering the Patran system to set up each additional problem, invoke the fatigue preprocessor routine FEFAT by typing the symbol **fefat** from the system prompt and enter the following keystrokes.

Operation	Comments
fefat	Invoke FEFAT from the system prompt.
Preprocessing and analysis	Go into the preprocessing portion of the code.
Input file (CODE_CI):	Accept this as the default. Press OK.
Node to process (ALL):	Accept the default.
Output Results filename (code):	Accept the default or enter a new name. You will be prompted for overwrite permission if you accept the default.
Edit load cases	Set this to Yes.
Full analysis (N)	Answer no to this question. Press OK.
Time History name: CODE2	Enter CODE2 for the second time history. Press OK.

We can now use our calibration scale factor.

Operation	Comments
Design Optimization	Enter the design optimization portion of FEFAT.
Input file (CODE_CI):	Accept this as the default. Press OK.
OK	Press OK to rerun the analysis. Don't forget to enter a design life if needed.
End	Press end to exit the result summary page.
Change parameters...	Select the Change parameters.
Scaling Factor: 2.2	Enter the scale factor that you noted earlier. Press OK.
Recalculate	Recalculate. Note the new life.
End / exit	Quit from FEFAT.

Table 15-10 tabularizes the measured and predicted lives. MSC Fatigue used three different mean stress correction factors, STW, Morrow, and no correction. Figure 15-8 plots the predicted vs. measured lives. Repeat the last few operations (**fefat**) for each different problem you wish to review.

Note that problem #1 is not listed in the results table. This is because problem #1 was used to "calibrate" MSC Fatigue, as was done for the in-house code as well. A "K" factor was determined to make the results for the first problem fall exactly in line with the test data. This factor was then applied to the remaining six problems.

In general, MSC Fatigue and the in-house code are in good agreement with the measured lives, falling well within the scatter band of Figure 15-8. However, the strain-life durability is very sensitive to the effects of positive mean stress and is highly dependent of surface finish and surface treatment.

Table 15-10 Measured and Predicted Lives

Problem No.	Test	In-House	MSC Fatigue No Correction	STW	Morrow
1	47.647		(calibrated with Kt=2.2		
2	1028	1182	~1200	~200	~330
3	309	450	~550	~130	~230
4	129	173	~280	~100	~170
5	64	59	~130	~60	~90
6	31	35	~85	~40	~60
7	32	26	~100	~16	~27

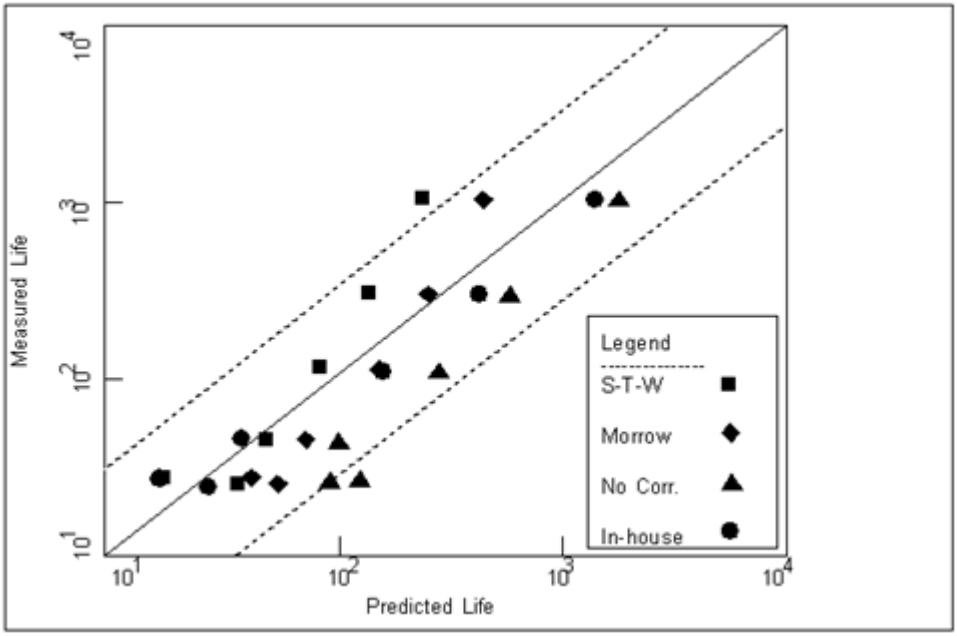


Figure 15-8 Measured and Predicted Lives of In-House Code

Problem 4: Ten Simple Notched Geometries

Problem Description

The problems here are described in Further Reading (App. 17). They consist of ten simple notched geometries which were tested at various constant amplitude load levels. Four different material types were used, whose properties are given later. In some cases, the measured life was to failure while for others it was to crack initiation. Figure 15-9 shows the various notched geometries.

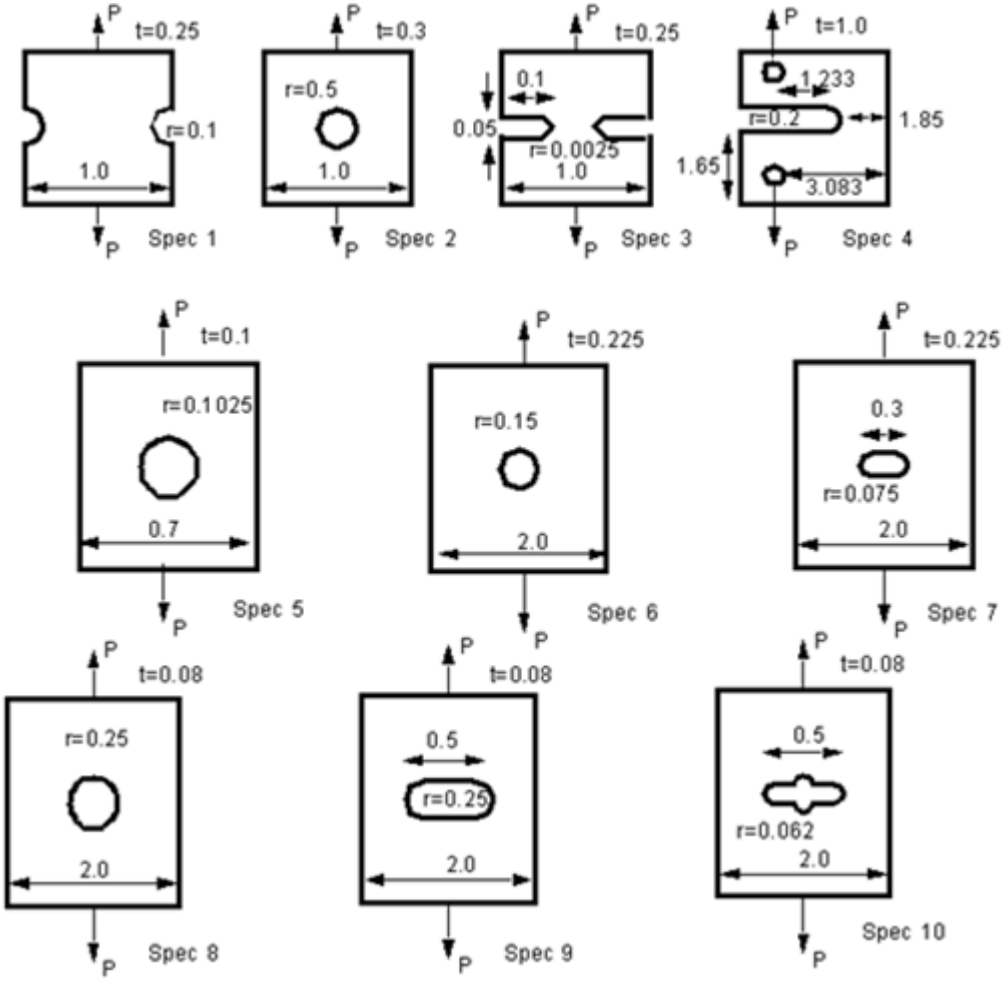


Figure 15-9 Ten Simple Notched Geometries (Dimensions in Inches)

Objectives

1. To predict the initiation life of these ten different specimens using the indicated load levels and material properties.
2. To validate MSC Fatigue against the obtained measured test data relative to these ten specimens.

Step 1: Geometry and FEA Results

For this series of benchmark problems, the geometries were created in Patran. Due to symmetry, only half of each specimen was actually modeled. Patran FEA was used as the analysis code using plane stress elements. The Patran neutral files and Patran FEA results files can be found in the examples directory delivered with your MSC Fatigue system. The neutral files containing the geometries are in files `notch`***`xx`***`.out` where *xx*

= 01 - 10. The stress-strain results are in files `notchXX.txt` which need to be converted to binary format using the Patran utility program RESTXT. See previous examples. Files can be found in

```
<install_dir>/mscfatigue_files/examples
```

Step 2: Material Characterization

Table 15-11 shows the four materials used in the study of these 10 specimens. They must be input to the MSC Fatigue materials database manager PFMAT. Follow the method used in Problem 3: Comparison to Another Code, 1276, Step 2: Material Characterization, 1278 to input the four material properties and name them MATA, MATB, MATC, and MATD.

Hint: You can set the units preference to KSI so that you do not have to convert the values in Table 15-11 to SI units. Use the Preferences option from the PFMAT main menu.

Table 15-11 Fatigue material properties

Material A:	Medium strength steel
Material B:	High strength steel
Material C:	Low strength steel
Material D:	High strength aluminum

Material Properties	Material			
	A	B	C	D
Strain Properties				
Fatigue strength coefficient, Sf '(KSI)	169	267	162	241
Fatigue strength exponent, b	-0.081	-0.09	-0.11	-0.15
Fatigue ductility coefficient, Ef'	1.142	0.79	0.338	0.158
Fatigue ductility exponent, c	-0.67	-0.73	-0.48	-0.83
Cyclic strain hardening exponent, n'	0.123	0.112	0.226	0.04
Cyclic strength coefficient, k' (KSI)	154	235	194	101
Standard error, Se	0	0	0	0
Monotonic Properties				
Yield strength, Sy, (KSI)	94	158	51	78
Young's Modulus, E (KSI)	30E3	30E3	30E3	10E3
UTS (KSI)	114	168	78	85

Step 3: Loading Histories

Loading was performed with constant amplitude load levels. The first seven specimens used a stress ratio of R = -1 (fully reversed) while the last three specimens used a R ratio of 0.1. The magnitudes of the loads are

given in Table 15-12. They were created in PTIME by making just one triangular cycle with a max. and min. value of ±20kips/in for R=-1 and entering X-Y points with max=20000, and min=2000 for R=0.1. These time histories were then scaled in FEFAT for the rest of the load levels and specimens. All Patran FEA loads are in pounds/inch and the scale factors used properly convert the time history loads from ±20kips/in. to the appropriate kip/in depending on the specimen geometry and load level. (Specimen 4 used kip loading as opposed to kip/in.)

The following keystrokes create these two simple constant amplitude load histories. This assumes you are starting in a clean, empty directory:

Operation	Comments
ptime	Invoke PTIME either from the system prompt or from the MSC Fatigue Loading form.
Wave creation	Select the Wave creation option.
Filename: 20KIPR1	Give it a filename called *20KIPR1*.
Description 1: Fully reversed	Give it a description.
Load type: Force	Select the load type to be Force.
Units: lbs force	Select the units to be lbs force.
OK	Accept this form.
Triangular	Create a triangular wave. Use the defaults of all values except what is described below.
Amplitude: 20000	Set the amplitude to *20,000*.
OK / Finish	Accept this form and select Finish for the next action. The percentage of positive slope should be 50.
Plot an entry	Select the Plot an entry option and accept the default. Close the graphics when done.
Add an entry...	Select the Add option.
X-y time series	Choose the X-Y point creation option to create the other time history.
Filename: 20KIPR01	Give it a filename called *20KIPR01*.
Description 1: Not fully reversed	Give it a description.
Load type: Force	Select the load type to be Force.
Units: lbs force	Select the units to be lbs force.
OK	Accept this form.
2000 / 20000 / 2000 / OK / OK	Enter these three values and use the RETURN key each time. Click the OK with a blank entry when done.
End	Press the End button when done defining the X-Y points.
Plot an entry	Select the Plot option and accept the default. Close the graphics when done. Then exit from PTIME.

Step 4: Setup MSC Fatigue Job

Enter Patran and use the following operations to setup the MSC Fatigue job for the first specimen. Accept all defaults unless otherwise specified:

Operation	Comments
patran	Invoke Patran (or MSC Fatigue) if you have not already done so.
File / New...	Open a new database from the File pull -down menu. Call it "notch." Set the analysis preference to Patran FEA if asked. Ignore any warning messages.
File / Import...	Import the neutral file `notch01.out` into the database. At this time you may wish to manipulate the model, turn labels off, or other Patran operations. When ready, go on to the next step.
Tools / FATIGUE... **(Analysis)**	Invoke Patran's FATIGUE interface by selecting it from under the Tools pull-down menu (or the Analysis application switch in MSC Fatigue).
General Setup Parameters:	
Analysis: Initiation	Set the analysis type to Crack Initiation on the main form.
Units: PSI	Set the Stress Units to PSI.
Jobname: notch1a	Give the job a name. Use *notch1a*.
Title: Notch specimen one with material A	Give the job a title.
Solution Parameter Form: Accept all defaults.	
Materials Information Form:	
Material: mata	Place the cursor in the cell under the word Material and click on the mouse. A listbox will appear. Select the material MATA from this listbox.
Finish: Polished	Select Polished from the option menu that appears. The word polished appears in the Finish cell. All ten specimens were polished specimen with no surface treatment.
Treatment: No Treatment	Select No Treatment from the option menu that appears.
Region: default_group	Select the default group as the region. It contains all the nodes of the entire model for which the fatigue analysis will be applied to. The Materials Information form can be closed down now by clicking the OK button.
Loading Information Form:	
Results From: Patran FEA	The results are from an external Patran FEA results file.
Select a Patran FEA Job: notch01	Press the Select File button and select the `notch01.res` file from the listbox.

Operation	Comments
Load Case ID: 1	Place the cursor in the Load Case ID cell and click the mouse button. A databox appears in which the load case ID from the FE analysis is to be entered. Press RETURN to accept the default (1).
Time History: 20KIPR1	Select a time history from the list that appears. Our time history is the one that was previously created and scaled, SAETRN15.
Load Magnitude: 20000	Enter 20,000 lbs as the load magnitude. This is used to normalize the stresses from the FE analysis. Press RETURN to enter this value into the cell. This form can now be closed down by clicking the OK button.
Job Control Form:	
Partial Analysis	Set the action to Partial Analysis and click the Apply button. Open the main FATIGUE and Job Control forms again when the translation is complete.
Monitor Job	At this point the job has been submitted and can be monitored as to its progress if desired. Set the Action to Monitor and click Apply each time you wish to see the progress state of the job.

At this point, the job has been submitted and can be monitored as to its progress if desired. Once the message *Preprocessing completed successfully* appears as the status message, the results can be determined by FEFAT. Note that we did not run the analysis to completion and only went through the preprocessor stage.

Step 5: Evaluate Results

Table 15-12 and Figure 15-10 show results for measured vs. MSC Fatigue predicted lives. Node 1 for all notch specimens has the most damage. To duplicate the results presented in Table 15-12 for the first specimen using FEFAT, use the following keystrokes.

Operation	Comments
Results Form:	
Optimize / Node 1	Set the Action to Optimize and click the Apply button in invoke FEFAT. Put "Node 1" in the databox as the node to be analyzed.
Design Life / OK	Press the OK button when FEFAT appears after entering a design life of 1000.
End	Close the summary page down.
Sensitivity Analysis...	Select the sensitivity analysis option.
Scaling Factors	Enter some scaling factors separated by commas from Table 15-12.
Recalculate	Recalculate
End	Clear the summary page.
Results Display...	Select the results display option.
Sensitivity Plot	Plot the scaled fatigue lives just calculated. Select the Close option from the File menu when done.
Change Parameters...	Select the Change parameters option.
Mean Stress Correction: Morrow	Change the mean stress correction method to Morrow.
Recalculate	Recalculate. You can repeat these operations for no mean stress correction. Compare the values to those in Table 15-12.
End / eXit	Quit from FEFAT when done.

You may repeat this entire exercise for all other notch configurations if you wish to reproduce all the results in Table 15-12. If you do, do not forget to use the correct material (A,B,C, or D) and the correct load time history (R=1, or R=0.1).

It might be pointed out that the lives at high load levels do not appear to give good results compared to those at lower loading levels. This is due to the fact that at these higher loading levels the specimens are being literally torn apart where wide areas are experiencing stresses exceeding the yield stress. When this happens the fatigue theory begins to break down since it is based on a local strain approach. There are other correction methods in MSC Fatigue, namely the Mertens-Dittmann and the Seeger-Beste, that can better compensate for high plasticity.

Table 15-12 Measured vs. Predicted Lives

Specimen No. (mat.)	Load Amplitude (KIP)	Scale Factor	Initiation Life (cycles)	MSC Fatigue STWMorrowNone		
1(A) R=-1	20	1.0	62	~300	~280	~280
	16	0.8	635	~630	~560	~560
	14	0.7	1300	~1,000	~880	~880
	12	0.6	2400	~1,750	~1,500	~1,500
	10	0.5	6000	~3,600	~3100	~3,100
	8	0.4	14000	~10,400	~8,700	~8,800
	7	0.35	19000	~22,400	~18,350	~18,600
	6	0.3	100000	~6.5E4	~5.27E4	~5.38E4
	6	0.3	8850	"	"	“
	5.4	0.27	1.8e6	~1.55E5	~1.27E5	~1.31E5
	5.2	0.26	500000	~2.19E5	~1.81E5	~1.87E5
2(A) R=-1	13.99	0.6995	68	~490	~440	~440
	12.65	0.6325	190	~690	~600	~600
	12.11	0.6055	265	~770	~680	~680
	10.65	0.5325	1250	~1,230	~1,080	~1,080
	9.02	0.4510	3600	~2,300	~2,000	~2,010
	9.03	0.4515	2400	~2,290	~1,990	~2,000
	7.00	0.3500	11500	~6,940	~5,880	~5,940
	5.68	0.2840	5540	~21,700	~17,940	~18,250
	4.95	0.2475	160780	~5.53E4	~44,550	~45,740
	4.70	0.2350	188000	~8.26E4	~6.76E4	~6,98E4
3(A) R=-1	18	0.90	4	~30	~25	~25
	14	0.70	47	~50	~50	~50
	11.4	0.57	133	~100	~90	~90
	9	0.45	1000	~190	~170	~170
	6.6	0.33	2000	~500	~450	~450
	5	0.25	1000	~1,290	~1,130	~1,140
	3	0.15	11150	~11,840	~9,900	~10,020
	2	0.10	62180	~2.16E5	~1.81E5	~1.89E5
	1.8	0.09	77000	~6.1E5	~5.34E5	~5.65E5

Table 15-12 Measured vs. Predicted Lives (continued)

Specimen No. (mat.)	Load Amplitude (KIP)	Scale Factor	Initiation Life (cycles)	MSC Fatigue STWMorrowNone		
4(B) R=-1	40	2	17	~35	~30	~30
	27.5	1.375	150	~100	~90	~90
	18	0.9	742	~440	~370	~370
	12.5	0.625	3600	~2,280	~1,920	~1940
	9.5	0.475	32500	~13,880	~12,120	~12,480
	8	0.4	600000	~6.17E4	~5.82E4	~6.11E4
	8	0.4	97000	"	"	"
	7.5	0.375	120000	~1.15E5	~1.12E5	~1.19E5
	7	0.35	190000	~2.31E5	~2.31E5	~2.46E5
5(B) R=-1	4.2	0.3	15000	~530	~450	~450
	3.5	0.25	35000	~1,160	~980	~990
	2.8	0.2	92000	~3,800	~3,240	~3,300
	2.3	0.164	250000	~15,420	~13,620	~14,140
6(C) R=-1	17.64	0.441	3603	~2,380	~2,230	~2,240
	17.64	0.441	2800	"	"	"
	14.70	0.3675	13340	~5,073	~4,760	~4,790
	11.76	0.294	51960	~13,540	~12,750	~12,850
	10.59	0.265	129000	~21,940	~20,700	~20,890
	9.41	0.2353	230460	~39,030	~36,930	~37,330
	8.82	0.2205	424750	~5.41E4	~5.13E4	~5.19E4
	8.24	0.206	589510	~7.7E4	~7.32E4	~7.41E4
	7.75	0.1938	1180700	~1.07E5	~1.02E5	~1.03E5
7(C) R=-1	20.25	0.5	924	~490	~450	~450
	18.00	0.45	2400	~730	~670	~680
	15.75	0.3938	5400	~1,220	~1,130	~1,140
	12.80	0.32	14000	~2,800	~2,610	~2,620
	11.25	0.2813	25000	~4,800	~4,480	~4,500
	9.00	0.225	87315	~12,800	~11,960	~12,040
	7.43	0.1858	170000	~31,500	~29,500	~29,760

Table 15-12 Measured vs. Predicted Lives (continued)

Specimen No. (mat.)	Load Amplitude (KIP)	Scale Factor	Initiation Life (cycles)	MSC Fatigue STWMorrowNone		
8(D) R=0.1	4.8	0.12	4850	~5,560	14,110	~41,040
	4.0	0.1	8850	~11,930	~41,330	~1.37E5
	3.2	0.08	22600	~41,640	~2.04E5	~6.02E5
	2.4	0.06	27500	~2.79E5	~1.83E6	~4.09E6
	1.6	0.04	2650000	~4.16E6	~3.61E7	~6.1E7
9(D) R=0.1	4.8	0.12	2190	~1,950	~3,240	~6,570
	4.0	0.1	5900	~3,790	~8,120	~21,340
	3.2	0.08	10800	~9,100	~28,430	~9.26E4
	2.4	0.06	27500	~41,700	~2.08E5	~6.25E5
	1.6	0.04	2100000	~6.13E5	~4.53E6	~9.31E6
10(D) R=0.1	4.8	0.12	945	~620	~650	~840
	4.0	0.1	1940	~1,320	~1,900	~3,180
	3.2	0.08	4380	~2,960	~5,670	~13,290
	2.4	0.06	10000	~9,280	~28,260	~8.77E4
	1.6	0.04	70500	~1.02E5	~5.35E5	~1.3E6

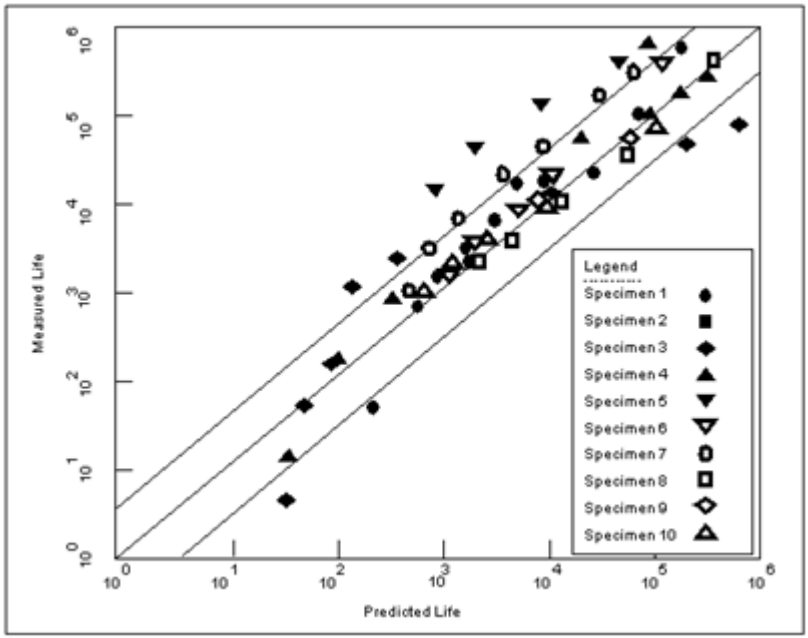


Figure 15-10 Measured and Predicted Lives of Notched Specimens

Problem 5: Transient Results

Problem Description

In this example, we investigate the use of transient FE results in a MSC Fatigue job. This example is broken up into two separate examples, one using external results files, and the other using transient results from Patran FEA. The first has no specific geometry associated with it while the second is the familiar keyhole model. In both cases, we investigate the difference between the transient approach and the normal pseudo-static method in which a load time history is associated with each static loading case. All things being equal, the two should produce the same results.

Simple Transient Comparison

Objectives

1. To show how to set up a MSC Fatigue job using external transient results files.
2. To compare this method with results from the pseudo-static method.

Step 1: Geometry and FEA Results

In this particular example, there is no geometry and we have synthesized some results to simulate a transient analysis that may have come from any analysis code which the Patran system supports.

During the job setup stage, we will create a single node to which we will associate these stress results. There are nine time steps which reside in nine separate files called `transN.gps` where `N` corresponds to the time step number. These file can be found in the examples directory delivered with your MSC Fatigue system.

```
<install_dir>/mscfatigue_files/examples
```

For comparison purposes, we will also use the stresses from the first of these files (`trans1.gps`) with an equivalent load time history to run a pseudo-static fatigue calculation to compare with fatigue life from the transient analysis. This is possible due to the simplicity of this example.

Step 2: Material Characterization

The material chosen is the MANTEN steel alloy with a polished surface finish and no surface treatment. The material already resides in the materials database.

Step 3: Loading Histories

The loading time history is defined by the transient analysis and therefore it is not necessary to create one. However, we also wish to run a fatigue analysis from pseudo-static results for comparison purposes.

The time history will be created by scaling the stress results from the first time step by the ratio relative to the other time steps. These happen to be -2, 2, -1, 0.5, -0.5, 1.5, 1, and 2. In other words the stresses from the second time step are twice (negative) that of the first time step, the third are twice the first, the forth are negative the first, the fifth are half the first, etc. Figure 15-11 shows the actual load history. The following

PTIME keystrokes will reproduce this time history using the X-Y point option. This exercise assumes you are starting in a clean, empty directory.

Operation	Comments
ptime	Invoke PTIME either from the system prompt or from the MSC Fatigue Loading Information form.
Enter X-Y points	Choose the X-Y point creation option to create the other time history.
Filename: TRANSIENT	Give it a filename called *TRANSIENT*.
Description 1: Transient example	Give it a description. Accept the defaults for all other fields. Press OK when done.
0 / 1 / -2 / 2 / -1 / 0.5 / -0.5 / 1.5 / 1 / 2 / 0	Enter these values and use the RETURN key each time.
End	Press the End button when done defining the X-Y points.
Plot an entry	Select the Plot option and accept the default. Close the graphics when done.
File / Exit / eXit	Close the graphical display by selecting Exit under the File menu and then select the Exit option to leave PTIME.

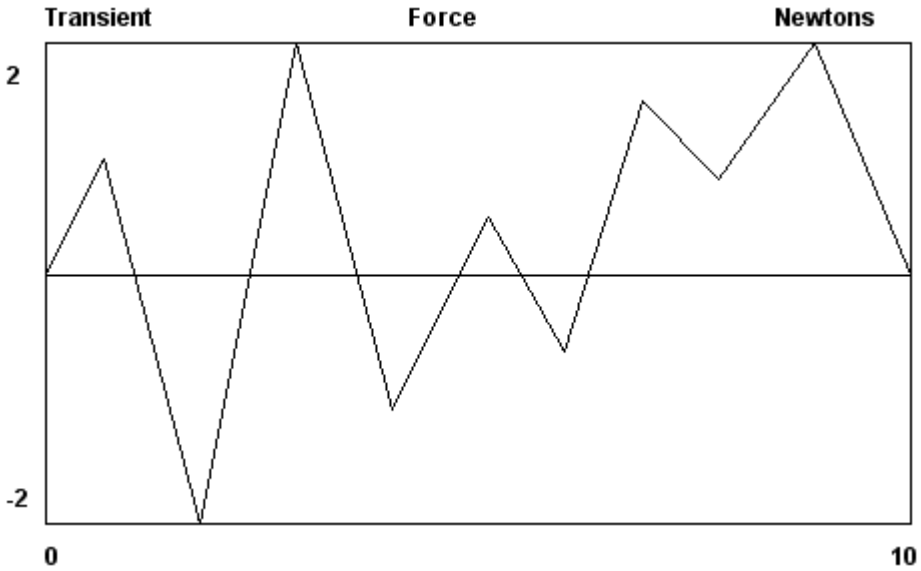


Figure 15-11 Transient Example Time History Equivalent

Step 4: Setup MSC Fatigue Job

Enter Patran and use the following keystrokes to set up both jobs. Accept all defaults unless otherwise specified.

Operation	Comments
patran	Invoke Patran (or MSC Fatigue) if you have not already done so.
File / New...	Open a new database from the File pull -down menu. Call it "transient." Set the analysis preference to MSC.Nastran if asked. Ignore any warning messages.
Finite Elements	Open the Finite Elements application from the main form of Patran.
Action: Create **Object: Node** **Method: Edit**	Set the Action, Object, and Method accordingly.
Node Location List: [0,0,0]	Input "[0,0,0]" as the location of a node to be created and click the Apply button. If the Associate with Geometry toggle is ON, set it to OFF before clicking Apply. If you are using MSC Fatigue you can create the node by typing in the command window: STRING create_nodes[VIRTUAL] fem_create_nodes("Coord 0","Coord 0",TRUE,"1","[0,0,0]",create_nodes)
Tools / FATIGUE... **(Analysis)**	Invoke Patran's FATIGUE interface by selecting it from under the Tools pull-down menu (or the Analysis application switch in MSC Fatigue).
General Setup Parameters:	
Analysis: Initiation	Set the analysis type to Crack Initiation on the main form.
Jobname: Transient	Give the job a name. Use *transient*.
Title: Transient Example	Give the job a title.
Solution Parameters Form: Accept all defaults.	
Materials Information Form:	
Material: MANTEN	Place the cursor in the cell under the word Material and click on the mouse. A listbox will appear. Select the material MANTEN from this listbox.
Finish: Polished	Select Polished from the option menu that appears. The word polished appears in the Finish cell. The SAE specimen was a polished specimen with no surface treatment.
Treatment: No Treatment	Select No Treatment from the option menu that appears.
Region: default_group	Select the default group as the region. This contains the node we created at the beginning of this exercise. The Materials Information form can be closed down now by clicking the OK button.

Operation	Comments
Loading Information Form:	
Results Type: Transient	FE results are from a Transient analysis.
Results From: External	The results are from an external Patran results file.
Surface: Top	This parameter is not applicable for this example.
External File Names: trans#.gps Columns: 1,2,3,4,5,6	Input the `trans#.gps` file name. Also indicate which columns the component stresses can be found in the files. Each time step comprises one file where the wildcard (#) will be replaced by the time step number.
Scale Factor: 1	Leave this at unity.
No. of Time Steps: 9	The number of time steps is 9. There are nine external stress results files in your local directory. Press OK.
Job Control Form:	
Full Analysis	Set the action to Full Analysis and click the Apply button. Let this job run and set up the static analysis for which we will compare against.
General Setup Parameters:	
Jobname: static	Change the jobname on the main form to *static*.
Title: Static Example	Give the job a different title.
Loading Information Form:	
Results Type: Static	Change the FE results from Transient to Static.
Results From: External	This should stay the same.
External File Names: trans#.gps Columns: 1,2,3,4,5,6	This stays the same.
Load Case ID: 1	Place the cursor in this cell and press RETURN to accept load case one from the displayed databox.
Time History: TRANSIENT	Select the TRANSIENT time history from the spreadsheet that appears when this cell is active.
Load Magnitude: 1.0	Accept the default for this cell. Make sure you press RETURN in the databox to accept the value in the cell.
Job Control Form:	
Full Analysis	Set the action to Full Analysis and press the Apply button.
Monitor	Monitor the job if you wish.

The following notes are made about the job setup:

1. Notice that when setting up the transient job, no time history and/or load case data was requested. This makes the setup much easier and quicker.

2. When specifying the time history files you are required to indicate the number of time steps. In this case there were nine. The program then looks for files called `trans1.gps`, `trans2.gps`, ..., `trans9.gps`. The # wild card is substituted for the time step numbers. For transient cases, these must be sequential starting as 1.
3. For static cases, the load case ID is substituted for the # wild card. It does not have to be sequential. For instance, we could have indicated `trans5.gps` to be the only load case by specifying the load case id as 5. In this case only `trans1.gps` was used and was scaled according to the externally defined time history `transient.dac`.

Step 5: Evaluate Results

The last portion of this exercise is to simply look at the results and see that they are the same between the two different fatigue jobs (~550 repeats).

Operation	Comments
Results Form:	
List Results	Set the Action to List Results. The separate MSC Fatigue module PFPOST will be spawned. Press the Apply button.
OK	Press OK twice when the PFPOST form comes up to accept the default jobname (static).
Most Damaged Nodes	Select the first option to view the most damaged elements. Press OK when done.
Cancel	Press the Cancel button to page up to the initial form.
Jobname: transient	Input the transient jobname.
Most Damaged Nodes / OK / eXit	Repeat the operations above to look at the transient results and then exit.

MSC Nastran Transient Results

Objectives

1. To show how to set up a MSC Fatigue job using MSC Nastran static and transient results.
2. To compare this method with results from the pseudo-static method as done in the previous example.

Step 1: Geometry and FEA Results

The geometry is the keyhole model that you have used in some of the previous problems. However, you will need to use a different neutral file that has been set up specifically for a modal transient analysis. The geometry and results are in the `key_stat.op2` and `key_tran.op2` files which can be found in the `examples` directory delivered with your MSC Fatigue system.

```
<install_dir>/mscfatigue_files/examples
```

Unlike previous examples you will need to import these results into the database.

Enter Patran or MSC Fatigue and use the following keystrokes:

Operation	Comments
patran	Invoke Patran (or MSC Fatigue) if you have not already done so.
File / New...	Open a new database from the File pull down menu. Call it “keyhole.” Set the analysis preference to MSC.Nastran if asked. Ignore any warning messages.
Analysis / Import	Open the Analysis application from the main form of Patran or the Import application in MSC Fatigue .
Action: Read Output2 **Object: Both** **Method: Translate**	Set the Action, Object, and Method accordingly.
Select Results File...	Select the result file `key_stat.op2`. Click the Apply button to import the model and static results.
Action: Read Output2 **Object: Result Entities** **Method: Translate**	Set the Action, Object, and Method accordingly.
Select Results File...	Select the result file `key_tran.op2`. Click the Apply button to import th transient results.
Tools / FATIGUE... **(Analysis)**	Invoke Patran’s FATIGUE interface by selecting it from under the Tools pull-down menu (or the Analysis application switch in MSC Fatigue).

The static load case results were determined using a 30 Newton loading at the same point of application as the original keyhole problem, the results from which, when scaled by the load time history should give roughly equivalent stress time histories for all nodes as does the modal transient analysis. This of course does not take into account any dynamic effects that the mass distribution may have on the dynamic behavior and resulting stress results, however with this simple model and a very evenly distributed mass, there should not be a large difference. The previous example we expected to obtain exactly the same answer between the pseudo-static and transient examples. This will not be the case for this problem but the answers should be statistically equivalent.

Step 2: Material Characterization

The material chosen is the MANTEN steel alloy with a polished surface finish and no surface treatment. The material already resides in the materials database.

Step 3: Loading Histories

The loading time history is defined by the transient analysis and therefore, is not necessary to create one. However, we also wish to run a fatigue analysis from pseudo-static results for comparison purposes.

The time history will be created by reading in an ASCII version of this time history that is equivalent to the force time history used in the transient analysis. This file can be found in the examples directory delivered with your MSC Fatigue system and is called `key_tran.asc`. The following PTIME keystrokes will accomplish this for you assuming you started in a clean, empty directory.

Operation	Comments
Loading Information Form:	
ptime	Invoke PTIME either from the system prompt or from the MSC Fatigue Loading form by pressing the Time History Manager button.
Add an entry / ASCII convert + load	Select the ASCII load option.
ASCII Filename: key_tran.asc	Enter the name of the file.
OK	Accept this form.
Description 1	You must enter at least one descriptive title. Accept the defaults for the rest of the fields and click the OK button.
Plot an entry...	Select the Plot a time history option to plot the new time histories. Use the List button to select one for viewing. Repeat this operation as many times as you wish to view the time histories.
File / Exit / eXit	Close the graphical display by selecting Exit under the File menu and then select the Exit option to leave PTIME.

Step 4: Setup MSC Fatigue Job

Re-enter Patran (if you left) and issue these commands to set up the fatigue analyses using both the transient and the static results from the MSC Fatigue main form accepting all defaults unless otherwise specified.

Operation	Comments
General Setup Parameters:	
Analysis: Initiation	Set the analysis type to Crack Initiation on the main form.
Jobname: key_tran	Give the job a name. Use *"key_tran."*
Title: Transient Example	Give the job a title.
Solution Parameters Form: Accept all defaults.	
Materials Information Form:	
Material: MANTEN	Place the cursor in the cell under the word Material and click on the mouse. A listbox will appear. Select the material MANTEN from this listbox.

Operation	Comments
Finish: Polished	Select Polished from the option menu that appears. The word polished appears in the Finish cell. The SAE specimen was a polished specimen with no surface treatment.
Treatment: No Treatment	Select No Treatment from the option menu that appears.
Region: default_group	Select the default group as the region. This contains all nodes in the model. The Materials Information form can be closed down now by clicking the OK button.
Loading Information Form:	
Results Type: Transient	FE results are from a Transient analysis.
Get/Filter Results...	Open this form from the Loading Information form. Select the only Result Case that appears in this top of this form. It may already be selected for you (LOAD_CASE.1, 131 subcases).
Filter Method:Global Variable - Variable:Time	Turn this option menu to Global Variable and set the Variable to Time. This may already be the default settings.
Filter/Add/Close	Press the Filter button. This will filter only subcases with time step information from the Result Case. Remember there is a static load case in these also. Click the Apply button. This will load all the time steps into the listbox on the Loading Information form which will be used in the analysis.
Results Time Steps / Select a Stress/Strain Tensor / Select a Layer	Select one of the time steps in the first listbox. Since there are only stresses and the top layer is the default this is all you need to do. Everything else is automatically selected. All the time steps in the list box will be used in the analysis. Click OK to accept the Loading form.
Job Control Form:	
Full Analysis	Set the action to Full Analysis and click the Apply button. Let this job run and set up the static analysis for which we will compare against.
General Setup Parameters:	
Analysis: Initiation	Set the analysis type to Crack Initiation on the main form.
Jobname: key_stat	Give the job a name. Use *"key_stat."*
Title: Static Example	Give the job a title.
Solution Parameters Form: Accept all defaults.	
Materials Information Form: This remains the same.	
Loading Information Form:	
Results Type: Static	Change the FE results from Transient to Static.
Load Case ID:	Place the cursor in this cell and click the mouse to reveal the database result listboxes.
Get/Filter Results...	Open this form from the Loading Information form.

Operation	Comments
Select All Results Cases	Turn this toggle on and click the Apply button. This will fill the listbox with available results from time step analyses.
Results Time Steps / Select a Stress/Strain Tensor / Select a Layer	Select the only "Static Subcase" in the first listbox (the time steps from the transient analysis also appear but don't select any of them). Since there are only stresses and the top layer is the default this is all you need to do. Everything else is automatically selected.
Fill Cell	Press the Fill Cell button to fill the Load Case ID cell with the internal Result Case IDs of the selected static results.
Time History: KEY_TRAN	Select the KEY_TRAN time history from the spreadsheet that appears when this cell is active.
Load Magnitude: 1.0	Accept the default for this cell. Make sure you press RETURN in the databox to accept the value in the cell.
Job Control Form:	
Full Analysis	Set the action to Full Analysis and click the Apply button.
Monitor	Monitor the job if you wish after reopening the forms.

Step 5: Evaluate Results

The last portion of this exercise is to simply look at the results and see that they are approximately the same between the two different fatigue jobs (~2400 repeats). There is a difference between the two which is most likely due to the fact that the modal transient takes into account the dynamic effects of the mass distribution whereas the pseudo-static does not, as mentioned earlier.

Operation	Comments
Results Form:	
List Results	Set the Action to List Results. The separate MSC Fatigue module PFPOST will be spawned. Press the Apply button.
OK	Press OK twice when the PFPOST form comes up to accept the default jobname ("key_stat").
Most Damaged Nodes	Select the first option to view the most damaged elements.
OK	Clear the summary page.
Cancel	Press the Cancel button to page up to the initial form. Repeat this for the "key_tran" job.

Problem 6: Spot Weld Analysis

H-Profile Spot Weld

The following example is intended as an illustration of the spot weld method. The component is a laboratory test specimen made from a mild steel V-1147 with two rows of five spot welds. The test piece is illustrated in Figure 15-12. The specimen is loaded in tension so that the spot welds are subject to shear loadings. A series of constant amplitude tests were carried out with a load ratio (minimum to maximum load) of 0.1.

Note that the FE model is a half model due to the plane of symmetry. The model consists of two sheets of shell elements joined by five bar elements representing the spot weld nuggets. The bars are stiff and have a length equal to half the sum of the sheet thicknesses.

The spot weld fatigue analysis system was used to predict load-life curves for these test specimens. The calculation was based on the S-N curves for spot welds in (Ref. 32.) derived for spot welds in St1403 with thicknesses from 0.66 mm to 2.5 mm and spot weld nugget diameters from 3.5 mm to 6.5 mm.

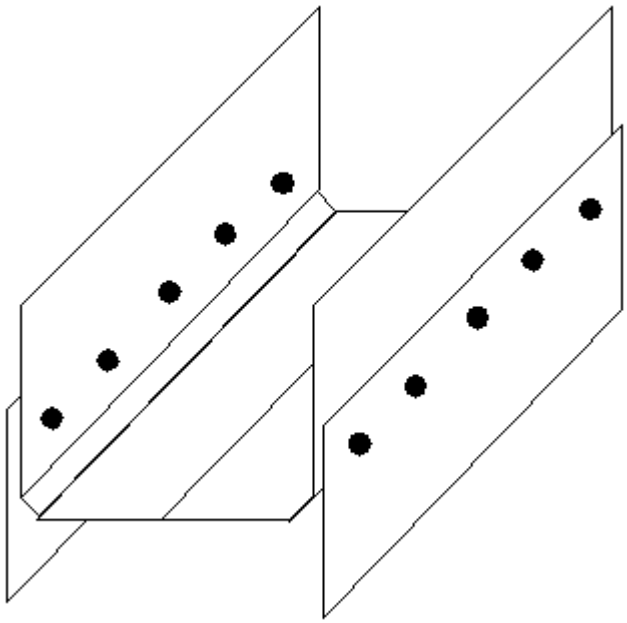


Figure 15-12 H-Profile Spot Welded Laboratory Specimen

This particular example assumes knowledge of the MSC Fatigue job setup.

Step 1: Geometry and Results

The geometry and results are provided in the file called `spotweld.op2` which can be found in the examples directory provided with MSC Fatigue.

```
<install_dir>/mscfatigue_files/examples
```

Step 2: Material Characterization

The Material Info. form needs a bit more information than for other analysis types. Namely, there will be an S-N curve defined for each sheet that connect the spot welds as well as for the spotweld nugget itself. These are provided in the MSC Fatigue materials database as spot_sheet_generic and spot_nugget_generic respectively. In addition, you must define the thickness of the sheets and the diameter of the spot weld nugget. There are also a number of other spot weld S-N curves defined in the materials database.

Step 3: Loading Histories

Define a time history with R=0.9 by entering XY points in PTIME using three points at 100, 1000, and 100 in a similar manner to previous problems. Call the load history name "spotweld." The loading is Force, in Netwons.

Step 4: Setup MSC Fatigue Job

Read in the old fatigue job setup. You will need the file called `spotweld.fin`. We will not actually run the fatigue analysis in this example as it is quite time consuming. Rather we will read in the job setup and discuss it briefly.

Operation	Comments
patran	Invoke Patran (or MSC Fatigue) if you have not already done so.
File / New...	Open a new database from the File pull -down menu. Call it "spotweld." Set the analysis preference to MSC.Nastran if asked. Ignore any warning messages.
Analysis / Import	Open the Analysis application from the main form of Patran or the Import application in MSC Fatigue .
Action: Read Output2 **Object: Both** **Method: Translate**	Set the Action, Object, and Method accordingly.
Select Results File...	Select the result file `spotweld.op2`. Press the Apply button to import the model and static results.
Group / Create...	Create a group with only the bar elements in it (the spot welds). These are elements 201, 202, 208, 209, and 210. Call the group "bars."
Tools / FATIGUE... **(Analysis)**	Invoke Patran's FATIGUE interface by selecting it from under the Tools pull-down menu (or the Analysis application switch in MSC Fatigue .
Jobname: spotweld	Enter the jobname "spoweldt" and press the RETURN or ENTER key. This will read in the job parameters. Inspect the job setup forms to see how they are set up.
Job Control...	Go to job control and click Apply with the Action set to Full Analysis and the job will run.

Step 5: Evaluate Results

The results of the fatigue analysis can be read in using the appropriate option from the fatigue Results form.

The spot weld fatigue results can be postprocessed in a number of ways:

1. By listing the global results file.
2. By plotting the results in Patran. Patran's Insight application is particularly good for this, allowing the fatigue life of each spot weld to be clearly visualized in the form of color coded spheres attached to each beam element.
3. Detailed information may also be obtained about the life, damage, stresses and forces at each calculation point (i.e. 108 sets of results per spot weld). Use the SPOTW module in interactive mode. Type **spotw** at the system prompt or invoke it from the Results or Job Control form.
4. Damage may also be visualized in the form of a polar plot as in Figure 15-14. A polar plot shows damage at discrete points around the spot weld. There will be three curves on each plot corresponding to each location of the spot weld, i.e, the nugget and the two locations where the spot weld connects to the sheets. Use the module SPOTW in interactive mode.

The studies so far indicate that in a typical application such as an automobile, the system correctly identifies all spot weld likely to give rise to durability problems. Using generic S-N curves as in this problem, life predictions tend to be conservative as shown in Figure 15-13. When S-N curves are used which are more representative of the material from which the component is constructed, more accurate life predictions can be made.

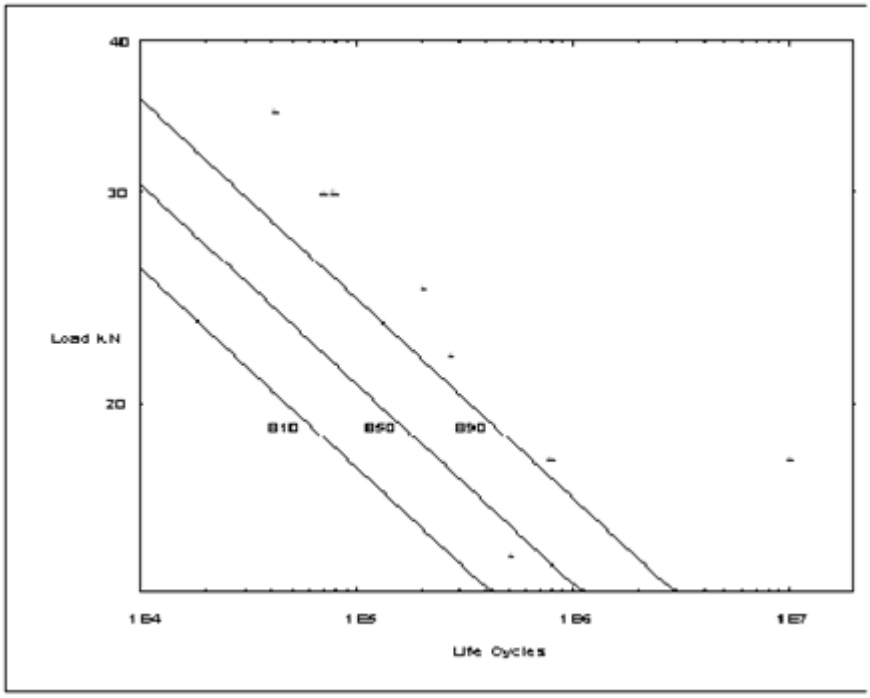


Figure 15-13 Comparison of Test Data with Scatter Band

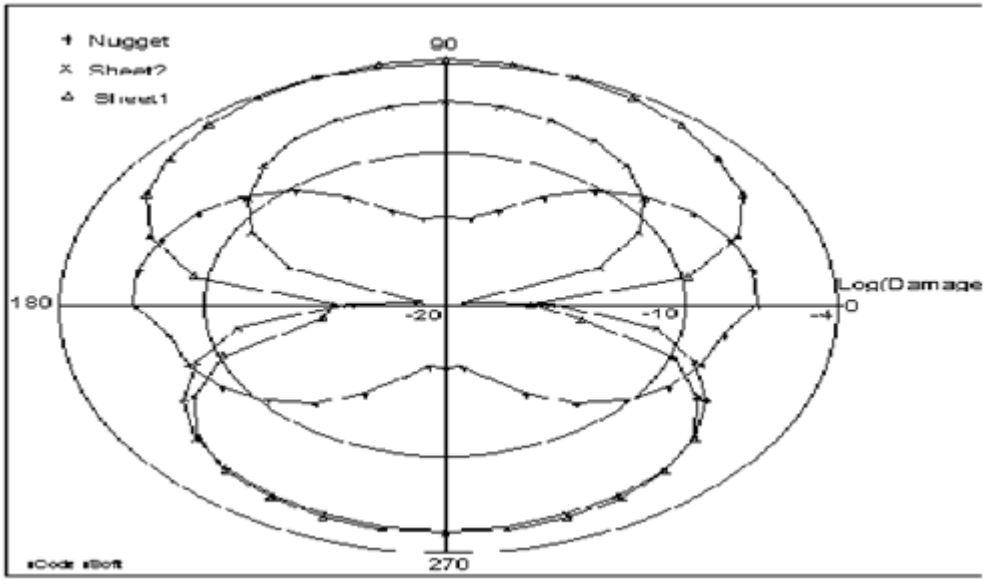


Figure 15-14 Polar Plot of Damage in Most Damaged Spot Weld

T-Beam Specimen

The second example concerns a spot welded T-piece. It is composed of sheets of thickness 1.0 and 0.8 mm. The finite element model of this structure is illustrated in Figure 15-15 below.

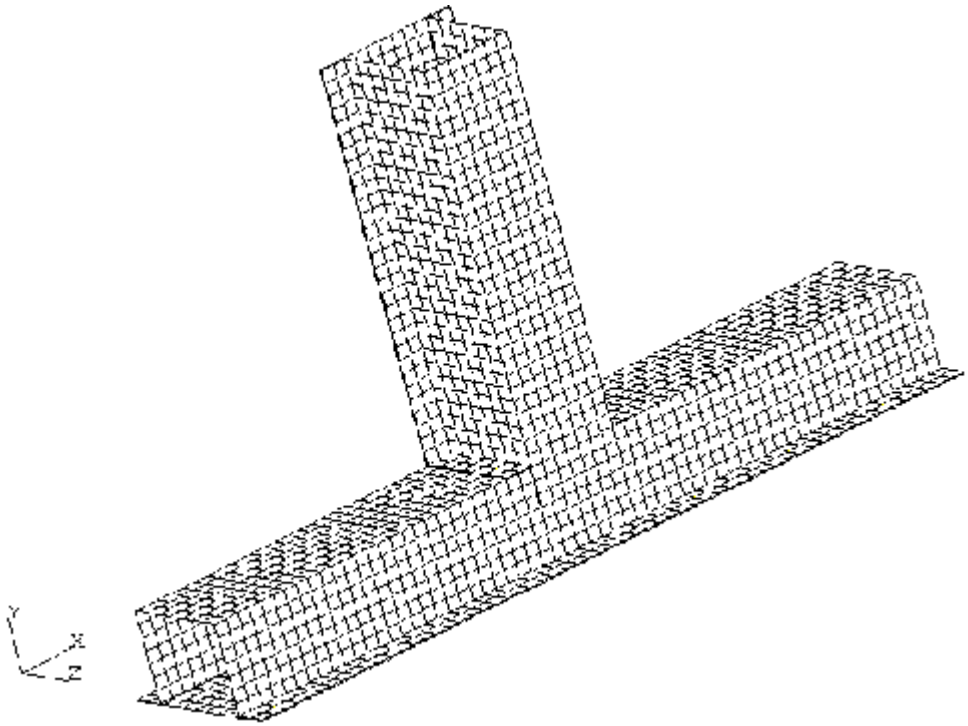


Figure 15-15 Finite Element Model of T-beam Specimen

The T-beam specimen was clamped at the two ends of the cross-piece and could be loaded in either the x or z direction at the end of the vertical. The following cases have been tested and analyzed in the table below, once again using the generic spot-weld S-N curves.

Table 15-13 Measured vs. Predicted Lives

Case	Experimental Life (cycles)	Predicted Life (cycles)
F(x) amplitude 157.5 N R = 0.1	460,000	~500,000
F(z) amplitude 503 N R = -1	70,000	~35,020
F(z) amplitude 400 N R = -1	290,000	~145,500

Step 1: Geometry and Results

The geometry and results are provided in the files called `t_beam_x.op2` and `t_beam_z.op2` which can be found in the examples directory provided with MSC Fatigue.

```
<install_dir>/mscfatigue_files/examples
```

Step 2: Material Characterization

The material is the same as defined in the previous spotweld example.

Step 3: Loading Histories

Define three time histories, one with R=0.1 and the other two with R=-1 with the appropriate amplitudes as in Table 15-13. This is most easily done by creating sine waves under the Wave Creation option with Max/Min values of 350/35, 503/-503, and 400/-400 respectively. Call the load history names "tbeam1," "tbeam2" and "tbeam3" respectively.

Step 4: Setup MSC Fatigue Job

Read in the old fatigue job setup. You will need the file called `tbeam.fin`. We will not actually run the fatigue analysis in this example as it is quite time consuming. Rather we will read in the job setup and discuss it briefly.

Operation	Comments
patran	Invoke Patran (or MSC Fatigue) if you have not already done so.
File / New...	Open a new database from the File pull -down menu. Call it "spotweld." Set the analysis preference to MSC.Nastran if asked. Ignore any warning messages.
Analysis / Import	Open the Analysis application from the main form of Patran or the Import application in MSC Fatigue .
Action: Read Output2 **Object: Both** **Method: Translate**	Set the Action, Object, and Method accordingly.
Select Results File...	Select the result file `t_beam_x.op2`. Press the Apply button to import the model and static results. You can do the same for the other load case `t_beam_z.op2`, except import the results only.
Group / Create...	Create three groups with only the bar elements in them (the spot welds). These groups should contain elements 3080:3095, 6267:6274, and 6237:6266. Call the groups "0.8_to_0.8,", 0.8_to_1.1,", "1.1_to_1.1", respectively.
Tools / FATIGUE... **(Analysis)**	Invoke Patran's FATIGUE interface by selecting it from under the Tools pull-down menu (or the Analysis application switch in MSC Fatigue .
Jobname: tbeam	Enter the jobname "tbeam1" and press the RETURN or ENTER key. This will read in the job parameters. Inspect the job setup forms to see how they are set up.
Job Control...	Go to job control and press Apply with the Action set to Full Analysis and the job will run. You can repeat this and the previous step for jobs "tbeam2" and "tbeam3."

Step 5: Evaluate Results

The results of the fatigue analysis can be read in using the appropriate option from the fatigue Results form as described in the previous spot weld example.

There appears to be reasonable correlation between the tests and the analysis, given the assumptions made. Clearly the predicted load-life curve is somewhat conservative compared to the experimental data, particularly for shorter lives. There are a number of possible reasons for this:

1. There may be a significant difference between the fatigue strength of spot welds in V-1147 and those in St1403 which were used to build the S-N curves used in the calculation.
2. A better spot welding technique may have been used in the specimens.
3. When the S-N data for shear loadings were processed(Ref. 32.), it seems that the effect of axial forces in the bar elements may have been neglected. In practice there is some axial force in shear tests which is quite sensitive to the geometry of the test assembly.
4. Because the forces and moments on each spot weld are based on FE calculations, S-N curves depend not only on the tests carried out to derive them, but also on the FE calculations used to obtain the forces. The same standard of FE modelling must be used in the S-N curve derivation and any subsequent life prediction. In particular it seems that changing the mesh density changes joint stiffness and the relative importance of the forces and bending moments on individual beams. More research is needed to establish best practical FE modelling techniques.
5. The failure criterion used in the test data plotted here was 30% decrease in stiffness. The failure criterion used to derive the basic S-N curve is not known.

Problem 7: Vibration Fatigue Analysis

This section is concerned with the validation of the vibration fatigue methodologies described in Vibration Fatigue (Ch. 9). These techniques presently employ the S-N method of fatigue life estimation as validated in Problem 1: Analysis of a Keyhole Specimen, 1223. Here we are concerned not with the validation of the S-N method, but with the comparison of time domain results with frequency domain results. Three examples are provided ranging from a simple hand calculation, to measured responses, to finite element model frequency versus time domain results.

A Simple Worked Example

These pages describe a simple worked example for a wide band spectrum comprising 1 Hz and 10 Hz frequency components. A predicted fatigue life is hand calculated for both a frequency and a time domain approach, and compared with an equivalent calculation performed using MSC Fatigue tools. This brings together many of the topics discussed in Vibration Fatigue (Ch. 9) and shows how the theory is applied using MSC Fatigue programs.

The worked example is based upon a PSD comprising two spectral components, 10,000 MPa^2/Hz at 1 Hz and 2,500 MPa^2/Hz at 10 Hz, with a 1 Hz spectral bandwidth. The stress life (S-N) material properties used are for a typical steel, of the form,

$$N = KS^b$$

b		4.2
K		1.0 x 1015

And in the form required by MSC Fatigue materials data manager, MDM,

$$N^{-b1} \times S = SRI1$$

b1		-0.238
SRI1		3728 MPa

Hand Calculation - From a PSD in the Frequency Domain

The idealized form of the input PSD is shown in Figure 15-16 below.

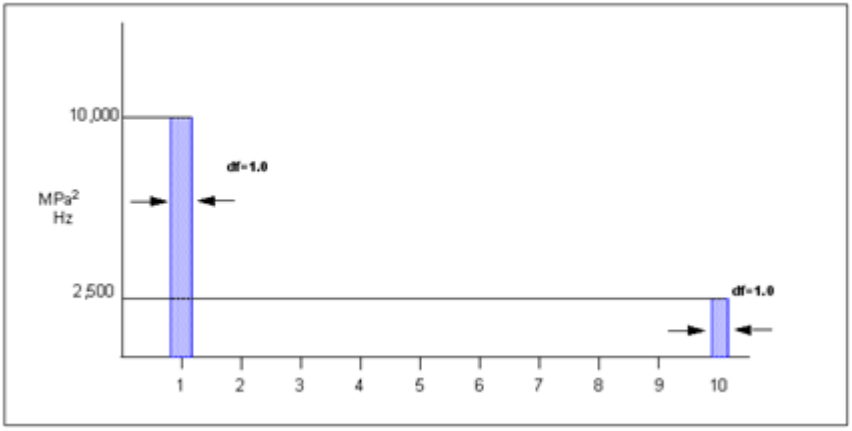


Figure 15-16 An Idealized PSD

In the time domain the PSD shown above is equivalent to a summation of two sine waves:

sine wave 1 at 1 Hz with stress range $\sqrt{10{,}000} \times 1.41 \times 2 = 282$ MPa
sine wave 2 at 10 Hz with stress range $\sqrt{2{,}500} \times 1.41 \times 2 = 141$ MPa

The following hand calculation establishes the necessary moments of this PSD to calculate the narrow band (NB) and Dirlik predicted life.

$$M_n = \sum f^n G(f) \delta f$$

$M_n = 40{,}000 \times 1^n + 10{,}000 \times 10^n$
Mo = 12,500
M1 = 35,000
M2 = 260,000
M4 = 25,010,000

$$E[0] = \left[\frac{M_2}{Mo}\right]^{1/2} = 4.561 \quad \text{Zero crossings per second}$$

$$E[P] = \left[\frac{M_4}{M_2}\right]^{1/2} = 9.807 \quad \text{Peaks per second}$$

$$\gamma = \frac{E[0]}{E[P]} = 0.465 \qquad \text{Irregularity factor}$$

$$\text{RMS} = \sqrt{Mo} = 112 \quad \text{MPa}$$

Equivalent sine wave magnitude = 112 x 1.41 x 2 = 315 MPa

P-M damage $E[D] = \frac{9.8}{3.2\text{E+4}} = 3.1\text{E-4}$

This corresponds to a fatigue life of 3,265 seconds or 0.9 hours

Narrow-Band Computation

$$\text{Damage NB} = \frac{E_p \cdot 1}{A} \cdot (2 \cdot \sqrt{2 \cdot M_0})^m \cdot \Gamma\left(\frac{m}{2} + 1\right) = 6.816 \cdot 10^{-4}$$

Life NB = 1,472 seconds (0.409 hours)

Dirlik Computation

$$Z = \frac{1}{2 \cdot \sqrt{M_0}} = 0.004$$

$$X_m = \frac{M_1}{M_0} \cdot \sqrt{\frac{M_2}{M_4}} = 0.285$$

$$D_1 = \frac{2 \cdot (X_m - \gamma)}{1 + \gamma} = 0.114$$

$$R = \frac{\gamma - X_m - D_1^2}{1 - \gamma - D_1 + D_1^2} = 0.384$$

$$D_2 = \frac{1 - \gamma - D_1 + D_1^2}{1 - R} = 0.704$$

$$D_3 = 1 - D_1 - D_2 = 0.184$$

$$Q = \frac{1.25 \cdot (\gamma - D_3 - D_2 \cdot R)}{D_1} = 0.142$$

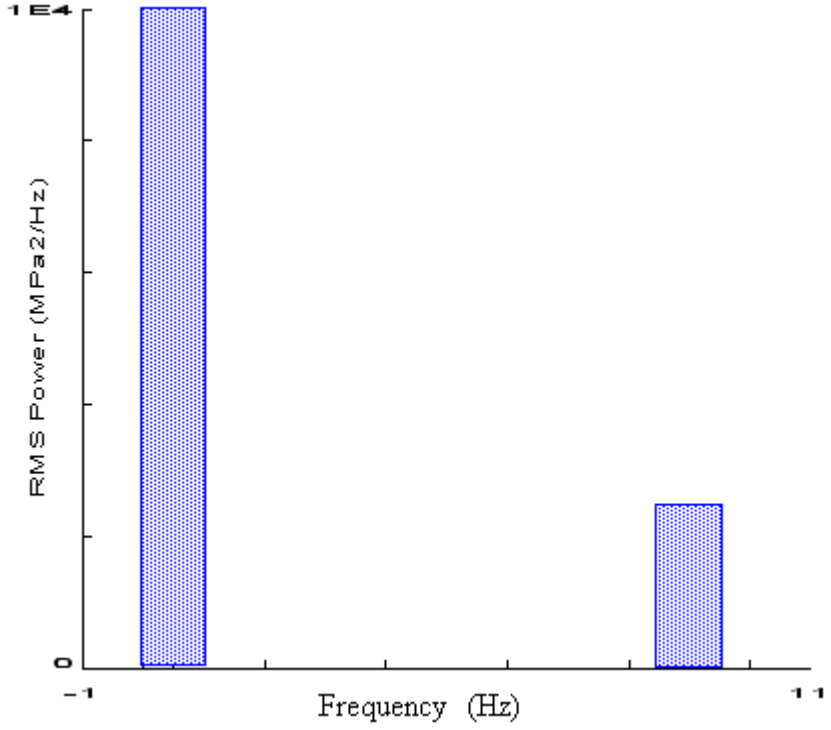
$$f_s(S) = \frac{D_1}{2 \cdot \sqrt{M_0} \cdot Q} \cdot e^{\frac{-Z}{Q} \cdot S} + \frac{D_2 \cdot Z \cdot S}{2 \cdot \sqrt{M_0} \cdot R^2} \cdot e^{\frac{-Z^2}{2 \cdot R^2} \cdot S^2} + \frac{D_3 \cdot Z \cdot S}{2 \cdot \sqrt{M_0}} \cdot e^{\frac{-Z^2}{2} \cdot S^2}$$

Damage Dirlik = $\frac{E_p \cdot 1}{A} \cdot \int_0^\infty S^m \cdot f_S(S) dS = 1.326 \cdot 10^{-4}$

Life Dirlik = 7,650 seconds (2.125 hours)

MSC Fatigue Calculation - From a PSD in the Frequency Domain

The MSC Fatigue module MCOE is used to create the PSD described above. Define the Y-axis label as "RMS Power" with units "MPa^2/Hz". With a sample rate of 1.0, define the value at 1 Hz = 10,000 and 10 Hz = 2,500, all other points should remain zero, and exit when the two values have been defined. Use MQLD in tower display mode to obtain the output shown below.



The steps are outlined in the table below:

Operation	Comments
mcoe	Invoke MCOE from the system prompt.
Create	Select the Create option.
Input Filename: simple.psd **OK/OK**	Enter a file name such as `simple.psd`. Then click the OK button twice to accept it. The file will be created on completion of the exercise in MCOE.
Sample Rate: 1	Enter 1 as the sample rate.
X-axis Label: Frequency	Change the X-axis label to Frequency.
X-axis Units: Hz	Change the X-axis label to Hz.
Y-axis Label: RMS Power	Change the Y-axis label to RMS Power.
Y-axis Units: MPa2/Hz	Change the Y-axis label to MPa^2/Hz.
OK	Press OK. A spreadsheet will appear.
1 Hz: 10000; 10 Hz: 2500	Enter 10,000 in the 1 Hz row and 2,500 in the 10 Hz row for the Y value. Enter zero for all other up to say 11 Hz.
File/OK	Under the File pulldown menu, select OK. This will save the file.
mqld	Invoke MQLD from the system prompt. The default file to display should be simple.psd.
Display | Towers	Change the display to plot as towers as opposed to joined. You should see a plot similar to that in the above figure.
File | Exit	Quit from MQLD.

Now perform the frequency life analysis in MFLF. The file created above in MCOE, is the input filename. The "Cut-off Frequency" will correctly default to 11 Hz, and ensure that the units selected are "MPa^2/Hz". Choose "128" for the number of bins required, this high number will help to reduce rounding errors due to bin widths enabling a more accurate comparison with the above hand calculation. Use the List file browser button to select any material data set and request to edit the material the data. Edit the material data and insert the values defined above, SRI1=3728 and b1 = -0.238. The transition life NC1 can remain at 1E8, and the second slope b2 can remain as is since no input cycles exist below the stress at the 1E8 transition life. The surface finish is polished, and there is no surface treatment.

The steps are as follows:

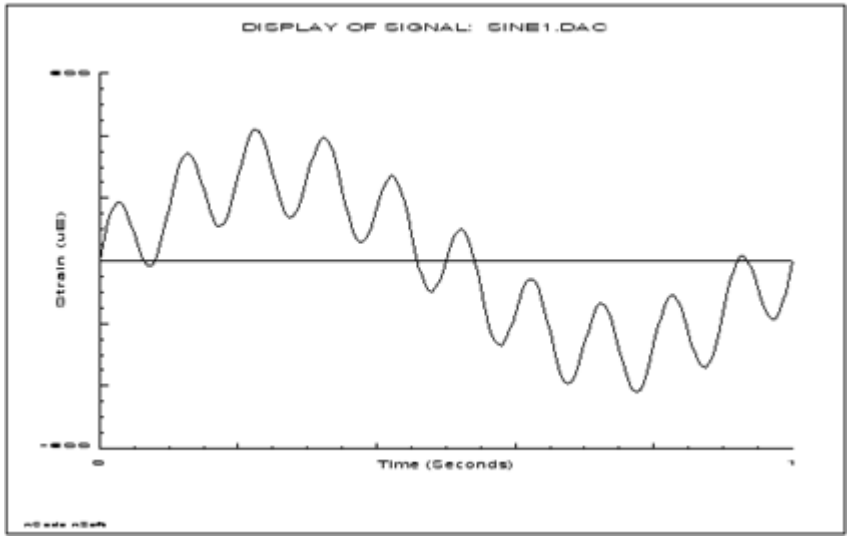
Operation	Comments
mflf	Invoke MFLF from the system prompt.
Input Filename: simple.psd	Select the file just created and click the OK button twice.
Bins Required: 128	Change the bins to 128 and click OK.
Dataset Name:	Press the List button here and select any dataset.
Edit Data: Yes	Turn on the Edit Data switch by setting it to Yes and then press the OK button.
SRI1: 3728	Change Stress Range Intercept to this value.
b1: -0.238	Change the First Slope to this value and then press the OK button twice. The calculation will proceed. Press the End button after the results are displayed.
Analysis method	Press the Analysis method button from the main menu options.
All	Select All as the analysis method. Press OK when the next form is presented. The number of bins should still be set to 128.
Recalculate life	Press the Recalculate button. The results from all analysis types will be presented.
End | eXit	Quit from MFLF when you are ready.

After performing the analyses a results screen is presented with results for each analysis option. The first thing that should be noticed is that the Tunna method predicts a significantly longer life (unconservative) than all the other methods which are within a factor of 4 of each other.

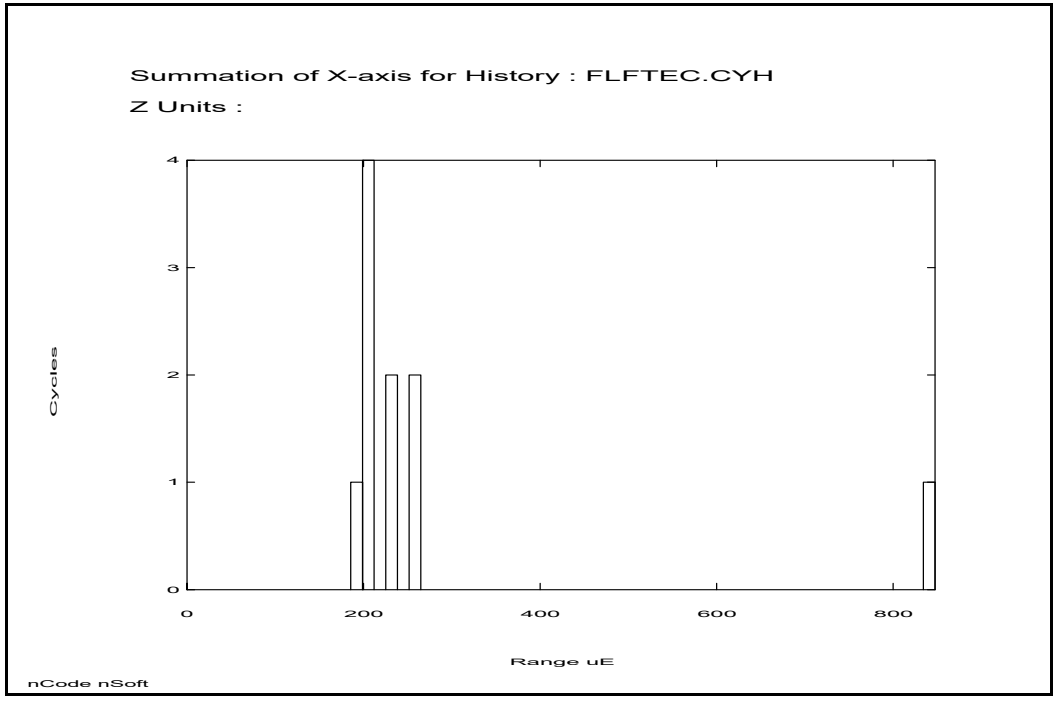
The two important results to note are the Narrow Band = 0.4091 hours, and the Dirlik = 2.104 hours, which compare to 0.409 hours and 2.125 hours respectively from the above hand calculation.

Hand Calculation - From a PSD in the Time Domain

In the time domain, the PSD described above is equivalent to a summation of two sine waves; with stress range 282 MPa at 1 Hz and 141 MPa at 10 Hz. The actual fatigue calculation is done by hand, though MSC Fatigue tools are used to Rainflow count the cycles. With a sample rate of 200 Hz, and a time period of one second, use PTIME to create and sum these sine waves. Use MQLD to obtain the output shown below.



This time series is rainflow counted using PTIME, into 10 cycles, and displayed using MP3D in "view left" mode to obtain the output shown below.



The histogram editor is used to obtain the range and the mean for the 10 cycles in one second of this time series. Using these values in the MSC Fatigue S-N materials equation give the following life values at the various bin center stress levels. No mean stress correction is applied.

N(range,mean)= life
N(129,-134)= 1.375E+6
N(116,-115)= 2.148E+6
N(103,-82)= 3.539E+6
N(103,-43)= 3.539E+6
N(96,-3.3)= 4.757E+6
N(420,3.3)= 9.642E+3
N(103,43)= 3.539E+6
N(103,82)= 3.539E+6
N(116,115)= 2.148E+6
N(129,134)= 1.375E+6

A conventional Palmgren-Miner damage calculation on these cycles gives:

E[D] = 1.074E+4

which corresponds to a fatigue life of 9308 seconds, or 2.59 hours.

MSC Fatigue Calculation - From a PSD in the Time Domain

Use the MSC Fatigue S-N time series fatigue analyzer (MSLF), with the following inputs/parameters:

- input file is the single parameter file created by summing the sine waves
- equivalent units are 0.016667 and hours (to convert results in respect of a one second file to hours)

Define the material information in a similar manner as described above and run the analysis.

The predicted fatigue life should be 2.6 hours.

The steps are as follows:

Operation	Comments
ptime	Invoke PTIME from the system prompt.
Add an Entry: Waveform creation	Select the Waveform creation option.
Filename: sinewaves	Enter a new filename such as sinewave. A file called `sinewaves.dac` will be created.
Description 1	Enter an descriptions you wish.
Number of fatigue equivalent units: 0.0166667	This is the conversion factor for reporting a one second signal in hours.
Fatigue equivalent units: Hours	Change this to hours and click the OK button. Do not worry about the units of the actual time series. MPa will be assumed even though the load type is set to Force and the units to Newtons. These are really just labels in most cases.
Sample Rate: 200	Change the sample rate to 200.
Frequency: 1	Change the frequency to 1.

Operation	Comments
Amplitude: 282	Enter 282 for the amplitude of the first sine wave at 1 Hz and press the OK button.
Summation	Press the Summation button.
Next Frequency: 10	Enter 10 as the next frequency.
Next Amplitude: 141	Enter 141 for the amplitude at 10 Hz. Press the OK button when done.
Finish	Press the Finish button to indicate that no more are to be added.
Plot an entry	Plot the new time history. Select `sinewaves` as the input.
File | Exit | eXit	Exit from MQLD and exit from PTIME.
mslf	Invoke the S-N single shot fatigue analyzer MSLF.
Jobname: test	Enter a jobname. It can be anything.
Filename: sinewaves	Enter the newly created time history as the filename. Press OK twice.
Dataset Name	Again enter any valid S-N dataset name.
Edit Data: Yes	Again turn on the Edit Data switch.
SRI1: 3728	Enter 3728 as the Stress Range Intercept value.
b1: -0.238	Enter this value as the first slope. Press the OK button three times. The results will be displayed.
End | eXit	Quit from MSLF when you are ready.

Measured Responses

In this section we look at actual measured stress responses. From these responses we predict fatigue lives using the S-N method. The responses are then converted to response PSDs of stress with a subsequent vibration fatigue analysis performed using the S-N method. The predicted lives from the time domain and the frequency domain approaches are then compared for the various vibration fatigue methodologies.

Stress Response Descriptions

Three time domain stress responses are used. They are aptly named `1pk.asc`, `2pk.asc` and `3pk.asc` and can be found in your delivery in

```
<install_dir>/mscfatigue_files/examples
```

These signify one peak, two peaks, and three peaks. These names are chosen since if you convert these three time signals to the frequency domain you will see that they have 1, 2, and 3 peaks in the resulting PSDs which is an indication of the type of frequency content they contain. These are stress time histories which could be representative of the stress time histories at the critical location of a FE model.

To import these responses do the following in PTIME from an empty directory:

Operation	Comments
ptime	Invoke PTIME either from the system prompt (or do it from the MSC Fatigue Loading Info form by clicking on the Database Manager button).
ASCII convert + load	Select the "ASCII convert + load" option to read in the ASCII response files.
ASCII Filename	Use the List button to show the available time histories stored as ASCII files. Select `1pk.asc`.
Sample Rate: 50	Enter 50 as the sample rate. This is important that you enter the proper sample rate. Otherwise the subsequent conversion to a PSD will be erroneous.
Equally Spaced Data: X-y pairs	Change this entry to X-y pairs. Press OK.
Description 1	Enter a description such as "1 Peak Stress Responses."
Load Type: Pressure	This is a stress response time history so enter Pressure as the load type. Even though it is not actually a load type, this is the best equivalent.
Units: MPa	Give SAETRN the new name, *SAETRN15*, and also new description. Press OK. The time history SAETRN will be duplicated and will be called SAETRN15.
Fatigue Equivalent Units: Hours	One repeat of the response is equivalent to 1 hour. So enter Hours as the equivalent units.
OK	Press the OK button to start the conversion. This operation could take some time since it is a very large response history.
Plot an entry	Select the Plot option and accept the default.
eXit	Repeat this exercise for 2pk.asc and 3pk.asc if desired. Then exit from PTIME.

Time Domain Solution

The fatigue life from the three time domain signals can be determined quite easily if we select a material - say RQT501 from the MSC Fatigue materials database. To calculate fatigue lives from these responses, we use the MSC Fatigue single location stress fatigue life analyzer MSLF. This module accepts a response history directly as opposed to combining loads and FE stress results to determine the response history.

Operation	Comments
mslf	Invoke MSLF from the system prompt.
Jobname	Enter a jobname, say "1pk" for the first response history. Since it does not exist you will be prompted to create it.
Filename	On the next screen that appears, enter 1pk as the file name. This is the binary converted response time history. Accept all defaults on the rest of this input page.
OK	Accept all defaults on the next page, Model Parameters, also.
Database Name: RQT501	On the Material S-N Analysis form enter RQT501 as the S-N curve. Accept all other defaults.
OK	Accept all defaults of the next page, Geometry.
OK / End / eXit	Accept all defaults on the next page, Results Setup. At this point the analysis will proceed. Permit overwrite If asked for overwrite permission. The analysis results should give approximately 3600 repeats. You can repeat this for 2pk and 3pk or exit from MSLF.

PSD Responses of Stress

The corresponding PSDs of these three signals are shown in Figure 15-17. A single peak is generally called a narrow band signal since it only has one predominate frequency. The other signals are called wide band since they have more frequency content.

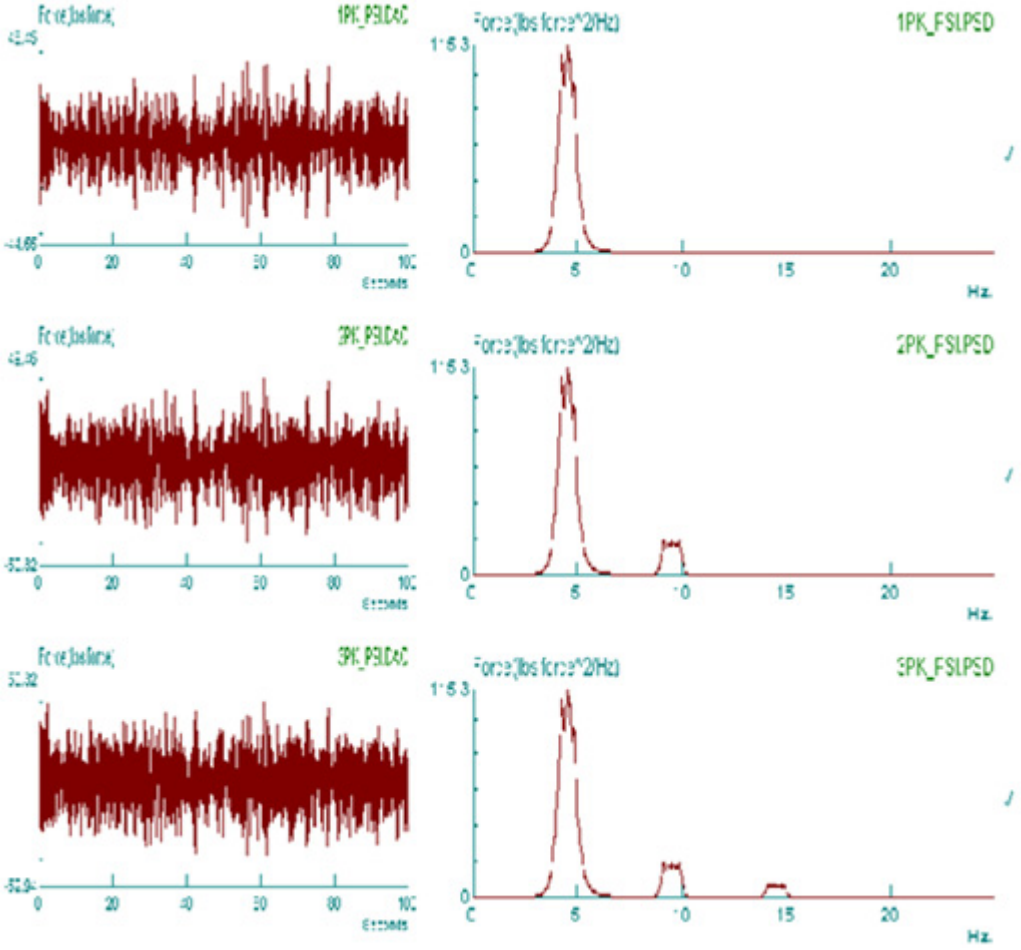


Figure 15-17 Stress Responses and Corresponding PSDs

To create these PSDs of stress from the response time histories, use the MSC Fatigue module MASD. Follow these instructions:

Operation	Comments
masd	Invoke MASD from the system prompt.
Input Filename	Use the List button to select the file `1pk.dac`. Accept all other input on this form by clicking the OK button.
OK	Accept all defaults on the next input screen, Parameter Input.
OK	Accept all defaults on the next input screen, Output Parameters, also.
End	At this point the PSD has been calculated by the program. By pressing the End button the PSD will be plotted.
File/Exit	Quit from the graphics program, MQLD, and repeat for the other responses, 2pk and 3pk, if desired.

Vibration Fatigue Solution

With the vibration fatigue analyzer, the fatigue life from the PSDs can easily be calculated also with a variety of different methods, and very quickly relative to the time domain. Follow these steps to reproduce results:

Operation	Comments
mflf	Invoke MFLF (frequency life fatigue) from the system prompt.
Input Filename	Use the List button to select the file `1pk.psd`. Accept all other input on this form by clicking the OK button.
OK	Accept all defaults on the next input screen, Intermediate Rainflow Options.
Dataset Name: RQT501	On the Material S-N Analysis form enter RQT501 as the S-N curve. Accept all other defaults.
OK	Accept all defaults on the next input form, Geometry.
End	At this point the program proceeds with the calculations. You should see a life of approximately 3800 hours.
eXit	You can quit from MFLF and then repeat the exercise for 2pk and 3pk, if desired.

Comparison of Results, 1pk, 2pk, 3pk

Table 15-14 a chart comparing the 1pk, 2pk, and 3pk with the time domain from all the frequency vibration methods. The Dirlik method is the default and generally gives the best results. The Narrow Band gives good results for the narrow band signal (1pk), but becomes too conservative when the signal is wide band. Tunna breaks down completely for a wide band signal. Wirshing is unconservative and then too conservative, as is Steinberg. Hancock and Kam & Dover do reasonably well but not as well as Dirlik. One of the major problems with most of these methods except Dirlik is that they are modifications of the Narrow Band method. When the signal is wide band, the Narrow Band method tends to turn any signal into a narrow band signal making the resulting fatigue prediction very and sometimes overly conservative.

Also it should be noted that there is nothing magical about the time signals or corresponding PSDs in this example. The only consideration should be when using with other material types that the relative intensity of the signal is compatible with the chosen material and should be scaled accordingly.

Table 15-14 Time vs. Frequency Predicted Lives (hours)

	Time Domain	Dirlik	Narrow Band	Tunna	Wirshing	Hancock	Kam & Dover	Steinberg
1pk	3626	3815	3762	4485	5771	3839	3352	5765
2pk	1549	1360	1015	15637	1695	1303	1367	1648
3pk	1032	1027	570	62440	964	870	1077	936

Comparison of Results, Wind Energy

Another example is given of responses from of a wind turbine blade. In Table 15-15 below the various methods are compared to the time domain by normalizing the frequency domain result by the time domain result. Therefore a value of unity would signify a one-to-one correspondence between frequency and time domain solutions.

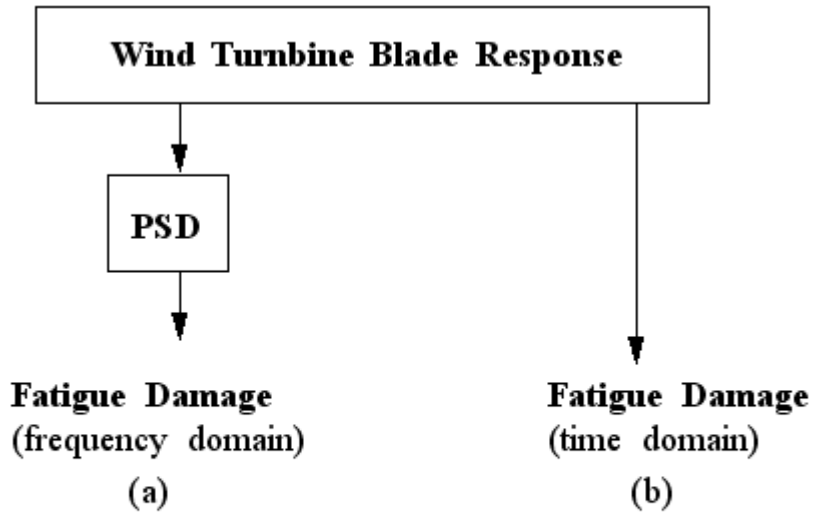


Figure 15-18 Frequency/Time Domain Normalization

Table 15-15 Wind Turbine Response Time vs. Frequency Predicted Lives (hours)

Load Case	Narrow Band	Dirlik	Wirshing	Bishop	Chaudhury	Hancock
y12a	5.14	1.03	3.91	1.52	2.13	2.75
y19a	5.15	1.00	3.92	1.54	2.14	2.77
y27a	14.34	1.59	10.91	1.74	5.12	5.83
y35a	81.87	2.34	62.23	1.95	30.08	25.08
y12b	1.91	0.77	1.46	1.13	0.98	1.25
y19b	1.98	0.81	1.50	1.22	1.04	1.31
y27b	3.67	1.07	2.79	1.29	1.47	1.92
y35b	18.34	1.48	13.95	1.84	5.68	6.10
y12c	1.98	0.76	1.51	0.86	0.95	1.25
y19c	1.87	0.73	1.43	0.86	0.92	1.20
y27c	2.03	0.74	1.54	0.72	0.87	1.14
y35c	3.22	0.76	2.45	0.66	1.15	1.42
y12d	2.09	0.84	1.59	1.15	1.03	1.33

Table 15-15 Wind Turbine Response Time vs. Frequency Predicted Lives (continued) (hours)

Load Case	Narrow Band	Dirlik	Wirshing	Bishop	Chaudhury	Hancock
y19d	2.03	0.83	1.54	1.17	1.02	1.31
y27d	2.92	1.01	2.22	1.15	1.23	1.62
y35d	7.50	1.12	5.70	1.23	2.75	3.29
y12e	2.80	0.99	2.13	1.27	1.50	1.95
y19e	3.06	1.01	2.33	1.27	1.50	1.95
y27e	3.50	1.03	2.67	1.53	1.65	2.16
y35e	8.81	1.11	6.71	1.99	3.31	4.15
y12f	3.86	0.98	2.93	1.43	1.66	2.18
y19f	3.97	1.00	3.02	1.61	1.78	2.33
y27f	3.96	1.01	3.01	1.57	1.76	2.31
y35f	5.59	0.98	4.25	1.65	2.17	2.80
Average:	7.98	1.04	6.08	1.36	3.08	3.32

The following is noted about the results from Table 15-15:

1. The worst load case is out by approximately 81 times for the Narrow Band method whereas the Dirlik method is out by only two times.
2. Dirlik does the best overall with an average of 1.04.
3. The Bishop method is a theoretical based method used to validate the Dirlik method which is an empirically based solution. The reason that Dirlik does better than the theoretical equivalent is that the response signals are assumed to be random, stationary and gaussian in nature for the theoretical solution. Not all these measured signals completely achieve these assumptions making the empirically fit solution give better answers.

Comparison of Results, Sine Waves

This is another simple example that can be reproduced quite easily by following the steps below. The exercise creates a time history by summing sine waves of very nearly the same frequency. We want to create a time history that will give a corresponding PSD with a single peak near 10 Hz. To accomplish this:

Operation	Comments
ptime	Invoke PTIME from the system prompt.
Waveform Creation	Select the Waveform Creation selection under Add an Entry or if no entries currently exist select if from the main menu.
Filename: wave	Give it a name. Call it "wave." Also give it some description.
Load Type: Pressure	Give it a load type of pressure to signify stress.

Operation	Comments
Units: MPa	Give it units of MPa.
Number of Fatigue Equivalent Units: 5.5556e-3	Enter 0.005556 as the number of fatigue equivalent units for a single block of the time history that will be created.
Equivalent Units: Hours	Enter hours as the equivalent units. Twenty seconds of time history will be defined below but we want the life to be reported to us in hours. Press OK when done.
Sample Rate: 200	Enter 200 as the sample rate. This will ensure enough points will be made to define a 10 Hz signal with the proper amplitude.
Total Time of Signal: 20	Enter 20 seconds as the total time of the signal.
Frequency: 10	Enter 10 Hz as the frequency. Press OK and accept all other defaults.
Summation Next Frequency: 9.9	Press the Summation toggle. This will allow you to define another sine wave with a different frequency to be added to the one just created. Enter 9.9 Hz, press OK accepting the defaults.
Summation Next Frequency: 10.1	Do it again with 10.1 Hz as the next frequency.
Finish	Press the Finish toggle to end.
Plot an Entry	You may plot it to see a varying sine wave very similar to a narrow band signal.
Change an Entry	Now we will scale the load up to some realistic stress level. Since we are using MPa as the stress units and with the material to be used, RQT501, a scale factor of 50 will do the trick.
Filename	Accept “wave” as the file name and overwrite it.
New Value =	For the multiplication value enter 50. This is the second entry. Leave the others at zero. Press OK twice to accept the new scale factor. Note the RMS of the signal is about if you plot it again 61.23.
eXit	Exit from PTIME.

Now convert the time history to a PSD using the MASD program.

Operation	Comments
masd	Invoke MASD from the system prompt.
Filename	Enter wave as the input file. Press OK four times accepting the default each time. The resulting PSD will be plotted after accepting the result page. Note that the RMS is around 55, very close to the original time signal.
File/Exit	Exit from MASD.

Now run the single location stress-life fatigue analyzer MSLF using the material RQT501 and the input file `wave.dac`.

Operation	Comments
mslf	Invoke MSLF from the system prompt.
Filename: wave	Enter "wave" as the input file. Press OK twice accepting the default each time.
Dataset Name: RQT501	Enter RQT501 as the material on this form. Press OK three times and allow overwrite. The result should be calculated. Note that it is a very large number (~2E8 Hours).
End	Leave MSLF running for now and move on to the next step.

Now run the single location frequency-life fatigue analyzer MFLF using the material RQT501 and the input file `wave.psd`.

Operation	Comments
mflf	Invoke MFLF from the system prompt.
Filename: wave	Enter "wave" as the input file. Press OK twice accepting the default each time. Ignore any warning message.
Dataset Name: RQT501	Enter RQT501 as the material name. Press the OK button accepting the defaults until the analysis runs. Note the life is close to 6 years.
End	Leave MFLF running for now and move on to the next step.

The comparison between the time domain and frequency domain results in this case does not look good (1.25E6 Hours or 142 years versus 6 years). This was done on purpose to illustrate a point. The problem is in the material data. For the time domain solution the number of cycles has become so high that we have gone right off the curve or the valid portion of the curve. In other words the stress levels we are dealing with are not realistic for the material chosen. We are past the endurance limit. Valid regions on the S-N curve for this type of analysis range between 10^3 and 10^7. To verify this go back to both MSLF and MFLF and scale up the loads as follows:

Operation	Comments
mflf or mslf	Both these programs should still be running.
Loading Environment	Enter the Loading Environment menu.
Scale Factor	Change the scale factor to 2 and accept all other defaults.
Recalculate	Press the recalculate button.
eXit	Leave MSLF/MFLF or repeat this exercise for the table of scale factors in Table 15-16.

Note that the lives are beginning to converge. Repeat the above process for each scale factor if desired.

Table 15-16 Time vs. Frequency Lives (hours)

Scale Factor	Time Domain	Dirlik	Narrow Band
1	1.25E6	51,290	67,381
2	20	13	13
2.25	5.4	3.5	3.4
2.5	1.6	1.06	1.04
2.75	2025 Seconds	1303 Seconds	1279 Seconds
3.0	762 Seconds	490 Seconds	1.7 Seconds
3.1	451 Seconds	339 Seconds	1.7 Seconds
3.2	221 Seconds	237 Seconds	1.7 Seconds

You may find it interesting to compare the other frequency domain solution types also. This signal composed of three similar sine waves summed together does not satisfy the requirement that the signal be random. Even without fully satisfying this, the Dirlik method still gives excellent correlation for realistic stress ranges showing the robustness of the technique. As a final verification the cycle histogram plots are shown for the time and frequency domain solutions in Figure 15-19 and Figure 15-20 below. Note that the maximum stress range between the two is very similar (~1000 MPa) for the load case with the highest scale factor (3.2).

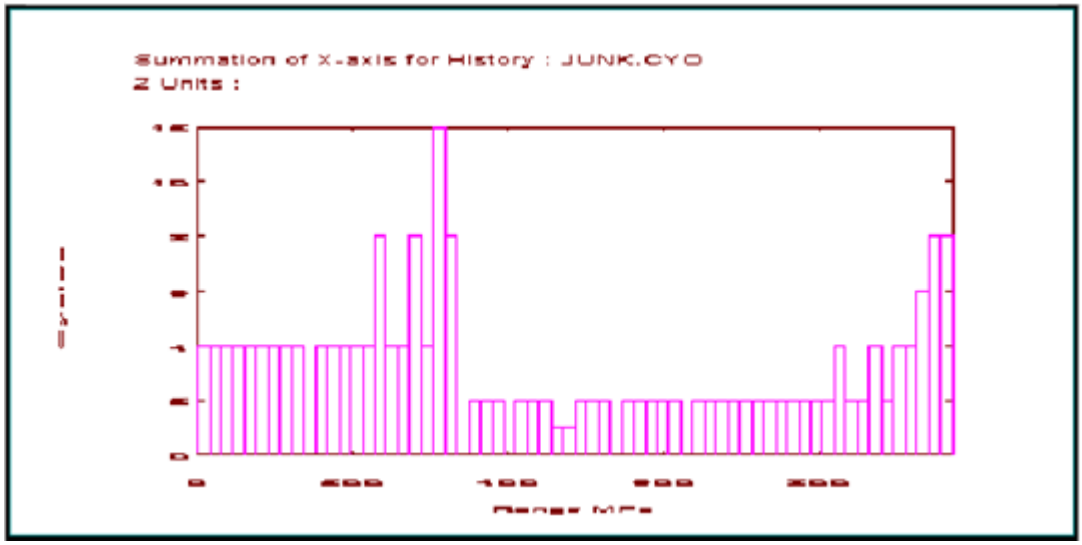


Figure 15-19 Cycle Histogram Results of Time Domain

Just as a note of point, this exercise would not work with a pure sine wave. This is because of the fact that any PSD analysis must realistically use a random signal. A pure sine wave is not a random signal. As an exercise you could perform the same operations as above but on a pure sine wave at 10 Hz. When you compare the resulting cycle histogram plots you will note that the stress ranges predicted by the PSD solution are twice as large as they actually are from the time domain solution.

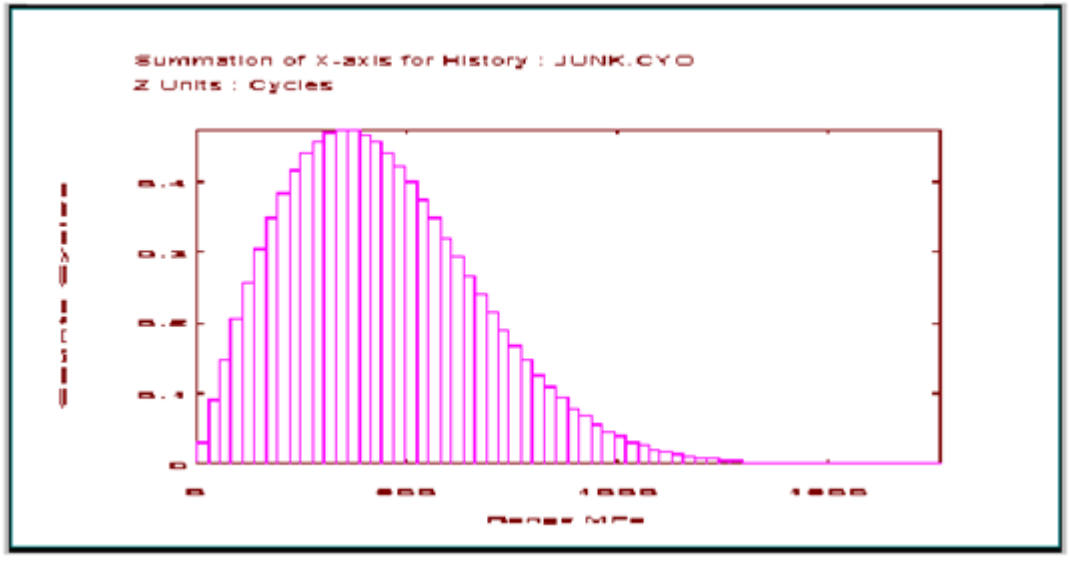


Figure 15-20 Cycle Histogram Results of Frequency Domain

Finite Element Model Responses

Now we compare fatigue life predictions in a similar manner as done in the previous section but the responses are determined from finite element modeling. Again the time domain responses and fatigue predictions will be compared to the frequency domain responses and life predictions.

A simple model will be used first in a quasi-static method by running a static analysis of a single load case and associate it's time variation via an external time history. The same model will then be run through a frequency response analysis and the time signal used in the quasi-static method will be converted into an input load PSD that will be applied to the same location of the model. The fatigue predictions from each will then be compared.

The model is a simple bracket shown in Figure 15-21 subject to random excitation in the direction of the applied loading. MSC.Nastran is used as the solver and two FE analyses are performed. For the static analysis a unit load is applied at the indicated location and direction. The bracket is fixed at the circular hole. A frequency response analysis is also performed with a unit load at the same location and subject to a frequency range from around zero to 25 Hz.

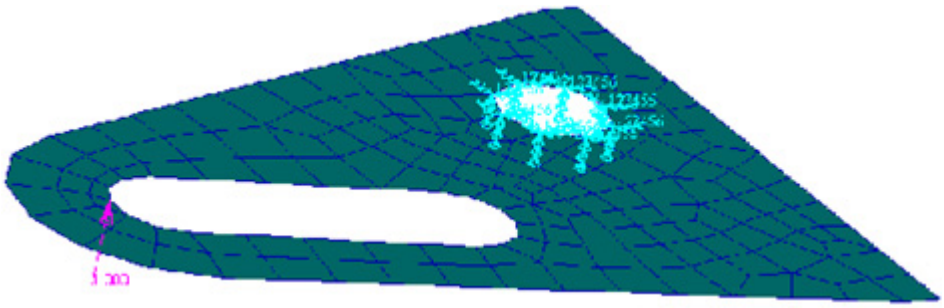


Figure 15-21 A Simple Bracket Subject to Random Excitation

Step 1 - Quasi-Static Model

The first step is to perform a fatigue analysis using the quasi-static S-N method to establish a baseline. The excitation is provided by the 1pk response previously used in this section, except it is scaled and the units are modified. It is assumed that this response signal is available. The following operation modifies the signal:

Operation	Comments
ptime	Invoke PTIME from the system prompt or from Patran. It is assumed that the 1PK, 2PK, and 3PK signals are loaded into PTIME already from the previous exercise. If not, load them now or import the ASCII versions as described earlier.
Change and entry | Polynomial transform	Select this option from the main menu.
Database Entry to Transform:1PK	Select 1PK as the signal to modify.

Operation	Comments
Target Filename: 1PK_PSI	Enter a new name such as 1PK_PSI. We are going to change the descriptive units to PSI since they are the units of the FE model.
New Value = 0.15 x <old value>	Enter 0.15 in the second databox to act as a multiplier and scale loading history. Press OK.
Units: lbs force	Change the units to lbs force.
Number of fatigue equivalent units: 1.3889	Change this to 1.3889 which is how many hours are in 5000 seconds, the length of the signal.
Fatigue equivalent units: Hours	Change this to report the life in hours. Press the OK button.
eXit	Exit from PTIME.

Now set the fatigue analysis up using either Patran or MSC Fatigue .

Operation	Comments
patran	Invoke Patran (or MSC Fatigue) if you have not already done so.
File / New...	Open a new database from the File pull -down menu. Call it "vibration." Set the analysis preference to MSC.Nastran if asked. Ignore any warning messages.
Analysis	Open the Analysis application (or Import in MSC Fatigue) from the main form of Patran.
Action: Read Output2 Object: Both Method: Translate	Set the Action, Object, and Method accordingly.
Select Results File...	Select the MSC.Nastran OUTPUT2 file to import for the static analysis. It is called bracket_stat.op2.
Apply	Press the Apply button to import the model into the database.
Tools / FATIGUE... **(Analysis)**	Invoke Patran's FATIGUE interface by selecting it from under the Tools pull-down menu (or the Analysis application switch in MSC Fatigue .)
General Setup Parameters:	
Analysis: S-N	Set the analysis type to S-N on the main form.
Res. Units: PSI	Set the units to PSI.
Jobname: static	Give the job a name. Use "static."
Title: Static Example	Give the job a title.
Solution Parameters Form:	
Mean Stress Correction: None	Set the parameter to None. The mean of the random response loading is essential zero, so no mean stress correction is necessary.

Operation	Comments
Materials Information Form:	
Material: MANTEN	Place the cursor in the cell under the word Material and click on the mouse. A listbox will appear. Select the material MANTEN from this listbox.
Finish: Polished	Select Polished from the option menu that appears. The word polished appears in the Finish cell. The SAE specimen was a polished specimen with no surface treatment.
Treatment: No Treatment	Select No Treatment from the option menu that appears.
Region: default_group	Select the default group as the region. This contains the node we created at the beginning of this exercise. The Materials Information form can be closed down now by clicking the OK button. You may wish to create a group with only Node 47 in it to expedite the analysis since this is the critical node which we will be looking at in more detail later. If you do, reference this group here.
Loading Information Form:	
Results Transformation: Transform to Basic	Set the transformation to Basic so that proper averaging of the element nodal stress results occurs.
Load Case ID	Activate this cell to make the widgets at the bottom of the form appear.
Get/Filter Results	Open this form to get the results.
Select All Results Cases	The easiest way to get the results is to press this toggle and then press the Apply button.
Select a Results Load Case:	Select the only Results Case that appears in this list box.
Select a Stress Tensor:	Select the result that says “Stress Tensor.”
Fill Cell	Press this button to fill the spreadsheet cell with the Results Case ID.
Time History: 1PK_PSI	Select 1PK_PSI which we created earlier.
Load Magnitude: 1.0	Accept the default here by pressing the RETURN key.
Job Control Form:	
Full Analysis	Set the action to Full Analysis and press the Apply button. This could take some time if you analyzer the entire model.
Monitor	Monitor the job if you wish.

Step 2 - Vibration Model

The second step is to perform a fatigue analysis using the vibration capability and then compare against the baseline. The excitation is provided by the 1pk response previously used in this section, except it is converted to a loading input PSD. The following operation modifies the signal:

Operation	Comments
ptime	Invoke PTIME from the system prompt.
Add an entry... | creaTe psd from time	Select this option from the main menu. This will invoke a separate module called MASD for auto-spectral density calculations.
Input Filename: 1PK_PSI	When MASD starts, select 1pk_psi as the input filename.
OK/OK/OK/End	Press the OK button three times to invoke the action. The time signal has now been converted into a loading input PSD. Press the End button to continue.
Description 1	Enter any description here.
Units: lbs force	Change the units. The rest of the form does not need be modified., so click the OK button. A new entry and file called `1pk_psi.psd` exists.
eXit	Exit from PTIME.

Now set the fatigue analysis up using either Patran or MSC Fatigue . It is assumed that Patran is still running and that the database is still opened.

Operation	Comments
patran	Invoke Patran (or MSC Fatigue) if you have not already done so.
File / Open...	Open the old database from the File pull -down menu if necessary.
Analysis	Open the Analysis application (or Import in MSC Fatigue) from the main form of Patran.
Action: Read Output2 **Object: Results Entities** **Method: Translate**	Set the Action, Object, and Method accordingly.
Select Results File...	Select the MSC.Nastran OUTPUT2 file to import for the static analysis. It is called `bracket_freq.op2`.
Apply	Press the Apply button to import the model into the database.
Tools / FATIGUE... **(Analysis)**	Invoke Patran's FATIGUE interface by selecting it from under the Tools pull-down menu (or the Analysis application switch in MSC Fatigue .)
General Setup Parameters:	
Analysis: Vibration	Set the analysis type to Vibration on the main form.
Res. Units: PSI	Set the units to PSI.
Jobname: vibration	Give the job a name. Use "vibration."
Title: Vibration Example	Give the job a title.

Operation	Comments
Solution Parameters Form:	
Mean Stress Correction: None	Set the parameter to None.
Materials Information Form: This should be setup identically to the pseudo-static job.	
Loading Information Form:	
Results Transformation: Transform to Basic	Set the transformation to Basic so that proper averaging of the element nodal stress results occurs.
Frequency Resp	Activate this cell to make the widgets at the bottom of the form appear.
Get/Filter Results...	Open this form from the Loading Information form. Select the only Result Case that appears in this top of this form. It may already be selected for you (LOAD_CASE, 58 subcases).
Filter Method:Global Variable - Variable:Frequency	Turn this option menu to Global Variable and set the Variable to Frequency. This may already be the default settings.
Filter/Add/Close	Press the Filter button. This will filter only subcases with frequency step information from the Result Case. Remember there is a static load case in these also. Press the Apply button. This will load an abbreviation of the frequency steps into the listbox on the Loading Information form which will be used in the analysis. Close the form down when done.
Select a Results Load Case:	Select the only Results Case abbreviation that appears in this list box.
Select a Stress Tensor:	Select the result that says “Stress Tensor.”
Fill Cell	Press this button to fill the spreadsheet cell with the Results Case IDs.
Time History: 1PK_PSI.PSD	Select 1PK_PSI.PSD which we created earlier.
Job Control Form:	
Full Analysis	Set the action to Full Analysis and press the Apply button. This could take some time if you analyzer the entire model.
Monitor	Monitor the job if you wish.

Comparison of FE Fatigue Results

Use PFPOST to view the results from the two analyses:

Operation	Comments
Results...	Open the Results... form from the main MSC Fatigue form.
List Results	Set the Action to List Results and click the Apply button. PFPOST will be invoked.
OK/OK	Press the OK button twice accepting the defaults.
Most damaged nodes	Select this option to display a list of the most damaged nodes. Take note of the life at node 47.
Cancel/Cancel	Press the Cancel button twice.
Jobname: static	Enter "static" as another jobname to view the results from the quasi-static fatigue analysis.
OK/OK	Press OK twice.
Most damaged nodes	Again note the life at node 47.
Cancel	Close the report down.
eXit	Exit from PFPOST.

The table below (Table 15-17) shows results from the two analyses. In all cases only the Dirlik method is compared to the time domain. Note that only the 1pk response was analyzed in this example, but the 2pk and 3pk are also reported. You can repeat this exercise using the 2pk and 3pk signals if you wish. Also reported in the table are the results using different frequency resolutions. This example corresponds to the 57f job which signifies 57 frequencies were analyzed in the frequency response analysis. Note that as the resolution of the frequency steps becomes greater, the answers tend to converge. The MSC.Nastran input deck (`bracket_freq.dat`) is provided if you wish to run the analysis with different frequency steps to reproduce the table below.

Table 15-17 Time vs. Frequency Lives

Job	1pk	2pk	3pk
static	295 Hrs.	136 Hrs.	97 Hrs.
vibration (15f)	450 Hrs.	150 Hrs.	111 Hrs.
vibration (29f)	261 Hrs.	99 Hrs.	76 Hrs.
vibration (57f)	234 Hrs.	94 Hrs.	71 Hrs.
vibration (141f)	229 Hrs.	90 Hrs.	68 Hrs.
vibration (ALLf)	241 Hrs.	94 Hrs.	71 Hrs.

The job ALLf was run with identical frequency content as the loading input PSDs, 1pk, 2pk, and 3pk. The following notes are made about the results:

1. The results are always conservative relative to the time domain but usually with in a factor of two or less.

2. Only if the frequency resolution of the transfer function around the excitation frequencies of the loading input PSD or the natural frequencies of the model (if they are excited) is not sufficient do answers tend to be non-conservative.
3. This particular example is stiffness dominated since the excitation frequencies are well below the natural frequencies of the model. When this is the case. This gives the same structural response as a static analysis at all the analyzed frequencies in this example. For this reason you would expect to achieve the same or close to the same fatigue results as the pseudo-static method in the time domain.
4. Very high cycle tends to be misleading because of large scatter in data. The table below shows this behavior. For lower cycle fatigue the answers between frequency and time domain do tend to converge. Although the validity of the S-N method then comes into question for very low cycle problems.

Table 15-18 Time vs. Frequency Lives (hours) of 1pk Response

Scale Factor	Time Domain	Dirlik
0.5	1,290,000	518,767
0.6	78,900	57,728
0.7	13,977	10,468
0.8	3,130	2,505
0.9	906	719
1.0	295	236
1.1	108	87
1.2	43	35
1.3	15	15
1.4	3.1	6.8
1.5	0	3.3

16 Fatigue Theory

- Introduction 1340
- Total Life (S-N) Analysis 1351
- Crack Initiation/Strain-Life (e-N) Analysis 1384
- The Statistical Nature of Fatigue 1421
- Estimating Material Cyclic Properties From UTS & E 1432

Introduction

Fatigue theory for basic stress--life (S-N) and strain-life (ε -N) analyses are presented in this chapter. This corresponds to the technical background for Total Life and Crack Initiation (Ch. 6). Theory and technical information for other analysis types are given in their respective chapters. However a good understanding of the other analysis types is dependent on an understanding of the basic theories presented here. For this reason, an entire chapter is dedicated to this theoretical discussion.

Background

Purely static loading is rarely observed in modern engineering components or structures. By far, the majority of structures involve parts subjected to fluctuating or cyclic loads. For this reason, design analysts must address themselves to the implications of repeated loads, fluctuating loads, and rapidly applied loads. Such loading induces fluctuating or cyclic stresses that often result in failure of the structure by fatigue. Indeed, it is often said that from 80% to 95% of all structural failures occur through a fatigue mechanism.

It is worth noting at the outset that the term fatigue, coined more than a hundred years ago by the French engineer Monsieur Poncelet, may not be the best choice of terminology today, since many aspects of the phenomenon are distinctly different from the biological counterpart. For example, it is next to impossible to detect any progressive changes in material behavior during the fatigue process, and therefore failures often occur without warning. Also, periods of rest with the fatigue stress removed do not lead to any measurable healing or recovery of the material. Thus, the damage done during the fatigue process is cumulative, and generally unrecoverable. From this standpoint, the German term *Betriebsfestigkeit* (operational strength) is a better descriptor of the phenomenon. However, since Betriebsfestigkeit involves 17 characters, and fatigue only 7, we shall continue to use the term fatigue!

Fatigue, although a complex subject, has not been neglected by the research community. Estimates indicate that if one wished to keep up to date with all the literature published about fatigue by reading a paper each working day, one would fall behind by more than a year for each year of reading. Furthermore, attempting to catch up with the backlog would be virtually impossible. Yet the design analyst and engineer is increasingly challenged by the demands of higher performance, lower weight, and longer life, and all this at a reasonable cost and in as short a time as possible! These apparently conflicting demands can only be overcome through a consideration of the problems associated with fatigue resistant designs. Up until recently, these problems were summarized as the following:

- Life calculations are usually less accurate than strength calculations. Order of magnitude errors in life estimates are not unusual.
- Fatigue properties cannot be accurately deduced from other mechanical properties; they need to be measured directly.
- Full-scale prototype testing is usually necessary to assure an acceptable life.
- Laboratory results of tests carried out under identical conditions may differ widely, requiring statistical interpretation.
- Materials and design geometries must often be selected to provide slow crack growth, and if possible, detection of cracks before they become dangerous.

- "Fail-safe" design concepts must often be implemented in order to achieve acceptable reliability. That is, even if a structural element fails, the structure must remain intact and able to support the loads in the short term.

Modern advances in fatigue life estimation techniques have, to some extent, mitigated these problems. For example, these days it is usual to consider life estimates to be within a factor of two or three rather than ten. Furthermore, computerized analysis of thousands of laboratory data sets do point to acceptable empirical correlations between monotonic tensile data and fatigue parameters.

The History of Fatigue

For centuries, it has been known that a piece wood or metal can be made to break by repeatedly bending it back and forth with a large amplitude. However, it came as something of a surprise when it was discovered that repeated loading produced fracture even when the stress amplitude was apparently well below the elastic limit of the material. The first fatigue investigations seemed to have been reported by a German mining engineer, W. A. S. Albert, who in 1829, performed some repeated loading tests on iron chain. Some of the earliest fatigue failures in service, occurred in the axles of stage coaches. When railway systems began to develop rapidly in the middle of the nineteenth century, fatigue failures of railway axles became a widespread problem that began to draw attention to cyclic loading effects. This was the first time that many similar components had been subjected to millions of cycles at stress levels well below the monotonic tensile yield stress. As is often the case with unexplained service failures, attempts were made to reproduce the failures in the laboratory. Between 1852 and 1870, the German railway engineer August Wöhler setup and conducted the first systematic fatigue investigation. From this point of view, he may be regarded as the grandfather of modern fatigue thinking. He conducted tests on full-scale railway axles and also on small scale bending, torsion, and axial cyclic loading specimens for different materials. Some of Wöhler's data for Krupp axle steel were plotted in terms of nominal stress amplitude versus cycles to failure. This presentation of fatigue life has become very well known as the S-N diagram. Each curve on such a diagram is still referred to as a Wöhler line.

At about the same time, other engineers began to concern themselves with the problems associated with fluctuating loads in bridges, marine equipment, and power generation machines. By 1900, over 80 papers had been published on the subject of fatigue failures. During the first part of the twentieth century, more effort was placed on understanding the mechanisms of the fatigue process rather than just observing its results. This activity finally led, in the late 1950s and early 1960s, to the development of two approaches to fatigue life estimation. One method, known as the Manson-Coffin local strain approach, attempts to describe and predict crack initiation whilst another is based on linear elastic fracture mechanics, LEFM, and was developed to explain crack growth. Most recently, Miller and his colleagues at Sheffield University, England, have been working on ways of finding a unified theory of metal fatigue, describing crack growth on a microscopic, macroscopic, and structural level.

From this vast wealth of knowledge, one thing has become clear; modern design analysts and engineers will not create more fatigue resistant components and structures by indulging in more experimentation, although the need for more research is ever present. From a practical point of view, a more profitable approach is the implementation and efficient use of the knowledge which is available today.

High Cycle versus Low Cycle Fatigue

Over the years, fatigue failure investigations have led to the observation that the fatigue process actually embraces two domains of cyclic stressing or straining that are distinctly different in character. In each of these domains, failure occurs by apparently different physical mechanisms: one where significant plastic straining occurs and the other where stresses and strains are largely confined to the elastic region. The first domain involves some large cycles, relatively short lives and is usually referred to as l*ow-cycle fatigue*. The other domain is associated with low loads and long lives and is commonly referred to as *high-cycle fatigue*. Low-cycle fatigue is typically associated with fatigue lives between about 10 to 100,000 cycles and high-cycle fatigue with lives greater than 100,000 cycles.

The rules for distinguishing between these two domains are discussed later. For now, it is enough to recognize that remedies for extending fatigue lives in each domain are different. In the high-cycle fatigue domain, measures such as shot peening and other surface hardening treatments or the use of higher strength materials are beneficial. For low-cycle fatigue, where ductility and resistance to plastic flow are important, these measures are inappropriate.

Summary

In summary, fatigue analysis may be thought of as a process of initiating and then growing a crack which finally causes the structure to break into two or more pieces. This process can be represented by the following equation:

Total Life = Crack Initiation + Crack Growth

or

$$N_f = N_i + N_p \tag{16-1}$$

N_f		Total fatigue cycles to failure
N_i		Number of cycles to initiate an engineering crack
N_p		Number of cycles to propagate a crack to final fracture

The mathematical models used to simulate the initiation and propagation processes are quite different. The initiation phase is usually modeled using strain-life and cyclic stress-strain curves while the propagation phase uses crack growth rate versus stress intensity curves.

The original work on fatigue life estimation did not make a distinction between crack initiation and growth. Stress was used as the control parameter and stress-life (or S-N) curves characterized the material or component response to cyclic loading. This approach was used widely until the 1970s when the new methods started to be used by industry. Unfortunately, there are quite a few inherent problems with the stress-life approach which cause a large amount of scatter in the experimental fatigue life results. In some companies, due to the extensive use of the stress-life method in past years, a large quantity of S-N curves and experience have been accumulated. In this case, the stress-life method offers compatibility with previous work and may therefore be more appropriate.

The stress-life approach is still used today for situations which are not easily modeled using the local strain methods, such as welded structures and cast irons. Due to the inherent defects in these materials which act more like cracks, fatigue damage should really be modeled with LEFM crack propagation tools. Another group of materials which appear to be best modeled using the stress-life approach are anisotropic and inhomogeneous materials such as composites. However, there are complications even with composites, since the mean stress correction models used in connection with metals do not apply to all composites.

Inputs to Fatigue Life Estimation Models

The fatigue life estimation process requires three main inputs and the process is usually illustrated with a diagram known as the "Five Box Trick" as shown in Figure 16-1.

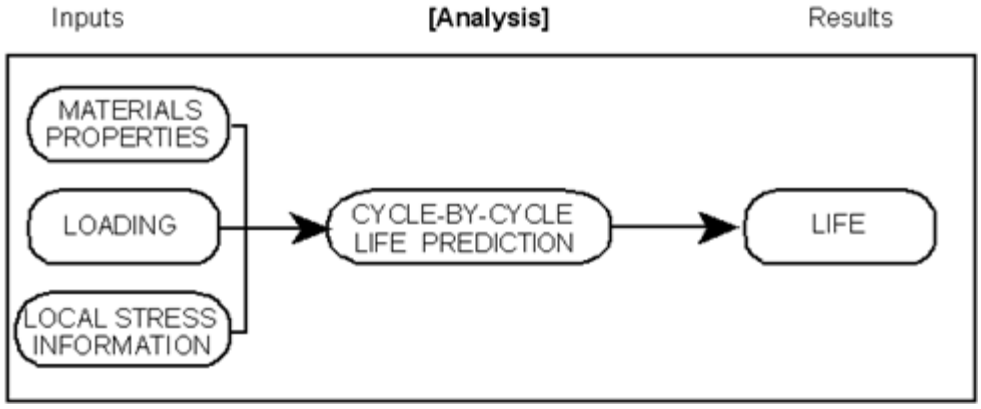


Figure 16-1 Schematic Illustration of Fatigue Life Estimation

From this diagram, it is easy to see the necessary inputs:

- Materials properties
- Loading in the form of load time histories
- Local stress-strain information from a linear elastic analysis

These inputs are then processed using various fatigue life estimation tools which are described in more detail below. It is important to understand the nature of these inputs so that the fatigue life estimation is meaningful.

Material Properties

When considering fatigue, it is not sufficient to characterize a material purely in terms of the Young's modulus and Poisson's ratio. The chemical composition, heat treatment, and microstructure will all change the way in which the material responds to cyclic loading. A summary of the materials data types for fatigue life estimation is given below.

Stress-Life Data

For this model, the fatigue response for a material used in a particular construction is the stress amplitude versus log cycles to failure curve. In addition, the Young's modulus may be required to convert stress to strain or vice versa. The S-N curve parameters are listed in Table 16-1 and typical S-N curve is shown in Figure 16-2.

Table 16-1 Stress-Life Data Parameter Description

Parameter Name	S.J. Units	Imperial Units	Parameter
Stress range intercept	MPa or MN/m2	PSI	SRI1
First fatigue strength exponent	none	none	b1
Fatigue transition point	cycles	none	NC1
Second fatigue strength exponent	none	none	b2
Stress range fatigue limit	MPa or MN/m2	PSI	FL
Standard error of Log(N)	none	none	SE

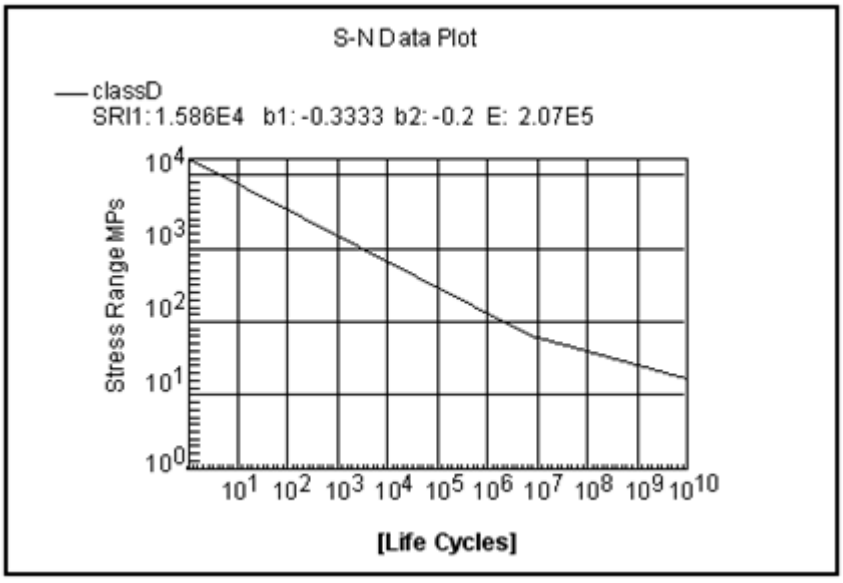


Figure 16-2 Example of a Stress-Life Plot

Crack Initiation Data

There are two basic sets of materials properties used for this fatigue life estimation model. They are the cyclic stress-strain curve and the strain-life curve. These are characterized by the parameters in Table 16-2 with a typical strain-life plot shown in Figure 16-3.

Table 16-2 Local Strain Data Parameter Description

Parameter Name	S.J. Units	Imperial Units	Parameter
Fatigue strength coefficient	MPa or MN/m2	PSI	σf'
Fatigue strength exponent	none	none	b
Fatigue ductility coefficient	none	none	εf'
Fatigue ductility exponent	none	none	c
Young's modulus	MPa or MN/m2	PSI	E
Cyclic strain hardening exponent	none	none	n'
Cyclic strain coefficient	MPa or MN/m2	PSI	K'
Cut off in reversals	none	none	Rc

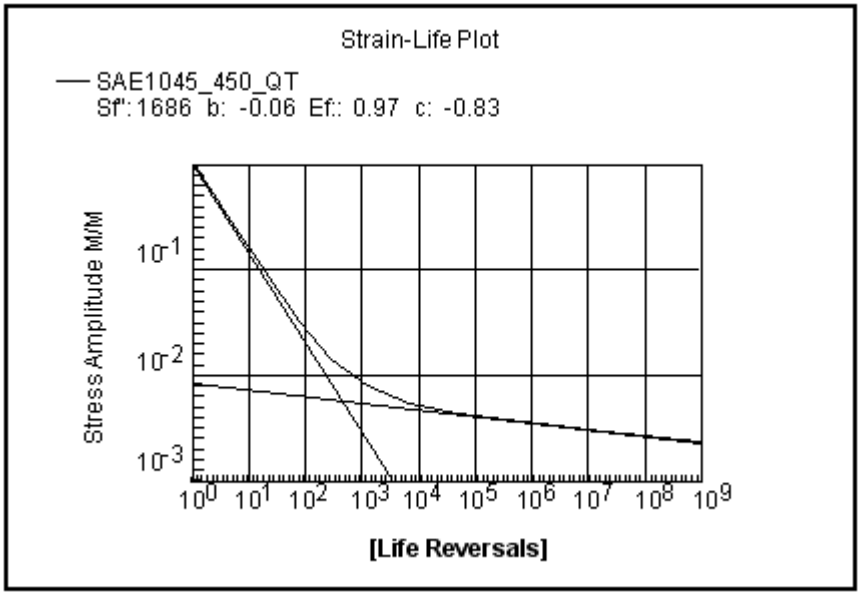


Figure 16-3 Example of a Strain-Life Plot

Crack Growth Data

The primary piece of information used in this model is the Paris curve which defines crack growth rate as a function of the stress intensity range. In addition, the fracture toughness and the threshold stress intensity range is required. These parameters are listed in Table 16-3 with a typical da/dN-Delta K plot in Figure 16-4.

Table 16-3 Crack Growth Data Parameter Description

Parameter Name	S.J. Units	Imperial Units	Parameter
Paris law coefficient	m/cycle	in/cycle	C
Paris law exponent	none	none	m
Delta K threshold at R=0	MPa $m^{1/2}$	KSI $in^{1/2}$	D0
Delta K threshold at R-1	MPa $m^{1/2}$	KSI $in^{1/2}$	D1
Fracture toughness	MPa $m^{1/2}$	KSI $in^{1/2}$	K1C
Stress ratio at threshold knee	none	none	Rc
Stress corrosion threshold	MPa $m^{1/2}$	KSI $in^{1/2}$	K1SCC
Unnotched fatigue strength	MPa	PSI	FL

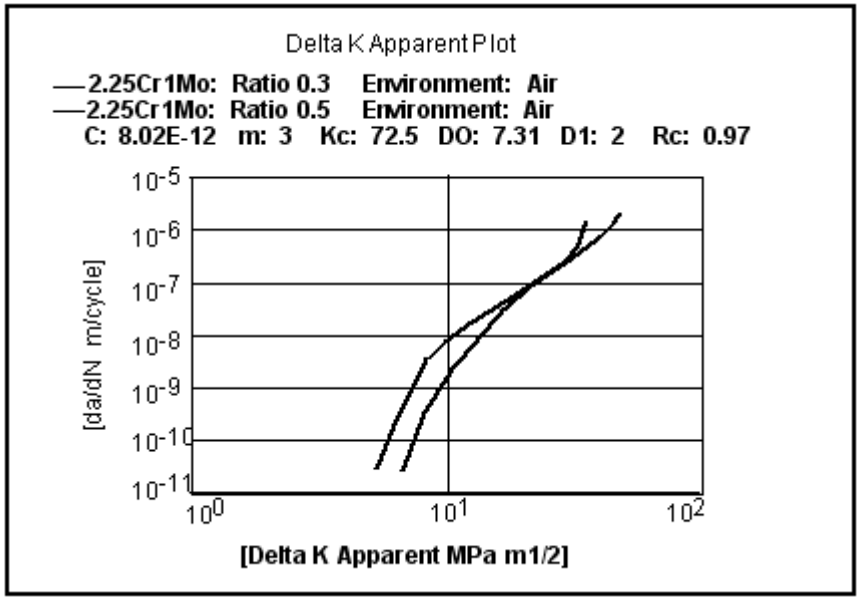


Figure 16-4 Example of a Delta K Apparent Plot

Loading

Loading time histories are measurements of loading for a period of time which must be long enough to ensure that the measurement reflects a typical duty cycle. The loading measurements will be made in the same attitude as the loading applied in the FEA. For example, an axle may experience bending loads in at least two

planes together with a torsion load, or a pressure vessel may experience both pressure and temperature variations in time. Simultaneous measurement of these loads for a typical or worst case event would be made.

Measurement of any physical phenomenon requires a transducer, the electronics to drive the transducer and a recording device. A typical combination would be a load cell, amplifier and FM tape recorder. Most engineering companies will have some capabilities and experience in doing this kind of work even if the group carrying out the finite element analysis does not. It is important that these two groups work together in order to achieve the best advantage of the MSC Fatigue package.

If it is not feasible to measure load or a related parameter, it may be necessary to synthesize a load time history. There are tools in the MSC Fatigue package to allow you to create, manage, and graphically edit time histories (PTIME). In some cases) the load variation with time may be trivial and easily created artificially, (e.g., the variation of pressure in a vessel). However, it is important to remember that many fatigue problems occur because an unexpected load combination occurred. For this reason, it is a truism that "a measurement is worth a thousand guesstimates."

Rainflow Cycle Counting

In reality, components are rarely subjected to purely constant amplitude loading, and so for many years various methods for extracting "equivalent" constant amplitude cycles from a random loading sequence were devised. Methods such as level crossing, range-pair, range-mean, and rainflow all attempt to reduce a random sequence of peaks and valleys to a set of equivalent constant amplitude cycles.

The concept of an equivalent constant amplitude cycle is perhaps, at first, difficult to understand. On the one hand, the most obvious cycle to be found in any time series will be the one that includes the maximum and minimum data values. On the other hand, the problem is how to define all the other cycles which are present.

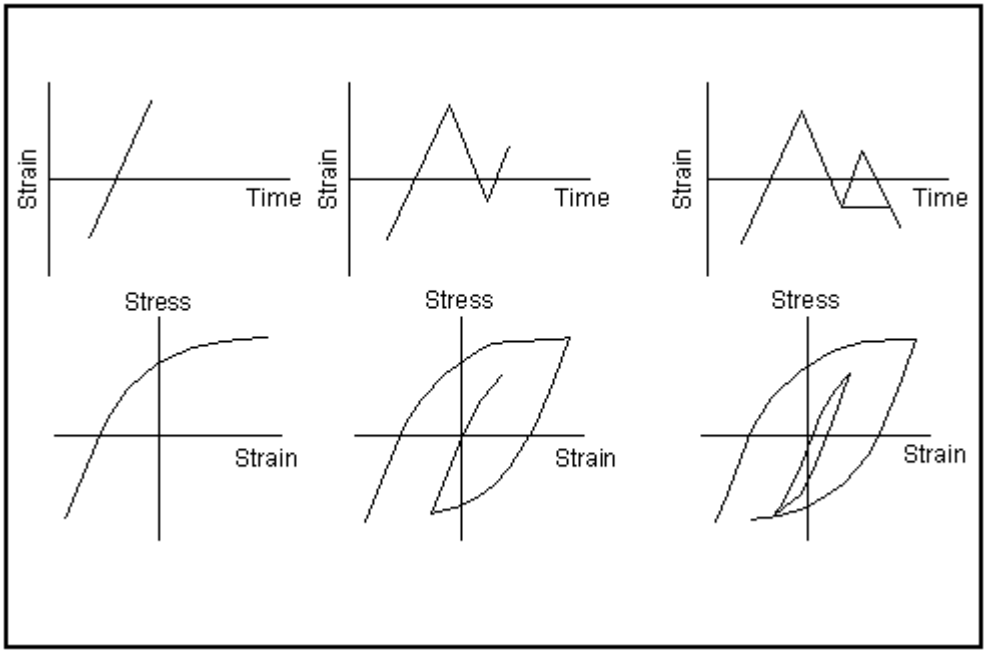


Figure 16-5 The Extraction of a Fatigue Cycle

A reversal can be easily understood as a change in the sign of the loading direction. Thus, a constant amplitude signal would consist of two equal and opposite reversals repeated continuously. In the simplest terms, therefore, a random time series may be considered to be a pair of reversals: the reversal from the maximum to the minimum in the signal and the reversal from minimum to the maximum in the signal, with all the other reversals effectively "interrupting" these two.

From the point of view of a material, this phenomenon is often referred to as "material memory", because a material subjected to a sequence of reversing loads, apparently interprets each closed cycle (matching pair of reversals), as a temporary interruption of a larger strain range, and "remembers" which hysteresis limb applies for this larger event.

Figure 16-5 shows a strain sequence of four turning points, and the stress-strain response of a material to this sequence. The closed hysteresis loop is a cycle, which may be characterized in terms of its strain range and mean strain. If the stress axis of this diagram is ignored and only successive strain ranges are considered then an algorithm can be developed which will extract cycles from a signal whatever its units. The rainflow algorithm is able to extract cycles in the way described above, classify them in terms of their range and mean value and store them in a range mean matrix.

The term "rainflow" is derived from an algorithm in which cycles are extracted through a consideration of rain drops flowing down a pagoda roof (the originators of this algorithm were Japanese). Modern algorithms no longer use this concept although the generic name "rainflow" still persists.

The Rainflow Procedure

The rainflow algorithm used by MSC Fatigue is based on the standard practice for cycle counting in fatigue analysis as defined by the ASTM designation E 1049-85, (See ASTM standards Vol. 03.01). Note that the form of the algorithm used relies on the fact that the analysis starts at the absolute maximum value in the data set. Under these circumstances, the rainflow cycle count will be identical to a range pair cycle count which itself starts at the largest value. Differences in the results produced by the two procedures only arise if processing starts at some value other than the absolute maximum.

It is possible to illustrate the rainflow cycle counting procedure used by considering the simple strain time series shown in Figure 16-6.

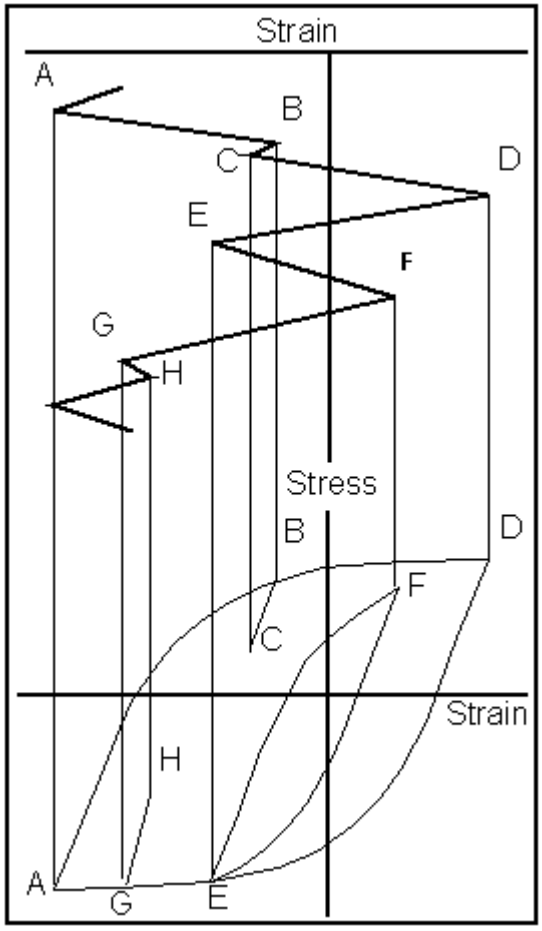


Figure 16-6 Illustration of the Rainflow Procedure

The local stress-strain response to the nominal strain time record, shown to the left, reveals that four equivalent constant amplitude cycles are present (i.e., `B-C-B, E-F-E`, `G-H-G` and the outside loop `A-D-A`). The small event `B-C-B` is treated as a interruption to the large overall event `A-D-A`. It is important to note that although, the events `B-C-B` and `G-H-G` appear very similar in the nominal strain record and would be counted as equally damaging by say the range-mean method, the local mean stresses and plastic strains are quite different and so the damage contribution from each would also be different. It should also be noted that local mean stress cannot be calculated directly from nominal mean strains. The main thrust behind the rainflow cycle counting method, therefore, is to treat small cycles as interruptions of larger ones.

Local Stress Information

The finite element analysis (FEA) provides a link between applied loads and the stress response at regular locations across the structure. This obviates the need for an engineer to obtain an approximate stress concentration factor as is normally the case in fatigue analysis. Moreover, the FEA allows the engineer to investigate the fatigue performance for a range of load combinations.

Linear Static Stress Analyses

One of the principles of the MSC Fatigue package is that the load cases being considered must be analyzed separately unless certain conditions apply (i.e., a stress analysis will be required for each discrete load applied to the structure). The results from the stress analysis provide a calibration between the applied load and the local stresses across the structure.

Linear elastic stress analysis is used because the fatigue analyzers in MSC Fatigue carry out various manipulations which include superimposing the FEA stress analysis results and carrying out the elastic-plastic transformations where necessary. From this, local strain- or stress-time histories at each location representing the strain response due to the simultaneous combination of all the load time histories are obtained.

Types of Load Cases in FEA

It is important that the units of the loads applied in the FEA correspond to those of the load time histories. To understand the reasoning below, it is necessary to define the following terms:

LOAD	a value of an applied force in units of force (kN,N, etc).
LOADING	a generic term for any external or internal effect which causes straining of a structure.

FEA is not acutely dependent on the actual units of force and length, and hence of stress. The calculation of life in MSC Fatigue is carried out in S.I. units. However, MSC Fatigue allows you to work in a range of unit systems.

Nevertheless, it is still the responsibility of the engineer to ensure the inputs are provided in a consistent manner.

The time history used in a fatigue calculation must be a representation of the time variation in the loading applied in the FEA. For simple cases, this implies a force time history corresponding to a time variation in the point loading used in the FEA.

There are a number of different kinds of loading possible, each one requiring a different type of time history as shown in Table 16-4.

Table 16-4 Possible Loading Types

Loading	Time History Used
Point Loads	Load time history at point load application position
Pressure	Pressure time history
Temperature	Temperature time history
Accelerations	Acceleration time history
Distributed Load	A related parameter time history*
Moment	Moment time history
Scalar	Any time history to define a scaled variation of the actual FEA stress-strain

*The distributed load is a special case which must be treated by considering a time history of a related quantity such as deflection. In this case you must specify the magnitude of the deflection at a given location as calculated by the FEA and the corresponding deflection time history.

A load time history will not be required if you wish to define a FE load case as a simple static offset of the stress-strain responses since this does not vary with time though the load case must be isolated and dealt with in a separate FEA.

Pressures may be applied and pressure time histories used in the fatigue analysis. Pressures and loads may be mixed in a fatigue analysis as long as the units are the same.

Any loads which vary in proportion to and in-phase with each other may be combined together in one FEA. However, the load time history must be a scalar which represents the proportion of these loads acting on the structure at each increment of time.

Note that, if you plan on using linear elastic results from a transient dynamic or forced vibration FE analysis, none of this applies since you have already defined your time variation in your FE analysis.

Total Life (S-N) Analysis

It has been recognized, since 1830, that a metal subjected to a repeated or fluctuating load will fail at a stress level lower than that required to cause fracture on a single application of the load. The nominal stress (S-N) method was the first approach developed to try to understand this failure process and is still widely used in applications where the applied stress is nominally within the elastic range of the material and the number of cycles to failure is large. From this point of view, the nominal stress approach is best suited to that area of the fatigue process known as *high-cycle fatigue*. The nominal stress method does not work well in the low-cycle

region where the applied strains have a significant plastic component. In this region, a strain-based methodology must be used.

Stress Cycles

Before looking in more detail at the nominal stress procedure it is worth considering the general types of cyclic stresses which contribute to the fatigue process. Figure 16-7 details some typical fatigue stress cycles.

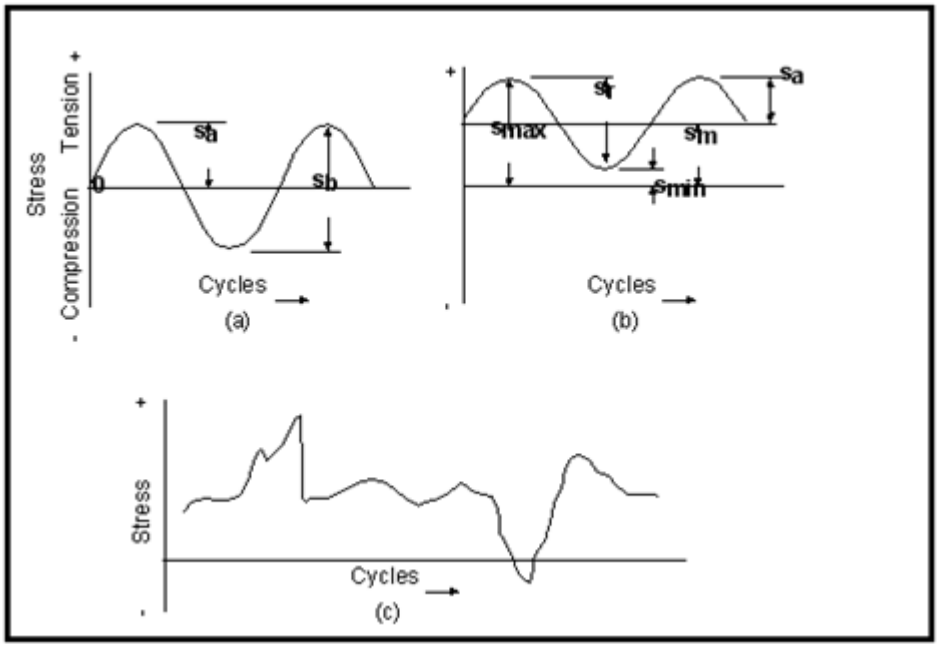


Figure 16-7 Typical Fatigue Stress Cycles,
(a) Fully Reversed (b) Offset, (c) Random

In Figure 16-7(a) illustrates a fully-reversed stress cycle with a sinusoidal form. This is an idealized loading condition typical of that found in rotating shafts operating at constant speed without overloads. For this kind of stress cycle, the maximum and minimum stresses are of equal magnitude but opposite sign. Usually tensile stress is considered to be positive and compressive stress negative. Figure 16-7 (b) illustrates the more general situation where the maximum and minimum stresses are not equal. In this case they are both tensile and so define an offset for the cyclic loading. Figure 16-7 (c) illustrates a more complex, random loading pattern which is more representative of the cyclic stresses found in real structures.

From the above, it is clear that a fluctuating stress cycle can be considered to be made up of two components, a *static* or steady state stress σ_m, and an alternating or variable stress *amplitude*, σ_a. It is also often necessary to consider the stress *range*, σ_r, which is the algebraic difference between the maximum and minimum stress in a cycle.

$$\sigma_r = \sigma_{max} - \sigma_{min} \tag{16-2}$$

The stress amplitude, σ_a, then is one half the stress range.

$$\sigma_a = \sigma_r / 2 = (\sigma_{max} - \sigma_{min}) / 2 \tag{16-3}$$

The mean stress, is the algebraic mean of the maximum and minimum stress in the cycle.

$$\sigma_m = (\sigma_{max} + \sigma_{min}) / 2 \tag{16-4}$$

Two ratios are often defined for the representation of mean stress, the stress or R ratio, and the amplitude ratio A.

$$R = \sigma_{min} / \sigma_{max} \tag{16-5}$$

$$A = \sigma_a / \sigma_m = (1 - R) / (1 + R) \tag{16-6}$$

Table 16-5 illustrates some R values for common loading conditions.

Table 16-5 R Ratio for Common Loading Conditions

R ratio	Loading Condition
R > 1	Both σ_{max} and σ_{min} are negative. Negative mean stress.
R = 1	Static loading.
0 < R < 1	Both σ_{max} and σ_{min} are positive; Positive mean stress, $\lvert\sigma_{max}\rvert > \lvert\sigma_{min}\rvert$.
R = 0	Zero to tension loading, $\sigma_{min} = 0$
R = -1	Fully-reversed loading, $\lvert\sigma_{max}\rvert = \lvert\sigma_{min}\rvert$; zero mean stress.
R < 0	$\lvert\sigma_{max}\rvert < \lvert\sigma_{min}\rvert$, σ_{max} approaching zero.
R *infinite*	σ_{max} equal to zero.

The S-N Curve

Between 1852 and 1870, the German railway engineer August Wöhler set up and conducted the first systematic fatigue investigation.

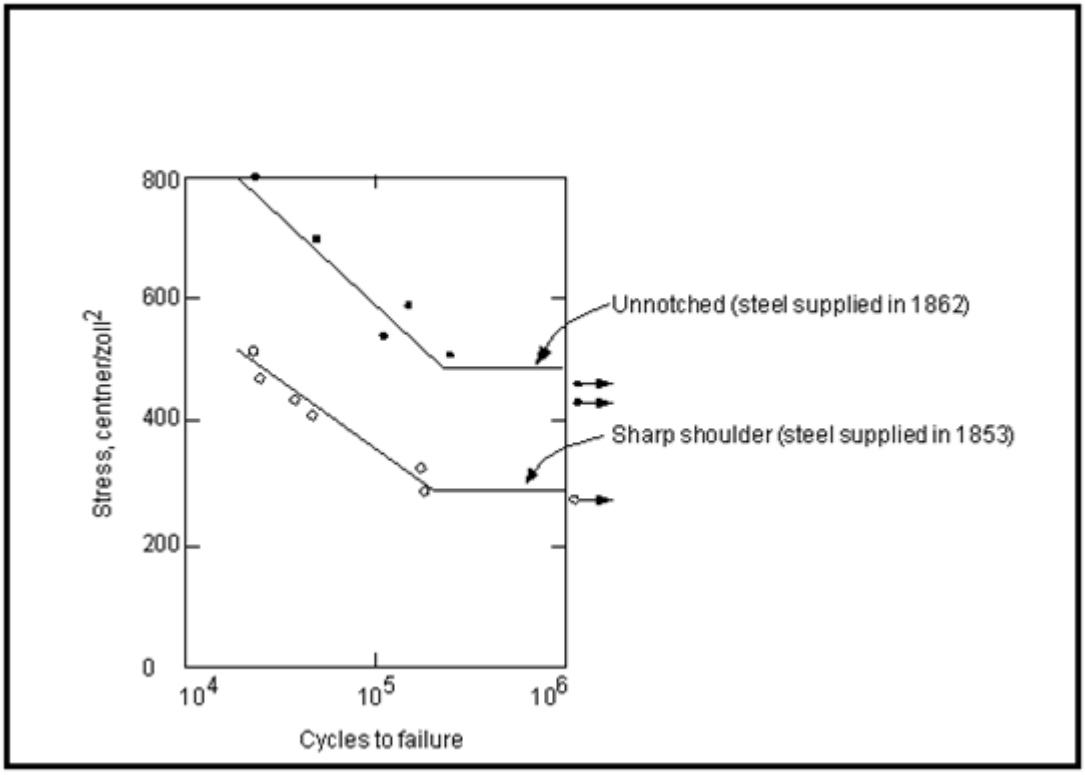


Figure 16-8 S-N Data Reported by Wöhler

(Note, 1 centner = 50 Kg, 1 zoll = 1 inch, 1 centner / $\text{zoll}^2 \approx 0.75\,\text{MPa}$.)

Wöhler conducted cyclic tests on full-scale railway axles and also on small-scale bending, torsion, and push-pull specimens of several different materials. Some of Wöhler's data, shown in Figure 16-8, are for Krupp axle steel and are plotted, in terms of nominal stress vs. cycles to failure, on what has become known as the S-N diagram. Typically, the S-N relationship is determined for a specific value of S_m, R or A. Note that in dealing with the nominal stress approach, the convention is that nominal stress is usually referred to as S and localized stress by the Greek counterpart, σ .

Procedure for Determining the S-N Curve

Most determinations of fatigue properties have been made in completely reversed bending (i.e., R = –1), by means of the so-called rotating bend test. One example is the R. R. Moore test, which uses four-point loading to apply a constant moment to a rotating (1750 rpm) cylindrical hourglass-shaped specimen. See Figure 16-9. Specimens, which are typically between 6 to 8 mm in diameter in the test section, are usually polished to a mirror finish prior to testing.

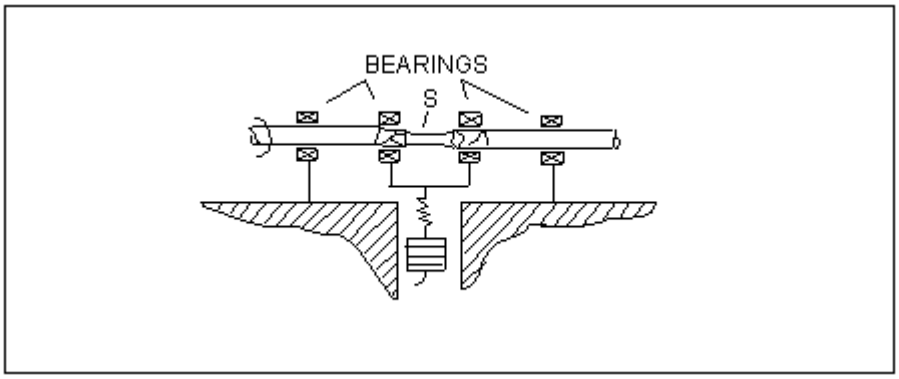


Figure 16-9 The R.R. Moore Fatigue Testing Machine

The stress level at the surface of the specimen is calculated using the elastic beam equation, even if the resulting value exceeds the yield strength of the material.

$$S = Mc/I \tag{16-7}$$

where:

S		the nominal stress acting normal to the cross-section
M		the bending moment
c		the distance of the surface from the neutral axis
I		the moment of inertia

For the circular section of the R. R. Moore specimen, the beam equation reduces to:

$$S = \frac{32M}{\pi d^3} \tag{16-8}$$

where:

d		the diameter of the specimen.

The usual laboratory procedure for determining an S-N curve is to test the first specimen at a high stress, about two thirds of the static tensile strength of the material, where failure is expected in a fairly small number of cycles. The test stress is decreased for each succeeding specimen until one or two specimens do not fail

before at least 10^7 cycles. For materials which exhibit it, the highest stress at which no failure occurs, a runout is taken to be the fatigue limit. For situations where an infinite life design requires a probability of survival to be associated with it, more complex testing and analysis procedures, such as the Prot and staircase methods, have been developed to determine the mean and variance of the fatigue limit. For materials which do not exhibit a fatigue limit, tests are usually terminated between 10^7 and 10^8 cycles, and the concept of an *endurance limit* at either 10^7 or 10^8 cycles defined.

The S-N curve is usually determined through the use of about 15 specimens. However, it is generally found that results are accompanied by a large amount of scatter and some form of statistical analysis should be applied, see The Statistical Nature of Fatigue, 1421 for more details.

S-N data are nearly always presented in the form of a log-log plot of alternating stress, amplitude S_a or range S_r, versus cycles to failure, with the actual Wöhler line representing the mean of the data. Certain materials, steels for example, display a fatigue limit, S_e, which represents an alternating stress level below which the material has an *infinite* life. For most engineering purposes, infinite is taken to be 1 million cycles. Great care must be exercised when designing on the basis of a fatigue limit, since it has a nasty habit of disappearing due to periodic overloads, corrosion, and elevated temperature.

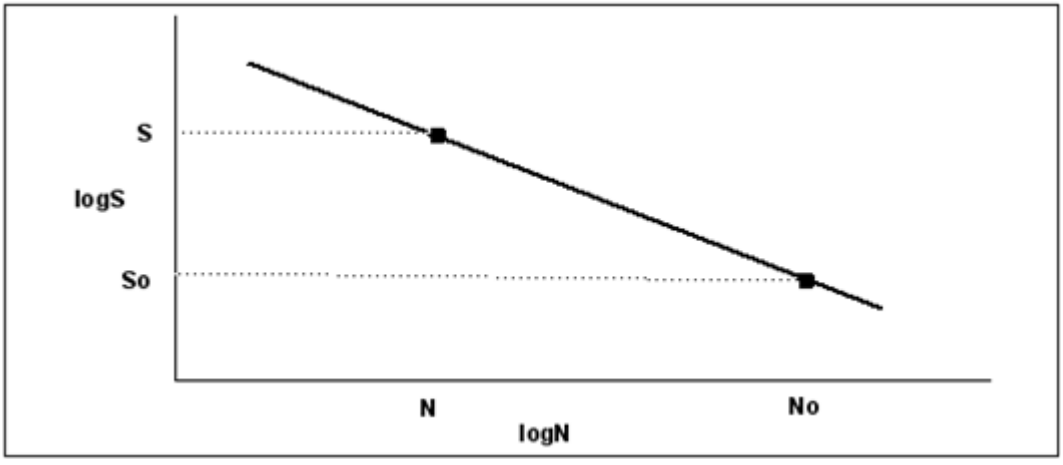


Figure 16-10 Idealized Form of the S-N Curve

When plotted on log-log scales, the relationship between alternating stress, S, and number of cycles to failure, N can be described by a straight line, Figure 16-10. The slope of the line, b, (after Basquin, a prominent worker who first proposed the law) can be derived from the following:

$$b = \frac{-(log S - log S_o)}{(log N_o - log N)} \tag{16-9}$$

$$log\mathrm{N}_{\mathrm{o}} log\mathrm{N} = -\frac{1}{\mathrm{b}} log\left(\frac{\mathrm{S}}{\mathrm{S}_{\mathrm{o}}}\right) \tag{16-10}$$

$$log N = log N_o + \frac{1}{b} log\left(\frac{S}{S_o}\right) \tag{16-11}$$

$$\mathrm{N} = \mathrm{N}_{\mathrm{o}}\left(\frac{\mathrm{S}}{\mathrm{S}_{\mathrm{o}}}\right)^{\frac{1}{\mathrm{b}}} \tag{16-12}$$

Sometimes, for convenience, the term 1/b is replaced by the letter k,

$$\mathrm{N} = \mathrm{N}_{\mathrm{o}}\left(\frac{\mathrm{S}}{\mathrm{S}_{\mathrm{o}}}\right)^{\mathrm{k}} \tag{16-13}$$

The above equation says that if we know the Basquin slope, b, and any other coordinate pair (N_o,S_o), then for a given stress amplitude S, the number of cycles can be calculated directly. Typically, N_o is taken to be 10^6 cycles and the corresponding stress amplitude is taken to be an endurance limit, usually denoted as S_e or S_6, so that the above equation may be rewritten as:

$$\mathrm{N} = \left(\frac{\mathrm{S}}{\mathrm{S}_{\mathrm{e}}}\right)^{\mathrm{k}} \times 10^6 \tag{16-14}$$

Example 1: A Simple Life Estimation

For a material with an endurance limit of 250 MPa, and a Basquin slope, b, of -0.1, calculate number of cycles to failure at a stress amplitude of 300 MPa. Under these conditions, (16-14) gives:

$$\mathrm{N} = \left(\frac{300}{250}\right)^{-10} \times 10^6 = 161,000 \mathrm{cycles}$$

Limits of the S-N Curve

As previously mentioned, the S-N approach is applicable to situations where cyclic loading is essentially elastic.

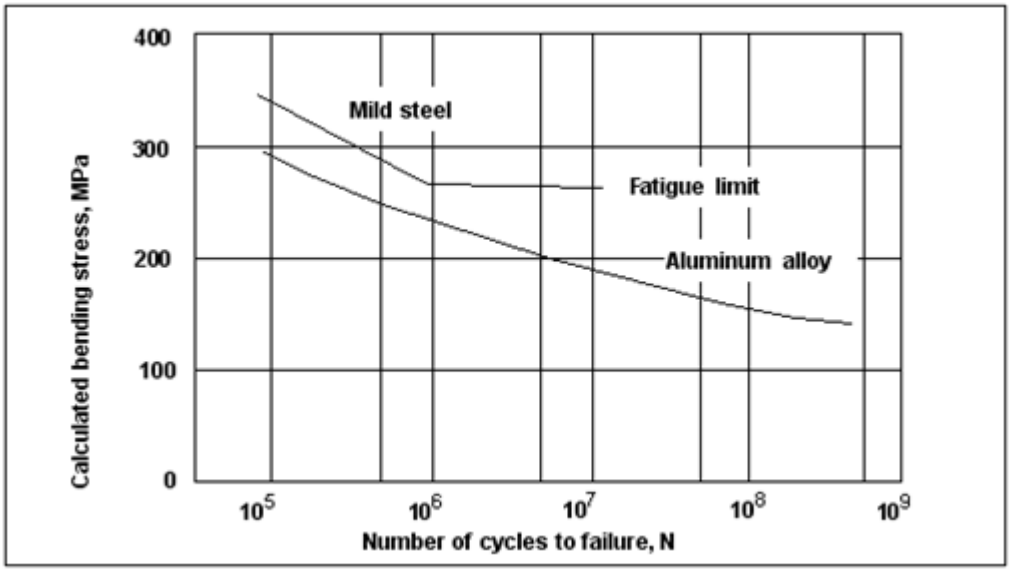


Figure 16-11 Typical S-N Curves for Ferrous and Nonferrous Metals

This means that the S-N curve should be confined on the life axis to numbers greater than about 10,000 cycles in order to ensure no significant plasticity is occurring.

Indeed, great care must be taken in using the above S-N equations in situations where lives less than 10,000 cycles are being estimated. Figure 16-11 shows typical S-N curves for both ferrous and nonferrous metals. The points to note in Figure 16-11 are the limits of the logN axis, the presence of a fatigue limit for the mild steel and the absence of a fatigue limit for the aluminum alloy. Because both materials represented in Figure 16-11 have relatively low yield stresses, the life axis is confined to begin at 10^5 cycles at which point the alternating stress is about 350 and 300 MPa respectively for the two alloys.

Tensile Properties and the S-N Curve

Through many years of experience, particularly with steels, empirical relationships between fatigue and tensile properties have been developed. These relationships are not soundly based in science. However, they remain useful tools for engineers for assessing fatigue performance. When the S-N curves for a number of different steels of varying strengths are plotted as the ratio of endurance limit (i.e., the stress amplitude at 10^6 cycles), S_6, to ultimate tensile strength, S_u, all the curves tend to all fall onto a single curve which implies that:

$$S_6 = S_e \approx 0.5 S_u \text{ for } (S_u < 1400 \text{MPa}) \tag{16-15}$$

and

$$S_6 = S_e \approx 700 MPa \text{ for } (S_u > 1400 \text{MPa}) \tag{16-16}$$

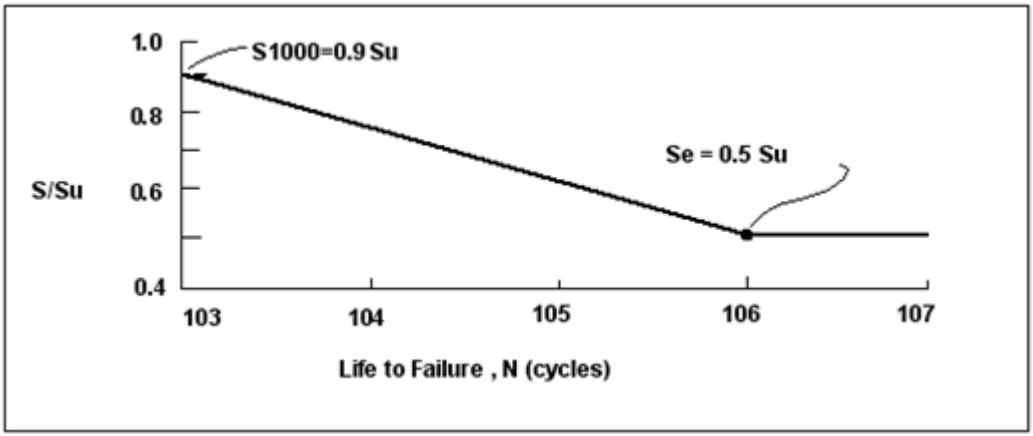


Figure 16-12 Generalized S-N Curve for Wrought Steels

In addition to this, the stress at 10^3 cycles, S_3, can be approximated by 0.9 S_u and so, utilizing these approximations, a generalized S-N curve can be generated for wrought steels, see Figure 16-12.

Methods of representing the S-N curve in the range 1 to 10^3 cycles have been developed but they must be treated with extreme caution. They usually use some percentage of the ultimate strength, S_u, or true fracture stress, σ_f, as a measure of the stress amplitude at either 1 or 1/4 cycles. The main difficulty with employing this approach is that the deduced S-N curves are extremely flat in the low-cycle region, and this makes estimates of life particularly inaccurate. The reason for this apparent flatness is the large plastic strain which results from the high load levels. Low-cycle fatigue analysis is best treated by strain-based procedures which account for rather than ignore the effects of plasticity.

The Influence of Mean Stress

As previously mentioned, most basic fatigue data are collected in the laboratory by means of testing procedures which employ fully reversed loading (i.e., R = –1). However, most realistic service situations involve nonzero mean stresses. It is, therefore, very important to know the influence that mean stress has on the fatigue process so that the fully reversed laboratory data can be usefully employed in the assessment of real situations.

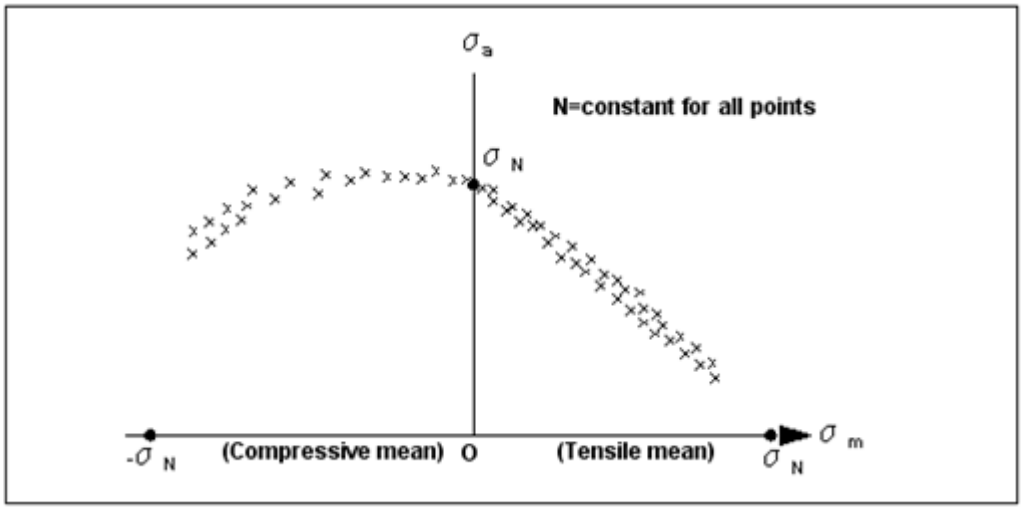


Figure 16-13 High-Cycle Fatigue Data Showing the Influence of Mean Stress

Fatigue data collected from a series of tests designed to investigate different combinations of stress amplitude and mean stress are characterized in Figure 16-13 for a given number of cycles to failure. The diagram plots the mean stress, both tensile and compressive, along the x-axis and the alternating constant stress amplitude along the y-axis. This kind of representation was first proposed by Haigh and is therefore commonly referred to as the Haigh diagram.

The stress amplitude at zero mean stress, S_n, corresponds to the stress amplitude at N cycles to failure as measured by the fully-reversed fatigue test. The failure data points tend to follow a curve which if extrapolated would pass through the ultimate tensile strength, S_u, on the mean stress axis. Notice that the influence of mean stress is different for compressive and tensile mean stress values. Failure appears to be more sensitive to tensile mean stress, than compressive mean stress. When available, data of the type illustrated above are collated into what are commonly referred to as master diagrams for a particular material. Figure 16-14 illustrates the master diagram for SAE 4340 from the U.S. Department of Defense MIL Handbook-5.

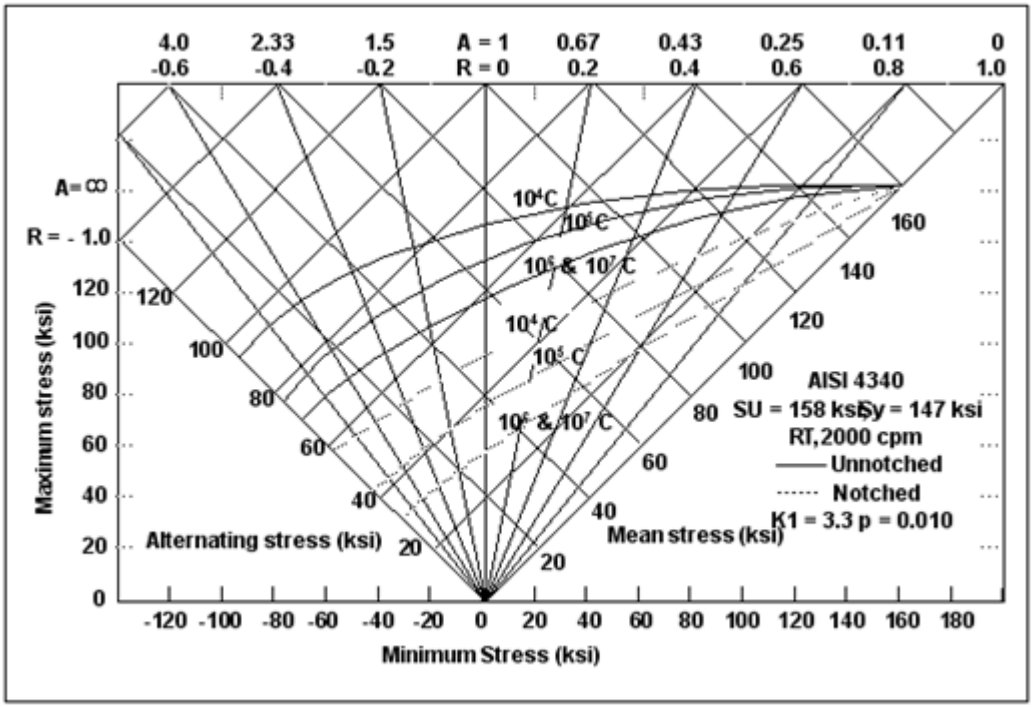


Figure 16-14 Master Diagram for SAE 4340

Since the tests required to generate a Haigh or master diagram are quite expensive, several empirical relationships which relate alternating stress amplitude to mean stress have been developed. These relationships characterize a material through its ultimate tensile strength, S_u, and are very convenient. For infinite life design strategies, the methods use various curves to connect the endurance limit, S_e, on the alternating stress axis to either the yield stress, S_y, ultimate strength, S_u, or true fracture stress, σ_f, on the mean stress axis. Of all the proposed relationships, two have been most widely accepted (i.e., those of Goodman and Gerber).

$$\text{Goodman: } \left(\frac{S_a}{S_e}\right) + \left(\frac{S_m}{S_u}\right) = 1 \tag{16-17}$$

$$\text{Gerber: } \left(\frac{S_a}{S_e}\right) + \left(\frac{S_m}{S_u}\right)^2 = 1 \tag{16-18}$$

Experience has shown that actual test data tends to fall between the Goodman and Gerber curves (Goodman joining S_e to S_u by means of straight line and Gerber by means of a parabola). For most design situations where R « 1 (i.e., small mean stress in relation to the alternating stress), there is little difference between the two relationships. However, when R approaches 1 (i.e., nearly equal mean and alternating stresses, the two relationships show considerable differences). Unfortunately, little or no experimental data exist to support one approach over the other and so typically, the recommendation would be to select the approach which provides the most conservative lives in a given situation.

Example 2: Correcting for Mean Stress Effects

A component is subjected to a maximum cyclic stress of 750 MPa and a minimum of 70 MPa. The steel from which it is manufactured has an ultimate tensile strength, S_u, of 1050 MPa and a measured endurance limit, S_6, of 400 MPa. The fully reversed stress at 1000 cycles is 750 MPa. Using both the Goodman and Gerber mean stress correction procedures calculate the component life.

The first step is to calculate the stress amplitude, S_a, and the mean stress, S_m, using equations (16-3) and (16-4).

$$S_a = \frac{(S_{max} - S_{min})}{2} = \frac{(750 - 70)}{2} = 340 \text{MPa} \tag{16-19}$$

$$S_m = \frac{(S_{max} + S_{min})}{2} = \frac{(750 + 70)}{2} = 410 \text{MPa} \tag{16-20}$$

A Haigh diagram for the Goodman correction procedure can now be constructed for constant lives of 10^6 and 10^3 cycles. This is done by connecting the endurance limit, S_6, and the stress at 1000 cycles, S_3, respectively, on the alternating stress axis with the ultimate tensile strength, S_u, on the mean stress axis (see Figure 16-15). The stress conditions on the component calculated above, S_a = 340 and S_m = 410, can be plotted on the diagram and a line drawn from S_u to the alternating stress axis. This line represents the constant life line for the component at all combinations of stress amplitude and mean stress. The line intersects the fully reversed axis at a stress S_n = 558 MPa.

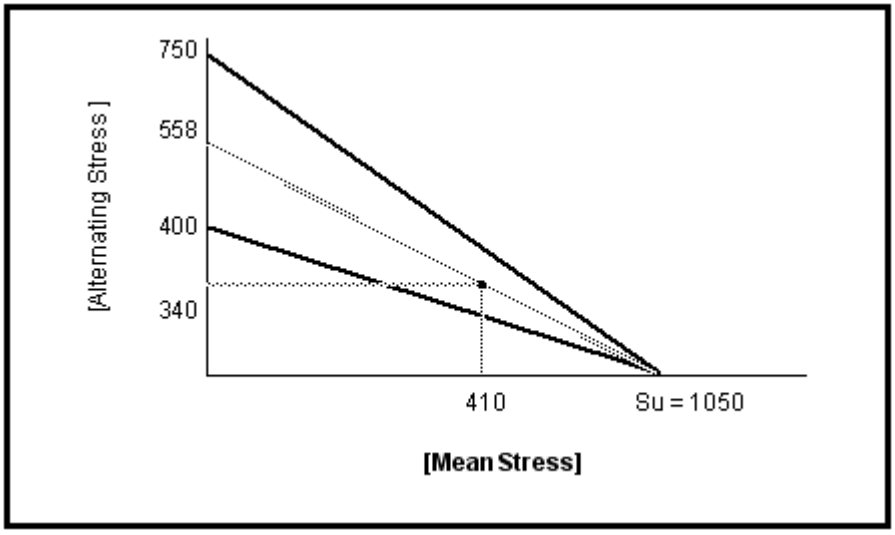


Figure 16-15 The Haigh Diagram

It should be noted that this stress can also be calculated directly from the Goodman equation (16-17):

$$\left(\frac{S_a}{S_n}\right) + \left(\frac{S_m}{S_u}\right) = 1 \tag{16-21}$$

$$\left(\frac{340}{S_n}\right) + \left(\frac{410}{1050}\right) = 1$$

$$S_n = 557.8\text{MPa}$$

It will be recalled that the S-N curve is given by equation (16-13),

$$N = N_o\left(\frac{S}{S_o}\right)^{\frac{1}{b}}$$

and so for the conditions defined at 10^3 and 10^6 cycles,

$$b = -\frac{\log S - \log S_o}{\log N_o - \log N}$$

$$b = -\frac{1}{3}\log\left(\frac{S_3}{S_6}\right)$$

and so

$$b = -\frac{1}{3}log\left(\frac{750}{400}\right) = -0.091$$

and for S_n = 557.8, the life can be calculated from:

$$N = N_o\left(\frac{S}{S_o}\right)^{\frac{1}{b}}$$

$$N = \left(\frac{558}{400}\right)^{-11} \times 10^6$$

$$N = 26,000 cycles$$

The Gerber correction can be used in a similar way, i.e.,

$$\left(\frac{S_a}{S_n}\right) + \left(\frac{S_m}{S_u}\right)^2 = 1$$

$$\left(\frac{340}{S_n}\right) + \left(\frac{410}{1050}\right)^2 = 1$$

$$S_n = 401.2 MPa$$

and

$$N = \left(\frac{401}{400}\right)^{-11} \times 10^6$$

$$N = 973,000 cycles$$

Factors Influencing Fatigue Life

A standardized rotating bend test (such as the R.R. Moore test) is used to determine a baseline S-N relationship for a polished specimen of approximately 6mm diameter loaded under conditions of fully reversed bending. If the fatigue or endurance limit measured by these means be denoted by S'_e then the actual limit for a real component, S_e, must reflect all the modifications that come about in moving from a laboratory specimen to a component. For steels in particular, several empirical relationships have been developed which can account for the variation in S_e as a result of the following:

- Component size
- The type of loading
- The effect of notches
- The effect of surface finish
- The effect of surface treatment

The usual way to account for these effects is through the calculation and application of specific modifying factors so that:

$$S_e = S'_e C_{notch} C_{size} C_{load} C_{sur} \cdots \quad (16\text{-}22)$$

where reciprocal of the product, C_{notch} C_{size} C_{load} C_{sur}, is collectively known as the *fatigue strength reduction factor* K_f, i.e.,

$$K_f = \frac{1}{(C_{notch} C_{size} C_{load} C_{sur} \cdots)} \quad (16\text{-}23)$$

The approach tends to be conservative and the corrections are usually only applied at the endurance limit, the modifications required for the rest of the S-N curve are ill-defined. Typically, the procedure is to pivot the S-N curve about the 1000 cycles point, see Figure 16-16.

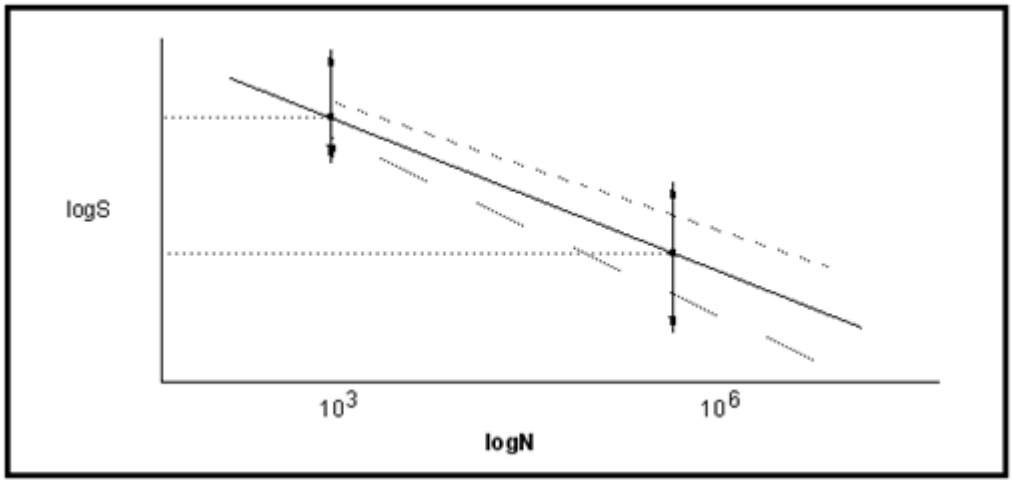


Figure 16-16 Modification of the S-N Curve

It is very important to remember that all the modification factors are empirical, conservative and mostly only applicable to steels. They provide little or no fundamental insight into the fatigue process itself other than providing approximate trends. In particular, they should not be used in areas outside their measured applicability.

The Influence of Component Size

Fatigue in metals results from the nucleation and subsequent growth of crack-like flaws under the influence of an alternating stress field. This view leads to the concept of failure commencing from the *weakest link*, the most favorably orientated metal crystal for example, and then growth through less favorably orientated grains until final failure. Intuitively, it would seem reasonable to suppose that the larger the volume of material subjected to the alternating stress, the higher the probability of finding the weakest link sooner. Actual test data do confirm the presence of a size effect particularly in the case of bending and torsion.

The stress gradient built up through the section, in bending and to a lesser extent in torsion, concentrates more than 95% of the maximum surface stress to a thin layer of surface material. In large sections, this stress gradient will be less steep than in smaller ones, and so the volume of material available which could contain

a critical flaw will be greater leading to reduced fatigue strength. The effect is quite small for axial tension where the stress gradient is absent. The value for C_{size} can be estimated from one of the following, if the diameter of the shaft is < 8mm:

$$C_{size} = 1 \tag{16-24}$$

if the diameter is between 8 mm and 250 mm:

$$C_{size} = 1.189 d^{-0.097} \tag{16-25}$$

The effect of size is particularly important for the analysis of rotating shafts such as might be found in vehicle powertrains.

For situations where components do not have a round cross section, an equivalent diameter, d_{eq}, can be calculated for a rectangular section width, w and thickness, t, undergoing bending from:

$$d_{eq}^2 = 0.65 wt \tag{16-26}$$

The Influence of Loading Type

Fatigue data measured according to one regime, axial tension for example, may be "corrected" to represent the data that would have been obtained had the test been carried out in some other loading methodology such as torsion or bending. Recall that the R.R Moore test calls for fatigue tests to be carried out under conditions of fully-reversed bending.

The values of C_{load} to be used in conjunction with the endurance limit, S_e, in moving from one loading condition to another are detailed in Table 16-6.

Table 16-6 Modification Factor at 10^6 Cycles for Various Loadings

Measured Loading	Target Loading	C_{load}
Axial to	Bending	1.25
Axial to	Torsion	0.725
Bending to	Torsion	0.58
Bending to	Axial	0.8
Torsion to	Axial	1.38
Torsion to	Bending	1.72

In addition to influencing the endurance limit, loading conditions can also influence the Basquin slope, b. This effect is usually taken into account by modification of the stress at 10^3, S_3, as well as S_e. The following factors can be used to define C'_{load}, the S_3 modification factor, see Table 16-7.

Table 16-7 Modification Factor at 10^3 Cycles for Various Loadings

Measured Loading	Target Loading	C_{load}
Axialto	Torsion	0.82
Bendingto	Torsion	0.82
Torsionto	Axial	1.22
Torsionto	Bending	1.22

The Influence of Surface Finish

A very high proportion of all fatigue failures nucleate at the surface of components and so surface conditions become an extremely important factor influencing fatigue strength. The usual standard by which various surface conditions are judged is against the polished laboratory specimen. Normally, scratches, pits, machining marks influence fatigue strength by providing additional stress raisers which aid the process of crack nucleation. Broadly speaking, high strength steels are more adversely affected by a rough surface finish than softer steels, for this reason the surface correction factor, C_{sur}, is strongly related to tensile strength. The surface finish correction factor is often presented on diagrams that categorize finish by means of qualitative terms such as polished, machined or forged, see Figure 16-17.

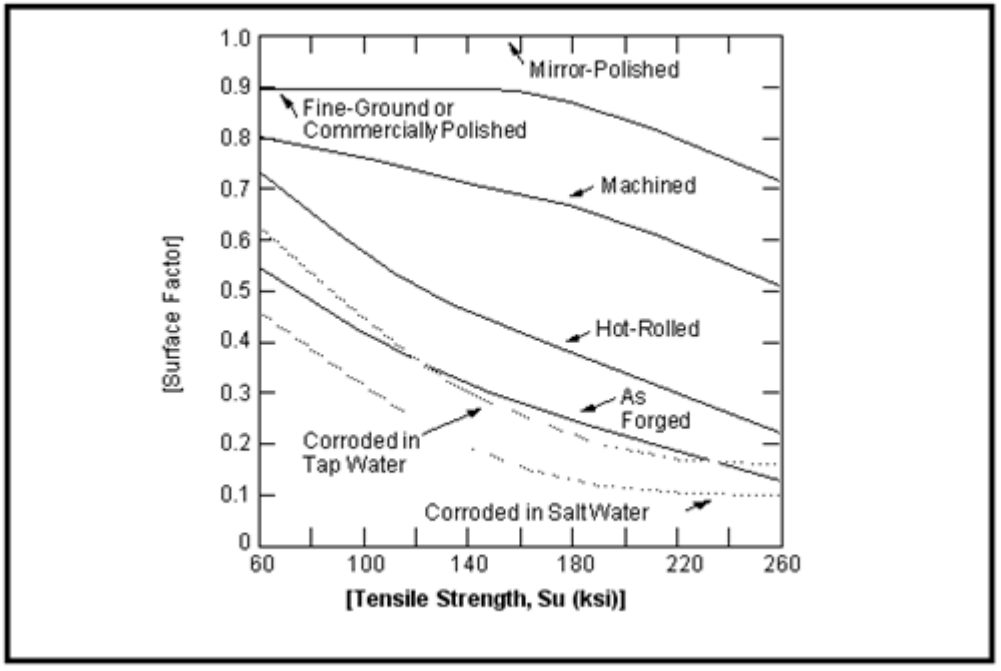


Figure 16-17 Surface Finish Correction Factor for Steel Components

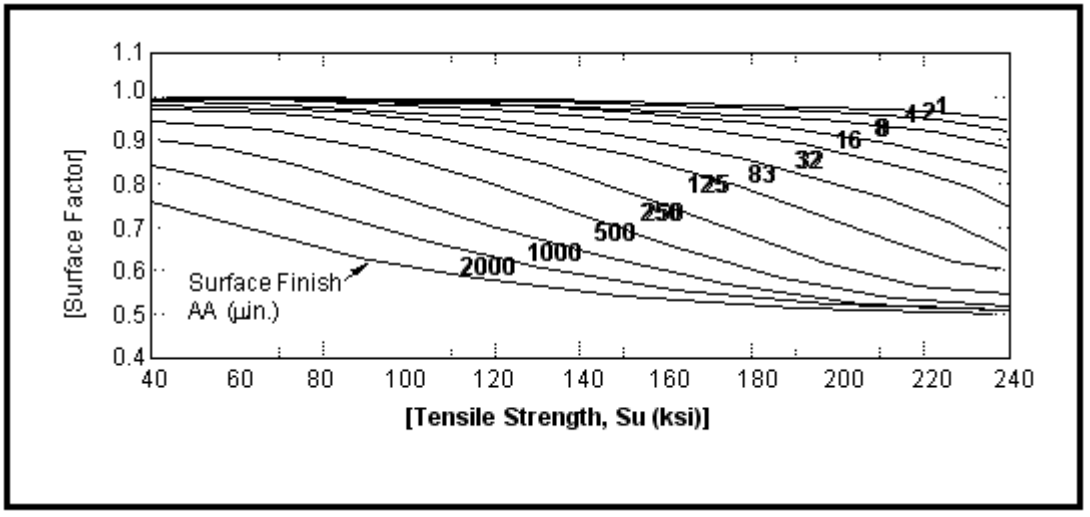


Figure 16-18 The Effect of Surface Roughness on Surface Finish Factor

It is worth noting that some of the curves presented in Figure 16-17 include effects other than just surface finish. For example, the forged and hot-rolled curves include the effect of decarburization.

Other diagrams present the surface factor in a more quantitative way by using a quantitative measure of surface roughness such as R_A, the root mean square, or AA the arithmetic average, see Figure 16-18. Values of surface roughness associated with each of the manufacturing processes are readily available in handbooks, as an example consider Table 16-8.

Table 16-8 Values of Surface Roughness for Various Processes

Type of finish	Surface Roughness (microns)
Lathe-formed	2.67
Partly hand polished	0.15
Hand Polished	0.13
Ground	0.18
Superfinished	0.18
Ground and polished	0.05

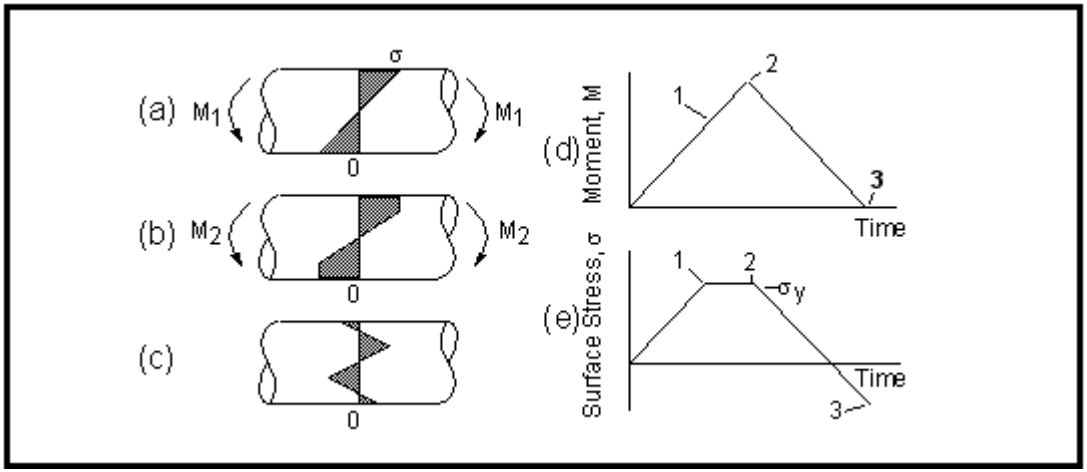


Figure 16-19 Residual Stress in a Beam

The Influence of Surface Treatment

As in the case of surface finish, surface treatment can have a profound influence on fatigue strength, particularly the endurance limit. Surface treatments can be divided broadly into mechanical, thermal, and plating processes. The important point to note with all three is that the net effect of the treatment is to alter the state of residual stress at the free surface. In the first two processes, by providing a compressive layer, and in the case of plating, by providing a tensile residual stress.

Residual stresses arise when plastic deformation is not uniformly distributed throughout the entire cross-section of the component being deformed. Figure 16-19 represents a metal bar whose surface has been deformed in tension by bending so that part of it has undergone plastic deformation. When the external force is removed, the regions which have been deformed plastically prevent the adjacent elastic regions from complete elastic recovery to the unstrained condition. In this way, the elastically deformed regions are left in residual tension, and the plastically deformed regions must be in a state of residual compression. For many purposes, residual stress can be considered identical to the stresses produced by an external force. Thus, the presence of a compressive residual stress at the surface of a component will have the effect of decreasing the likelihood of fatigue failure.

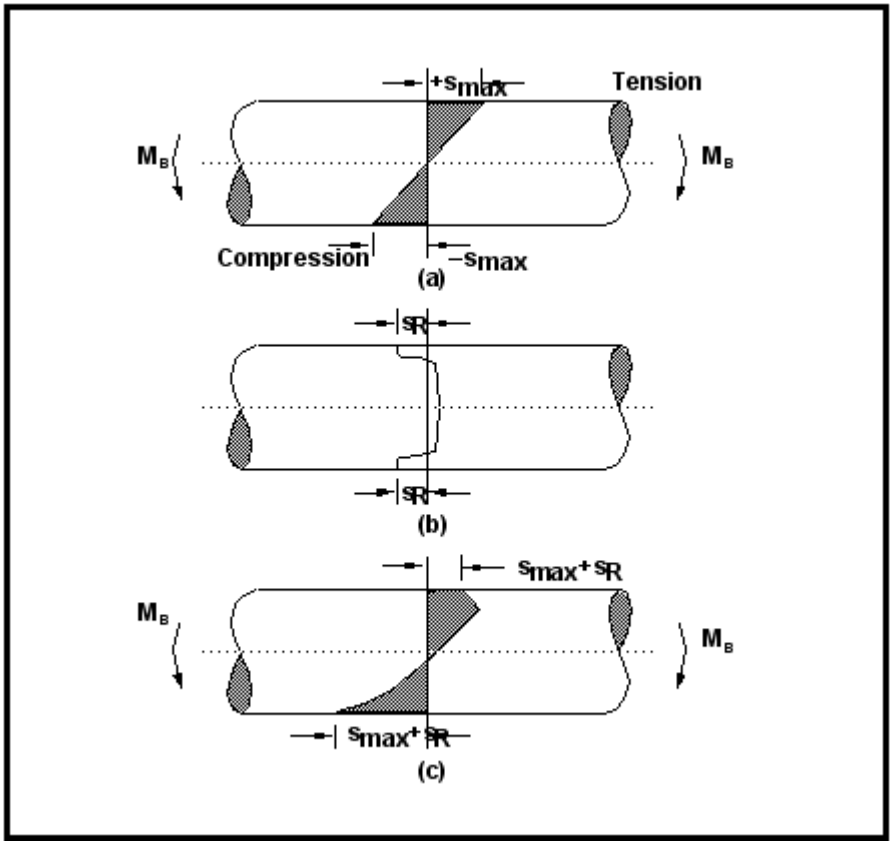


Figure 16-20 Superposition of Applied and Residual Stresses

Figure 16-20 illustrates the effect schematically. Figure 16-20(a) shows an elastic stress distribution in a beam with no residual stress. The typical residual stress distribution associated with shot peening is detailed in Figure 16-20(b). Note that the compressive stress at the surface must be compensated by an equivalent tensile stress over the interior of the cross section. In Figure 16-20(c), the distribution due the algebraic summation of the residual and applied stresses is shown. Observe that the maximum tensile stress at the surface has been reduced by the amount of the residual stress. Furthermore, note that the peak tensile stress has now been moved to the interior of the beam. The magnitude of this stress will depend on the gradient of the applied stress and the residual stress distribution. Also note that under these conditions, subsurface crack initiation becomes a possibility.

Mechanical Treatments

The main commercial methods for introducing residual compressive stresses are cold rolling and shot peening. Although some alteration in the strength of the material occurs as a result of work hardening, the improvement in fatigue strength is due mainly to the compressive surface stress. Surface rolling is particularly suited to large parts and is frequently used in critical components such as crankshafts and the bearing surface of railway axles. Bolts with rolled threads typically possess twice the fatigue strength of conventionally machined threads.

Shot peening, which consists of firing fine steel or cast iron shot against the surface of a component, is particularly well suited to processing small mass produced parts.

It is important to remember that cold rolling and shot peening have their greatest effect at long lives. At short lives they have little or no effect.

As with other modifying factors, the effect of these mechanically induced compressive stresses can be accounted for by the use of correction factors which can be used to adjust the endurance limit S_e. Typically the factor associated with peening is about 1.5-2.0.

Plating

Chrome and nickel plating of steel components can more than halve the endurance limit due to the creation of tensile residual stresses at the surface. These tensile stresses are a direct result of the plating process itself. As in the case of mechanically induced surface stresses, the effect of plating is most pronounced at the long life end of the spectrum and also with higher strength materials.

The deleterious effect of plating can be reduced by introducing a compressive residual stress prior to the plating process by either shot peening or nitriding. An alternative approach might be to anneal components after plating and thereby relieve the tensions.

Electroplating may also cause a shortening of the fatigue life through the mechanism of hydrogen embrittlement. Hydrogen ions are produced at the surface of the metal during plating. For high-strength steels, these ions enter the large steel microstructure and then combine into the molecular hydrogen, H_2, causing high stresses under cyclic loading and accelerating fatigue failure. Such effects are difficult to model analytically. In practice, the preventative action of baking the component after plating to release the hydrogen is often used.

Thermal Treatments

Thermal treatments are processes which rely on the diffusion of either carbon, *carburizing*, or nitrogen, *nitriding*, onto and into the surface of a steel component. Both species of atoms are interstitial (i.e., they occupy the spaces between adjacent iron atoms), and thereby both increase the strength of the steel and, through volumetric changes, cause a compressive residual stress to be left on the surface. Carburizing is commonly carried out by packing the steel components within boxes which contain carbonaceous solids, sealing to exclude the atmosphere and heating to about 900° C for a period of time which depends on the depth of the case required. Alternatively, components may be heated in a furnace in the presence of a hot carburizing gas such as methane. This process has the advantage that it is quicker and more accurate. In addition, the carburizing cycle may be followed up by a diffusion cycle, with no carburizing agent present, which allows some of the carbon atoms to diffuse further into the component and so reduce gradients.

The nitriding process is very similar in nature to gas carburizing, except that, in this case, ammonia gas is used and the components are soaked at lower temperatures. Typically 48 hours at about 550 degrees centigrade will provide a nitrided case depth of about 0.5 mm. Nitriding is particularly suited to the treatment of finished notched components such as gears and slotted shafts, the effectiveness of the process is illustrated in Table 16-9.

Table 16-9 The Effect of Nitriding on Endurance Limit

Geometry	Endurance Limit (MPa)	
	not nitrided	nitrided
Unnotched	310	620
Semicircular notch	175	600
V notch	175	550

Example 3: To Shot Peen or Not?

From a particular vehicle, as heat treated, low alloy leaf springs (manufactured to a tensile strength of 1500 MPa), are failing in service, albeit after some considerable time, but nevertheless within the lifetime of the vehicle. Service measurement has shown that the loading is of the type from zero to a maximum stress of about 770 MPa (i.e., R = 0). The question to be answered is, will shot peening the springs prior to installation make the problem go away?

The springs have a section measuring 40 mm by 5 mm. From tables of surface roughness, C_{sur} has been determined to be about 0.75 for the "as heat treated" condition and about 0.58 for the "shot peened" condition. Measurements have shown that shot peening introduces a compressive residual stress of about 550 MPa.

From the ultimate strength, S_u = 1500 MPa, S'_e can be estimated to be 750 MPa.

For the "as heat treated" condition the modification factor required to correct for the section size of the springs can be estimated by first calculating the equivalent diameter, d_{eq}, associated with the rectangular cross section from (16-26):

$$d_{eq}^2 = 0.65\ w\ t = 0.65 \times 40 \times 5 = 130$$
$$d_{eq} = 11.4 \text{ mm}$$

and since $d_{eq} > 8$ mm, C_{size} can now be computed from (16-25):

$$C_{size} = 1.189 \times 11.4^{-0.097} = 0.94$$

Since, in operation, the leaf springs are loaded only in bending, C_L is equal to 1.0.

We are now in a position to calculate the modified magnitude of endurance limit, S_e.

$$S_e = S' \text{ x the product of all the modifying factors}$$
$$S_e = 750 \times 0.94 \times 1.0 \times 0.75$$
$$S_e = 529 \text{ MPa}$$

For the loading condition defined by R = 0, the amplitude of the alternating stress, S_a, and the mean stress, S_m, must be equal. By setting $S_a = S_m$, the allowable stress level, S, for the as heat-treated material can now be calculated from the Goodman equation (16-17).

$$(S_a / S_e) + (S_m / S_u) = 1$$
$$(S / 529) + (S / 1500) = 1$$
$$S = 391 \text{ MPa}$$

and the maximum allowable stress for infinite life would be

$$S_{max} = S_a + S_m = 2\ S = 782 \text{ MPa}$$

This figure is uncomfortably close to the operational maximum stress of 770 MPa and so clearly we do not have an infinite life design; we must expect to have some failures in service.

The procedure so far has been useful since it has helped us verify the calculation procedure being used and provided some confidence in the methodology. We are now in a position to estimate the effect of shot peening.

In the shot peened condition, the modified magnitude of endurance limit, S_e is given by:

$$S_e = S' \text{ x the product of all the modifying factors}$$
$$S_e = 750 \times 0.94 \times 1.0 \times 0.58$$
$$S_e = 409 \text{ MPa}$$

As in the as heat-treated case, for R = 0, the amplitude of the alternating stress, S_a, and the mean stress, S_m, are equal and the Goodman equation can be used to calculate the allowable stress. By subtracting the compressive residual stress from the mean stress, the peening operation is taken into account,

$$\left(\frac{S_a}{S_e}\right) + \left(\frac{S_m}{S_u}\right) = 1$$

$$\left(\frac{S}{409}\right) + \left(S - \frac{550}{1500}\right) = 1$$

S = 439 MPa

and the maximum allowable stress for infinite life would be

$$S_{max} = S_a + S_m = 2\ S = 878 \text{ MPa}$$

The calculated maximum working stress for shot peened springs is significantly higher than both the equivalent as heat treated stress and the operational stress. From this, we could justifiably conclude that shot peening to the extent which produces residual compression of 550 MPa would be sufficient to provide the leaf spring with an infinite service life.

Application in MSC Fatigue

MSC Fatigue supports two types of stress-life curve. The first is the nominal or component S-N curve where the stress plotted on the Y-axis is not the actual stress where failure occurs (i.e., there is a built-in geometric stress concentration factor, K_t). The use of this type of S-N curve is only valid when the stress in a nominal or reference area is used to compute lives. This approach takes no account of localized plasticity effects such as might be found at a stress-raiser. The effects of such discontinuities must be taken into account through the S-N damage curve itself. Each S-N curve, therefore, must reflect the overall "structural" characteristics of the detail under investigation.

The second type of S-N curve is the local stress or material S-N curve. This curve has associated with it a Kt value of 1.0 (i.e., the stress on the Y-axis is a local stress which causes fatigue). No reference area is needed as the results at each node or element may be used directly.

The S-N approach must be used when basic material data are unavailable, or one or more of the basic assumptions embodied in the more fundamental local stress-strain approach are considered invalid. Typical examples would be the treatment of welded structures or complex joints, rivets or spot welds and composites. In addition) some companies, due to the extensive use of the stress-life method in past years, a large quantity of S-N curves and experience has been accumulated. In this case, the stress-life method offers compatibility with previous work and so may be more appropriate.

The Measured Loading Environment

MSC Fatigue requires a representation of a load time history in order to define the loading environment to which a component or structure is subject, such as shown in Figure 16-21.

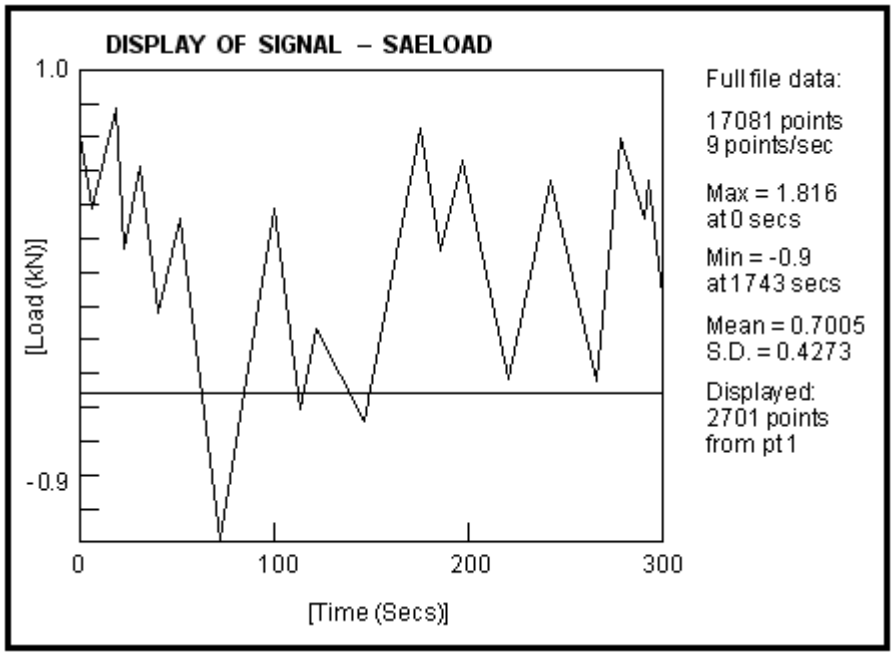


Figure 16-21 A Load Time Series

It is very important to ensure that these measured load histories are indeed representative of the service environment experienced by the component in terms of both amplitudes and sequence. If, for example, the main source of fatigue damage is due to occasional "overloads" such as the bump of a wheel on a pothole or the "cold-hot-cold" (startup-run-shutdown) cycle of a large engine, as opposed to the cyclic loads associated with continuous smooth operation, then large errors and misinterpretations can arise.

Definition of the S-N Curve

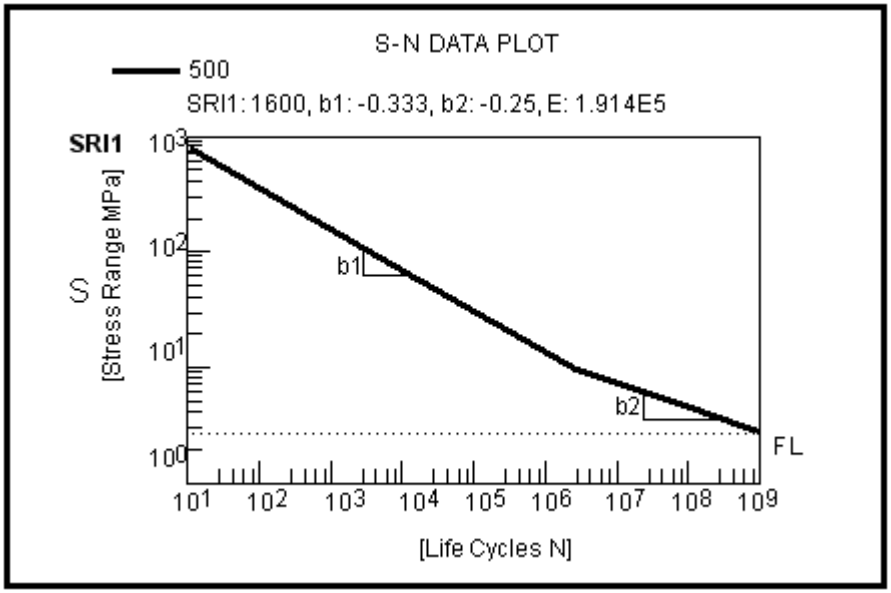


Figure 16-22 The S-N Fatigue Damage Curve

The basic equation for the S-N curve is:

$$\Delta S = SRI1(N_f)^{b1} \tag{16-27}$$

which may be written as:

$$log(\Delta S) = log(SRI1) + b_1 log(N_f) \tag{16-28}$$

ΔS		Nominal stress range
SRI1		Stress range intercept of the life line
SRI2		Stress range intercept of second life line
b_1		Slope of the first life line
b_2		Slope of the second life line

Note that in Figure 16-22 the slope of the life line changes, with the second slope, b_2, being shallower than the first. This "knee" or fatigue transition point, as it is sometimes called, punctuates the change in slope and typically occurs at about 10^7 cycles. The second slope can have any value between the first slope, b_1, and zero. In the case that it is zero, the transition point represents a fatigue limit below which no further damage will accumulate.

The equation of the second life line will of course be defined by:

$$\Delta S = SR12(N_f)^{b2} \tag{16-29}$$

If b_2 is non-zero, a stress-based fatigue limit, FL, may be defined to cutoff any further fatigue damage; all three features can be distinguished in Figure 16-22.

Because of the complexity and often the variability of the test pieces used, together with the difficulties of controlling local stresses and strains, the degree of scatter in the measured lives can be quite high. It is usual, therefore, to apply some statistical methods to account for this variability. MSC Fatigue takes this scatter into account through a consideration of the standard error of log (N_f).

The standard error of log (N_f) is calculated in the following way:

$$SE = SD_x\sqrt{(1 - r^2)} \tag{16-30}$$

SE		Standard error of log(N_f)
SD_x		Standard deviation of log(N_f) from the regression analysis
r		Correlation coefficient from the regression analysis

By the use of this statistic, a whole family of life curves, centered around the “mean life line”, may be constructed. It is usual to consider three standard deviations on either side of the mean. The lower limit represents 99.9% confidence that, for a given stress range, failure will not occur after less cycles than those which correspond to that stress. The mean life line provides 50% confidence and in the case of the upper line only 0.15% confidence that the component will survive.

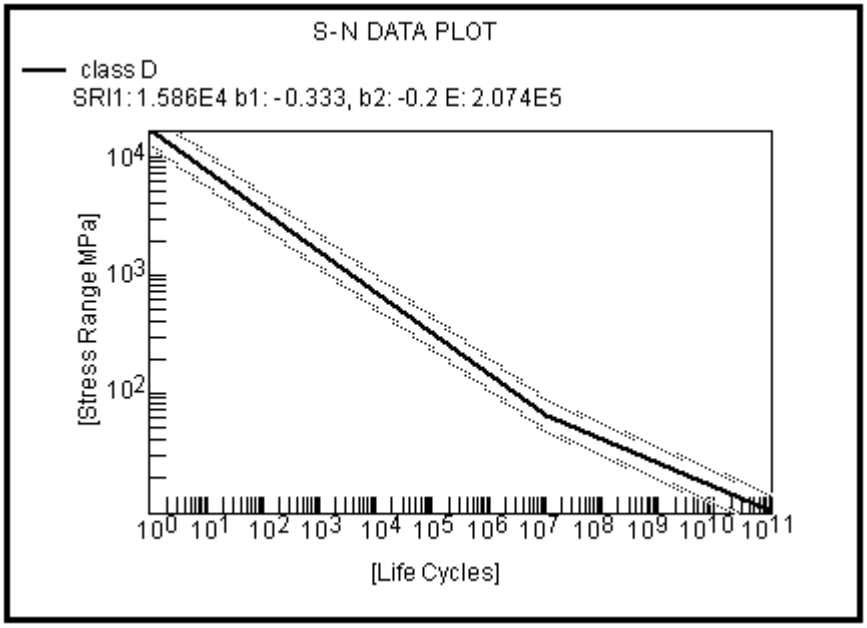


Figure 16-23 The 3-SD Confidence

Analytical Procedure

Since the stress-life method in MSC Fatigue makes no attempt to model localized plasticity, the analytical procedure used is relatively straightforward and may be summarized as follows:

1. By means of linear static FEA, derive the local stress time history from the load time histories. This includes superpositioning of multiple FEA/load time history load cases (or use stress time history directly from linear transient or forced vibration FE analysis). However, it is important to ensure that the S-N data applies to the situation being modeled; most S-N curves are for nominal stress, not local stress.
2. Extract the fatigue cycles in the local stress time history by means of the rainflow algorithm.
3. Assess the damage contribution of each cycle by referring to the selected damage curve.
4. Linearly sum the damage associated with each cycle by using Miner's rule.

Account for Nominal Mean Stress

Nominal S-N curves can be either structure, component or specimen based and are used to provide damage data for complex situations which otherwise could not be characterized. This gives rise to different procedures for accessing the S-N curve, depending on the situation being considered. Indeed, the British Standards Institution has tried to formalize some of these procedures in the document BS5400 Pt. 10. The main differences between the various procedures lie in the treatment of mean stresses and the damage contribution of cycles with "small" ranges. The total life analyzer in MSC Fatigue offers five different procedures:

1. Ignore the Effect of Mean Stress

   In this mode of operation, the stress-life analyzers in MSC Fatigue estimate the damage for each cycle and takes no account of its mean stress. If the stress range of the cycle is below the fatigue limit then no damage will be accumulated irrespective of the relative magnitude of any other cycles which may be present.

2. Account for Mean Stress According to BS5400 Pt.10, Welded

   In this mode of operation, the stress-life analyzers in MSC Fatigue estimate the damage for each cycle and do not take into account the effects of mean stress.

   Specific S-N curves are used for the BS5400 welded analysis and these are referred to as weld classes. These damage curves are known to MSC Fatigue by the following names:

   CLASSB, CLASSC, CLASSD, CLASSE, CLASSF,

   CLASSF2, CLASSG, CLASSW, CLASSS and CLASST

3. Account for Mean Stress According to BS5400 Pt.10, Non-Welded

   In this mode of operation, the stress-life analyzers in MSC Fatigue estimate the damage for each cycle and takes account of the effects of mean stress by reducing compressive stress amplitudes to 60% of their measured values.

   Specific S-N curves are used for the BS5400 non-welded analysis. These are referred to as classes and are known to MSC Fatigue by the following names:

   CLASSB, CLASSC, CLASSD, CLASSE, CLASSF,

   CLASSF2, CLASSG, CLASSW, CLASSS and CLASST

   These S-N curves are in fact the same as those used in the welded analysis. The difference in the non-welded analysis lies in the treatment of compressive stresses.

4. Account for Mean Stress According to Goodman

   In this mode of operation, the stress-life analyzers in MSC Fatigue estimate the damage for each cycle and account for the effects of mean stress by reducing the allowable applied stress amplitudes in a linear way for both positive and negative mean stresses. This procedure was first proposed by Goodman and can be summarized as follows:

$$S_a = S_o\left\{1 - \left(\frac{S_m}{S_u}\right)\right\} \tag{16-31}$$

S_a		The allowable stress amplitude.
S_o		The allowable stress amplitude at zero mean stress.
S_m		The mean stress.
S_u		The ultimate tensile stress.

If the stress range of the cycle is below the fatigue limit, as defined by the FL parameter, then no damage will be accumulated irrespective of the relative magnitude of any other cycles which may be present.

Account for Mean Stress According to Gerber

In this mode of operation, the stress-life analyzers in MSC Fatigue estimate the damage for each cycle and account for the effects of mean stress by reducing the allowable applied stress amplitudes in a quadratic way for both positive and negative mean stresses. This procedure was first proposed by Gerber and can be summarized as follows:

$$S_a = S_o\left\{1 - \left(\frac{S_m}{S_u}\right)^2\right\} \tag{16-32}$$

S_a		The allowable stress amplitude.
S_o		The allowable stress amplitude at zero mean stress.
S_m		The mean stress.
S_u		The ultimate tensile stress.

If the stress range of the cycle is below the fatigue limit, as defined by the FL parameter, then no damage will be accumulated irrespective of the relative magnitude of any other cycles which may be present.

Linear Damage Summation

A constant amplitude, S-N curve represents a set of tests at constant stress range (ΔS) together with associated lives. Operation at a stress range (ΔS_1) will result in failure in say N_1 cycles. Operation at the same stress range for a number of cycles less than N_1, N_j say, will result in a smaller fraction of damage, D_j, which is often referred to as a partial damage. Operation over a spectrum of different stress ranges, results in a partial damage contribution Di from each stress cycle, ΔS_i. Failure is then predicted when the sum of these partial damage fractions reaches unity so that:

$$D_1 + D_2 + \ldots + D_{i-1} + D_i \geq 1 \tag{16-33}$$

The Palmgren-Miner rule asserts that the partial damage at any stress range ΔS_1, is linearly proportional to the ratio of the number of cycles of operation to the total number of cycles that would produce failure at that stress level, $\Sigma\ (F(n_1,N_1))$. For example:

$$D_1 = \frac{n_1}{n_2} \tag{16-34}$$

Failure is predicted then if:

$$\frac{n_1}{N1} + \frac{n_2}{N_2} + \ldots + \frac{n_i}{N_i} + \frac{n_{i-1}}{N_i} \tag{16-35}$$

or

$$\Sigma\left(\frac{n_1}{N_1}\right) \geq 1 \tag{16-36}$$

The above equations represent a statement of the linear damage rules used by both the stress-life and local strain fatigue life processors in MSC Fatigue. Experience shows that linear damage summation is somewhat of an over simplification of reality. The most important shortcoming is that no account is made of the

sequence in which strain levels are experienced, and damage is assumed to accumulate at the same rate for a given strain level regardless of prehistory. In particular, it appears that large strain amplitudes which precede smaller ones, cause the smaller cycles to become more damaging than expected. The net result is that in the first instance, the Miner's sum Σ (F(n,N)) is measured to be less than 1 and in the latter case it becomes greater than 1. Since most service environments involve quasi-random loading sequences, the use of the Palmgren-Miner linear damage rule summing to a constant of 1 is mostly satisfactory.

Important:	BS 5400 Pt 10 attempts to cope with these effects by asserting that if there are no cycles with a stress range greater than the stress corresponding to 1 x 107 cycles, then no damage will be accumulated. If, on the other hand, even one cycle exceeds this stress, then damage will be accumulated for ALL other cycles. This correction takes into account the observed behavior when a crack growing cycle occurs, subsequent previously non-damaging cycles start to increment the crack size (i.e., cause damage). This is particularly true for welded structures which are always already cracked. The choice of the 1E7 cycles point reflects the fact that the stress at this point is close to the threshold for crack advance. If, on the other hand, even one cycle exceeds this stress, then damage will be accumulated for ALL other cycles.

Factor of Safety Analysis

When fatigue was first recognized in the mid 1800s, the solution to fatigue problems was to find a load below which fatigue failures didn't occur. This worked well as far as eliminating fatigue failures, but it made structures heavy and expensive. The direction of fatigue research since then has been to allow engineers to make products lighter and more highly optimized. The safe life and fail-safe philosophies came from this recognition by engineers that product weight and cost must be reduced without increasing risk of unexpected failure. This was especially true for aircraft structures which would never leave the ground if they were designed for no cracks (i.e., no fatigue damage). In other words, the fatigue life of some products must be finite.

However, there is still a class of structures where fatigue lives are essentially infinite or at least beyond any available fatigue test data. Typically these include components from high speed rotating machines such as internal combustion engines and gas turbines. In order to assess the fatigue life of "infinite life" structures, it has been found most useful if a measure is obtained of the difference between the working stress and the fatigue limit stress (a stress below which no fatigue damage is incurred). Often this is expressed as a ratio of fatigue limit/working stress and this ratio is known as the "factor of safety." This simple methodology is ideal for hand calculations using idealized loading but for realistic variable amplitude loading, a more realistic approach is required.

Stress Based Calculation

Taking the basic factor of safety concept as a ratio of the applied stress to the fatigue stress, it is possible to introduce a mean stress correction based on the Goodman or Gerber rules.

Goodman Based Factor

$$\text{Factor of Safety} = \frac{\sigma}{\sigma_a\left[1-\left(\frac{\sigma_m}{\sigma_u}\right)\right]} \tag{16-37}$$

This equation may be modified to include corrections for surface finish (C_{surf}) and size effects (C_{size}). These factors take values less than 1.0. An additional notch sensitivity factor may be included (K_f) which is a multiplier of the applied stress amplitude.

$$\text{Factor of Safety} = \frac{C_{surf}\ C_{size}\ \sigma_n}{k_f \sigma_a \left[1 - \left(\frac{\sigma_m}{\sigma_u}\right)\right]} \tag{16-38}$$

σ_n		is the stress at life n
σ_a		is the stress amplitude of the cycle
σ_m		is the mean stress of the cycle
σ_u		is the ultimate strength of the material

The normal way to select S_n is to choose a target life and to obtain the corresponding stress from an S-N curve, extrapolating the curve to the required life if necessary.

The factor of safety can be illustrated graphically on the Goodman diagram as shown in Figure 16-24.

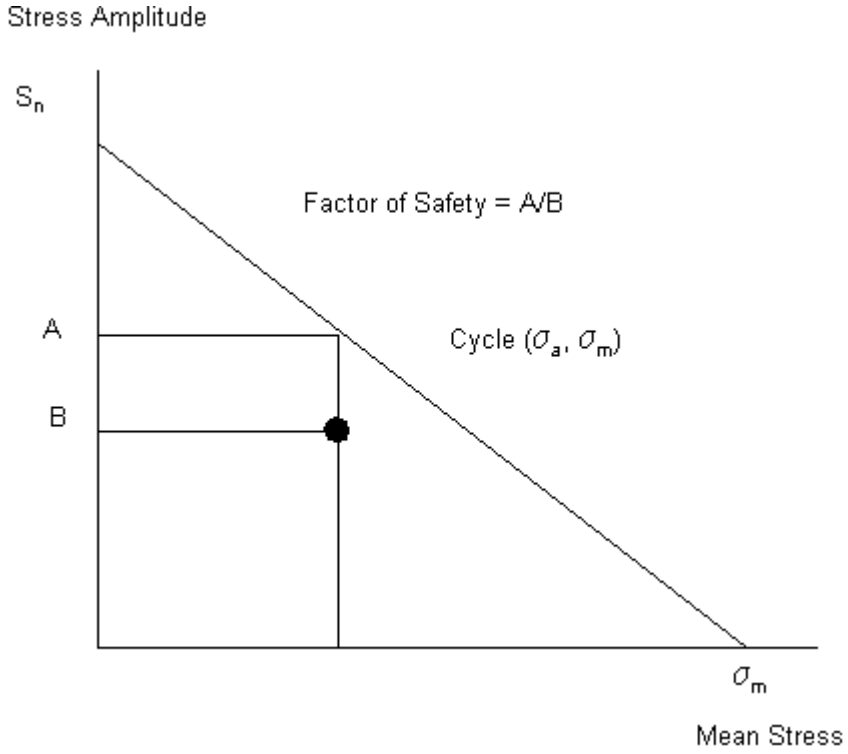


Figure 16-24 Factor of Safety Illustrated on the Goodman Diagram

Gerber Based Factor

This is the same as the Goodman except the Gerber mean stress correction is applied.

$$\text{Factor of Safety} = \frac{S_n}{\sigma_a\left[1-\left(\frac{\sigma_m}{\sigma_u}\right)^2\right]} \tag{16-39}$$

By also calculating the factor of safety for each cycle in a random loading history, the stress-based factor of safety may be used with realistic loading in a meaningful way.

MSC Fatigue calculates factors of safety for all cycles in a time history at each location (node or element) and reports the lowest value computed. The correction factors for surface treatment and finish are obtained from the Lipson (9) empirical data and are related to the ultimate strength of the material.

Life-Based Calculation

The life based calculation is in fact a calculation to find the magnitude of stress scaling factor which will cause failure in a specific design life. Since the scaling factor is applied to all cycles it is an overall factor of safety. This approach is far better suited to calculating meaningful factors of safety for random loading.

This approach to calculating a scaling factor is used in MSC Fatigue in the Factor of Safety module for multi-location computation and also in Design Optimization where the scaling parameter optimization calculates the value of a factor at one location (node/element).

Other Factors Influencing Safety

The calculations described above compute a stress scaling factor either using a stress based or life based methodology. It should be remembered that there are other factors contributing to the life of the product. For example, a change in the material may reduce the factor of safety. This may be explored more fully using the Design Optimization tools in MSC Fatigue or by submitting a new job with edited input data.

Crack Initiation/Strain-Life (ε-N) Analysis

The nominal stress approach has been used extensively in the study of premature failures of components subjected to fluctuating loads. Traditionally, the magnitude of the observed cyclic stresses were observed to be less than the tensile elastic limit and the lives long (i.e., greater than about 10^5 cycles). This pattern of behavior has become known as *high-cycle fatigue*.

As duty cycles have became more severe and components more complicated, another pattern of fatigue behavior has emerged. In this regime, the cyclic loads are relatively large and have significant amounts of plastic deformation associated with them together with relatively short lives. This type of behavior has been commonly referred to as *low-cycle fatigue* or more recently strain-controlled fatigue. The transition from low-cycle to high-cycle fatigue behavior generally occurs in the range 10^4 to 10^5 cycles. A more precise definition of this transition will be developed later.

The analytical procedure evolved to deal with strain-controlled fatigue is called the strain-life, local stress-strain or critical location approach. In MSC Fatigue, it is referred to as the *crack initiation* method for reasons that will become clear later.

The Microscopic Aspects of Fatigue Failure

It is almost universally agreed that fatigue failures usually start at the surface of a fatigue specimen or component. This is true whether the test is made in a rotating-beam machine where the maximum stress is always at the surface, or in a push-pull machine which gives a simple tensile-compressive stress cycle. Furthermore, fatigue failures start as microscopic cracks and accordingly are very sensitive to even minute stress raisers. It is quite apparent from these considerations, that a fatigue specimen will give results which are representative of the metal tested only if its surface is free of defects. Tool or grinding marks left on the surface make the formation of fatigue cracks easier and may result in low apparent values of fatigue limit or strength.

Early research into the mechanisms of fatigue clearly demonstrated that the failure process is linked to reversed plastic flow (i.e., the forward and backward motions of dislocations). (Dislocations are imperfections

in the crystal lattice of a metal and are responsible for nearly all aspects of plastic deformation along the slip planes of metallic crystals.) Under cyclic loading, the direction of the strain is reversed over and over again; the slip lines that appear on the surface reflect this fact.

When the strain is monotonic, the slip steps that appear on a crystal surface have a relatively simple topology, see Figure 16-25(a). On the other hand, under cyclic loading, the slip bands tend to group into packets or striations. The surface topology of these striations is more complex and is indicated schematically in Figure 16-25(b). Note that both ridges and crevices tend to be formed. There is good evidence that the crevices are closely associated with the initiation of cracks. Whether or not crevices are formed in a particular specimen or a specific grain of a particular specimen is largely a function of the crystal orientation. If the shear direction in the striation is nearly normal to the surface, crevice formation will be favored. However, if the slip direction is parallel to the surface, crevices are not normally formed. It is possible, even in this case, for damage to occur inside a striation, since pores or holes can also open up inside the slip-band packet.

A further consequence of these repeated dislocation movements is that small localized deformations called *extrusions* may occur in the slip bands. An extrusion is a small ribbon of material which is apparently squeezed out from the surface of a slip band. The inverse of extrusions, which are narrow crevices called *intrusions* have also been observed, Figure 16-25(b). Typically these surface disturbances are approximately 1 to 10 microns in height and constitute embryonic cracks.

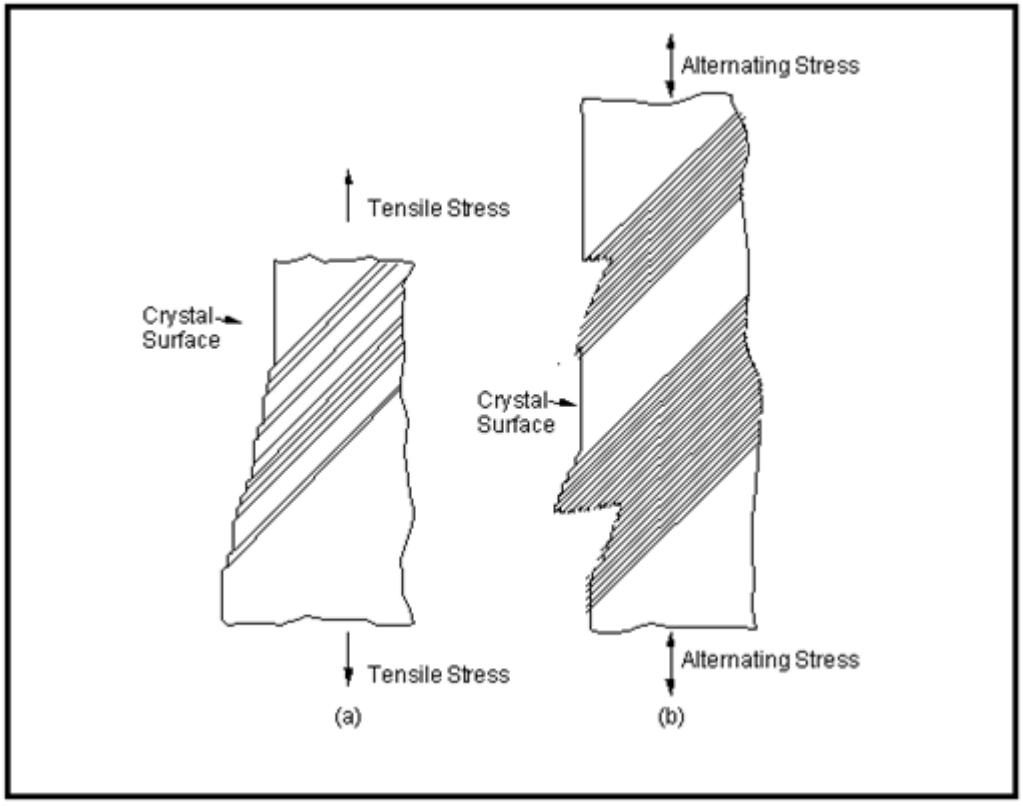


Figure 16-25 Surface Contours Where Slip Bands Intersect a Surface

Another way of describing the above observations is that *without some plastic deformation there can be no fatigue damage*! Given this assertion, how could it be that the nominal stress approach, which specifically overlooks plasticity, could have been so successfully applied for so many years?

The reason is that, in the high-cycle regime loading, levels are low and so stress and strain are almost linearly related. Under these circumstances, load and strain-controlled cycling are equivalent. Consequently, in the area of the endurance limit, load-controlled rotating beam fatigue data based on notionally elastic nominal stresses can be used to describe adequately fatigue behavior. However, as the load levels become larger, the method breaks down and a more generalized, strain-based approach, which accounts for plasticity, must be adopted. Indeed, it should be clear by now that the nominal stress approach is in reality a special case of strain-based analysis, as the plastic strain tends to zero.

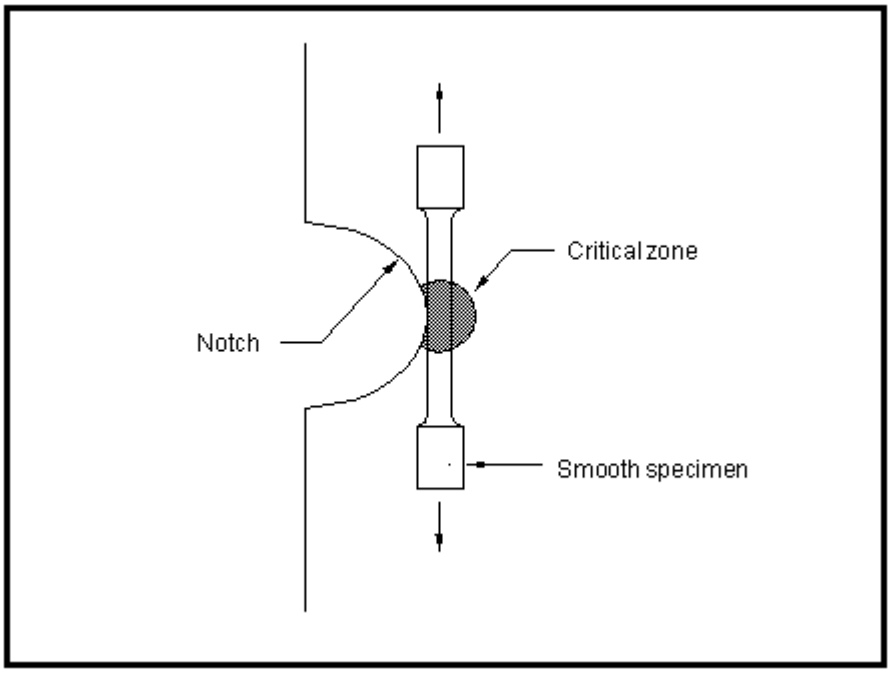


Figure 16-26 Similitude Between a Laboratory Specimen and the Root of a Notch

The Strain-Life Methodology

The strain-life methodology is based on the observation that in many critical locations such as notches the material response to cyclic loading is strain rather than load controlled. This arises from the fact that whilst most components are designed to confine nominal stresses to the elastic region, stress concentrations such as notches often cause plastic deformation to occur locally. The material surrounding the plastically deformed zone remains fully elastic and so the deformation at the notch root is considered to be strain controlled.

The strain-life method assumes similitude between the material in a smooth specimen tested under strain control and the material at the root of a notch Figure 16-26. For a given loading sequence, the fatigue damage in the specimen and the notch root are considered to be similar and so their lives will also be similar.

The cyclic stress-strain response of the material at the critical location is determined by characterizing the behavior of smooth specimens subjected to similar loading, the *local stress-strain history*. The local stress-strain history must be determined, either by analytical or experimental means. Stress analysis procedures such as finite element modelling, or experimental strain measurements are usually required.

In performing smooth specimen tests which characterize fatigue performance, it must be recognized that fundamental material properties are being measured which are independent of component geometry. Phenomena such as cyclic hardening or softening, cycle dependent stress relaxation, and loading sequence effects are all taken into consideration.

Monotonic Stress-Strain Behavior

The engineering tension test is widely used to provide basic information on the strength of materials and also as an acceptance test for the specification of alloys. In this test, a cylindrical specimen is subjected to a continually rising, *monotonic*, uniaxial load while simultaneously measuring its elongation.

The so-called engineering stress-strain curve can be constructed from the measured values of load and elongation, Figure 16-27.

The stress plotted on the engineering stress-strain curve is the average longitudinal stress in the test specimen and is obtained from:

$$S = \frac{P}{A_o} \tag{16-40}$$

S		the nominal or engineering stress
P		the applied load
A_o		the original, unloaded, cross-sectional area

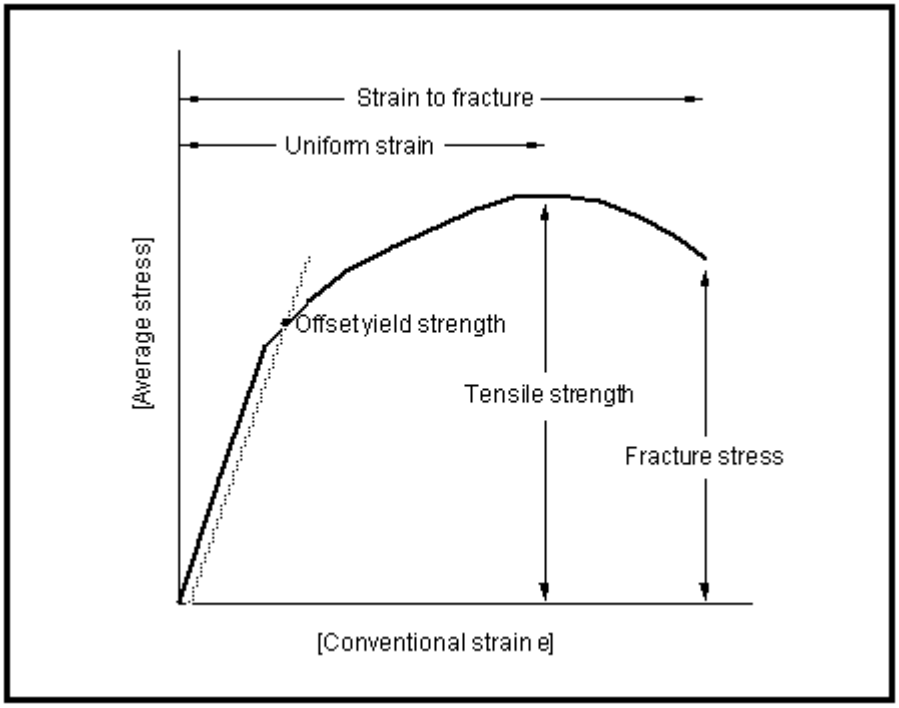


Figure 16-27 The Engineering Stress-Strain Curve

The engineering strain is the average linear strain obtained from:

$$e = \frac{\Delta l}{l_o} = \frac{(l - l_o)}{l_o} \tag{16-41}$$

e		the engineering strain
l_o		the unstrained specimen gauge length
l		the strained gauge length

The shape and magnitude of the engineering stress-strain curve goes a long way towards characterizing any particular material. The curve itself may be considered to be made up of three distinct regions. Uniform straining where stress and strain are linearly related, uniform straining where stress and strain are no longer linearly related, and a nonuniformly straining region where plastic instability, *necking*, sets in.

Certain parameters may be measured directly from the engineering stress strain curve. For example, yield strength, tensile strength, percentage elongation, and reduction in area.

The yield stress is used to define the limit of elastic behavior. In some steels, it may be easily defined as a sharp transition between elastic and elastic-plastic behavior. More commonly, it is defined, as in Figure 16-27, by a fixed offset (i.e., the stress corresponding to 0.2% plastic strain for example):

$$S_{0.2} = \frac{P_{0.2}}{A_o} \tag{16-42}$$

The tensile strength, sometimes referred to as the ultimate tensile strength (UTS or R_m), corresponds to the stress resulting from the maximum load carried by the specimen. It is calculated by dividing the maximum load by the original cross-sectional area of the specimen:

$$S_u = \frac{P_{max}}{A_o} \tag{16-43}$$

The tensile strength defines the limit of uniform plastic deformation within the tensile test. It is not actually a fundamental material property, but rather it is a function of the test itself. However, it has been and remains the most widely quoted material property, indeed entire national standards are based around it.

Both percentage elongation and reduction in area are measures of ductility, that is the extent to which a metal will deform prior to fracture. Typically, these properties are measured by putting a fractured tensile specimen back together and measuring its final length, l_f, and final area, A_f thence:

Percent elongation, EL% $= (l_f - l_o) / l_o \times 100$

Percent reduction in area, RA% $= (A_o - A_f) / A_o \times 100$

True Stress and Strain

All the parameters derived above have been based on the original dimensions of the test specimen, and as such are normally referred to as *engineering* stress and strain parameters and denoted by S and e respectively. A true

stress is defined as the load divided by the instantaneous area, P / A, and since the specimen is stretching and its cross-sectional area decreasing, the true stress is always *larger* than the engineering stress.

Up to the onset of necking, a true strain is based on instantaneous gauge length, and is defined as the integral from the original length, l_o, to the instantaneous length l, of 1 / l, otherwise:

$$\varepsilon = \ln\left(\frac{l}{l_o}\right) \tag{16-44}$$

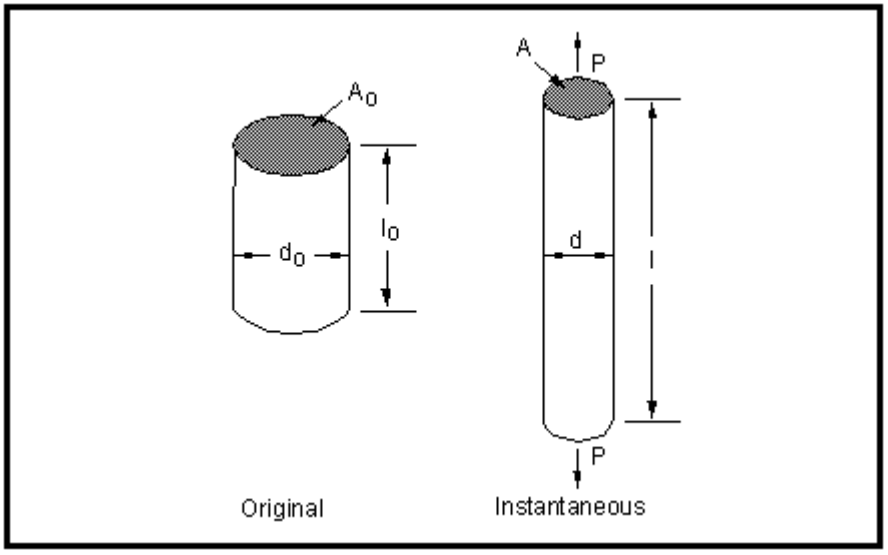


Figure 16-28 Original and Instantaneous Specimen Geometries

When engineering stress and strain are corrected to their true values and the data replotted, the nature of the stress strain diagram changes, Figure 16-29. True stress and strain may be related to engineering stress and strain (i.e., true stress, up to necking).

$$\sigma = \frac{P}{A_o}(l + e) = S(l + e) \tag{16-45}$$

S		the engineering stress
e		the engineering strain

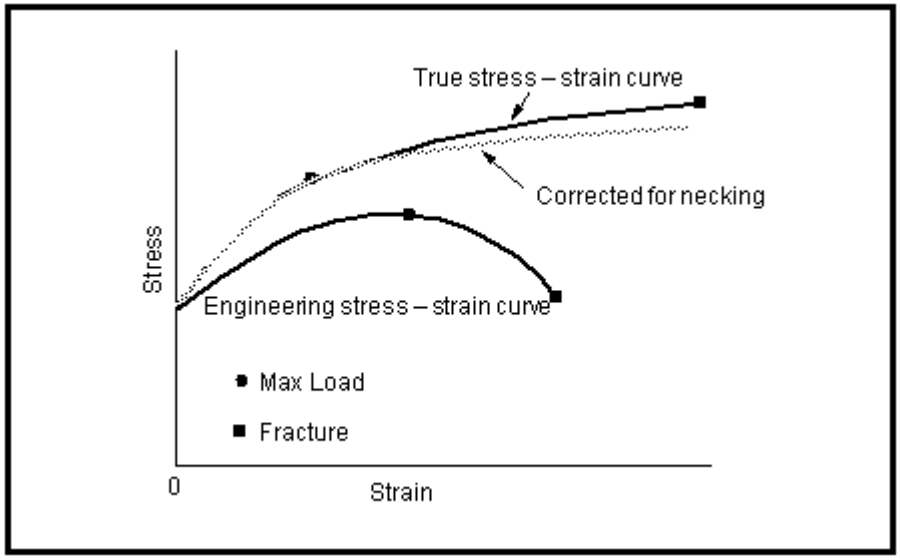


Figure 16-29 True Stress/True Strain Diagram

Beyond necking, true stress must be calculated from actual measurements of load and instantaneous cross-sectional area from:

$$\sigma = \frac{P}{A} \tag{16-46}$$

True strain, up to necking:

$$\varepsilon = ln(1 + e) \tag{16-47}$$

Beyond necking, true strain can be calculated from:

$$\varepsilon = 2ln\left(\frac{D_o}{D}\right) \tag{16-48}$$

D_o		the original diameter
D		the instantaneous diameter

The fracture stress corresponds to the highest stress carried by the specimen and is calculated by dividing the fracture load, P_f, by the final area, A_f

Stress-Strain Relationships

Figure 16-30 below illustrates various regions of a true stress/true strain curve. The curve has been generated by monotonically loading a specimen up to point B and then unloading down to point C and then reapplying the load up point D.

The portion of the curve defined by O-A represents the elastic, reversible, behavior of the material where stress and strain are linearly related.

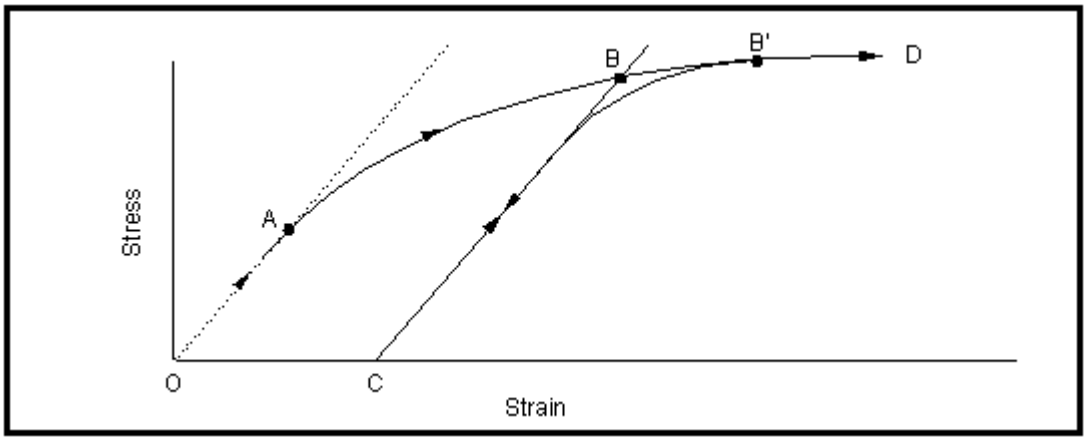


Figure 16-30 Various Regions of the True Stress/True Strain Curve

The constant of proportionality is called the *modulus of elasticity* or sometimes the *stiffness*. In this region:

$$E = \frac{\sigma}{\varepsilon} \tag{16-49}$$

and any elastic strain, ε_e may be written as:

$$\varepsilon_e = \frac{\sigma}{E} \tag{16-50}$$

The region B-D defines the elastic-plastic part of the true stress/true strain curve, and for most metallic materials, this region can be represented by a simple power law of the form:

$$\sigma = K(\varepsilon_p)^n \tag{16-51}$$

K		the constant of proportionality is known as the strength coefficient.
n		known as the work hardening coefficient.
ε_p		the plastic strain.

The plastic strain, ε_p, can now be written as:

$$\varepsilon_p = \left(\frac{\sigma}{K}\right)^{\frac{1}{n}} \tag{16-52}$$

Reversing the Loading Direction

If during the course of a monotonic tension test, point B in Figure 16-30 for example, the loading direction is reversed and the specimen unloaded, the stress-strain relationship will follow the line defined by B-C. The slope to the line will be the same as O-A, that is to say, equivalent to the modulus of elasticity, E. Indeed, this is the most accurate way of measuring E during the course of a tension test.

The distance O-C represents a permanent extension of the specimen as a result of loading it up to point B. This extension, the *plastic strain*, is sometimes referred to as the permanent set.

The distance C-C' represents the recoverable extension of the specimen as a result of loading it up to point B, and is known as the *elastic strain*.

The total strain induced into the specimen by loading it up to point B, therefore, is the sum of the elastic and plastic components:

$$\varepsilon_t = \varepsilon_e + \varepsilon_p \tag{16-53}$$

and utilizing the expressions derived above the total strain can be written as:

$$\varepsilon_t = \frac{\sigma}{E} + \left(\frac{\sigma}{K}\right)^{\frac{1}{n}} \tag{16-54}$$

which represents a good description of the true stress/true strain curve.

At fracture, point D in Figure 16-30, two further parameters can be defined — the *true fracture strength* and the *true fracture ductility*. True fracture strength, σ_f , is the true stress at final fracture and is defined as:

$$\sigma_f = \frac{P_f}{A_f} \tag{16-55}$$

The true fracture ductility, ε_f, is the true strain at final fracture and is defined in terms of the initial area and the area at fracture:

$$\varepsilon_f = In\left(\frac{A_o}{A_f}\right) = In\left(\frac{1}{(1 - RA)}\right) \tag{16-56}$$

where the reduction in area RA is defined as:

$$RA = \frac{(A_o - A_f)}{A_f} \tag{16-57}$$

Reversing the Loading Direction, Again

If, from point C in Figure 16-30 above, the loading direction is again reversed (i.e., the specimen is loaded back towards point B and beyond), the true stress/true strain curve will follow the trace C-B'-D. As in the case of unloading, most of the trace will be a straight line with slope equal to the modulus of elasticity, E. The interesting thing to note, however, is that the new stress-strain curve C-B'-D rejoins the original stress-strain curve at B'. It is almost as though the material remembered the nature of the original curve and continued to deform along it. Conversely, once beyond B' the material has apparently totally forgotten the excursion B-C-B' and goes on to fracture at D as though it had never happened! The excursion B-C-B' is, therefore, only an interruption in the primary loading process O-A-B-D, and in reality, represents a fatigue cycle. These observations are central to the local strain life methodology and we shall return to them later.

The Bauschinger Effect

Since 1886, Bauschinger was carrying out and reporting experiments of the sort described above. In reality, he went further in that he did not stop the unloading process at the zero stress level, point C in Figure 16-30, but rather continued on down into full compression. Figure 16-31 illustrates what he found.

In Figure 16-31(a), a specimen of material is loaded in tension to beyond the yield stress to a maximum stress of σ_{max}, point A. At this point the loading direction is reversed and the specimen is unloaded from point A through the zero stress level and on into compression to a stress equal to $-\sigma_{max}$, point B. The interesting point to note is that the material appears to yield at a stress level before $-\sigma_y$ is reached (Figure 16-31(b)). This observation has nothing to do with the fact that the yield strength in compression might be different from that in tension. The whole experiment could be repeated, but this time loading in compression first by going to $-\sigma_{max}$. Yield would take place at $-\sigma_y$ but would again occur before σ_y was reached when the loading direction was reversed. Unsurprisingly, this behavior is known as the Bauschinger effect and its importance cannot be overstated.

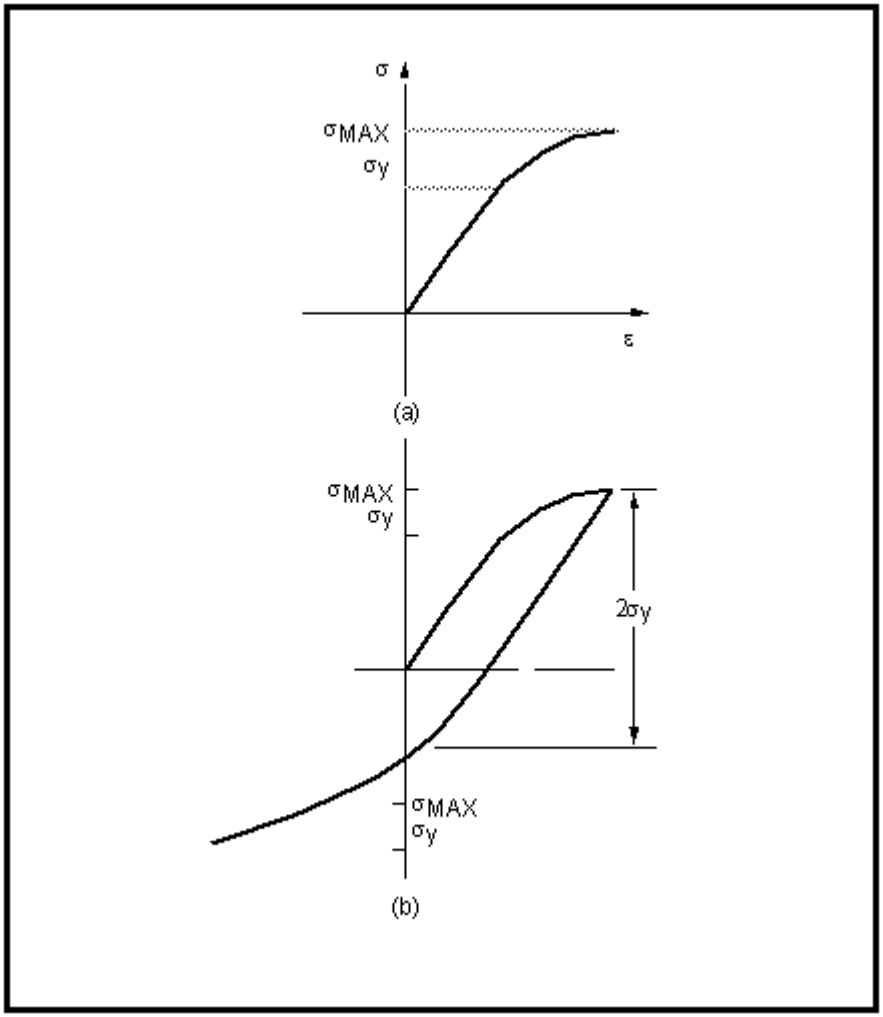


Figure 16-31 Reversed Loading Into Compression, the Bauschinger Effect

At this point, a valid question to ask would be: If reversed yield does not take place at $-\sigma_y$, then at what point does it take place? Figure 16-31(b) provides the necessary clue (i.e., at a stress level equal to $2\sigma_y$ from σ_{max}). It appears as though the stress-strain curve defined by A-B is geometrically twice the initial true stress/true strain curve O-A. This observation was first made by Masing, in 1926, and is today known as Massing's hypothesis.

Cyclic Stress-Strain Behavior

If the loading process started in Figure 16-31 is continued and at $-\sigma_{max}$, the loading direction is again reversed and the specimen is loaded back up to σ_{max}, then a complete loop will be defined (see Figure 16-32). The stress-strain loop illustrated in Figure 16-32 is called a hysteresis loop and defines, in stress-strain space, a single fatigue cycle. Typically such loops are generated in the laboratory under strain control; that is, by straining to specific strain limits. During the initial loading, the stress-strain response is according to the curve O-A-B.

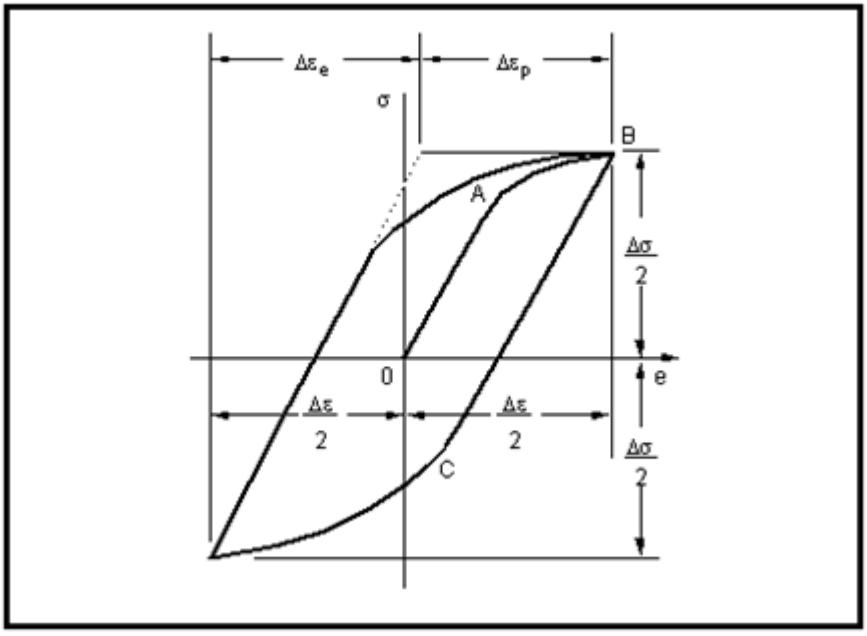


Figure 16-32 A Complete Stress-Strain Cycle, a Hysteresis Loop

On unloading, yielding begins in compression at point C due to the Bauschinger effect. In reloading, in tension the hysteresis loop develops. The dimensions of the loop are described by its total width, the total strain range $\Delta\varepsilon$, and its height, the total stress range $\Delta\sigma$.

The total strain range is made up of the elastic and plastic components:

$$\Delta\varepsilon_t = \Delta\varepsilon_e + \Delta\varepsilon_p \tag{16-58}$$

which may be written as:

$$\Delta\varepsilon_t = \frac{\Delta\sigma}{E} + \Delta\varepsilon_p \tag{16-59}$$

Cyclic Loading Under Strain Control

If a material is repeatedly cycled between fixed strain limits one of several things may happen depending on its nature and initial conditions of heat treatment.

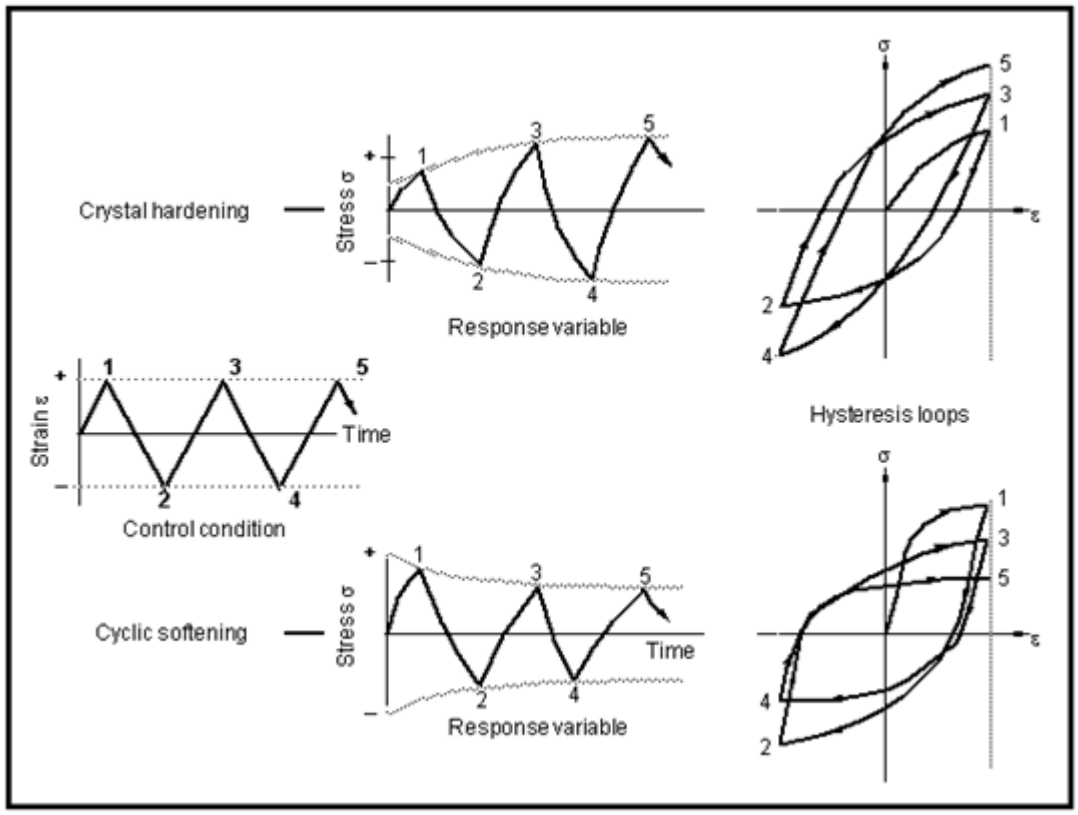


Figure 16-33 Cyclic Hardening and Softening Under Strain Control

The material could respond in any one of the following ways,

- it could cyclically harden
- it could cyclically soften
- it could remain stable
- it could soften or harden depending on strain range

Figure 16-33 illustrates the effects of cyclic softening and hardening where the first two hysteresis loops of two different materials are plotted. In both cases, the cycling of the specimen is between fixed-strain limits and the load is allowed to find its own level. It is clear that in the case of hardening, the maximum stress reached in each successive strain cycle increases with number of cycles and in the case of softening the maximum stress decreases with number of imposed cycles. This process does not continue indefinitely. In both cases, the stress will find a constant level and remain stable at that level until the first emergence of a fatigue crack.

The reason materials cyclically harden or soften is thought to be associated with dislocation substructures within the metal crystal lattice. Broadly speaking, soft materials such as aluminium, which contain low dislocation densities, tend to harden; whereas, hard materials, such as steels, tend to soften. As a rule of

thumb, the ratio of the ultimate tensile strength, S_u to the 0.2% proof strength, $S_{0.2}$ can be used to gauge whether a particular material will cyclically harden or soften, if

$$\frac{S_u}{S_{0.2}} > 1.4 \tag{16-60}$$

$$\frac{S_u}{S_{0.2}} > 1.2 \tag{16-61}$$

For ratios between 1.2 and 1.4, the estimate is difficult and anything may happen — hardening, softening or both.

Determining the Cyclic Stress-Strain Curve

As mentioned above, under strain control and early in life, the stress-strain response of most materials changes significantly with applied cyclic strains. However, after a relatively small number of cycles, typically no more than about 10% of total life, the hysteresis loops tend to stabilize so that the stress amplitude remains reasonably constant over the remaining portion of fatigue life. If the stress, strain coordinates of the tips from a number of stable hysteresis loops, of differing strain amplitudes, are plotted in stress-strain space, then the locus of these points defines the cyclic stress-strain curve, see Figure 16-34.

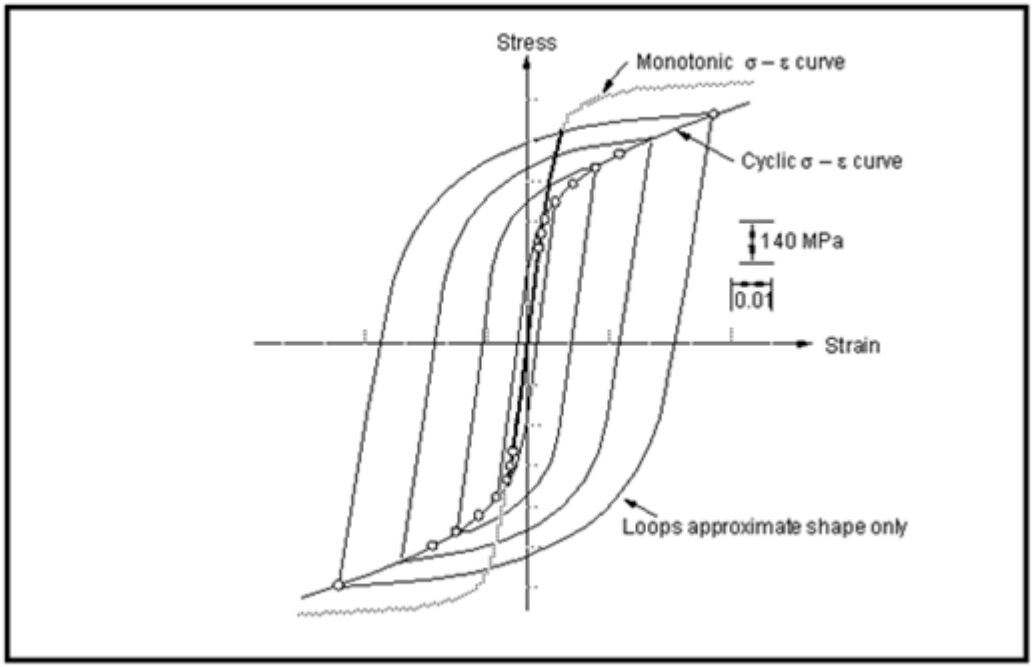


Figure 16-34 Definition of the Stable Cyclic Stress-Strain Curve

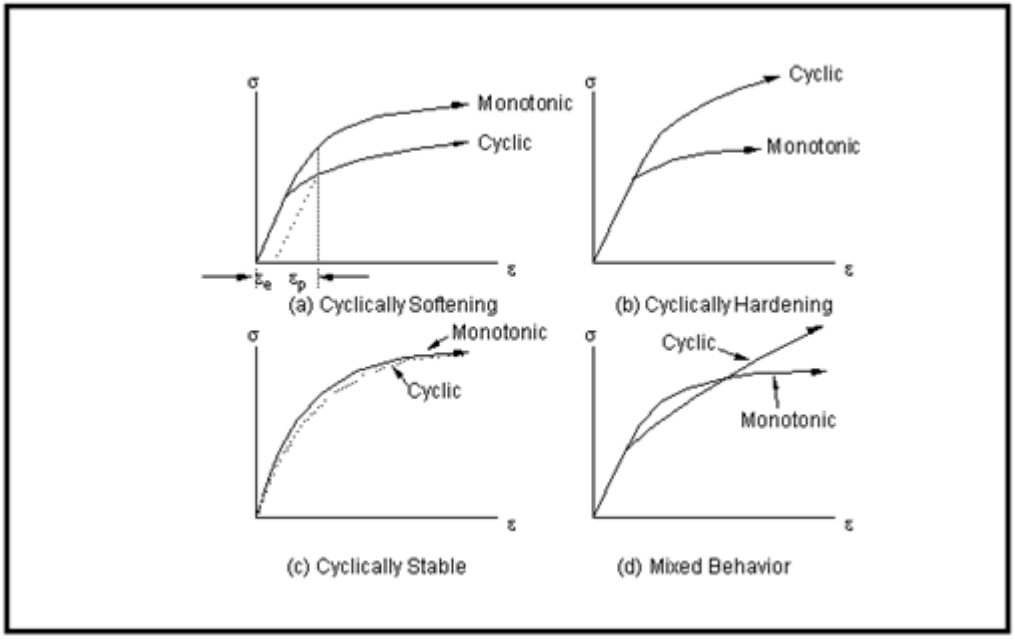


Figure 16-35 Comparison Between Cyclic and Monotonic Stress-Strain Curves

The cyclic stress-strain curve defines the relationship between stress and strain under cyclic loading conditions, just as the tensile true stress/true strain curve defines the conditions under monotonic loading. The cyclic stress-strain curve can be compared directly with the monotonic stress-strain curve to quantitatively assess the cyclically induced changes in material behavior.

Note that, in Figure 16-35(a), when a material cyclically softens the cyclic yield strength is considerably lower than the monotonic value. Using monotonic properties in cyclic applications can result in the prediction of fully-elastic strains when in reality considerable plastic strains are being generated.

It is also interesting to note that whilst stabilization of hysteresis loops can consume 10-20% of total life in a constant amplitude test, a single large overload *in service* can produce an immediate change from the monotonic curve to the cyclic. It is a sobering thought that the assembly of or using a machine or component could cause an instant reduction of yield strength of up to 50%.

Figure 16-35(d) illustrates a common behavior for low strength steels, whereby the material both hardens and softens depending on the strain amplitude. This phenomena is often known as mixed mode behavior.

As with the monotonic stress-strain curve, the cyclic stress-strain curve consists of both elastic and plastic strains:

$$\varepsilon_t = \varepsilon_e + \varepsilon_p \tag{16-62}$$

and can be fully characterized by an expression equivalent to that developed for the monotonic stress-strain curve. However, in this case, primes appended to the values of n and K to distinguish them from their monotonic counterparts:

$$\varepsilon_t = \frac{\sigma}{E} + \left(\frac{\sigma}{K'}\right)^{\frac{1}{n'}} \tag{16-63}$$

K'		called the cyclic strength coefficient.
n'		called the cyclic strain hardening exponent.

Typically, K' takes values in the range 1000 to 3000 MPa and n' varies between about 0.1 to 0.2. In general, metals with a high monotonic strain hardening exponent, n, will harden whilst those with a low monotonic strain hardening exponent will cyclically soften. Table 16-10 compares the cyclic stress-strain behavior of a number of well-known alloys.

Table 16-10 Cyclic Stress-Strain Behavior of Some Common Alloys

Material	Heat Treatment	0.2% Yield mono/cyclic	n/n' mono/cyclic	Cyclic Behavior
OFHCCopper	Annealed	20/138	0.40/0.15	Hardens
	Partial anneal	255/200	0.13/0.16	Stable
	Cold worked	345/235	0.10/0.12	Softens
Aluminium Alloys	T4	305/450	0.20/0.11	Hardens
2024	T6	470/520	0.11/0.11	Hardens
7075				
Titanium Alloy	Annealed	1000/790	0.078/0.14	Mixed
Ti-8A1-1Mo-1				
Steels	Q+T 1206 UTS	1170/760	0.066/0.14	Softens
SAE4340	Q+T 2050 UTS	1860/1720	0.071/0.14	Stable
SAE 1045	Q+T 1725 UTS	1690/1275	0.047/0.12	Softens
	Q+T 1550 UTS	1517/965	0.041/0.15	Softens
	Q+T 1345 UTS	1275/758	0.044/0.17	Softens
SAE 4142	Q+T 1930 UTS	1690/1725	0.092/0.13	Stable
	Q+T 1640 UTS	1725/1345	0.048/0.12	Softens
	Q+T 1550 UTS	1585/1070	0.04/0.17	Softens
	Q+T 1310 UTS	1380/825	0.051/0.18	Softens

Cyclic Yield and the Fatigue Limit

The close association between plasticity and fatigue damage has already been mentioned. This point can be further illustrated through a consideration of the cyclic stress-strain properties of some typical steels, see Table 16-11.

Table 16-11 Typical Monotonic vs. Cycle Material Properties

Monotonic Properties		
Material	Yield Stress (MPa)	Ultimate Tensile Stress (MPa)
Low strength	350	540
Medium strength	650	790
High strength	1090	1160
Cyclic Properties		
Material	Strength coefficient K', MPa	Hardening exponent n'
Low strength	1340	0.226
Medium strength	1062	0.123
High strength	1620	0.112

From the cyclic properties, it is possible to use the equation for the cyclic stress-strain curve,

$$\varepsilon_p = \left(\frac{\sigma}{K'}\right)^{\frac{1}{n'}} \tag{16-64}$$

$$\sigma = {}_nK'\varepsilon_p^{n'} \tag{16-65}$$

to calculate say the 0.02% cyclic yield stress (i.e., the stress corresponding to 0.02% plastic strain). Note that this is *one tenth* the amount of plastic strain usually used to determine an offset yield stress. Broadly speaking, this stress represents a limit below which the level of plasticity is sufficiently low to ensure little or no fatigue damage (i.e., a *fatigue limit*). Values of the calculated cyclic yield together with the ratio of cyclic yield to ultimate tensile strength are tabulated below for the three steels, see Table 16-12.

Table 16-12 Cyclic Yield/UTS Ratio

Material	Cyclic Yield (MPa)	Cyclic Yield / UTS
Low strength	195	0.36
Medium strength	372	0.47
High strength	624	0.53

It is interesting to note that the ratios of cyclic yield to UTS are in the region of 0.5. This is consistent with the observation from the nominal stress approach that the endurance limit for steels is about one *half* the UTS and also supports the view that endurance limit and cyclic yield are closely related if not the same.

Hysteresis Loop Shape

Massing's hypothesis states that the limb of a stress-strain hysteresis loop is geometrically similar to the cyclic stress strain curve but numerically *twice* it. Consequently, the equation for the limb can be derived directly from the equation of the cyclic stress-strain curve.

Consider any point on the cyclic stress-strain curve with coordinates $(\varepsilon_1, \sigma_1)$, it follows that:

$$\varepsilon_1 = \frac{\sigma_1}{E} + \left(\frac{\sigma_1}{K'}\right)^{\frac{1}{n'}} \tag{16-66}$$

From Massing's hypothesis, the same point can be located on the hysteresis loop curve and it will have coordinates $(\Delta\varepsilon_1, \Delta\sigma_1)$ where:

$$\Delta\varepsilon_1 = 2\varepsilon_1 \tag{16-67}$$

$$\Delta\sigma_1 = 2\sigma_1 \tag{16-68}$$

Substituting into the equation for the cyclic stress-strain curve:

$$\frac{\Delta\varepsilon_1}{2} = \frac{\Delta\sigma_1}{2E} + \left(\frac{\Delta\sigma_1}{2K'}\right)^{\frac{1}{n'}} \tag{16-69}$$

which in the general case reduces to:

$$\Delta\varepsilon = \frac{\Delta\sigma}{E} + 2\left(\frac{\Delta\sigma}{2K'}\right)^{\frac{1}{n'}} \tag{16-70}$$

The relationship between the cyclic stress-strain curve and hysteresis loop shape is illustrated in more detail in the following example.

Example 4: Calculating Cyclic Stress-Strain Response

A test piece with the following cyclic properties,

cyclic strength coefficient,	K' = 1200 MPa
cyclic strain hardening exponent,	n' = 0.2
modulus of elasticity,	E = 210,000 MPa

is to be cycled under strain control with a fully-reversed strain range of 0.03. Calculate the expected stress-strain response.

In calculating the stress-strain response, it will be necessary to make two assumptions.

Firstly, it will be assumed that the material will follow the cyclic stress-strain response rather than the monotonic one on initial loading from zero to the maximum strain amplitude of 0.015. While there is a case for the use of the monotonic response for the first half cycle, in practice after repeated loading, the resulting hysteresis loop tip always *migrates* towards the cyclic stress-strain curve.

Secondly, it will be assumed that the material will exhibit cyclically stable response from the initial loading. A more precise analysis would require accounting for the cyclic hardening or softening characteristics of the material. As in the case of the first assumption above, the cyclic modelling process is used to model the stable hysteresis behavior since in most cases this occupies the majority of fatigue life.

Figure 16-36 (**a**) illustrates a portion of the strain time sequence that is to be used to cycle the test specimen. The value of the stress corresponding to the first turning point, 1, can be calculated directly from the equation for the cyclic stress-strain curve,

$$\varepsilon_1 = \frac{\sigma_1}{E} + \left(\frac{\sigma_1}{K'}\right)^{\frac{1}{n'}}$$

$$0.015 = \frac{\sigma_1}{210,000} + \left(\frac{\sigma_1}{1200}\right)^{\frac{1}{0.2}}$$

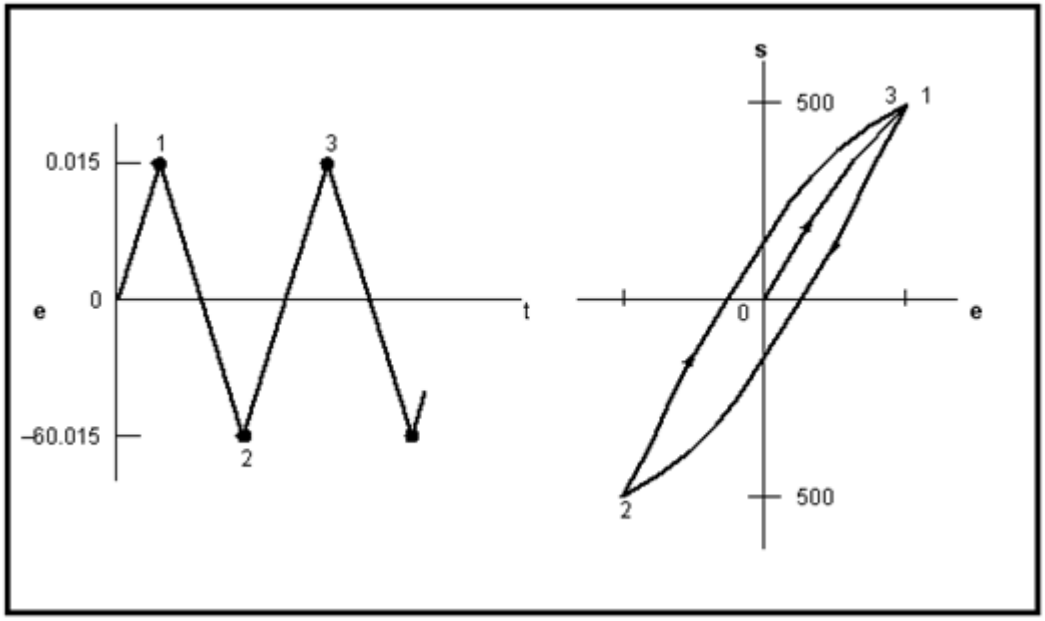


Figure 16-36 (a) The Driving Strain History. (b) The Stress-Strain Response

The value of σ_1 can be calculated from the above equation by using an iterative technique such as Newton Ralphson or interval halving. Such a procedure will provide a value of

$$\sigma_1 = 500\text{MPa}$$

The value of the stress at the next turning point, 2 (in Figure 16-36 (**a**)), can be calculated by considering the total strain range, 0.03, together with the equation for twice the cyclic stress-strain curve,

$$\Delta\varepsilon = \frac{\Delta\sigma}{E} + 2\left(\frac{\Delta\sigma}{2K'}\right)^{\frac{1}{n'}}$$

$$0.03 = \frac{\Delta\sigma}{210,000} + 2\left(\frac{\Delta\sigma}{2400}\right)^{\frac{1}{0.2}}$$

Solving this equation by iteration leads to a value of $\Delta\sigma = 1000$ MPa. The coordinates of the second turning point in stress-strain space can now be calculated from:

$$\varepsilon_2 = \varepsilon_1 - \Delta\varepsilon = (0.015 - 0.03) = -0.015$$

$$\sigma_2 = \sigma_1 - \Delta\sigma = (500 - 1000) = -500\text{MPa}$$

The coordinates of the third turning point can be calculated in the same way as for the second point, and for the fully reversed loading being considered here, they must be the same as for the first turning point. Figure 16-36 (**b**) illustrates the stress-strain hysteresis loop corresponding to the induced cyclic loading.

The Strain-Life Curve

A number of years ago, Basquin observed that stress-life data may be represented by a straight line relationship when plotted using log scales. The relationship could be expressed in terms of true stress as:

$$\sigma_a = \sigma'_f(2N_f)^b \tag{16-71}$$

σ_a		the true cyclic stress amplitude
σ_f'		the regression intercept called *fatigue strength coefficient*
$2N_f$		number of half cycles, *reversals*, to failure
b		the regression slope called the *fatigue strength exponent*

σ_f' and b are considered to be material properties with the fatigue strength coefficient being approximately equal to the monotonic fracture stress, σ_f, and b varies between -0.05 and -0.12.

The Basquin equation may be rewritten in terms of elastic strain amplitude:

$$\varepsilon_e = \frac{\sigma_a}{E} = \frac{\sigma'_f}{E}(2N_f)^b \tag{16-72}$$

ε_e		the elastic strain amplitude
E		the modulus of elasticity

In the 1950s, Coffin and Manson independently proposed that the plastic strain component of a fatigue cycle may also be related to life by a simple power law:

$$\varepsilon_p = \varepsilon'_f(2N_f)^c \tag{16-73}$$

ε_p		the plastic strain amplitude
ε_f'		the regression intercept called *fatigue ductility coefficient*
$2N_f$		number of half cycles, *reversals*, to failure
c		the regression slope called the *fatigue ductility exponent*

ε_f' and c are considered to be material properties with the fatigue ductility coefficient being approximately equal to the monotonic fracture strain, ε_f, and c varies between -0.5 and -0.8. More recent work, notably by Morrow, has indicated that the total strain amplitude, that is the sum of the elastic and plastic components, may be better correlated to life. Figure 16-37 illustrates schematically the nature of the total strain-life curve.

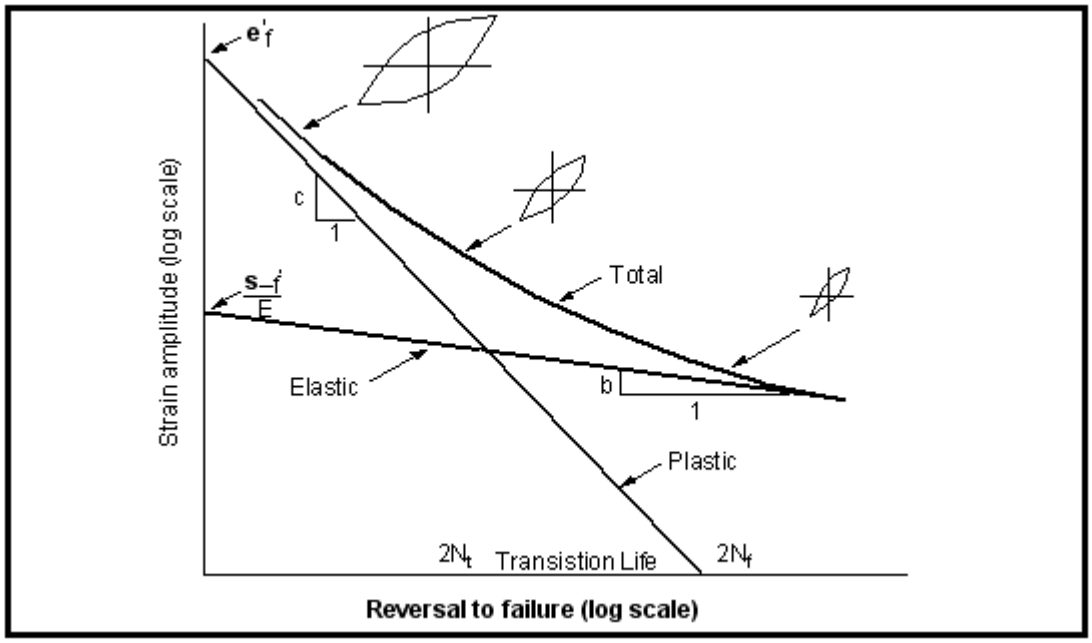


Figure 16-37 The Total Strain-Life Curve

Mathematically, this curve can be described by summing together the Basquin and the Coffin-Manson component curves:

$$\varepsilon_t = \varepsilon_e + \varepsilon_p \tag{16-74}$$

$$\varepsilon_t = \frac{\sigma'_f}{E}(2N_f)^b + \varepsilon'_f(2N_f)^c \tag{16-75}$$

Note that there is no fatigue limit defined by this equation and so it is usual to define a *cutoff* in terms of reversals which specifies an endurance limit. Typically, the cutoff is set to about 5×10^7 reversals.

Strain-Life vs. Stress-Life

In the realm of high-cycle fatigue, the equation for the S-N curve is usually written as:

$$N_f = N_o\left(\frac{S}{S_o}\right)^{\frac{1}{b}} \tag{16-76}$$

which may be rearranged as

$$S = \frac{S_o}{N_o}(N_f)^b \tag{16-77}$$

and at one cycle S_o is the intercept between the S-N line and the stress axis, approximately, the fracture stress, S_f and so

$$S = S_f(N_f)^b \tag{16-78}$$

The Basquin equation may be rearranged so that:

$$\sigma_a = \sigma'_f 2^b (N_f)^b \tag{16-79}$$

It is clear that these two equations are the same, particularly since magnitude of b is about -0.1 and 2^b is close to unity. The thing to note from this comparison is that the classical S-N curve is actually a subset of the total strain-life curve and also that the fundamental material S-N curve, as opposed to a component S-N curve, can be derived from the low-cycle fatigue parameters, σ_f' and b. It is important to remember that whilst the elastic S-N curve is defined all the way up to, σ_f', it should only be used within the high-cycle regime (i.e., above the *transition* life).

Transition Life

In Figure 16-37, the point where the plastic and elastic life lines intersect is called the *transition* life. The transition life represents the point at which a stable hysteresis loop has equal elastic and plastic components. At lives less than the transition, plastic events dominate elastic ones and at lives longer than the transition elastic events dominate plastic ones. From this point of view, therefore, the transition life represents a very convenient and important way of delineating between the low- and high-cycle fatigue regimes.

This distinction is important because the solutions which may be proposed to a particular fatigue problem depend entirely on the dominant loading regime. Problems of high-cycle fatigue are usually tackled through the selection of stronger, higher UTS, materials, or through the application of compressive surface stresses through shot peening or nitriding, etc. These solutions would be largely ineffective for the treatment of a low-cycle fatigue problem. Indeed, the selection of a material with a higher UTS, and presumably a lower ductility, could make the situation worse.

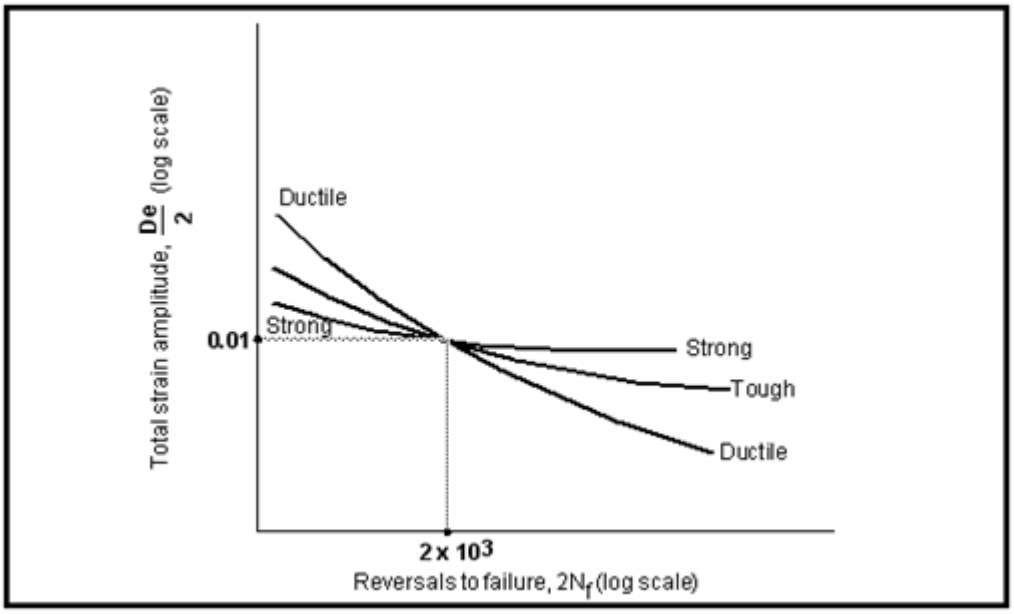


Figure 16-38 Idealized Representation of Several Strain-Life Curves

These observations are further emphasized in Figure 16-38 where three idealized materials are shown, one very high strength material (strong), one ductile material (ductile), and one whose properties lie between the two extremes (tough). The curve cross at about 2 x 10^3 reversals, 1000 cycles, at a strain amplitude of about 0.01. Thus, one would select the "strong" material for a design life requirement greater than 1000 cycles, pick the "ductile" material for design life requirements shorter than 1000 cycles, and "optimize" with the "tough" material for spectrum loading of a more complicated nature. It is interesting to note that all types of ductile materials seem to have about the same fatigue resistance for a strain amplitude of about 0.01, corresponding to a failure life of about 1000 cycles.

The transition life can be derived directly from the equation of the strain-life curve. At the transition, the elastic and plastic strain amplitudes are equal so that at $N_f = N_t$

$$\varepsilon_e = \varepsilon_p$$

$$\frac{\sigma'_f}{E}(2N_t)^b = \varepsilon'_f(2N_t)^c$$

and

$$2N_t = \left(\frac{\varepsilon'_f E}{\sigma'_f}\right)^{\frac{1}{(b-c)}}$$

Example 5: Transition Life and Shot Peening

A component is manufactured from a steel with the following cyclic properties:

fatigue strength coefficient,	σ_f'		1100 MPa
fatigue strength exponent,	b		-0.1 MPa
modulus of elasticity,	E		210,000 MPa
fatigue ductility coefficient,	ef'		0.6
fatigue ductility exponent,	c		-0.5

If the component were to be cycled under a fully reversed constant strain amplitude of 0.004, would you expect shot peening to influence observed fatigue life?

The first step is to calculate the transition life from:

$$2N_t = \left(\frac{\varepsilon'_f E}{\sigma'_f}\right)^{\frac{1}{(b-c)}}$$

$$2N_t = \left(0.6\left(\frac{210,000}{1100}\right)\right)^{\frac{1}{(-0.1+0.5)}}$$

$$2N_t = 114.5^{2.5} = 140,425 reversals$$

The next step is to calculate the total strain amplitude at the transition life. This can be done by calculating either the elastic or plastic strain components at the transition life:

$$\varepsilon_p = \varepsilon'_f(2N_t)^c = 0.6x140425^{-0.5}$$

$$\varepsilon_p = 0.0016$$

and so ε_t is given by $\varepsilon_t = 2 \times 0.0016 = \underline{0.0032}$

The working strain amplitude of 0.004, is significantly larger than the strain amplitude at the transition and so the loading can be considered to fall within the low cycle regime. Under these circumstances shot peening should *not* be considered and is not providing any real benefit.

The Effect of Mean Stress

As mentioned above, most basic fatigue data are collected in the laboratory by means of testing procedures which employ fully-reversed loading (i.e., R = –1). However, most realistic service situations involve nonzero mean stresses. It is, therefore, very important to know the influence that mean stress has on the fatigue process so that the fully-reversed laboratory data can be usefully employed in the assessment of real situations.

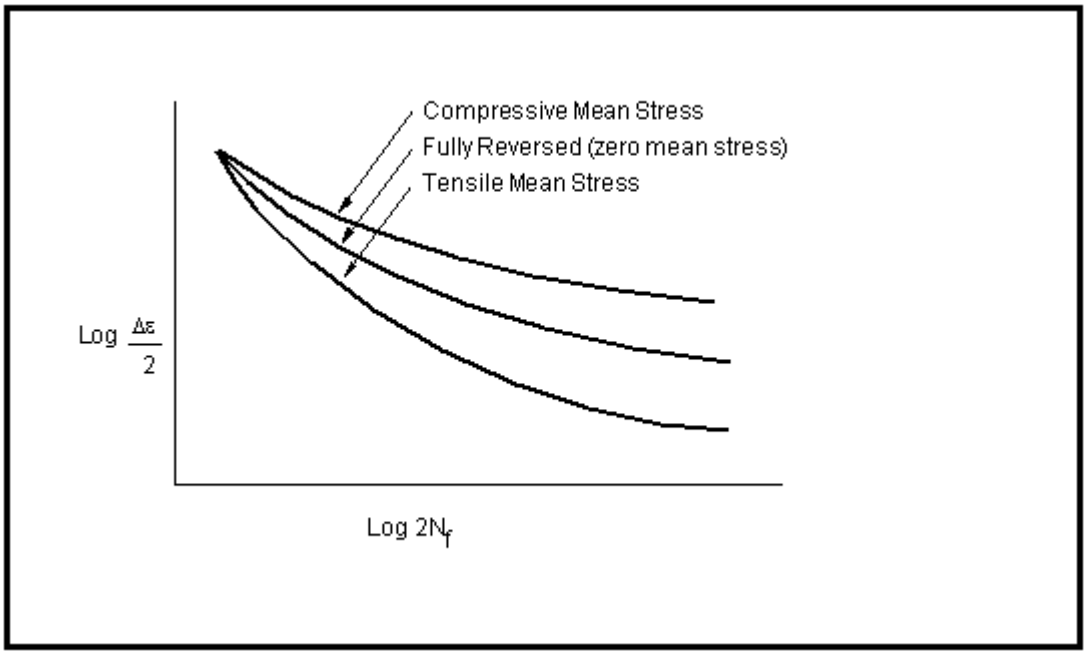


Figure 16-39 Effect of Mean Stress on the Strain-Life Curve

Figure 16-39 illustrates schematically the effect mean stress has on the strain-life curve. Typically, the effects are concentrated at the long life end of the diagram, with compressive means extending life and tensile means reducing it. At high strain amplitudes, mean stress relaxation occurs and any mean stress present tends towards zero. This effect is similar to the "washing out" of surface compressive stresses in the low-cycle regime.

The Morrow Mean Stress Correction

Morrow was the first to propose a modification to the baseline strain-life curve which can account for the effect of mean stress. He suggested that mean stress could be taken into account by modifying the elastic part of the strain-life curve by the mean stress, σ_o.

$$\varepsilon_e = \frac{(\sigma'_f - \sigma_o)}{E}(2N_f)^b \tag{16-80}$$

the entire strain life curve becoming:

$$\varepsilon_a = \frac{(\sigma'_f - \sigma_o)}{E}(2N_f)^b + \varepsilon'_f(2N_f)^c \tag{16-81}$$

The Morrow equation is consistent with observation that mean stress effects are significant at low values of plastic strain and of little effect at high plastic strains.

The Smith-Watson-Topper Mean Stress Correction

Smith, Watson, and Topper have proposed a slightly different approach to account for mean stress through a consideration of the maximum stress present in any given cycle. In this case, the damage parameter is taken to be the product of the maximum stress, σ_{max}, and the strain amplitude, ε_a of a cycle.

For fully reversed loading, the maximum stress is give by

$$\sigma_{max} = \sigma'_f(2N_f)^b \tag{16-82}$$

and by multiplying the strain-life equation by this term

$$\sigma_{max}\varepsilon_a = \frac{(\sigma'_f)^2}{E}(2N_f)^{2b} + \sigma'_f\varepsilon'_f(2N_f)^{b+c} \tag{16-83}$$

The Smith-Watson-Topper equation predicts that no fatigue damage can accrue when the maximum stress becomes zero or negative, which is not strictly true.

Morrow vs. Smith-Watson-Topper

It is difficult to categorically select one procedure in preference to the other. However, for loading sequences which are predominantly *tensile* in nature, the Smith-Topper-Watson approach is more conservative and is, therefore, recommended. In the case where the loading is predominantly *compressive*, particularly for wholly compressive cycles, the Morrow correction can be used to provide more realistic life estimates.

Factors Influencing Fatigue Life

A standardized, fully reversed, strain controlled, fatigue test is used to determine the baseline strain-life relationship for a polished specimen of approximately, 6mm diameter. If the fatigue or endurance limit measured by these means be denoted by σ'_e then the actual limit for a real component, σ_e, must reflect all the modifications that come about in moving from a laboratory specimen to a component. For steels in particular, several empirical relationships have been developed which can account for the variation in σ_e as a result of the following:

- Component size
- The type of loading
- The effect of surface finish
- The effect of surface treatment

The usual way to account for these effects is through the calculation and application of specific modifying factors so that

$$\sigma_e = \sigma'_e C_{size} C_{load} C_{sur} \cdots \tag{16-84}$$

where reciprocal of the product, C_{size} C_{load} C_{sur}, is collectively known as the *fatigue strength reduction factor* K_f, i.e.,

$$K_f = \frac{1}{(C_{size}C_{load}C_{sur}\ldots)} \tag{16-85}$$

The approach tends to be conservative and the corrections are usually only applied at the endurance limit, the modifications required for the rest of the elastic portion of the strain-life curve are ill-defined. Typically, the procedure is to pivot the elastic strain (Basquin) curve about the 1000 cycles point.

The magnitude of the modification factors required to account for the various effects can be taken to be the same as that required to modify the basic nominal stress-life S-N curve. The manner in which the modifications are implemented is the same as that required for the nominal stress approach.

Application in MSC Fatigue

The basic premise of the crack initiation or local stress-strain approach to fatigue life estimation is that the local fatigue response of a material at a critical location, for example the site of crack initiation at the root of a notch, is similar to that of a smooth specimen subjected to the same cyclic strains, see Figure 16-40.

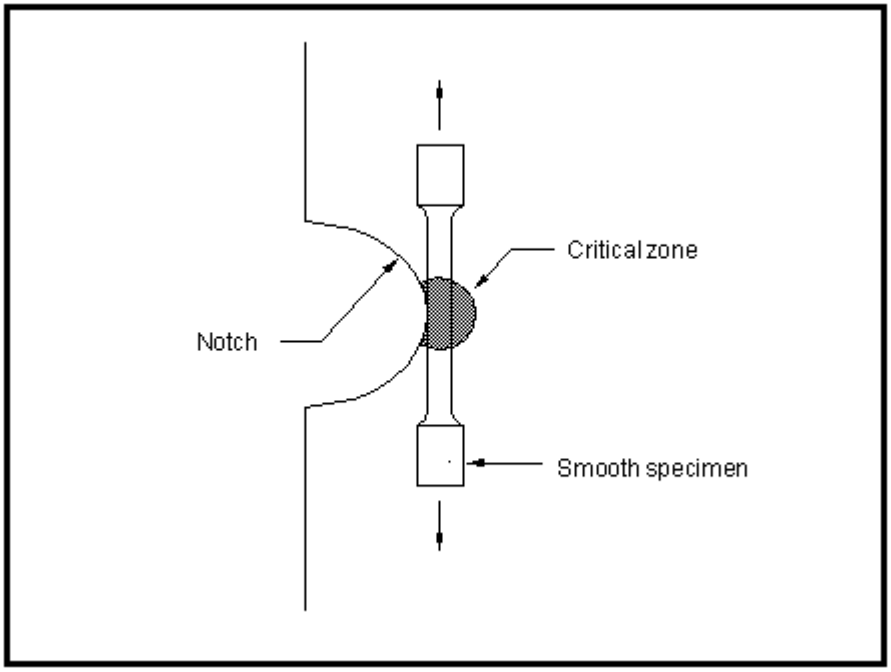


Figure 16-40 The Principle of Similitude

The Measured Strain Environment

MSC Fatigue requires representations of the load time histories in order to define the loading environment to which a component or structure is subject.

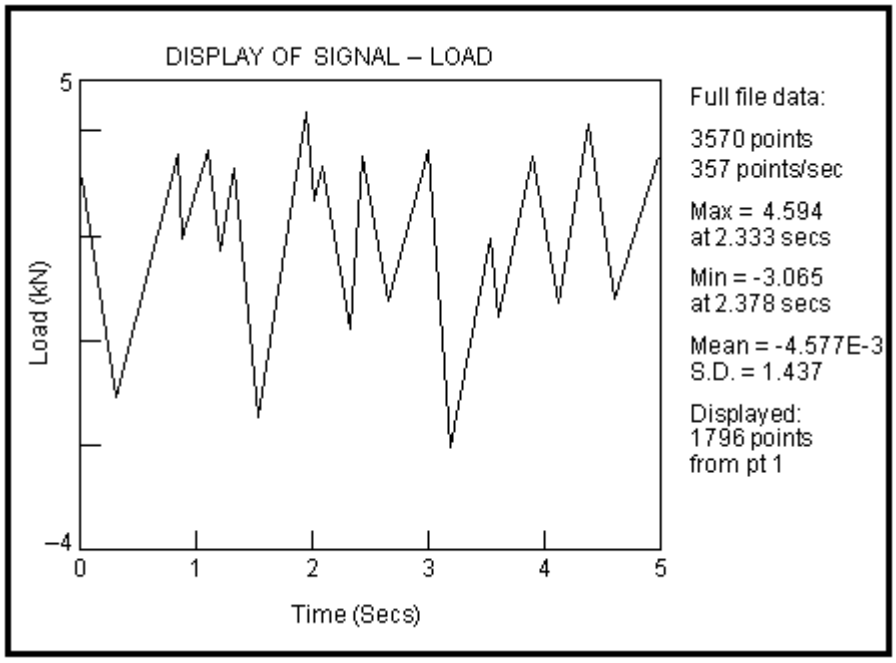


Figure 16-41 A Load Time Series

It is very important to ensure that these measured load histories are indeed representative of the service environment experienced by the component in terms of both amplitudes and sequence. If, for example, the main source of fatigue damage is due to occasional "overloads", such as the bump of a wheel on a pothole, or the "cold-hot-cold" (startup-run-shutdown) cycle of a large engine, as opposed to the cyclic loads associated with the continuous smooth operation, then large errors and misinterpretations can arise due to a misrepresentation of the important fatigue cycles.

Important:	In practice, in the ground vehicle industry in particular, experience has shown that 90 percent of fatigue damage accumulates from only 5 percent of induced fatigue cycles.

Component Geometry and Stress Concentration

Up until now, the values of stress and strain discussed have been local to the area where fatigue damage accumulates. The finite element analysis translates a load (or set of loads) on a structure to a stress distribution across the component. Consequently, the FEA provides the local stress information at all locations but especially the points of stress concentration where fatigue problems are likely to occur during the life of the structure.

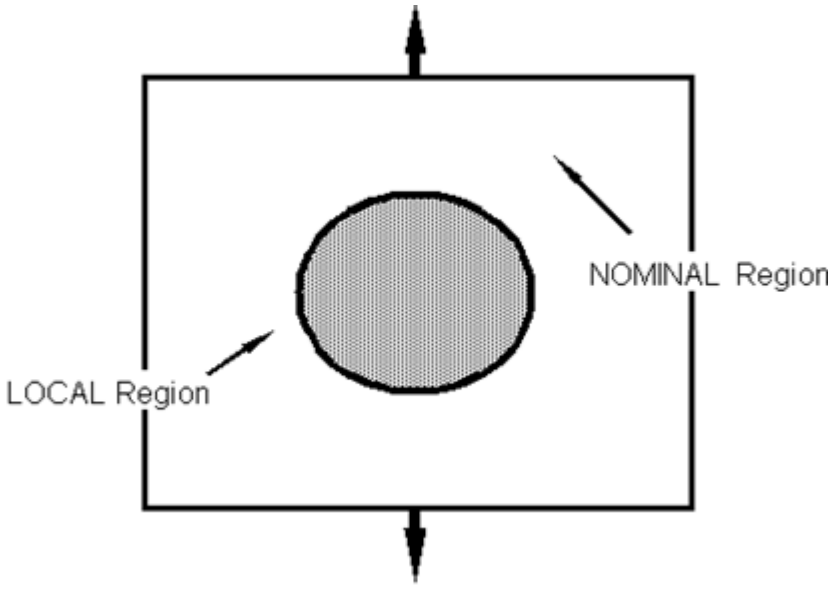


Figure 16-42 Nominal and Local Critical Regions

In the diagram above, the critical location is at the edge of the hole, where the stress concentration will be a maximum. If conditions at both the nominal and local positions remain purely elastic, then the theoretical elastic stress concentration factor, K_t is equal to both the local stress concentration factor, K_σ, $(\Delta\sigma/\Delta S)$ and the local strain concentration factor, $K_\varepsilon(\Delta\sigma/\Delta e)$, so that:

$$K_t = K_\sigma = K_\varepsilon \tag{16-86}$$

Under these conditions, NO damage can accumulate since fatigue only occurs as a result of reversed plastic flow. The basic rule for the crack initiation model is that fatigue damage, can only accumulate if at least some localized plasticity is present. In order to translate from the local elastic strains to the local elastic-plastic strains, MSC Fatigue uses Neuber's Rule.

Neuber's rule asserts that the theoretical elastic stress concentration factor, K_t, is equal to the geometric mean of the local stress concentration factor, K_σ and the local strain concentration factor, K_ε.

This relationship can be described as follows:

$$K_t^2 = K_\sigma K_\varepsilon \tag{16-87}$$

which may be rewritten as:

$$K_t^2 = \left(\frac{\Delta\sigma}{\Delta S}\right)\left(\frac{\Delta\varepsilon}{\Delta e}\right) \tag{16-88}$$

and finally:

$$K_t^2 \Delta S \Delta e = \Delta\sigma\Delta\varepsilon \tag{16-89}$$

Δe		The nominal strain range
$\Delta \varepsilon$		The local strain range
ΔS		The nominal stress range
$\Delta \sigma$		The local stress range

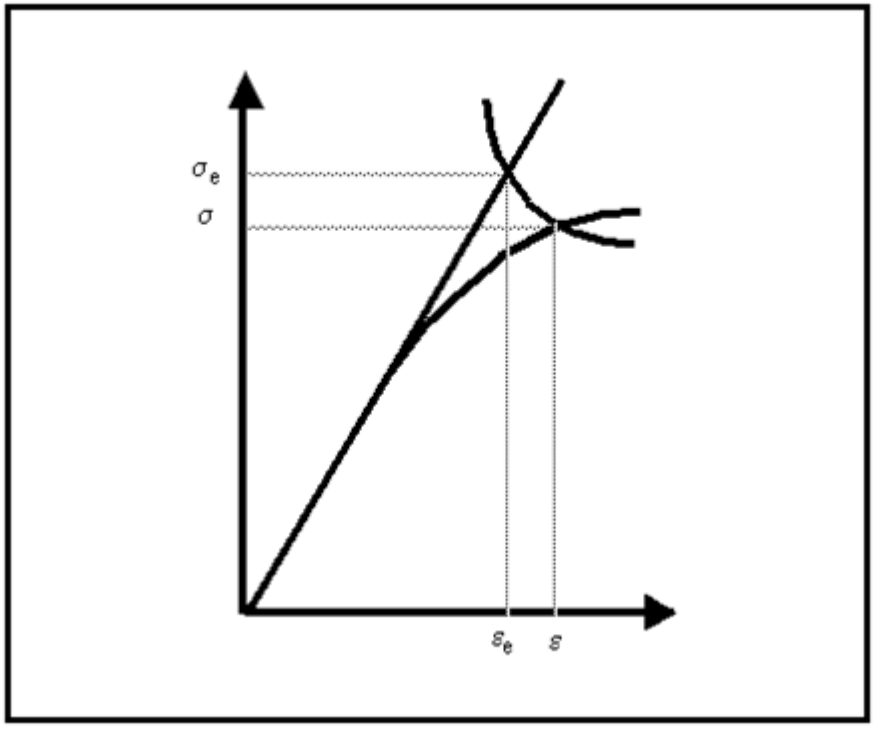
On its own, K_t, can only account for the effect of geometry on the stresses and strains visible at a given location. K_t is not required in MSC Fatigue since the linear static FEA has already calculated local elastic strains. However, in MSC Fatigue, an additional K_t, is used which allows you to include the effect of manufacturing and environmental factors such as casting, fretting, corrosion, etc., if they are present.

The surface finish and treatment of a component can also change the local stress concentration and hence affect the fatigue life. However, MSC Fatigue does not model this using a stress concentration factor since the effect is not equally important at all stress or strain levels. Instead, the high-cycle end of the life curve is modified to reflect a lowering of the endurance limit as reported by Lipson and Juvinall (Ref. 9). This is implemented by adjusting the slope of the elastic strain-life curve. See Factors Influencing Fatigue Life, 1364.

Elastic-Plastic Corrections

The Neuber method is commonly used to estimate elastic-plastic stresses and strains at the roots of notches on the basis of elastic stress analysis. It applies where the yielding is limited in extent; under these circumstances it provides a reasonable approximation for the redistribution of stress and strain. When yielding is more widespread, for instance when loads are very high, or notches very shallow (or absent) it may give non-conservative predictions. The following methods address this problem through a consideration of behavior in the limit as the component becomes fully plastic.

Mertens-Dittmann method - Neuber's rule is usually used to calculate elastic-plastic stress and strain from their wholly elastic theoretical equivalents such as might be estimated by an elastic finite element analysis.

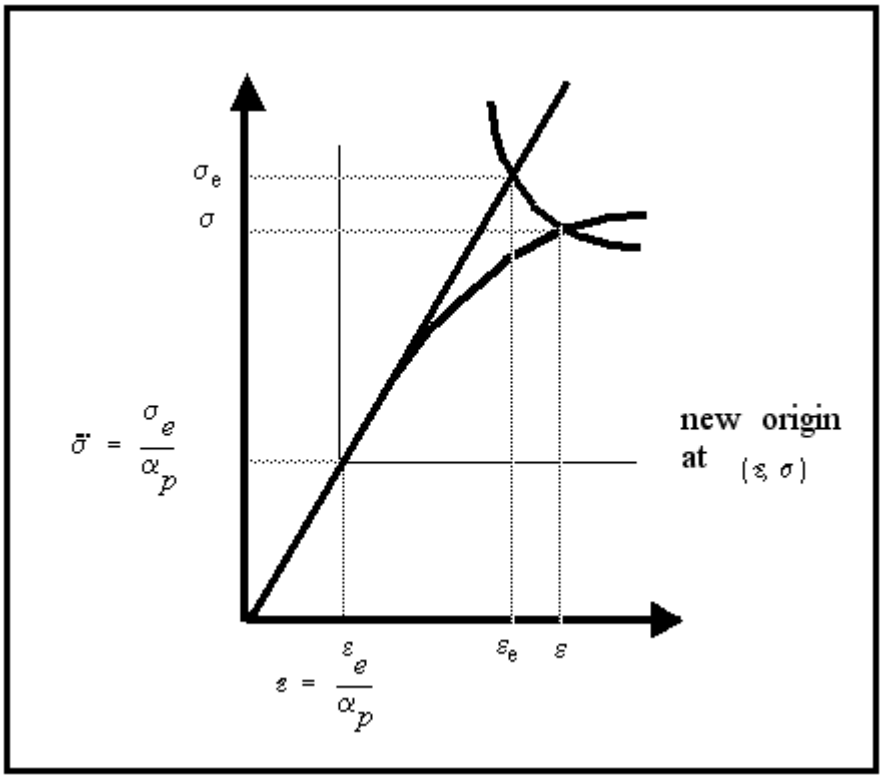


So that with K_τ taken as unity, i.e. using local elastic finite element stresses and strains:

$$\varepsilon_e \sigma_e = \varepsilon\sigma = constant \tag{16-90}$$

Both ε and σ can be calculated from the above equation together with the equations for the cyclic stress-strain curve and twice the cyclic-stress strain curve.

Dittmann (5) describes a modification to the conventional Neuber approach proposed by Mertens. He suggested that a better estimate of the local notch stress and strain can be obtained through a consideration of the minimum theoretical stress and strain which must exist, due to the applied load, in the absence of the notch. This approach requires that a new origin, at these minimum values, be used in the Neuber equation.



The modified form of the Neuber equation is then

$$(\sigma_e - \hat{\sigma})(\varepsilon_p - \hat{\varepsilon}) = (\sigma - \hat{\sigma})(\varepsilon - \hat{\varepsilon}) \tag{16-91}$$

The plastic concentration or shape factor is defined by:

$$\alpha_p = \frac{L_p}{L_y} \tag{16-92}$$

where L_y is the load that gives rise to first yielding and L_p the load required to cause plastic collapse. What this means in terms of the Neuber correction can best be understood by means of a example; consider a simple beam in bending. The bending moment required to produce yielding at the surface of a beam is given by:

$$M_y = \frac{\sigma_y I}{c} \tag{16-93}$$

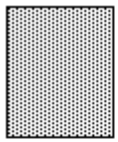
where I is the second moment of area of the beam and c is the distance of the extreme fibre from the neutral axis. As the moment increases the plastic layer will increase and the size of the elastic core will decrease. If the material is considered to be elastic-perfectly plastic, then the stress will not increase but stay fixed at the yield value. Eventually the plastic layer will penetrate right to the centre. The moment corresponding to this load is called the plastic moment and is given by:

$$M_p = \frac{\sigma_y A I (y_1 + y_2)}{2} \tag{16-94}$$

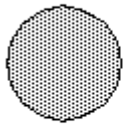
where y_1 and y_2 are the distances from the neutral axis to the centroids of the two yielded areas and A is the cross sectional area. For beams in bending then:

$$\alpha_p = \frac{M_p}{M_y} \tag{16-95}$$

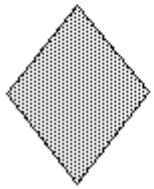
The value of the plastic strain concentration or shape factor α_p is a function of the shape of the beam. Some typical values for beams in bending are given below:



$\alpha_p = 1.5$



$\alpha_p = 1.7$

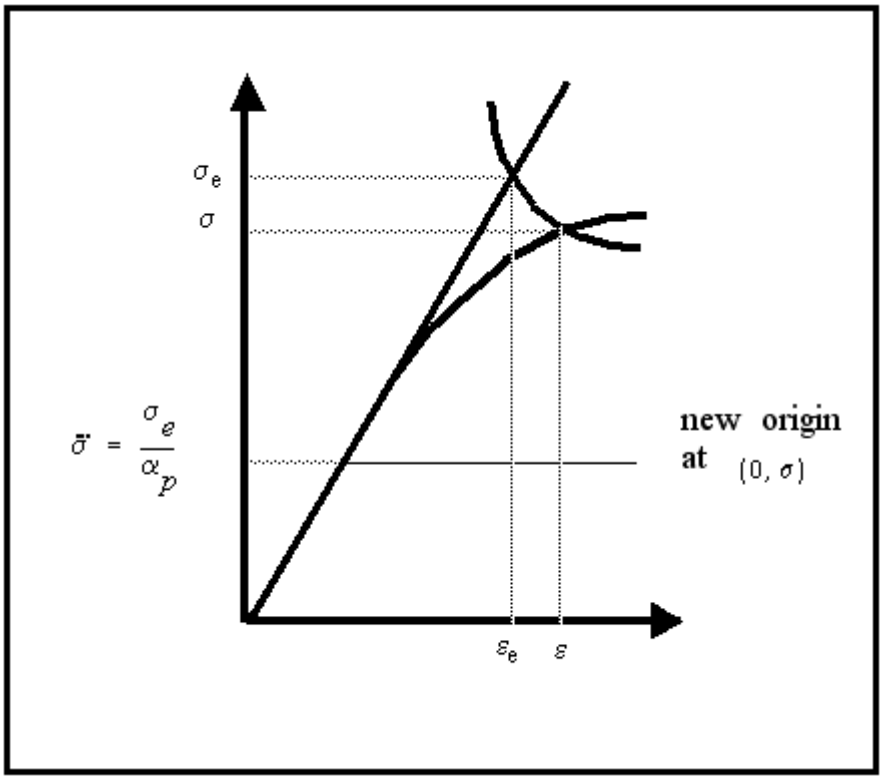


$\alpha_p = 2.0$

The normal Neuber correction will tend to underestimate the stress and strain at the surface of a beam in bending when there is more than a small degree of plasticity. When the origin used in the Neuber correction is shifted using the Mertens-Dittmann method, as the beam goes fully plastic, the Neuber hyperbola will be asymptotic to the plastic limit stress. The result is a more conservative prediction of stress and strain as plasticity spreads. Note the following points:

- The value of α_p is a function of the loading type.
- When dealing with a notched component, α_p can be estimated from the product of α_p for the unnotched component and a K_t value for the notch.
- As α_p approaches infinity, the Mertens-Dittmann method reduces to the standard Neuber method.

Seeger-Beste method - The Seeger-Beste (6) method is very similar to the Mertens-Dittmann method except that the origin is shifted only in the stress direction. This is best illustrated graphically:



Linear Damage Summation

A strain-based, constant amplitude, fatigue damage curve represents a set of tests at constant strain amplitude (ε_a) or STW parameter ($\sigma_{max}\varepsilon a$) together with associated lives, see Material Management (Ch. 4), for more details.

Operation at a strain amplitude ε_1 will result in failure in say N_1 cycles. Operation at the same strain amplitude for a number of cycles less than N1, Nj say, will result in a smaller fraction of damage, D_j, which is often referred to as a partial damage. Operation over a spectrum of different strain levels, results in a partial damage contribution D_i from each strain cycle, ε_1 . Failure is then predicted when the sum of these partial damage fractions reaches unity so that:

$$D_1 + D_2 + \ldots + D_{i-1} + D_i 1 \geq 1 \tag{16-96}$$

The Palmgren-Miner rule asserts that the partial damage at any strain amplitude ε_1 or STW value, is linearly proportional to the ratio of number of cycles of operation, n_i, to the total number of cycles that would produce failure at that strain level, N_i, i.e.,

$$D_1 = \frac{n_1}{n_2} \tag{16-97}$$

Failure is predicted then if:

$$\frac{n_1}{N_1} + \frac{n_2}{N_2} + \ldots + \frac{n_i}{N_i} \geq 1 \tag{16-98}$$

or

$$\Sigma\left(\frac{n_i}{N_i}\right) \geq 1 \tag{16-99}$$

The above equations represent a statement of the linear damage rules used by both the stress-life and local strain fatigue life processors in MSC Fatigue. Experience shows that linear damage summation is an over simplification of reality. The most important shortcoming is that no account is made of the sequence in which strain levels are experienced, and damage is assumed to accumulate at the same rate for a given strain level regardless of prehistory. In particular, it appears that large strain amplitudes which precede smaller ones, cause the smaller cycles to become more damaging than expected and vice versa. The net result is that in the first instance, the Miner sum Σ (F(n,N)) is measured to be less than 1. Since most service environments involve quasi-random loading sequences, the use of the Palmgren-Miner linear damage rule summing to a constant of 1 is mostly satisfactory.

Important:	The Design Optimizer (DESOPT) does allow you to change the miner sum from the default value of 1.0 to any value between 0.01 and 2.0.

Analytical Procedure

The strain-life method in MSC Fatigue may be summarized as follows:

1. By means of linear static FEA, derive the local stress-strain time history from the load time histories. This includes superpositioning of multiple FEA/load time history load cases (or use stress-strain time history directly from linear transient or forced vibration FE analysis).
2. Extract the fatigue cycles in the local stress time history by means of the rainflow algorithm.
3. Make the elastic-plastic correction using Neuber's rule.
4. Model the fatigue crack initiation process using hysteresis loop simulation based on the cyclic stress-strain curve.
5. Assess the damage contribution of each closed hysteresis loop by referring to the selected damage curve (strain-life, Morrow, STW curves).
6. Linearly sum the damage associated with each cycle by using Miner's rule.

The Statistical Nature of Fatigue

A considerable amount of time and effort has been expended in the statistical analysis of fatigue data and in exploring the reasons for variability in fatigue test results. Since both the fatigue life and fatigue limit are statistical quantities, as are most other measured parameters, it must be realized that considerable deviation from the average curve determined by only a few specimens is to be expected (see Figure 16-43). It is necessary

to think in terms of the probability of a specimen attaining a certain life at a given stress or strain level or the probability of failure at a given level near the fatigue limit. To do this requires the testing of considerably more specimens than might be expected so that the statistical parameters for estimating these probabilities can be determined.

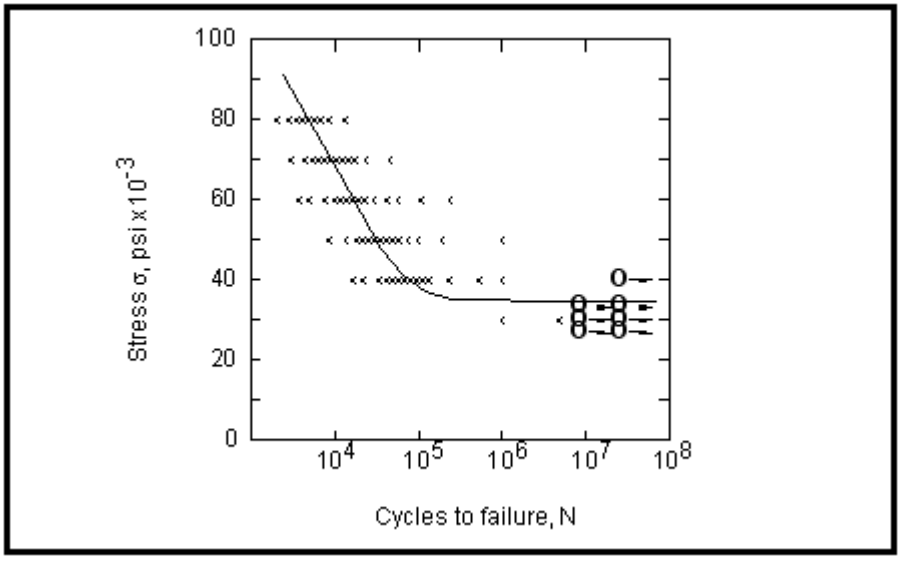


Figure 16-43 Plot of Stress-Cycle (S-N) Data
Collected by Laboratory Testing

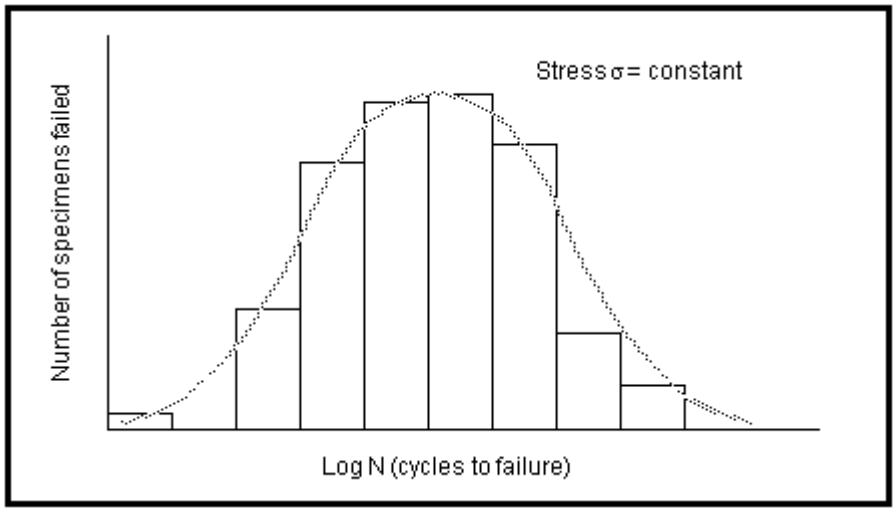


Figure 16-44 Distribution of Fatigue Specimens Failed at Constant Stress Level

Representation of Fatigue Data on a Statistical Basis

Therefore, the basic method for expressing fatigue data should be a Wöhler line which has been expanded to form a surface representing the relationship between stress (or strain), number of cycles to failure and probability of failure. A distribution of fatigue life at constant stress is illustrated schematically in Figure 16-45 and based on this, curves of constant probability drawn. Thus, at standard deviation from mean, σ_1, 1 percent of samples would be expected to fail after N_1 cycles, 50 percent after N_2 cycles, etc. The figure also indicates a decreasing scatter in fatigue life with increase in stress, which is usually found to be the case.

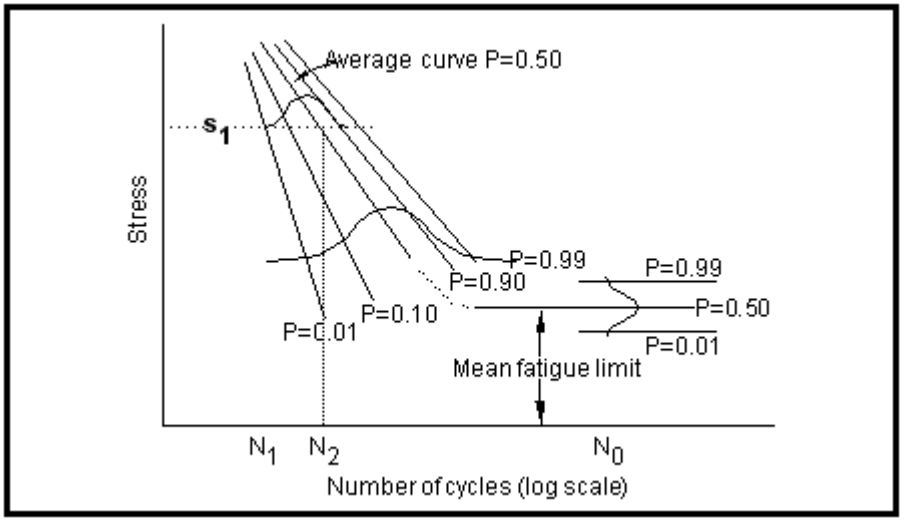


Figure 16-45 Representation of Fatigue Data on a Probability Basis

The Statistical Distribution Function

The statistical distribution function which describes the distribution of fatigue lives at a constant stress or strain level is not accurately known, since this would require the testing of over 1000 identical specimens under identical conditions at a constant stress. Researchers have tested up to 200 steel specimens at a single stress amplitude and have found that the frequency distribution of cycles to failure, N_f, followed the Gaussian, or normal, distribution if the fatigue life was expressed as $\log(N_f)$. For most engineering purposes, it is sufficiently accurate to assume a logarithmic normal distribution of fatigue life at constant stress in the region of the probability of failure of P = 0.10 to P = 0.90. However, it is frequently important to be able to predict fatigue life corresponding to a probability of failure of 1 percent or less. At this extreme limit of the distribution, the assumption of log-normal distribution of life is no longer justified, although it is frequently used. Alternative approaches have been proposed with the most commonly used one being the Weibull distribution.

Probability of Failure at a Finite Life

For a given stress or strain level, fatigue lives from a series of tests may be plotted on probability paper in order to estimate probability of failure. It is usual to use either log-normal or Weibull probability paper. To plot sample data in a meaningful way, the following procedure may be used:

- Order the array of data with the smallest value of life listed first.
- Assign a rank, r, to each data value (i.e., 1 to the first, 2 to the second, and n to the last — the longest life).
- Determine the plotting position for each data point by dividing its rank, r, by n+1, where n is the total number of measured lives.

- Plot the magnitude of the life at its corresponding plotting position on the chosen probability paper, either log-normal or Weibull.

Having plotted the data, the probability plot is examined for linearity. If the lives fall on a straight line, it may be concluded that the chosen distribution is appropriate.

As an example of the procedure, consider the following table of lives for constant amplitude fatigue tests at 345 MPa.

Rank (r)	Cycles to Failure ($x10^{-5}$)	Plotting Position 100r/(n+1)
1	4.0	11.1
2	5.0	22.2
3	6.0	33.3
4	7.3	44.4
5	8.0	55.6
6	9.0	66.7
7	10.6	77.8
8	13.0	88.9

These data have been plotted in Figure 16-46, from which the mean, 50%, or characteristic, 63.2% lives can be read off directly. Extrapolation of the line to lower probabilities of failure are also possible.

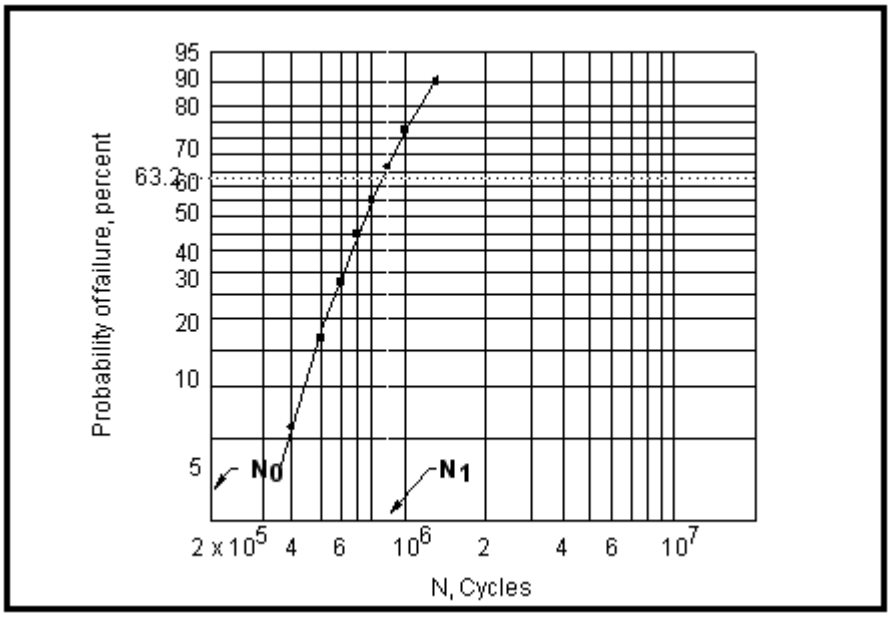


Figure 16-46 Weibull Plot for Fatigue Data

Probability of Failure for Infinite Life

Design for infinite life requires operating stresses which are all smaller than the fatigue limit. As with the main part of the Wöhler line, the fatigue limit itself can be treated as a statistical quantity. For a statistical interpretation of the fatigue limit, we are concerned with the distribution of stress at a constant fatigue life. The fatigue limit of steel was once considered to be a sharp threshold value, below which infinite life would prevail. However, it is now recognized that the fatigue limit is really a statistical quantity which requires special techniques for an accurate determination. For example, in a heat- treated alloy steel the stress range which would include the fatigue limits of 95% of all specimens tested could easily range from 280 - 360 MPa.

The statistical problem of accurately measuring fatigue limit is complicated by the fact that we cannot measure the individual value of fatigue limit for any given specimen. We can only test a specimen at a particular stress, and if it fails, then the stress was somewhere above the fatigue limit of the specimen. Since the specimen cannot be retested, even if it did not fail at the test stress, we have to estimate the statistics of the fatigue limit by testing groups of specimens at several stresses to see how many fail at each stress.

Several testing procedures have been evolved for making statistical estimates. Two of the most commonly used procedures are known as *Probit* analysis and the *staircase method*. It is beyond the scope of this course to detail these procedures, however, if the reader is interested a good account may be found in "Guide for Fatigue Testing and the Statistical Analysis of Fatigue Data" ASTM STP 91-A, 2 edition, 1963.

Handling Statistics Under Random Loading Conditions

The statistical concepts outlined above have been and remain mainly applicable to fluctuating stresses which have a constant amplitude, rather like the rotation of Wöhler's railway axles. However, the nature of the fatigue problems being addressed today requires the treatment and handling of random loading situations. Clearly, the log-normal or Weibull probability approaches involving probability paper would be totally inappropriate for use in fatigue life estimation of real field data consisting of several thousand, if not millions, of data values. Several factors have come together to meet this challenge of providing statistical information during the process of fatigue life estimation.

In the late 1960s, the switch away from the so-called nominal stress, S-N, based fatigue approach to the local stress-strain approach led workers to design fatigue testing procedures which are inherently much more precise. It became necessary to control and measure local stresses and strains during the fatigue test and so the "bubble of uncertainty" associated with these parameters in a nominal S-N test was dramatically reduced. Great improvements in laboratory testing procedures, compare Figure 16-47 and Figure 16-48, were required to achieve the necessary accuracy in the results, and this has gone a long way towards reducing the scatter in measured data and so mitigating the extent of the effect. That is not to say of course that such errors do not remain a problem, they are just less visible.

In addition to this, the need to process large amounts of data, together with the advent of low cost computing has meant that procedures have been developed for on-line statistical analyses, which whilst being firmly based on the principles outlined above, differ in their execution. Figure 16-49 illustrates the difference in approach.

Instead of considering the life at a particular stress or strain level to be distributed according to some chosen distribution, such as log-normal or Weibull, it is convenient to think of the stress or strain being distributed, at any life, according to the same distributions. In Figure 16-49 for example, the strain-life curve is made up of two components; the plastic strain ε_p, and the elastic strain, ε_e.

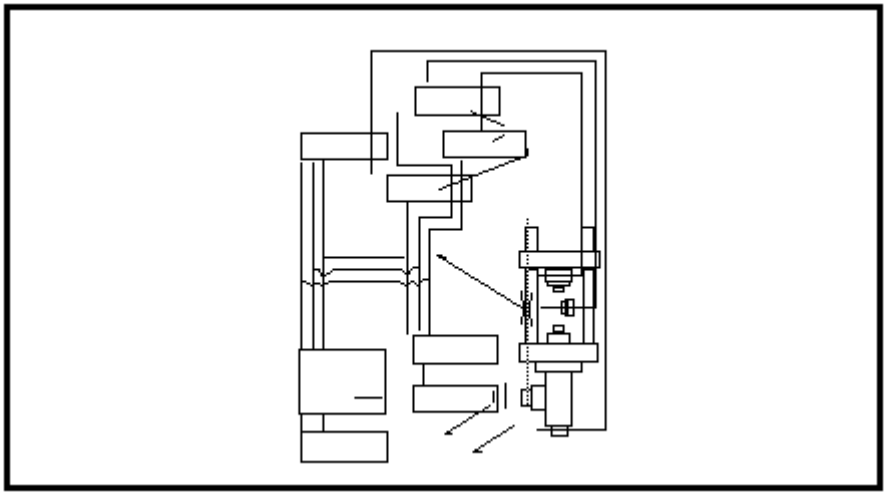


Figure 16-47 Schematic Diagram of a Computer-Controlled Closed-Loop Fatigue Testing Machine

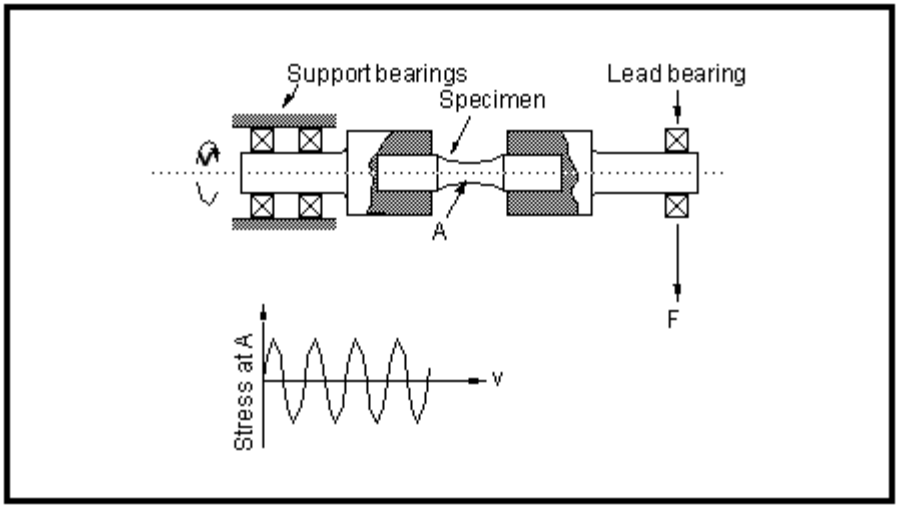


Figure 16-48 Rotating-Bending Fatigue Testing Machine of the Cantilever Bending Type

The equation of the strain-life curve may then be written as:

$$\varepsilon = \varepsilon_e + \varepsilon_p \tag{16-100}$$

otherwise

$$\varepsilon = \frac{\sigma'_f}{E}(2N_f)^b + \varepsilon'_f(2N_f)^c \tag{16-101}$$

Where $2N_f$ represents the number of reversals, half cycles to failure, and the other parameters are material constants. This equation can now be expressed statistically in terms of probability of survival:

$$\varepsilon = \frac{\sigma'_f}{E}(2N_f)^b(10)^{qe} + \varepsilon'_f(2N_f)^c(10)^{qp} \tag{16-102}$$

q		the number of standard errors from the mean life
e		the standard error of $2N_f$ for the elastic life line
p		the standard error of $2N_f$ for the plastic life line

Typically, the elastic and plastic component standard errors can be calculated directly from the results of the log elastic and log plastic vs. log reversals regression analyses, generally:

$$\text{Std. Error} = \sigma_\varepsilon\ (1 - r^2)^{0.5} \tag{16-103}$$

σ_ε		the standard deviation of log(ε) either elastic or plastic
r		the regression correlation coefficient.

For a normal distribution, the following values of 'q' can be assumed:

No. of Standard Errors from Mean, q	Probability of Survival (%
-3	99.87
-2.33	99
2	97.7
-1.28	90
-1	84
0	50
1	16
1.28	10
2	2.3
2.33	1
3	0.13

For example, if the following material parameters are assumed:

$=\varepsilon_f'$	=-	=0.4
$=\sigma_f'$	=-	=1326 MPa
=c	=-	=-0.528
=b	=-	=-0.139
=E	=-	=205000 MPa
$=s_e$	=-	=0.023
$=s_p$	=-	=0.067

Then the following values of reversals to failure, $2N_f$, can be calculated for a 10% and 90% probability of survival.

Specimen	Probability of Survival (%)		
Plastic, ε'_f	50	10	90
Elastic, σ'_f(MPa)	0.400	0.456	0.350
Reversals 2Nf	1326	1417	1240
	1769	2307	1355

which means that there is a 90% chance that the specimen will survive 1355 reversals and only a 10% chance it will survive 2307 reversals.

In the above examples, we have assumed values of q, the number of standard deviations from the mean, for a log-normal distribution. It would be equally valid to use values which represent the Weibull distribution. Indeed, for confidence levels greater than 90% it is recommended to use Weibull.

This approach is by no means confined to a treatment of strain-life curves, S-N curves can also be considered by these means. See, for example, Figure 16-50.

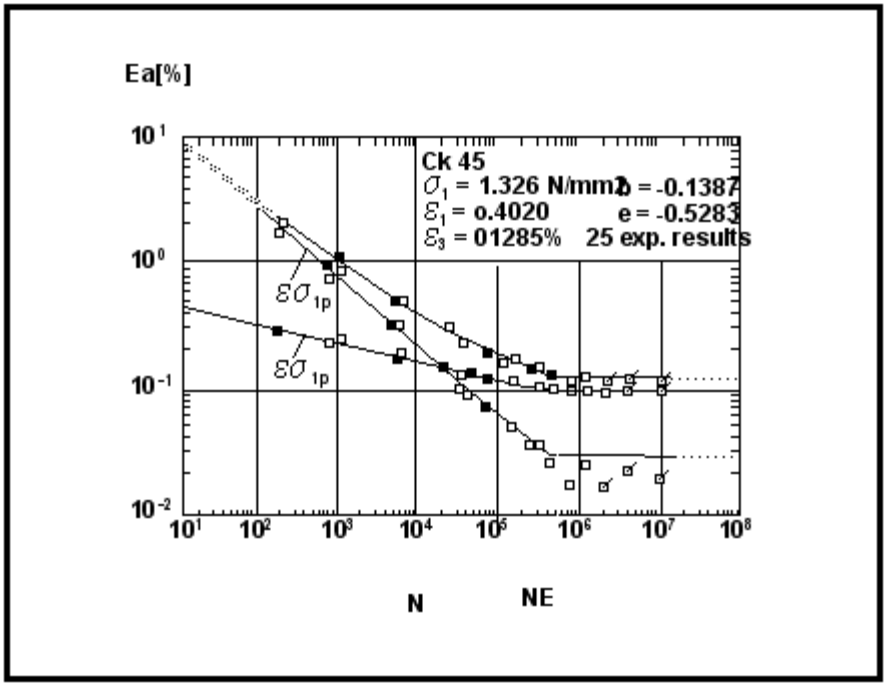


Figure 16-49 Strain-Life Curve Made Up of Elastic and Plastic Components

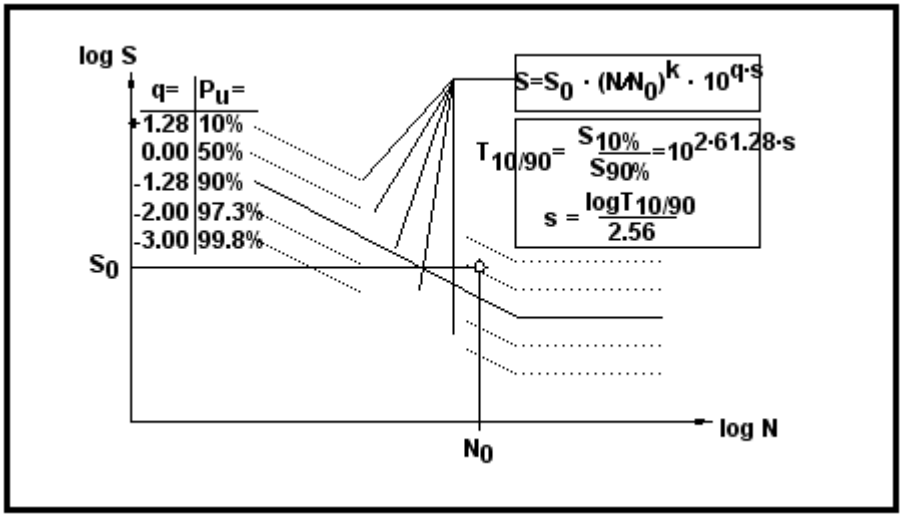


Figure 16-50 Weibull Approach With S-N Curves

The Absolute Accuracy of Fatigue Life Estimation

From the above discussion, it has been indicated that fatigue life is a rather statistical quantity to which it is very difficult, if not impossible to ascribe a unique value. It has been shown that the material law used to describe the fatigue response of a material, the Wöhler line (stress or strain), is itself not unique and must be treated as a statistical quantity. However, Wöhler lines are amenable to statistical treatment and so a measure of the variability resulting from the determination of material properties can be defined. In addition to this variability, other sources of error can and most often do arise.

Errors in the measured loading environment can also lead to errors in calculated life. As a rule of thumb, a 10% error in strain or stress measurement can either *halve* or *double* life, all else being equal. Similarly, errors in the description of component geometry can lead to significant variations. For this reason, and for complicated geometries in particular, finite element analysis may be used to describe the state of stress at any point within a structure with greater accuracy.

Taking all these factors into account, today the state of the predictive art with respect to fatigue life is to within between a factor of **two** or **three** of the *actual*, which itself is not a unique number. Expressing this mathematically it is possible to say, for any given calculated life, L_c, the largest scatter band we should expect is given by the expression:

$$\left(2.5L_c - \frac{L_c}{2.5}\right) \tag{16-104}$$

If our life estimates are significantly more widely scattered than this, then a major error in one or more of the inputs to the analysis has been made and must be sought out if the predictive process is to continue.

Estimating Material Cyclic Properties From UTS & E

Laboratory Testing

The cyclic material properties required to define the cyclic stress-strain curve and the strain-life curve are usually determined by carrying out tests, under strain control, on a series of smooth highly-polished hourglass specimens. Typically, about 15 tests need to be performed at differing strain amplitudes. The properties can then be calculated by regression analysis on the following curves:

K' and n' : from a log stress vs. log plastic strain regression

sf' and b : from a log elastic strain vs. log $2N_f$ regression

ef' and c : from a log plastic strain vs. log $2N_f$ regression

Estimation of Material Cyclic Properties

It is often difficult to gain access to measured cyclic properties. For this reason, a lot of effort has been put into finding ways of relating monotonic properties, of which there are an abundant supply, to cyclic properties. The approaches have all been empirical but have provided some approximations which are useful.

The first method of approximating the strain-life relationship from monotonic properties was proposed by Manson and later modified by Muralidharan. The procedure is usually referred to as the method of universal slopes and it has been suggested that this method can be applied to *any* metal. See Table 16-13.

More recently, Baumel Jr. and Seeger have compiled an alternative approach based on the results of more than 1500 fatigue tests. Currently, the approach is limited to plain carbon and low to medium alloy steels, aluminium, and titanium alloys.

The ductility factor α is calculated from:

$$\alpha = 1.0 \quad \text{for values of } R_m / E <= 3 \times 10^{-3}$$

$$\alpha = (1.375 - 125\, R_m / E) \quad \text{for } R_m / E > 3 \times 10^{-3}$$

Table 16-13 Strain Life Parameter Approximations

Parameter	Universal Slopes (Manson)	Modified Universal Slopes (Muralidharan)
sf'	$1.9\ R_m$	$0.623\ R_m^{0.823}\ E^{0.168}$
b	-0.12	-0.09
ef'	$0.76\ \varepsilon_f^{0.6}$	$0.0196\ \varepsilon_f^{0.155}\ (R_m / E)^{-0.53}$
c	-0.6	-0.56
K'	$sf' / (ef')^{0.2}$	$sf' / (ef')^{0.2}$
n'	0.2	0.2

R_m		the ultimate tensile strength
ε_f		the true fracture strain calculated from ln (1 / (1 - RA)) and
RA		the reduction in area.

Parameter	Uniform Material Law plain and low alloy steels (Baumel Jr & Seeger)	Uniform Material Law Aluminium and Titanium alloys (Baumel Jr & Seeger)
sf'	$1.5\ R_m$	$1.67\ R_m$
b	-0.087	-0.095
ef'	$0.59\ \alpha$	0.35
c	-0.58	-0.69
K'	$1.65\ R_m$	$1.61\ R_m$
n'	0.15	0.11

a		1 for R_m/E ð $3E10^{-3}$
a		$1.375 - (125 \times R_m/E)$ for $R_m/E > 3E10^{-3}$

Estimation of S-N Curve Data

S-N curves can be constructed purely on the basis of the ultimate tensile strength (UTS). The curve is constructed by fixing the stress axis intercept (one cycle) at the value of fracture stress, fixing the stress at 1000 cycles and the endurance limit according to the fraction of UTS detailed in the table below.

Life (cycles)	Stress (ksi)	Stress (MPa)
Ferrous		
1.0 x 10e6	0.357 x UTS	0.357 x UTS
1.0 x 10e6	0.357 x UTS	0.357 x UTS
1.0	Sf, (50.0 + UTS)	Sf, (345 + UTS)
Titanium		
1.0 x 10e6	0.307 x UTS	0.307 x UTS
1.0 x 10e6	0.8 x UTS	0.8 x UTS
1.0	Sf, UTS	Sf, UTS
Aluminium		
5.0 x 10e8	0.258 x UTS	0.307 x UTS
1.0 x 10e3	0.7 x UTS	0.8 x UTS
1.0	Sf, UTS	Sf, UTS
Other		
1.0 x 10e8	0.274 x UTS	0.274 x UTS
1.0 x 10e3	0.8 x UTS	0.8 x UTS
1.0	Sf, UTS	Sf, UTS

Calculation of Kf' (Kf at 1000 cycles) from Kf:

The effect of notches on the S-N curve are taken into account by calculating the values of Kf and Kf' and dividing the stress at the endurance limit and 100 cycles by these values respectively.

Kf' can be calculated from Kf by using the following table - in the UTS columns first values are in ksi, the second values (in parenthesis) are in MPa.

(Kf' - 1)/(Kf - 1)	UTS	UTS	UTS
0	48 (335)	16 (112)	11 (77)
0.2	98 (685)	33 (231)	22 (154)
0.4	158 (1104)	53 (370)	35 (244)
0.6	233 (1628)	78 (545)	51 (356)
0.8	343 (2396)	114 (796)	75 (524)

Note: 1 ksi = 6.985 MPa

17 References

- References 1438
- Further Reading 1443

References

1. "Steel, Concrete, and Composite Bridges, Part 10, Code of Practice for Fatigue," British Standards Institution, BS5400, ISBN 0-580-10567-9, 1980.
2. Peterson, R.E., "Stress Concentration Factors," John Wiley & Sons, New York, 1974, ISBN 0-471-68329-9.
3. Rooke and Cartwright, D. J., "A Compendium of Stress Intensity Factors," HMSO, 1974.
4. Murakame, Y., (Editor-in-Chief), "Stress Intensity Factors Handbook," Pergamon Press, 1987.
5. James, L. A. and Mills, W. J., "Eng. Fract. Mech.," Volume 30, No.5, 1988, page 641.
6. Gray, T. J. F., Int. Jnl. Fract., Volume 13, No.1, 1977, page 65.
7. Newman, J. C. Jr. and Raju, I. S., ASTM STP 791, ASTM, 1983, page I-238.
8. Walker, E. F. and May, M. J., BISRA Open Report No. MG/E/307/67, 1967.
9. Lipson, C., and Juvinall, R. C., Handbook of Stress and Strength — Design and Material Applications, Macmillan & Co., New York 1963.
10. Paris, P. C., Gomez, R. E., and Anderson, W. E., "The Trend in Engineering," Vol. 13, No. 1, Jan 1961.
11. Forman, R. G., Kearney, V. E., and Engle, R. M., "Jnl. Basic Eng.," Vol 89, 1967, p459.
12. Klesnil, M. and Lukas, P., Matl. Sci. Jnl., Vol 9, 1972, page 231.
13. Austen, I. M. and Walker, E. F., Proc. Int. Conf. "Steel in Marine Structures," Delft, Netherlands, June 1987, (Elsevier), page 859.
14. Austen, I. M. and Walker, E. F., Proc. Int. Conf. "Fatigue '87," Charlottesville, VA., USA, (EMAS), page 1155.
15. Austen, I. M. and Walker, E. F., Proc. Int. Conf. "Influence of Environment on Fatigue," London, UK, May 1977, (I. Mech. E.), page 1.
16. Austen, I. M., et al, "A Review of Fatigue Crack Growth Characterization by Linear Elastic Fracture Mechanics (LEFM) Part 3 — Corrosion Fatigue," (British Steel/The Welding Institute), 1989.
17. Austen, I. M. and Walker, E. F., Proc. Int. Conf. "Offshore Structures," Cambridge, UK, Nov. 1978, (DEn/CEC).
18. Webster, S. E., Austen, I. M., and Rudd, W. J., "Fatigue, Corrosion Fatigue, and Stress Corrosion of Steels for Offshore Structures," ECSC Contract 7210.KG/801, Final Report, (HMSO), April 1984.
19. Webster, S. E., Austen, I. M., and Rudd, W. J., "The Fatigue and Stress Corrosion Properties of Conventional and Higher Strength Steels for Use in Offshore Environments," ECSC Contract 7210.KG/802, Final Report, (HMSO), July 1988.
20. Austen, I.M., "Quantitative Understanding of Corrosion Fatigue Crack Growth Behaviour," ECSC Contract 7210.KE/806, Final Report, (HMSO), July 1982.
21. Austen, I. M., "Quantitative Assessment of Corrosion Fatigue Crack Growth Under Variable Amplitude Loading," ECSC Contract 7210.KE/810, Final Report, (HMSO), July 1987.
22. Garwood, S. J., and Boulton, C. F., "Cumulative Damage to Welded Steel Structures," ECSC Contract 6210.KD/8/801, Final Report, (HMSO), Sept. 1979.

23. Willenborg, J. D., Engle, R. M., Wood, H. A., Air Force Flight Dynamics Laboratory, Report TM-71-1-FBR, Wright-Patterson Airbase, USA, 1971.
24. Johnson, W. S., "Methods and Models for Predicting Fatigue Crack Growth Under Random Loading," ASTM STP 748, 1981, (ASTM), page 85.
25. Elber, W., "Fatigue Crack Growth Under Spectrum Loads," ASTM STP 595, 1976, (ASTM), page 236.
26. Neuber, H. (1961) "Theory of stress concentration for shear strained prismatic bodies with arbitrary nonlinear stress-strain law." Journal of Applied Mechanics, Vol. 28, pp. 544-551.
27. Hoffmann, M. and Seeger, T. (1989) "Estimating multi-axial elastic-plastic notch stresses and strains in combined loading." Biaxial and Multi-axial Fatigue, EGF3. Edited by M. W. Brown and K. J. Miller, MEP, pp. 3-24.
28. Hoffmann, M. and Seeger, T. (1989) "Stress-strain analysis and life predictions of a notched shaft under multi-axial loading." Multi-axial Fatigue, Analysis and Experiments, AE-14. Edited by G. E. Leese and D. Socie, SAE Inc., pp. 81-96.
29. Klann, D. A., Tipton, S. M. and Cordes, T. S. (1993) "Notch stress and strain estimation considering multi-axial constraint." SAE Technical Paper 930401.
30. Dittmann, K. J. (1991) "Ein Beitrag zur Festigkeitsberechnung un Lebensdauer fur Bauteile das Stahl unter mehrachsiger synchroner Beanspruchung." PhD Dissertation, TU Berlin.
31. Seeger, T. and Beste, A. (1977) "Zur Weiterentwicklung von Nherungsformeln fur die Berechnung von Kerbbeanspruchungen im elastisch-plastischen Bereich." VDI-Fortschrittsberichte, Reihe 18, Nr. 2, Dusseldorf.
32. Rupp, A., Störzel, K. and Grubisic, V. "Computer Aided Dimensioning of Spot-Welded Automotive Structures." SAE Technical Paper 950711, 1995.
33. Smith, R. A. and Cooper, J. F. "Theoretical predictions of the fatigue life of shear spot welds." Fatigue of Welded Structures, Ed. S. J. Maddox, pp. 287 - 293, The Welding Institute, 1988.
34. British Standards Institution. Code of Practice for Fatigue Design and Assessment of Steel Structures. BS 7608, 1993.
35. Radaj, D. "Local Fatigue Strength Characteristic Values for Spot Welded Joints." Engineering Fracture Mechanics, Vol. 37, No. 1, pp. 245 - 250, 1990.
36. Sheppard, S. D. and Strange, M. E. "Fatigue Life Estimation in Resistance Spot Welds: Initiation and Early Growth Phase." Fatigue and Fracture of Engineering Materials and Structures, Vol. 15, No. 6, pp. 531 - 549, 1992.
37. Sheppard, S. D. "Estimation of Fatigue Propagation Life in Resistance Spot Welds." ASTM STP 1211, Advances in Fatigue Life Prediction Techniques, M. R. Mitchell and R. W. Landgraf, Eds., pp. 169 - 185, ASTM Philadelphia, 1993.
38. Heyes, P., Dakin, J. and StJohn, C. "The Assessment and Use of Linear Static FE Stress Analyses for Durability Calculations." SAE Technical Paper 951101, 1995.
39. A.K.Khosrovaneh and N.E.Dowling "Fatigue Loading History Reconstruction based on the Rainflow Technique." International Journal of Fatigue,Vol 12 no. 2 pp90-106

40. J.Kowalewski "On the relationship between Component Life Under Irregularity Fluctuating and Ordered Load Sequences." Part 2, DVL Report 249
41. Thomson, W. T., "Theory of Vibration with Applications", 2nd Edition, George Alle and Unwin, 1981.
42. Newland, D. E., "An Introduction to Random Vibrations and Spectral Analysis", 2nd Edition, Longman, 1986.
43. Cooley, J. W., and J. W. Tukey, "An algorithm for the Machine Calculation of Complex Fourier Series", Math. Comput., 19 (April 1965): 297.
44. "Programs for Digital Signal Processing", Published by IEEE, 1979.
45. Mròz Z., (1967), "On the Description of Anisotropic Work Hardening," Journal of Mechanics and Physics of Solids, vol. 15, pp. 163-175
46. Garud Y. S., (1981), "A New Approach to the Evaluation of Fatigue under Multiaxial Loading," Journal of Engineering Materials and Technology, vol. 103, pp. 118-125
47. Wang C. H. and Brown M. W., (1996), "Life Prediction Techniques for Variable Amplitude Multiaxial Fatigue - Part 1: Theories," Journal of Engineering Materials and Technology, vol. 118, pp. 367-370
48. Wang C. H. and Brown M. W., (1996), "Life Prediction Techniques for Variable Amplitude Multiaxial Fatigue - Part 2: Comparison with Experimental Results," Journal of Engineering Materials and Technology, vol. 118, pp. 371-374
49. Heyes P. J., Milsted M. G. and Dakin J., (1996), "Multiaxial Fatigue Assessment of Automotive Chassis Components on the basis of Finite-Element Models, Multiaxial Fatigue and Design," ESIS 21 (Edited by A. Pineau, G. Cailletaud and T. C. Lindley) MEP London, pp. 461-475
50. Heyes P., Dakin J. and St.john C., (1995), "The Assessment and Use of Linear Static FE Stress Analyses for Durability Calculations," Proceedings of the Ninth International Conference on Vehicle Structural Mechanics and CAE, pp. 189-199
51. Heyes P. and Fermér M., (1996), "A Program for the Fatigue Analysis of Automotive Spot-Welds Based on Finite Element Calculations," Proceedings of the Symposium on International Automotive Technology, SAE Technical Paper 962507
52. Basquin O. H., (1910), "The Exponential Law of Endurance Tests," Proceedings of the American Society for Testing Materials, vol. 10, pp. 625-630
53. Manson S. S., (1953), "Behaviour of Materials under Conditions of Thermal Stress," Heat Transfer Symposium, University of Michigan Engineering Research Institute, pp. 9-75
54. Coffin L. F., (1954), "The Problem of Thermal Stress Fatigue in Austenitic Steels at Elevated Temperatures," ASTM STP No. 165, p.31
55. Coffin L. F., (1954), "A Study of the Effects of Cyclic Thermal Stresses on a Ductile Metal," Trans. American Society for Testing and Materials, vol. 76, pp. 931-950
56. Matsuishi M. and Endo T., (1968), "Fatigue of Metals Subjected to Varying Stress," Presented to Kyushu District Meeting, JSME.
57. Miner M. A., (1945), "Cumulative Damage in Fatigue, Journal of Applied Mechanics," vol. 12, pp. A159-A164

58. nCode International Ltd., (1997), "nSoft-E FATIMAS software manual"
59. Glinka G. and Buczynski A., (1997), "Elastic-Plastic Stress-Strain Analysis of Notches under Non-Proportional Cyclic Loading Paths," Proceedings of the 5th International Conference on Biaxial/Multiaxial Fatigue and Fracture, Krakow, Poland
60. Chu C. C., (1989), "A Three-Dimensional Model of Anisotropic Hardening in Metals and its Application to the Analysis of Sheet Metal Formability," Journal of Mechanics and Physics of Solids, vol. 22, no. 3, pp. 197-212
61. Armstrong P. J. and Frederic C. O., (1966), "A Mathematical Representation of the Multiaxial Bauschinger Effect," CEGB Report RD/B/M731, Berkeley Nuclear Laboratories
62. Wang C. H. and Brown M. W., (1993), "Inelastic Deformation and Fatigue under Complex Loading, Proceedings of the 12th International Conference on Structural Mechanics in Reactor Technology," vol. L, pp. 159-170
63. Wang C. H. and Brown M. W., (1993), "A Path-Independent Parameter for Fatigue under Proportional and Non-Proportional Loading, Fatigue and Fracture of Engineering Materials and Structures," vol. 16, pp. 1285-1298
64. Brown M. W., Suker D. K. and Wang C. H., (1996), "An Analysis of Mean Stress in Multiaxial Random Fatigue," Fatigue and Fracture of Engineering Materials and Structures, vol. 19, no. 2/3, pp. 323-333
65. Bannantine J. A., (1989), "A Variable Amplitude Multiaxial Fatigue Life Prediction Method," PhD thesis, University of Illinois at Urbana-Champaign
66. Smith K. N., Watson P. and Topper T. H., (1970), "A Stress-Strain function for the Fatigue of Metals," Journal of Materials, vol. 5, no. 4, pp. 767-778
67. Fatemi A. and Socie D. F., (1988), "A Critical Plane Approach to Multiaxial Fatigue Damage Including Out-of-Phase Loading," Fatigue and Fracture of Engineering Materials and Structures, vol. 11, no. 3, pp.149-165
68. Dreßler K., Köttgen V. B. and Kötzle H., (1995), "Tools for Fatigue Evaluation of Non-Proportional Loading," Proceedings of Fatigue Design 1995 (Edited by Gary Marquis and Jussi Solin), vol. 1, pp. 261-277
69. Andrzej Buczynski and Grzegorz Glinka (1997), "Elastic-Plastic Stress-Strain Analysis of Notches under Non-Proportional Loading," Proceedings of the 5th International Conference on Biaxial/Multiaxial Fatigue and Fracture, Cracow, Poland, Sept. 97.
70. Diboine, A. (1996) Fatigue Assessment of Components under Complex Loadings, *Multiaxial Fatigue and Design*, ESIS 21, Mechanical Engineering Publications, London, pp. 425-443.
71. McDiarmid, D.L. (1991) A General Criterion for High Cycle Multiaxial Fatigue Failure, *Fatigue and Fracture of Engineering Materials and Structures*. Vol. 14, No. 4, pp. 429-453.
72. McDiarmid, D.L. (1994) A Shear Stress based Critical-Plane Criterion of Multiaxial Fatigue Failure for Design and Life Prediction, *Fatigue and Fracture of Engineering Materials and Structures.* Vol. 17, No. 12, pp. 1475-1484.

73. Dang Van, K., Cailletaud, G., Flavenot, J.F., le Douaron, A., Lieurade, H.P. (1989) Criterion for High Cycle Fatigue Failure Under Multiaxial Loading, *Biaxial and Multiaxial Fatigue*, EGF 3, Mechanical Engineering Publications, London, pp. 459-478.

74. Dang Van, K., Griveau, B., Message, O. (1989) On a New Multiaxial Fatigue Limit Criterion: Theory and Application, *Biaxial and Multiaxial Fatigue*, EGF 3, Mechanical Engineering Publications, London, pp. 479-496.

75. D.E.Newland, An introduction to random vibrations and spectral analysis (2nd edition), Longman Inc., New York, (1984).

76. N W M Bishop, Dynamic fatigue response of deepwater offshore structures subjected to random loading, Structural Engineering Review SER, 76/11, 63-78, (1991).

77. N W M Bishop and F Sherratt, Fatigue life prediction from power spectral density data. Part 1, traditional approaches and Part 2, recent developments. Environmental Engineering, 2,(1989).

78. P.H.Wirsching and M.C.Light, Fatigue under wide band random loading, J Struct. Div., ASCE, 1593-1607, (1980).

79. G.K.Chaudhury and W.D.Dover, Fatigue Analysis of Offshore Platforms Subject to Sea Wave Loading, Int J Fatigue, 7, (1985).

80. J.C.P.Kam and W.D.Dover, Fast fatigue assessment procedure for offshore structures under random stress history, Proc. Instn. Civ. Engrs., Part 2, 85, 689-700, (1988).

81. T.Dirlik, Application of computers in Fatigue Analysis, University of Warwick Thesis, (1985).

82. N.W.M.Bishop, The use of frequency domain parameters to predict structural fatigue, PhD thesis, Warwick University, (1988).

83. I.Rychlik, A new definition of the rainflow cycle counting method, Int J Fatigue, 9, pp 119-121, (1987).

84. N.W.M.Bishop and F.Sherratt, A theoretical solution for the estimation of rainflow ranges from power spectral density data. Fat. Fract. Engng. Mater. Struct., 13, 311-326, (1990).

85. J.Kowalewski, On the relationship Between Component Life Under Irregularly Fluctuating and Ordered Load sequences. Part 2, DVL Report 249, (1963).

86. P Bonfield and MP Ansell, Fatigue testing of wood composites for aerogenerator rotor blades. Part V. Life Prediction and Hysteresis. Wind Energy Conversion, eds TD Davies et al. (1990)

87. N.W.M.Bishop and F.Sherratt, Signal processing for fatigue in both the time and frequency domains, IMA Conference on Mathematics in the Automotive Industry, University of Warwick, 315-331, (1989).

88. N W M Bishop, Z Hu, R Wang and D Quarton, Methods for the rapid evaluation of fatigue damage on the Howden HWP330 wind turbine, British Wind Energy Conference, York, (1993).

89. N.W.M.Bishop , F Sherratt and Hu Zhihua, The Analysis of Non-Gaussian Loadings from Wind Turbine Blades Using Frequency Domain Techniques, British Wind Energy Conference 13, Swansea, 317-321, (1991).

90. N.W.M.Bishop and Hu Zhihua, The Fatigue Analysis of Wind Turbine Blades Using Frequency Domain Techniques, European Wind Energy Conference, EWEC '91, Amsterdam, 246-250, (1991).

91. K.A.Digre, L.K.Brasted and P.W.Marshall. The Design of The Bullwinkle Platform, Offshore Technology Conference OTC # 6050, (1989).
92. D.I.Karsan. Design of Jackets in Deeper Gulf of Mexico Waters, Journal of Waterway, Port, Coastal and Ocean Engineering, ASCE, 112, 427-446 (1986).
93. J.E.Dailey, D.I.Karsan and D.A.Kypke. Comparative Study of Offshore Structure Concepts For Deepwater Applications, API North Pacific Section Meeting, Ventura, Calif, 1-19, (1979).
94. W.J. Pierson and L. Moskowitz. A Proposed Spectral Form For Fully Developed Wind Seas Based on the Similarity Theory of S.A. Kitaigorodskii, Journal of Geophysical Research, 69, (1964).
95. K.S. Bathe and E.L. Wilson. Solution Methods for Eigenvalue Problems in Structural Mechanics, Int. J. Numerical Methods in Engineering, 2, (1973).
96. J M Tunna. Random load fatigue: theory and experiment. Proc Inst Mech Engrs 199(C3), pp249-257,1985.
97. J M Tunna. Fatigue life prediction for gaissian random loads at the design stage. Fatigue Fact Engng Mat Struct, 9(3), pp169-184, 1986.
98. Steinberg, D.S Vibration Analysis for Electronic Equipment. John Wiley and Sons, New York, p223, 2nd edition, 1988.
99. So Rice, Mathematical Analysis of Random Noise, in selected papers on noise and stochastic processes, (N.Wax, ed) Dover, New York 1956.
100. Kreyszig E. "Advanced Engineering Mathematics. Seventh Edition." John Wiley & Sons, inc.
101. Cooley, J.W and Tukey, J.W (1965). "An algorithm for the machine calculation of complex Fourier series." Math. Comp., 19, 297-301.
102. Priestley M.B (1981). "Spectral Analysis and Time Series." Academic Press.
103. Spence P. "What is 'Power Spectral Density'?" The Analyst.
104. ESDU 85020 (1993). "Characteristics of atmospheric turbulence near the ground. Part II: single point data for strong winds (neutral atmosphere)." ESDU international plc. 27 Corsham Street, London, N1 6UA, England.
105. Newland D.E. (1993) "An introduction to Random vibrations, Spectral & Wavelet Analysis, (Third edition)." Longman Scientific & Technical.

Further Reading

1. “Wohler’s Experiments on the ‘Fatigue of Metals,” Engineering Editorial, March 24th, 1871, p199 et seq.
2. Bannantine, Julie A., Comer, Jess J., and Handrock, James L., “Fundamentals of Metal Fatigue Analysis,” Prentice Hall, ISBN 0-13-340191-X, 1990.
3. Fuchs, H.O., and Stephens, R.I., “Metal Fatigue in Engineering,” John Wiley & Sons, ISBN 0-471-052647, 1980.
4. Collins, J. A., “Failure of Materials in Mechanical Design — Analysis, Prediction, Prevention,” John Wiley & Sons, ISBN 0-471-05024-5, 1981.

5. "Fatigue Under Complex Loading — Analysis and Experiments," Society of Automotive Engineering, SAE AE-6, 1974.
6. Hertzberg, R. W., and Manson, J. A., "Fatigue of Engineering Plastics," Academic Press, ISBN 0-12-343550-1, 1980.
7. "Multi-axial Fatigue — Analysis and Experiments," Society of Automotive Engineering, SAE AE-14, ISBN 0-89883-780-4, 1989.
8. Pickard, A. C., "The Application of 3-Dimensional Finite Element Methods to Fracture Mechanics and Fatigue Life Prediction," EMAS, ISBN 0-947817-22-0, 1986.

18 Module Operations

- Terminal Definition 1446
- MASK (X) Mode 1446
- MOTIF (and Windows) Mode 1455
- Modifying the MSC Fatigue Environment (MENM) 1463

Terminal Definition

MSC Fatigue consists of many programs that can be accessed from the forms inside Patran or can be accessed by typing the proper symbol name at the system level. Some of these programs (such as PTIME and PFMAT) display their own graphics and require knowledge of the graphics terminal they are running on. This information is found by looking for an environment variable set in the user's home directory (except for Windows). MSC Fatigue checks the `envi.hom` file for this variable to determine if the Mask or Motif interface is to be used.

A program called MENM is provided which allows the user to create a terminal definition variable which indicates what mode of operation he/she wishes to run. Once a valid `envi.hom` file is created in the user's home directory, it is not necessary to run MENM again unless the user wishes to switch to another graphics device or run MSC Fatigue.

Patran automatically runs MENM whenever the user enters the MSC Fatigue forms. The default graphics display on Linux is the MOTIF driver. As long as the user accesses these program from form picks in Patran, he never has to run MENM manually. The Windows platform does not require any terminal definition.

Therefore, the only time it is necessary to run MENM is when running a MSC Fatigue module like PTIME in stand alone mode from the system prompt. If the user attempts to run a MSC Fatigue program outside of Patran, and there is no `envi.hom` file in the home directory, the Motif driver will be used by default.

To run MENM, type the symbol **menm** at the system prompt.

When MENM appears, modify the Home Environment by adding or altering a keyword. Add the keyword

`$trmtype`

The value that this keyword can take is either

`x`

for the MASK (X) driver, or

`motif`

for the MOTIF driver. This keyword can only be defined in the user's home directory. A file called `envi.hom` is created in the user's home directory. If this file does not exist, it will be created, and the MOTIF driver will be the default.

The MOTIF and MASK modes of operation are discussed in detail in this appendix. The Windows version does not require this environment variable. MSC Fatigue uses a native windows graphical interface common to most Windows programs. However many of the operations described in this appendix are applicable to the Windows graphical interface also and therefore it is suggested to at least familiarize yourself with the MOTIF operation. Actual operation of the MENM program is explained in Modifying the MSC Fatigue Environment (MENM), 1463.

MASK (X) Mode

The MASK mode is the original window (X) environment for the MSC Fatigue modules. Descriptions of the MASK mode of operation follows. Both the mouse and keyboard operations are described as well as graphics and text screens. To invoke the MASK driver, use MENM and add or alter the Home Environment keyword `$trmtype` and give it the value `x`.

$trmtype = x

When running the MSC Fatigue modules in MASK mode, many of the operations are similar, if not identical to the MOTIF mode. For this reason, it is advised to read the section in this appendix on the MOTIF driver also. Actual operation of the MENM program is explained in Modifying the MSC Fatigue Environment (MENM), 1463.

Description of the Text Screen

Certain programs in the MSC Fatigue system use an instruction protocol to handle data input, menu selection and results display. The programs that use this user interface can be recognized easily since the text screen displays use a top and bottom message line and the rest of the screen area is used for data input. An example of this kind of text screen display is shown in Figure 18-1, below. Most MSC Fatigue programs modules on Linux platforms can use this interface such as PFMAT, PTIME, PFPOST, PCPOST, FEFAT, FEFTRN, MMFD, PFXMUL, MCONFIL, PCRACK, SPOTW, PKSOL, etc.

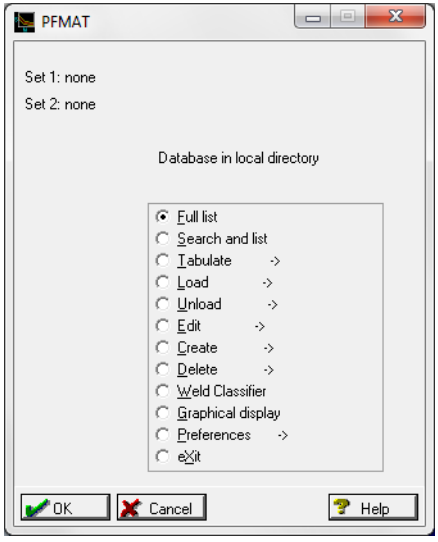


Figure 18-1 Example MSC Fatigue MASK Input Screen

The text screen is comprised of three main parts:

Screen	Description
HEADER	A description line with the following information: ■ Name of the program being used (center) ■ The right and left sides of the header will have the following cursor (mouse) selectable commands: OK - same as using <F1> key - accepts current screen BACK - same as using <F2> key - cancels and returns to previous screen LIST - same as using <F3> key- displays a listing of available choices to be selected HELP - same as using <F4> key - displays help for the current screen
FOOTER	A description line which reminds the user of the function of the <F1>, <F2>, <F3> and <F4> keys and that help for the currently active field is obtained at any point by typing a question mark (?).
MENU BAR	Any menu option from the vertical menu bar may be selected by highlighting the option with the cursor and pressing <RETURN>. Another method is to type the capitalized letter (hot key) in the menu option. (for example, the G in Graph).

Using the Keypad

In most cases, the mouse can be used to make most selections. Aside from using the mouse to make selections when using the MASK driver, other keyboard keys control the position of the cursor and enable the user to move around the text screen. These are described below.

Key	Description
F1	Accepts the current screen and proceeds to the next.
F2	Cancels current operation and pages back to the previous screen.
F3	Invokes a list of selectable items.
F4	Gives help on the currently displayed screen.
?	Gives help on the currently active field.
Arrow keys	Scrolls through a list of items or moves the input cursor when inputting data in a field. The horizontal arrow keys will skip over to another listbox if more than one appear on the screen.
Keypad PgUp / PgDown	The keypad 9 and 3 keys on some keyboards allow for paging up and down through a list.
Return	Caused the cursor to move down vertically in a menu bar.
Space bar	Causes the cursor to move across horizontally in a menu bar. In a toggle field, (indicated by diamonds at either end of the selection field), the space bar will toggle through the options.
+/-	These keys mimic the space bar.
q	Quits a listing and returns to the menu from which the listing was invoked.
#	Allows for input of a command in the bottom bar.
%	Allows for input of a Linux system command.
Hot keys	Hot keys are capital (bold) letters which appear on the screen as menu picks. By typing the letter that option will be selected. (Not case sensitive.)
Tab	The tab key clears a field from the current cursor position. By pressing the Tab key twice, the entire field is cleared.

Special Commands

There are special commands that allow for more functionality than that which is readily apparent from viewing the screen. These are:

Key	Description
%	Clears the bottom bar for entry of an operating system command.
#	Clears the bottom bar for entry of a MSC Fatigue command. The following is a list of the complete command that can be entered, and a brief description (only the capiTAlized letters need to be entered to achieve the response).
#UP	Takes the user up to the previous Input Field.
#HElp	Accesses On-Line Help.
#QUit	Causes the program to stop without saving any of the input data or results.
#ABOrt	Causes the program to abort all operations and stop executing.
#EXit	Causes the program to close all files open and shut down any computation in a clean manner.
#OVerwrite	Causes the screen input to operate in overwrite mode (MSC Fatigue programs normally operate in this mode).
#INSert	This option causes the screen input to switch to insert mode (overwrite is the normal mode of operation).
<Cntl-c>	Same as #ABO. Use with caution.

Correcting Input Errors on the Text Screen

If an error is made when typing data at the input line, it may be edited using the arrow keys to reposition the cursor and then typing the new input data. However, consider the following:

- If there are redundant characters ahead of the cursor, pressing the **Tab** key once will clear anything to the right of the cursor.
- Pressing the **Tab** key twice will clear the entire input field and place the cursor at the beginning of the field.
- MSC Fatigue screen input will normally operate in overwrite mode. To switch to insert mode type:

  #IN
- To return to overwrite mode type:

  #OV
- **Tab** is inoperative in toggle fields. Toggle fields are those that have multiple selections which are chosen by the **+** key or the **Space bar**.

Important:	By typing the % or # characters, the MSC Fatigue modules (such as shown in Figure 18-1) are put into the system level or command modes respectively. Input can then be typed at the bottom bar. To insert the % or # characters in a text string, it is necessary to type a \ character just before the % or #. When the second character % or # is typed, it will overwrite the \.

Inputting a Range of Numbers

When a range of numbers which are separated by an equal increment are to be defined, a short form may be used to save excessive amounts of typing. The required syntax is "(start value, end value, increment)." Note that the parentheses are required and the numbers may be separated by a space or a comma (e.g., 1, 5, 0.5) means values starting at 1.0 with increments of 0.5 to a maximum value of 5.0. This syntax is mainly used in the sensitivity analysis to define a range of values.

List Function Operation

The F3/List function is a quick and convenient way of tagging a large number of files for processing. The F3/List function removes the need for the user to type in the individual names of a large number of files.

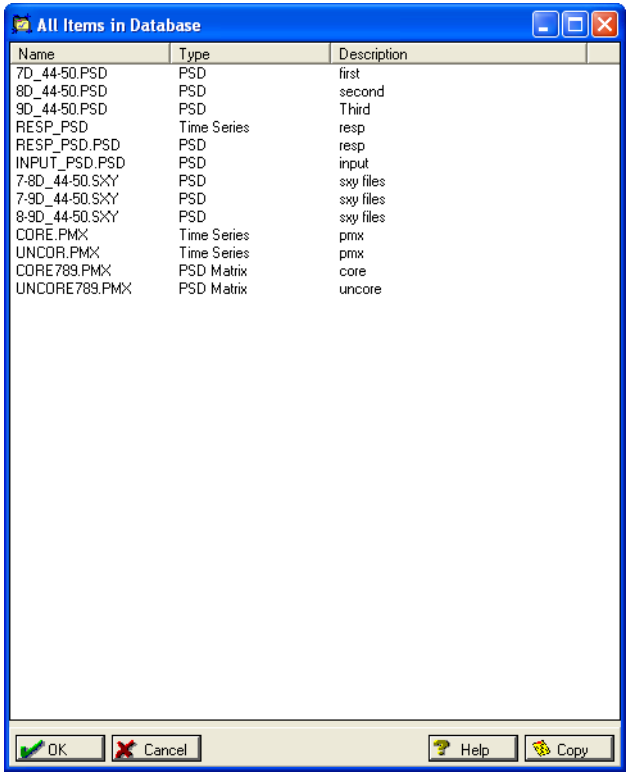


Figure 18-2 The F3/List Option and a Typical List of Files

Both access methods display a list of files on the screen and allow the user to select files by moving a cursor over the files and pressing **Enter** or **Return**, or by block selecting all the files listed, or by pointing and clicking on listed files with a mouse.

The F3/List function can be activated in one of two ways:

1. By clicking the mouse cursor on the List button in the top header bar.
2. By pressing the **F3** key (mainly for non-mouse systems).

The List button appears on the top header bar and clicking anywhere on the button will cause a menu and a list similar to the one in Figure 18-2 to pop-up.

In this case, the vertical list is a list of all the files available in the user directory with a `.DAC` extension (on some of the F3/List screens the current path can be specified).

The vertical scroll bar on the right of the file list indicates that there are too many files to fit in the list window. Press **PgUp** or **PgDn** (keypad keys **9** and **3**) to scroll up/down the list, or click the mouse cursor at the top or bottom of the scroll bar.

The horizontal menu bar details all the actions that can be carried out on the file list.

Options on the horizontal menu bar can be selected by clicking with the mouse, or by pressing the appropriate hot key on the keyboard (the hot key is the capital letter in the option name). An example hot key is the C in a**C**cept.

The menu options all act on the vertical listbox in the way indicated below. (Click the mouse pointer, or use the arrow keys, to move the highlight bar across the option functions.)

Selecting Tag/untag activates the tagging function. When Tag/untag is active the highlight bar can be moved up and down the list of files with the up/down arrows keys.

Pressing **Return** on any file will tag that file if it is untagged, or untag the file if it is tagged. More simply, files can be selected by clicking on the file name with the mouse cursor. Clicking on a file name will automatically make the highlight bar appear over the Tag/untag option.

When all the necessary files have been selected, press **F1** or click on the OK button on the top bar.

If there are no files available on the current drive, path, or directory, an error message to that effect will be displayed. It is possible to specify another drive, path, or directory by using the Path option on some F3/List screens.

Users, who prefer to use the keyboard or who do not have a mouse, can press **F3** to display a list of available files. The files can be tagged by using the keyboard as described above. If there are no files available on the current drive, path, or directory, an error message to that effect will be displayed. It is possible to specify another drive, path, or directory by using the Path option.

The **F3** and tag function is operated by using the up/down, left/right arrow keys to move the highlight bar, and **Enter** to select the highlight option. Pressing **F1**, or C (the hot key for a**C**cept) confirms the tagged choices from the list similar to the **Enter** key.

Graphical Screen Manipulation

The method of operation of the MSC Fatigue graphics screen menus is discussed below. This graphical user interface is used especially by PTIME, PFMAT, PCRACK,PKSOL, MMFD, SPOTW, and FEFAT.

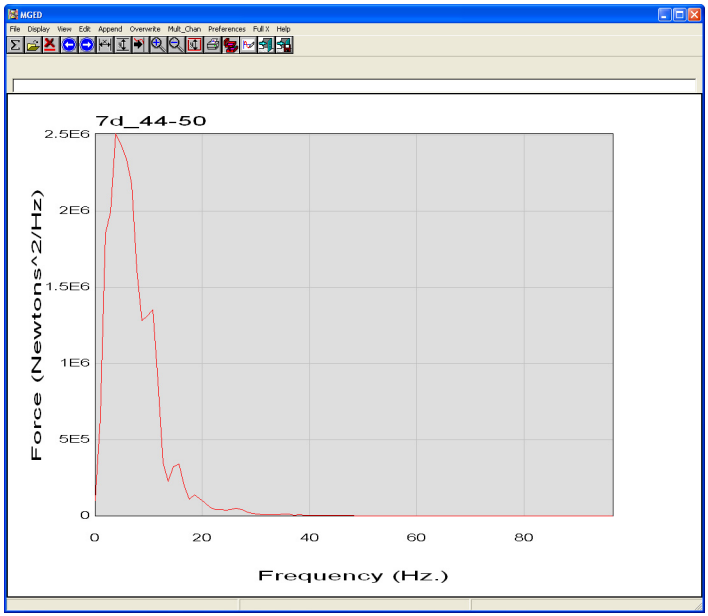


Figure 18-3 Graphical Display Example

Shown in Figure 18-3 is the screen layout of the PTIME graphical screen. However, similar screens are used in PFMAT, PKSOL, PCRACK, MMFD, SPOTW, and FEFAT.

The area at the top left of the screen is the Command Area. In this space, all input data, commands, questions, text output, and help are either entered or displayed. Below the command area is the graph of the signal.

The area down the right-hand side of the screen is the menu area. The top box contains the name of the current menu, and the boxes below show the options available within the current menu. The following section describes the use of the menu in driving the program.

Graphical Mouse Operations

The program is driven from the menu displayed on the right of the screen.

Operation	Description
Picking a Menu Option	When a mouse is being used, menu items are selected by positioning the mouse cursor over the required box and pressing the left mouse button. If the menu item contains a submenu as indicated by being enclosed in < > brackets, then if chosen the submenu is displayed in the menu area.
Defining an X-Y Position	Many options require points to be selected from the graph, for example start and end points for function generation. A point is selected by positioning the cursor at the required location and pressing the left mouse button. It is possible to enter any point as a coordinate pair at the keyboard instead of using the mouse, if required. This is achieved by pressing the key letter K (Keyboard), which allows all point selection for the current option to be specified at the keyboard. Note that coordinate pairs must be entered, even if only one coordinate is required. On completion of any option, the program automatically switches back to full mouse operation. To switch back to full mouse operation in the middle of the option, type UP at the prompt for the point selection.
Obtaining Coordinates	If the right mouse button is pressed with the cursor position anywhere to the left of the menu area, the coordinate value at that point is displayed, either for the main signal or the reference signal, depending on the actual cursor position.
Getting Help	If the right mouse button is pressed with the cursor positioned over the menu area, then help is displayed for the option over which the cursor is positioned. Also the ? key may be used or the F4 key to obtain additional help.
Terminating an Option	If the P key is pressed at any time, the current option is terminated and the whole screen is redrawn.
Terminating Graphical Display	In some instances if the Q key is pressed at any time, the current graphical display will be shut down and the user will be put back into the original calling window or module.
Entering Commands	Invoke the on-line help to obtain a list of command that are available or see below.

Graphical Command Line

By clicking the mouse in the upper left corner of a graphical screen, a command line will appear. A number of commands can be concatenated on the command line, separated by commas, and executed en masse. Commands can be edited. **Tab** clears text to the right of the cursor, left/right **arrow** keys move the cursor along the line of text, **backspace** deletes characters to the left of the cursor. It is error tolerant in that if a string of concatenated commands contains a faulty command then only the faulty command is represented to the user for amending (it is not necessary to type the entire string again). The commands that are valid in MASK mode are also valid in MOTIF mode. A complete list of valid commands are contained in the MOTIF mode section in Table 18-1.

Font Selection (PFSETFONT)

The MASK driver uses up to 10 different fonts. The availability of these fonts determines how many different screen sizes may be assumed by the different modules when the screens are resized. It is possible to change the fonts by running a small utility called PFSETFONT. It may be invoked from the system prompt by typing the symbol **pfsetfont**.

When PFSETFONT is invoked without any parameters a help page comes up explaining its usage. PFSETFONT is executed with the following parameters:

pfsetfont option, fontname, <save_location>

where option is ADD, DEL, LIST, FULL, or ZAP.

ADD adds a new font.

DEL deletes a font added using the ADD option.

LIST shows the current fonts defined.

FULL shows a full list of currently defined fonts.

ZAP will delete all the temporarily added fonts and restore the fonts to the original settings.

fontname is the name of a valid font for the X server that the modules are running under.

save_location is either Local or Global. The default is to save the font information in the local `envi.usr` file. To save it globally will require correct permissions to be set for the global `envi.sys` file.

MOTIF (and Windows) Mode

When running the MSC Fatigue modules in MOTIF mode, many of the operations are similar, if not identical to the MASK mode. For this reason, it is advised to read the section in this appendix on the MASK driver also. To invoke the MOTIF driver, use MENM and add or alter the Home Environment keyword `$trmtype` and give it the value `motif`.

```
$trmtype = motif
```

Actual operation of the MENM program is explained in Modifying the MSC Fatigue Environment (MENM), 1463.

Note:	In Windows, this is not necessary. There is only one basic difference between running under Windows and the Linux MOTIF mode. Instead of two windows, one being a utility window as in Figure 18-4, there is only one window with all the pulldown menus available from the one window. This section makes reference only to MOTIF with most of the discussion being interchangeable with the Windows version.

The MOTIF mode is the default mode when running the modules from within Patran unless changed at the bottom of the main MSC Fatigue form.

All modules running under MOTIF initiate by bringing up two forms initially.

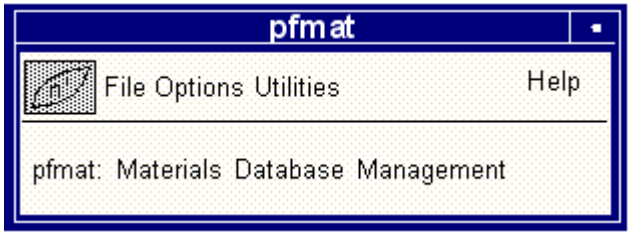


Figure 18-4 The Generic MOTIF Utility Form

The top, small form is a generic, utility form and allows for general program control. It consists of various pull down menus with their respective options. They are discussed below:

Screen Parts	Description
File	This pull-down menu gives the ability to exit from the program or to exit from the currently displayed screen. In general, *Exit* will quit from the program and save any changes. *Quit* will exit from the program without saving any changes. And *Abort* will exit unconditionally.
Options	Two options are available. The first, *Raise Current Window*, is self-explanatory in that the current window is brought to the front if other windows are covering it. The second, *Settings...* brings up a form from which the users can set the *Window History*, *File Access Diagnostics, and Option Box settings*. These setting affect the running of the program. Window History allows previous dialog windows to remain on screen where relevant. No Window History means that only the currently active dialog window remains on screen. The Diagnostics options allow diagnostic messages to be shown in a special window. The Option Box contains many of the commands found in the pulldown menus on the graphics screen and is configurable.
Utilities	Three options are available under this pull down menu: *Change Directory*, *O/S Command* and *Listing*. Change Directory simply brings up a form which allows specification of the working directory. Listing simply lists files in the working directory. O/S Command allows the entry of a operating system command from which a separate window will open showing the results of that command.
Help	To obtain online help at any time, this pull-down menu is available. Four types of help are available. Help *On Question* will explain the meaning of any input databox or listbox on the currently displayed form. Help *On Help* explains what is here. Help *On Window* gives help on the currently displayed window as a whole. Help *On Version* gives the program and library version numbers, plus other information. Help *On Question* is also available at any point with the standard help key, usually F1.

Unlike the MASK driver, which can be operated using the mouse and/or keyboard, the MOTIF driver is strictly mouse driven. Only when inputting data is the keyboard used.

Graphical Options

When graphical screens are displayed using the MOTIF driver, various pull down menus are available from the main form. These pull down menus will differ from module to module. However there are some applications and options which are generic to all modules. These are explained here. Those which are specific are dealt with throughout this guide when describing the individual modules.

A typical graphics form is shown in Figure 18-5. The pull down menus which are common to all graphics forms are the Command line, File, Preferences, and Help. The options available from each of these are explained.

Option	Description
Command Line	All options available under the pull down menus can also be invoked by using the command line. The commands are usually simple 2-4 letter phrases (such as PL for Plot). To learn what these commands are simply use the Help *on Option* under the *Help* menu. See below.
FILE **Hard Copy** **Close** **Exit**	By selecting this option, a `patran.hrd` file is created which can then be converted and sent to a printer for obtaining hard copies of plots. If an existing `patran.hrd` file already exists then the plot will be appended to the `patran.hrd` file. The option will close the existing graphics form. This option generally will exit the user from the program. To return to a non-graphical calling form use the Close option.
PREFERENCES **Pens** **Redraw On / Redraw Off** **Hc Bord On / Hc Bord Off** **Grid Style** **Save Plots**	Color control is accessed through this option. The components of the plot can be changed to different colors dependent on user interface. It is best to experiment with this option. Once a desirable setting of colors has been selected, it can be saved. With Redraw On add operations that cause a change in the graphics will redraw the plot. With Redraw Off, the user must manually have the plot redrawn using the Replot option under the Display menu. When doing a hardcopy, the border around the plot can be turned OFF or ON. Four types of grid styles can be selected: Solid, Fine, Medium, or Thick. Saves the plot as an X-Y data file.

Option	Description
HELP	When this option is selected the cursor turns into a question mark (?) from which the user must select one of the pull down menu options. Help on that particular option will then be given. This is the most effective way to obtain help or an explanation of the option.
On Option	
On Help	
On Window	Gives and explanation of how to use help.
On Version	Gives generic help on the currently displayed form.
	Displays the current versions of the software.

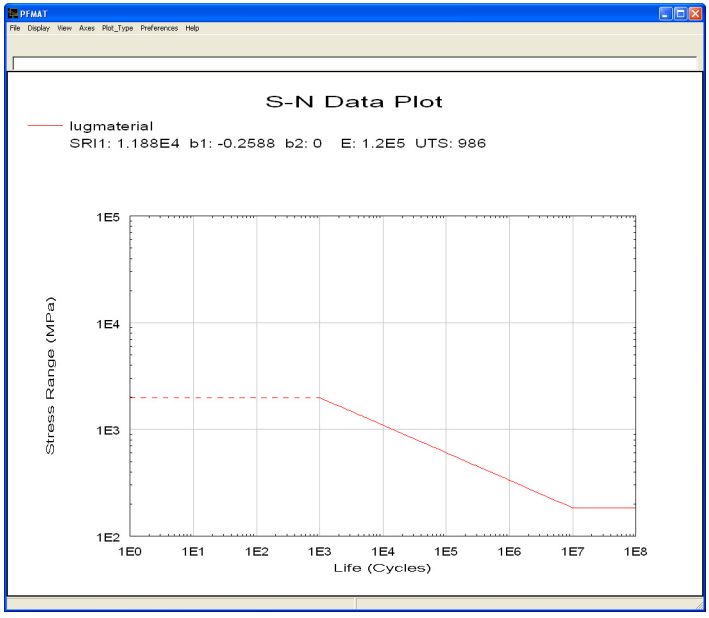


Figure 18-5 A Typical Graphics Screen in the MOTIF Driver

Graphical Command Line

A number of commands can be concatenated on the command line, separated by commas, and executed en masse. Commands can be edited. It is error tolerant in that if a string of concatenated commands contains a faulty command then only the faulty command is represented to the user for amending (it is not necessary to type the entire string again). A complete list of valid commands are contained in Table 18-1. Note that not all commands are available from all modules.

Table 18-1 Graphical CommandsFont Selection

Menu Option	Command	Meaning
FULL PLOT	FU	Plot the full signal. Useful is a zoomed or windowed plot needs to be replotted in full.
FULL X	FX	Plots the full data along the X-axis.
FULL Y	FY	Scales the Y-axis to the max and min of the file.
TOWER	HI	Plot the data as towers (histogram).
SCATTER	SC	Plot the data as a scatter plot.
JOIN	JO	Plots the data by joining points with a line.
POINTS	PO	Plot the data as a scatter plot.
HARDCOPY	HC	Send output to disk storage (`patran.hrd` file).
NEW FILE	NE	Enter a new file to display. Usually this sends the user back to the first (input) screen.
LOG X	LX	Log the X-axis. Log10 scaling may be set on either X or Y from the AXES menu.
LOG Y	LY	Log Y-axis.
LOG Z	LZ	Log Z-axis.
LINEAR X	NX	Normal X-axis, sets the X-axis to linear scaling.
LINEAR Y	NY	Normal Y-axis, sets the Y-axis to linear scaling.
LINEAR Z	NZ	Normal Z-axis, sets the Z-axis to linear scaling.
DB SCALE	DB	Use decibel scaling of the Y-axis. The formula is dB value = 20*log10 (actual value)/reference value. The dB scaling is a comparative value, and the user must decide the level with which the data is compared. The default is the maximum value. A negative dB indicates values smaller than the reference, a positive value indicates greater than the reference.
grid#	GR	Draw a solid grid on the graph.
DASH GRID	DG	Draw a dashed grid on the graph.
FINE GRID	GT1	Grid type 1 (Fine).
MEDIUM GRID	GT2	Grid type 2 (Medium).
THICK GRID	GT3	Grid type 3 (Thick).
NO GRID/TICK	TI	Mark the graph with ticks instead of a grid (both axes).
BORDEF OFF	HBOF	Stops a border being drawn around hard copies. HBON toggles the border back on.
BOX	BO	Place a box around the graph.

Table 18-1 Graphical CommandsFont Selection

Menu Option	Command	Meaning
NO BOX	NB	Remove the box from the graph.
ZERO LINES (ON)	ZE	Draws zero lines on both axes.
NO ZERO LINES (OFF)	NZ	Removes the display of zero lines on both axes.
(PAGE) LEFT	ML=<N>	Move left N seconds. Time plot length is unchanged. Occasionally just LEFT.
(PAGE) RIGHT	MR=<N>	Move right N seconds. Time plot length is unchanged. Sometime just RIGHT.
ZOOM OUT	ZO	Zoom out a factor of 2 or 5 (depending on module).
ZOOM IN	ZI	Zoom in by a factor of 2 or 5.
MOVE UP	MU	Move up.
MOVE DOWN	MD	Move down.
TRANSPOSE	TR	Switch the X and Y axes for sideways viewing.
TRACK	TK	Lock the cursor onto the joined signal trace and give continuous value reading.
STATS ON/OFF	SON/SOF	Statistics about the plot ON or OFF.
W STATS	WS	Display statistics for the current window of data.
F STATS	FS	Full statistics for the entire file, not just the current window of data.
X WINDOW	WX, XMAX=<xxxx>, XMIN=<xxxx>	Sets the maximum and minimum limits on the X-axis. If XMIN is greater than the maximum X, the values will be reversed. If log X is set, the value must be greater than 0. Press RETURN to leave the value unchanged. If XMAX is less than the minimum X, the values will be reversed. If Log X is set, the value must be greater than 0. Press RETURN to leave the value unchanged.
Y WINDOW	WY, YMIN=<xxxx>, YMAX=<xxxx>	Sets the upper and lower Y window as per X window above.
SET TITLE	ST	Set a title string. The string entered will form the title for the display. This title is saved in the extra details area. Once set, it will be shown unless removed using DT. Please note that DT does not remove the title from the extra details area.
DELETE TITLE	DT	Delete title string.

Table 18-1 Graphical CommandsFont Selection

Menu Option	Command	Meaning
REPLOT	PL	Replot the signal.
FULL X	FX	Plots the full data along the X-axis.
FULL Y	FY	Scales the Y-axis to the max and min of the file.
ADD TEXT	AN	Adds a text annotation at the current cursor position.
DELETE ANNOTATION	DA	Deletes any annotation at the current cursor position. Destructive actions such as this will prompt for confirmation before being carried out.
MOVE ANNOTATION	MA	Move an existing piece of text. This action will prompt the user to identify a text string to move and for its new position.
SAVE SETUP	SALL	Save current setup.
PEN (prompt)	PEPR	Set color of prompt.
PEN (Data)	PEDA	Set color of data plot.
PEN (Text)	PETX	Set color of text.
PEN (Axes)	PEAX	Set color of axes.
PEN (Annot)	PEAN	Set color of annotated text.
PEN (Grid)	PEGR	Set color of plot grid.
PEN (Background)	PEBA	Set color of plot background.
PEN (Error)	PEER	Set color of errors.
SAVE	PESA	Save pens settings.
RESET	RE	Removes all recent parameter setups and replots using initial settings.
REDRAW ON	RON	Redraw on. This command causes changes to be shown immediately by the redrawing of the plot.
REDRAW OFF	ROFF	Redraw off. This causes plot changes to be recorded but not displayed until REDRAW ON or RON or PL is carried out.
EXIT	EX	Exit the program — use NEW to plot a different file.
INFO SIDE/TOP/OFF	IF/IT/IO	Place informational data on side or top of plot or turn it OFF.
E-P LINES	EP	Plot elastic and plastic lines.
FATIGUE LIMIT	FL	Plot fatigue limit line.
LINES OFF	NL	Erase any extra lines on the plot.
JOIN POINTS	JP	Join data points together with lines.
HIDE ON/OFF	HID/NHID	Hide lines which would normally be out of view or allow viewing of all lines.
QUADRANT	QD	Change the quadrant the plot is in.

Table 18-1 Graphical CommandsFont Selection

Menu Option	Command	Meaning
ROTATE	ROT	Rotate the plot by a predetermined angle.
TILT	TIL	Tilt the plot by a predetermined angle.
SURFACE	SU	Plot as a surface (three-dimensional plots).
DAMAGE	DM	Plot the damage histogram.
CYCLES	CY	Plot the cycles histogram.
BACK PLANE/NO BACK PLANE	BP/NBP	One three-dimensional plots the back planes may be take on or off.

Fonts and other options for the MOTIF driver are controlled through a resource file delivered with the MSC Fatigue system. A listing of this file is given below.This file is located in the `nssys` directory of the MSC Fatigue system and is called `P3fatigue`. To make use of and change these settings, use the following command

```
xrdb -merge P3fatigue
```

This is generally done during installation of the product and only necessary if changes are desired.

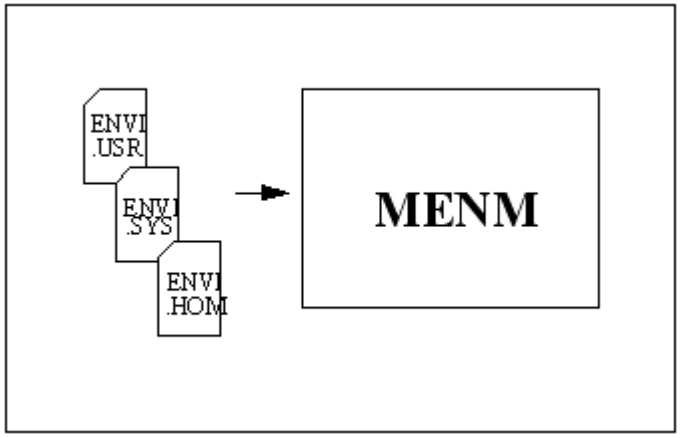
Option	Setting	Description
P3fatigue*Geometry:	+0+120	This is the X-Y position at which the program control window will be initially positioned.
P3fatigue*borderWidth:	0	This is the width of window and menu borders.
P3fatigue*highlightThickness:	2	This is the thickness of the highlight box which marks the field with the current input focus.
P3fatigue*keyboardFocusPolicy:	explicit	Indicates that the pointer must be clicked onto a field for the input focus to move to that field. The alternative value is 'pointer', in which case the input focus moves with the cursor.
P3fatigue*menuAccelerator:	<Key>KP_F2	This is the key that has to be used to activate the menu accelerator keys.
P3fatigue*menulogo*labelType:	pixmap	This is required to allow the display of our logo on the control window.
P3fatigue*monomaskform*fontLis:	iso1.20b	This is the font used for forms which require monospaced fonts.

Option	Setting	Description
P3fatigue*grphform*fontList:	iso1.20b	This is the form used for the graphics window.
P3fatigue*ttbox*fontList:	9x15bold	This is the form used for popped boxes offering lists to pick from. Normally, this is a monospaced font.
P3fatigue*fontList:	lucidasans-bold-14	This is the default font for all cases other than those explicitly set.
P3fatigue*helpbox*fontList:	9x15bold	This is the font for help boxes.
P3fatigue*foreground:	black	This is the foreground color.
P3fatigue*background:	white	This is the background color.
P3fatigue*menulogo*background:	SteelBlue	This is the background color of the logo icon.
P3fatigue*selectColor:	Green	This is the color used to fill the small diamonds which mark current menu settings.

Alternatively, you may take the file called `P3fatigue` that is located in the `nssys` directory of a typical MSC Fatigue installation and place it in `/usr/lib/X11/app-defaults` which will have the same effect.

Modifying the MSC Fatigue Environment (MENM)

The MSC Fatigue user environment manipulation module, MENM, can be used to create, edit, and maintain the environment under which MSC Fatigue modules execute.



The environment records and sets default file names, save types, plot labels, and much more. This guide details which environment keywords are applicable in each module; they can all be set from MENM.

MSC Fatigue modules function within the context of a user environment which determines their performance and behavior. For example, the name of the last time series (.dac) file used is saved in the environment and so when a particular module requires a .dac file as input, it can be offered as a default. In fact the entire MSC Fatigue default handling system is implemented through the user environment.

Most modules routinely access the environment either to read or write information, this process is normally transparent to the user. However, users can deposit specific instructions within the environment which will force program modules to behave in specific ways. For example, the location of plotting labels output by the quick look display (MQLD) module can be moved from the side to the top by setting the environment keyword **PLOTLABL** to **TOP**. The MENM module can be used to manipulate the MSC Fatigue user environment.

The Global, Home and Local Environments

The MSC Fatigue user environment functions on three levels, the **local**, the **home**, and the **global** level. This program allows the user access to the MSC Fatigue environment files, the local environment (`envi.usr`), the global, or central, environment (`envi.sys`) and the home environment (`envi.hom`). The global environment file is in the `nssys` directory on PC or Linux systems and may be accessed by many users.

Instructions saved at the local level will normally override instructions saved at the home level which will override instructions at the global level, so that for example, setting PLOTLABL to TOP locally will override a global PLOTLABL set to SIDE. This feature can be particularly useful on multi-user systems where some functionality needs to be defined globally for all users, but nevertheless, individuals can still redefine some functions locally for themselves. The local file is accessed only by programs running in the directory where the file resides. The environment is used for default settings (e.g. pen colours) and to pass information from program to program (e.g. last used file).

Physically, the MSC Fatigue user environment consists of three data files in which all the required information and commands are stored. The local environment file is called `envi.usr` and is always to be found in the user's current directory or subdirectory. Therefore, each directory from which any MSC Fatigue module is invoked will have a specific copy of `envi.usr` which reflects the usage within that directory. The Home environment is called `envi.hom`, the global environment file is called `envi.sys` and is located within the central MSC Fatigue system directory `nssys`.

The structure of the local, home and global environment files is identical and consists of a random access key-indexed format, which means that these files can only be accessed by the MENM module.

Information and instructions are stored within the environment files in the form of a keyword and an associated value. Modules access the environment by searching for keywords specific to them and then acting according to the associated value. The format of the keyword value combination, together with some examples, are detailed below:

KEYWORD **{8 characters max.}**	**VALUE** **{120 characters max.}**	**MEANING**
PLOTLABL	TOP	Plot Label position
$SIGNAL	TEST101.DAC	Last signal file accessed

$LASTPRG	MMFD	The last used program
$MINERS	1.0	Miners constant

Most of the keyword value pairs to be found in an environment file are of no direct interest to the user, they merely enable program modules to function in the expected way. However, as mentioned above, MENM can be used to deposit specific instructions, such as the value of Palmgren Miner's fatigue damage summation constant, in order to customize performance. The user definable keyword value pairs relevant to particular modules are detailed in the individual module documentation.

Module Operation

The MENM module can be run in one of the following ways:

- In stand alone mode by typing **menm** at the system prompt.
- By incorporating the MENM commands in a batch operation.

The first mode is interactive. Once running in interactive mode, MENM will display the following menu:

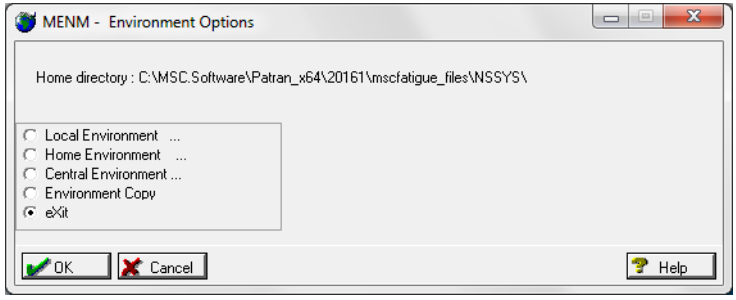


Figure 18-6 The MENM main menu.

All of the three main options (Local, Home, or Central) have the same sub menu, illustrated to the right of the main menu.

The functions of these submenu options are described below. The required option is selected by pressing the appropriate hot key or using the cursor. Alternatively, the highlight bar can be moved over the option and ENTER pressed OR the menu option can be selected by pointing and clicking the mouse pointer.

Option 1 - List environment to screen.

Selecting this option will cause MENM to list the contents of the selected environment file. The listing is directed to the screen, and takes the form of the keyword - value pairs.

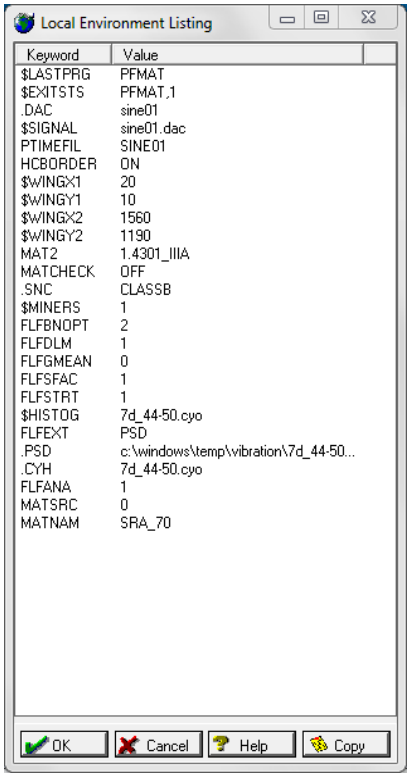


Figure 18-7 A Listing of Keywords

Option 2 - Output Environment to File

If File is chosen as the listing destination then a screen similar to Figure 18-8 below will appear.

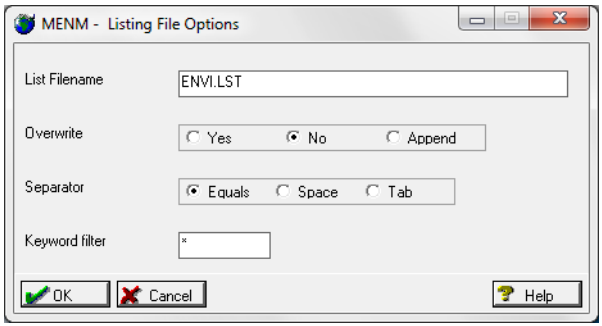


Figure 18-8 Listing destination is File

The fields are as follows:

Screen Part	Description
List File Name	The name of the ASCII file to which the keyword/value pairs will be written. Figure 18-8 above shows one of the default names, and if `envi.lst` should already exists then MENM will overwrite it. Change the file name if this is not the desired result.
Overwrite	This option specifies whether the output file should be overwritten or appended to. If any option is selected and the file does not exist then the file will be created.
Separator	Keyword/value pairs must be separated from each other by either an equals sign, a space, or a TAB character. This field allows the character to be selected.
Keyword filter	To write a subset of keywords to the output file, a keyword filter may be used. An asterisk (*) may be used at the end of the filter string and means, "any keyword beginning with". For example FRED* will list all keywords beginning with FRED. The default is that all keywords will be listed. Omitting the wildcard means that an exact match is required.

Option 3 - Add/Alter keyword

Selecting this option allows individual keyword value pairs to be added to or edited within the local environment file. After selection of this option MENM will display this screen:



Figure 18-9 Add/Alter local entry screen.

If the specified keyword exists within the local environment (use the pick list facility to select one) then the currently associated value is displayed and an alternative value may be entered.

The form of the environment is a set of keyword/value string pairs. The keywords may be up to 8 characters in length, the value up to 120 characters.

If the specified keyword does not exist within the local environment, then the new keyword and the value entered will be added.

Note:	Changes made to the local environment will only apply to the specific directory relevant to that environment.

Option 4 - Remove keywords

Selection of this option will cause MENM to ask for the name of keyword to be deleted.

If the specified keyword exists within the local environment then it together with its associated value will be deleted from the local environment file. If the specified keyword cannot be found then an appropriate error message will be output.

The asterisk (*) wildcard may be used. It must be the last character. For example FRED* will remove all keywords beginning with FRED.

The pick list can be used to tag a number of keywords for mass deletion.

Option 5 - Clear Environment

Selection of this option will cause MENM to prompt with a message such as this:

```
Are you sure you wish to overwrite the local environment
Yes/No?
```

The default to this option is N for No (retain the current local environment). Pressing the ENTER key will select this default. Pressing "Y "or clicking 'Yes' followed by ENTER will cause MENM to delete the local environment.

Note:	Deleting the global environment file affects all users. Use this function with extra care!

19 Limitations and Error Messages

- Program Limitations 1470
- Error Messages 1470

Program Limitations

The following table contains MSC Fatigue program limitations. Changes in these values are distributed to all customers as update pages to this guide and are contained in the Patran Technical bulletins.

Number of data values in a time history	No limit*
Number of load types	99
Number of load units	999
Number of material types	100
(The same material with two different surface conditions is counted as two different materials - 300 materials are possible with spot weld analysis however only 100 combinations are possible since three materials must be defined for each spot weld grouping)	
Number of load cases for strain-life	200
Number of load cases for stress-life	200
Number of load cases for crack growth	200
Number of load cases for spot weld analysis	200
Number of input PSDs for vibration fatigue	20
Number of Frequency Steps per Loading PSD	8192
Number of nodes/elements	No limit*
Number of time steps for transient analysis	No limit*

* Limited only by disk space, memory and swap space.

Error Messages

The following are possible error messages, their corresponding explanations and possible solutions. Only messages which are not self explanatory are elaborated upon. This list is not a comprehensive listing.

PCL Form Messages

Jobname "%A%" already exists. Do you want to overwrite (yes or no?)
Enter a jobname first!
Select a job first!
Error reading fin file!

Usually more information comes with this message. Check the `jobname.fin` file for improper parameter cards.

Error reading fnf file. Material information may not be complete!

When the `jobname.fnf` file is not read properly, materials spreadsheet on the Materials Info form does not get filled out completely.

Version number must be #2.5 or #3.0 on first line!

Check the jobname.fin file.

No equal sign in the following line:%A%

Check the jobname.fin file.

There is no 'ANALYSIS TYPE' defined!

Check the jobname.fin file.

The first parameter must be 'ANALYSIS TYPE'!

Check the jobname.fin file.

There is no 'FEA RESULTS LOCATION' defined!

Check the jobname.fin file.

The second parameter must be 'FEA RESULTS LOCATION'!

Check the jobname.fin file.

The %A%.fnf file does not exist. The material information will be incomplete!

When the jobname.fnf file is not read properly, materials spreadsheet on the Materials Info form does not get filled out completely.

File %A%.fnf is not a proper neutral file!

Check the format.

More materials defined in .fnf file than in .fin file!

Only five materials are allowed. The program keys off of the set ID of neutral packet types 10 or 11. When an ID changes during a sequential read, it assumes that a new material is being defined. The IDs must be sequential also. Otherwise this message can occur. Those IDs encountered after five materials have been detected are ignored.

Time History %A% not found!

Check that the appropriate filename.dac file exists and that it has been loaded properly into the PTIME module. A file called ptime.adb should have the header information for the time history in it.

File %A% does not exist.
Materials Information not properly defined!

A job submission is not allowable until the materials information is filled out properly in the Materials Info form.

Load Information not properly defined!

A job submission is not allowable until the loading information is filled out properly in the Loading Info form.

There are no nodes in region %A%.

A nodal fatigue analysis has been detected but the group defined for analysis has no nodes in it.

There are no elements in region %A%.

An elemental fatigue analysis has been detected but the group defined for analysis has no elements in it.

Warning: The system environment variable p3fpath has not been set properly. No job submission or module spawning will be possible! The MSC Fatigue executable is not loaded on this computer system. Check that the environment variable p3fpath is set properly!

Check with your system manager to determine if you are licensed to run MSC Fatigue on this machine and that all environment variables and paths have been set up properly.

No Patran FEA job has been specified!

No Patran FEA `job.res` file has been selected in the listbox on the Loading Info form.

Select a compliance function first!
No compliance function has been selected!

No `filename.ksn` file has been selected in the listbox on the Solution Parameters form for Crack Growth.

No compliance function template data exists. Run PKSOL!

Two files are created when plotting a compliance function. One contains the XY data (`filename.xyd`) and the other (`filename.tem`) contains PCL function calls to make the plot and plot the data. This is the template file and is created in PKSOL when a compliance function is created. PKSOL needs to be run and the compliance function redefined in order to create this file.

No compliance function XY data exists for %A%. Run PKSOL!

Two files are created when plotting a compliance function. One contains the XY data (`filename.xyd`) and the other (`filename.tem`) contains PCL function calls to make the plot and plot the data. If either one of these do not exist, PKSOL needs to be run and the compliance function redefined in order to create these files.

Plot template information not compatible with curve data. Plot may be in error!

When plotting compliance functions or sensitivity data two files are created. PKSOL creates an XY data file (`filename.xyd`) and a template file (`filename.tem`) containing PCL commands as does DESOPT. If this message occurs simply redefine the compliance function or replot the sensitivity data in DESOPT to recreate these files.

Select sensitivity data first!
The sensitivity x-y data file, %A%, does not exist. Run FEFAT!

Two files are created when plotting sensitivity data. One contains the XY data (`filename.fal`/rfl/...) and the other (`filename.tem`) contains PCL function calls to make the plot and plot the data. This is the template file and is created in FEFAT when sensitivity data is plotted. FEFAT needs to be run and the sensitivity data replotted in order to create these files.

Life results for a Component S-N job relate to the component or critical feature (e.g., a weld). Life contour plots may be misleading.

It is not recommended to make fringe plots of Component S-N jobs since component S-N results are based on an S-N curve of a specific geometry (such as a weld) whereas the rest of the structure's fatigue life (such as away from the weld) will not follow that predicted by the S-N curve.

Do you wish to read in existing job '%A%'?
Delete job '%A%' and all associated files?

The material %A% is not iron or steel and therefore surface finish and treatments are invalid. They will be ignored in the analysis.

Steel is the only material in which surface finish and treatments are valid.

It is not necessary to select Time Steps. All Time Steps appearing in the Listbox will be used in the analysis. Use the Filter form to modify the contents of the Listbox.

All time steps appearing in the listbox will be used in a fatigue analysis using transient data. Use the Filter form to eliminate unwanted time steps or other unwanted results appearing in the listbox.

It appears that Jobname "%A%" is still running from a previous submittal. Do you still want to submit this job (yes or no?)

On submission of a fatigue analysis throughout the PCL forms, a file is created to signal that a job is running. If a job aborts unceremoniously, this file will not be deleted properly. It may be necessary for the user to manually delete it.

The PAT3FAT translation was not successful. No input file was created. Check the message file for possible problems.
Warning: A normal vector file "%A%.vec" has been detected. Do you wish to do surface stress/strain extraction?
Shells and solids have been detected in the same group. Please separate them into distinct groups in order to calculate surface normals.

When calculating nodal normal vectors, only shell or only solid elements are allowed in the current group.

The "Current spectrum" as been set to "fatigue spectrum" for life contouring. Change it under "Display/Spectrums" if necessary.

When fatigue results are read into the database a new spectrum is created for plotting life results. Damage results and biaxial parameters should use the Standard Spectrum.

A biaxiality analysis has been requested but normal vectors have not been calculated to extract surface stress/strain.
("Calculate Normals" option in the "Control Job" form.) Are the stresses/strains already surface resolved (Y/N?)

Run the Calculate Normals option if surface stresses are required.

Translator (PAT3FAT) Messages

First argument must be the jobname!

PAT3FAT must have a jobname as the first argument. (e.g., **pat3fat myjob**)

End of PAT3FAT translation.
Messages output from the PDA %A% translator:%A% %A% on %A% at %A%. \n
Total execution time in real seconds: %G%
Total execution time in CPU seconds: %G%
*****Error: Not able to open message file.**
*****Error: Not able to open .fin file.**

PAT3FAT could not find a `jobname.fin` file for the submitted job.

*****Error: Not able to open .fnf file.**

PAT3FAT could not find a `jobname.fnf` file for the submitted job.

*****Error reading .fin file. Check message file.**
*****Error reading .fnf file. Check message file.**
*****Error writing .fes file. Check message file.**
*****Error: Not able to open .vec file.**
*****Job aborted due to user request.**
PAT3FAT reading the FIN file...
PAT3FAT reading the FNF file...
PAT3FAT reading the FE results...
PAT3FAT writing the FES input file...
PAT3FAT terminated normally.
*****Error: Invalid version number on the first line of the .fin file. It must be "#3.0".**

Check `jobname.fin` file.

*****Error: There is a line in the .fin file that is missing an equals sign.**

Check `jobname.fin` file.

*****Error: The first parameter in the .fin file must be the ANALYSIS TYPE.**

Check `jobname.fin` file.

*****Error: An invalid ANALYSIS TYPE has been detected. You must use CRACK INITIATION, NOMINAL STRESS COMPONENT, CRACK GROWTH, or NOMINAL STRESS MATERIAL.**

Check `jobname.fin` file.

*****Error: An invalid RESULTS LOCATION has been detected. You must use NODE or ELEMENT CENTROID**

Check `jobname.fin` file.

*****Error: Input %I% for %A% is outside the valid range (%I% to %I%).**

Check `jobname.fin` file.

*****Warning: Invalid parameter in the .fin file ignored for %A% = %A%: %A%**

Check `jobname.fin` file.

*****Error: Invalid TENSOR TYPE detected. You must use STRESS or STRAIN.**

Check `jobname.fin` file.

*****Error: Invalid TENSOR TYPE detected for the ANALYSIS TYPE selected. Set it to STRESS.**

Check `jobname.fin` file.

*****Error: Invalid STRESS UNITS detected in the .fin file. Use only MPA, PASCALS, PSI, KSI, or KG/M**2.**

Check jobname.fin file.

```
***Error:  Invalid STRESS or STRAIN COMBINATION. Use only MAX ABS
PRINCIPAL, SIGNED VON MISES, SIGNED TRESCA, or one of the
components.
```

Check jobname.fin file.

```
***Error:  Invalid MEAN STRESS CORRECTION method detected for
given ANALYSIS TYPE.
```

Check jobname.fin file. This could be the ANALYSIS METHOD also.

```
\n***Error: Input %F% for %A% is outside the valid range ( %F% to
%F% ).
```

Check jobname.fin file.

```
\n***Error reading the %A% parameter.
```

Check jobname.fin file.

```
\n***Error:  Zero (0.0) not allowed in parameter %A%.
```

Check jobname.fin file.

```
***Error: Invalid CRACK LENGTH UNITS have been detected. Use only
INCHES, METERS, MILLIINCHES, or MILLIMETERS.
```

Check jobname.fin file.

```
***Error:  Transient FE analysis results not allowed for CRACK
GROWTH analyses.
***Error:  Invalid FEA ANALYSIS TYPE detected.  It must be either
STATIC or TRANSIENT.
```

Check jobname.fin file.

```
***Error:  Invalid FEA RESULTS TYPE detected.  It must be either
DATABASE, Patran FEA, or EXTERNAL.
```

Check jobname.fin file.

```
***Error:  Invalid external RESULTS COLUMNS detected.  There must
be six separated by commas and they must be in the range 0 to
200.
```

Check jobname.fin file.

```
***Error:  Invalid SHELL SURFACE parameter as been detected.  It
must be set to either TOP or BOTTOM or not included at all.
```

Check jobname.fin file.

```
***Error:  FEA load case informational parameters must be
preceeded by a LOAD CASE parameter in the .fin file.
```

Check jobname.fin file.

```
\n***Warning:  An unrecognized parameter has been detected in the
.fin file: %A%
```

Check jobname.fin file.

*****Error: Invalid TRANSFORMATION value detected. It must be either NONE or BASIC.**

Check `jobname.fin` file.

*****Error: An invalid load case ID has been detected in a FE LOAD CASE ID card. The ID must look like "n.n-" when referencing database results.**

Check `jobname.fin` file.

*****Error: It appears that the .fnf file is not a properly formatted neutral file. The first record must be packet type 25.**

Check `jobname.fnf` file format.

*****Error: No data of set IDs %I% - %I% found in the neutral (.fnf) file. Make sure at least one group of nodes and/or elements has been defined for fatigue properties.**

Check `jobname.fnf` file.

*****Error: No nodes or elements appear in the .fnf neutral file packet 26 summary. Not able to allocate memory.**

Check `jobname.fnf` file. Check that proper regions (groups) have been set up for the fatigue analysis and that they have nodes or elements in them.

*****Error allocating memory in neutral file (.fnf) read routine.**
*****Error allocating memory in fin file (.fin) read routine.**

Call MSC Software Corporation.

*****Error reading the Patran FEA results file.**

Check the `job.res` file.

*****Error: There are no nodal stress/strain results in the indicated Patran FEA results file.**

Check the `job.res` file.

*****Error: There are no load cases in the indicated Patran FEA results file.**

Check the `job.res` file.

*****Error: There are no static results in the Patran FEA results file.**

Check the `job.res` file.

*****Error: There are no transient results in the Patran FEA results file.**

Check the `job.res` file.

*****Error: The number of time steps requested exceeds the number of time steps in the Patran FEA results file.**

Check the `job.res` file.

*****Error allocating memory in FE results read routine.**

Call MSC Software Corporation.

*****Error reading Patran FEA results for load case header for load case/step number %I%.**

Check the `job.res` file.

*****Error reading Patran FEA results for load case/time step number %I%.**

Check the `job.res` file.

*****Error: There are no Patran FEA results for shell elements in the results file.**

Check the `job.res` file.

*****Error opening the specified Patran database.**

Call MSC Software Corporation.

*****Error in external results file name(s). There must be a "#" in the place of the load case or time step number.**

Specify the file name as `sub#.nod` as an example, where `sub1.nod`, `sub2.nod`, and `sub3.nod` would be read if the job had been set up for three load cases.

*****Error reading an external results file. See the message file.**

Check the external file format.

*****Error opening external file %A%, for load case %I%.**

Check the external file format.

*****Error reading external file header in file %A% for load case %I%.**

Check the external file format.

*****Error reading external results in file %A% for load case %I%.**

Check the external file format.

*****Error closing external file %A%, for load case %I%.**
*****Error allocating memory while writing output (.fes) file.**

Call MSC Software Corporation.

*****Error creating (opening) the fatigue input file (.fes).**

Call MSC Software Corporation.

*****Error opening Patran FEA (.res) results file.**

Check the `job.res` file.

*****Error writing the header to the .fes file.**

Call MSC Software Corporation.

*****Error writing stress/strains data to .fes file.**

Call MSC Software Corporation.

*****Error closing the fatigue input file (.fes).**

Call MSC Software Corporation.

```
***Error reading results from the Patran database.
```

Call MSC Software Corporation.

```
***Error:  No global variables of type "Time" are associated with
results case %I%.%I%.
```

To use transient results from the data base in a fatigue analysis, they must have a tag (global variable) associated with them. This tag is "TIME." Call MSC Software Corporation if standard results importing techniques are being followed.

```
\n***Error:  There are no appropriate stress/strain tensor
results associated with results case %I%.%I%.
```

Results in the database do not have tensor stress or strain result types in them. Try re-importing the results and make sure the appropriate results types are not being filtered out inadvertently.

```
Using results type "%A%" of ID %I% for load case/time step %I%.
***Error allocating memory to read database results.
```

Call MSC Software Corporation.

```
***Error:  Results load case %I%.%I% is associated with Nodes yet
an Elemental based fatigue analysis has been selected.  This is
not possible at this time!
```

Change the analysis to be a nodal fatigue job.

```
***Error:  There are no elements in the model.  Cannot continue
with translation.
***Warning:  A results type "%A%" of ID %I% was encountered. If
you wish to use it, change the FEA RESULTS LOCATION to ELEMENTAL
or ELEMENT CENTROID.
```

Change the analysis to be an elemental fatigue job.

```
***Warning:  A results type "%A%" of ID %I% was encountered. If
you wish to use it, change the FEA RESULTS LOCATION to NODE or
NODAL.
```

Change the analysis to be a nodal fatigue job.

```
MSG_FAT_SURFACE_STRESS_EXTRACTION Surface stress/strain
extraction has been initiated by detection of normal vectors
file. Check results carefully since this has feature has not been
validated.
***Error allocating memory for surface stress/strain extraction.
```

Call MSC Software Corporation.

```
***Error:  There are no nodes in the model.
***Error reading the ".vec" file.
***Error:  There are more nodes in the ".vec" file than there are
in the model.
```

This should not happen. Check the regions (groups) defined for the fatigue analysis.

Analyzers (FEFAT, PCRACK)

Only FES files with a revision level of 2.0 or greater are supported.

The `jobname.fes` file format changed after MSC Fatigue version 1.0. Check to see that "File Revision = 2" in the ASCII format of `jobname.fes` file. Use FESTXT.

The number of load cases must be less than 40 and greater than zero.
This file contains one or more zero load values.

Something went wrong when defining the load cases. Redefine the job setup.

This file contains information for static load cases only. No damage will accumulate therefore processing is terminated. Static load cases are only allowed when mixed with dynamic load cases.

Make sure at least one load case has a load time history associated with it.

This file has an invalid analysis type. If you use the program FESTXT and examine the output file, the "Fatigue code" value in the header must be set to 1 for critical location analysis, 2 for component S-N analysis or 4 for material S-N analysis. Any other value is invalid.
Nodal strain data is only allowed in the "local strain" analysis. Stress data must be used for stress-life and critical location analyses.
The analysis type of this file is invalid. Only 0 for linear static and 1 for transient dynamic are allowed.

Check that the "Analysis Code = " either 0 or 1 for nodal data or 10 or 11 for elemental data in the ASCII form of the `jobname.fes` file. Use FEFAT.

Exiting due to lack of input.
Default not allowed.
Enter Yes or No !!
This is not a valid time history file.

Validate the DAC file using PTIME; check to see that there are cycles (peaks and valleys) in the file.

Maximum and minimum of this signal are the same. This file cannot be cycle counted.

There are no peaks and valleys in the time history.

The sample rate of all time histories must be identical.

Use PTIME to change the sample rate.

The number of values in all time histories must be identical.

Use PTIME graphical editor or X-Y point editor to adjust length of all P(t)s to be identical. This can be accomplished by just extending the length out to be identical.

The same time history cannot be used for different load cases.
Time history not found in database - run PTIME.
Incompatible loading type - run PTIME.

The load type is defined in the jobname.fes file for each load case. The load time history supplied must have the same load type as that expected by FEFAT. Load type can be changed using PTIME. In some cases, deleting the ptime.adb and ptime.mdb files and re-loading all the *.dac files in PTIME can clear this problem.

Incompatible units - run PTIME.

The unit type is defined in the jobname.fes file for each load case. The load time history supplied must have the same load type as that expected by FEFAT. Units can be changed using PTIME units conversion capability. In some cases, deleting the ptime.adb and ptime.mdb files and re-loading all the *.dac files in PTIME can clear this problem.

A value must be entered.
The number of fatigue equivalent units must be positive.
An unspecified error has occurred in the fef analyser.

Check executable and directory privileges and links/symbols. Try running directly from the system prompt.

An error has occurred loading the fatigue analyser.

Check executable and directory privileges and links/symbols. Try running directly from the system prompt.

A read/write error has occurred on either the input or output file.

Check directory and file privileges.

The fef analyser was unable to find one or more material data sets.

Check that material name exists in PFMAT databank and pfmat.adb file.

An calculation error occurred in the fatigue analyser.

Check scaling of inputs for realistic stress/strain values.

An error occurred calculating the surface finish stress concentration.

Check that the UTS value for the material is reasonable.

A linear elastic stress exceeds 1.0E10. This has been detected during the superposition calculation prior to calculating a combined strain parameter.

Check scaling of inputs for realistic stress-strain values.

An error has occurred opening the output file.

Check privileges.

An error has occurred opening a time history file.

Check privileges. Does the DAC file exist; validate using PTIME.

An error has occurred initialising the rainflow count.
An error has occurred reading the time series data.
An error has occurred in the rainflow counting process.
An error has occurred accumulating the cycles for processing.
Warning - no turning points at this node.

There are no peaks or valleys in the load time history.

No reference nodes have been found in this file.

Check the `jobname.fes` file that the reference node/element is there. Use FEFAT.

Error writing nodal information to the fpp file.

Check privileges.

S-N data set not found - #

Check that material name exists in PFMAT databank and `pfmat.adb` file.

Unable to open database to read material #

Check privileges.

Unable to read or close database searching for material #

Check privileges.

Material not found - #

Check that material name exists in PFMAT databank and `pfmat.adb` file.

No strain data available for material #

Check that the appropriate dataset exist for the material.

Invalid or unknown Young's Modulus or UTS for material #
Duplicate data sets found for material #

Delete one of them using the PFMAT delete option.

No component S-N data available for data set #

Check that the appropriate dataset exist for the material.

Unable to open database to read data set #
Unable to read or close database searching for data set #
Data set not found - #

Check that the appropriate dataset exist for the material.

No S-N data available for data set #

Check that the appropriate dataset exist for the material.

Invalid or unknown UTS for data set #
Duplicate data sets found for S-N data #
Error closing down rainflow count - Stack empty

Report to MSC.

Error closing down rainflow count - Failure in ZIP routine

Report to MSC.

Error closing down rainflow count - Too many cycles closed at once.

Report to MSC.

Error closing down rainflow count - Invalid stack

Report to MSC.

Error in preprocessor during input phase#

Check jobname.msg file.

Error during preprocessing - see message file for details#

Check jobname.msg file.

Unable to load fatigue analyser FEF#

Check jobname.msg file.

User abort of preprocessor. Job terminated.#

Check jobname.msg file.

An error has occurred setting up the tables for fatigue analysis.

Report to MSC.

Too many reference nodes have been specified. A maximum of 5 is allowed.

Only five (S-N) materials allowed; therefore, only five reference nodes/elements allowed.

Error calculating surface finish. Either UTS is unavailable, the surface file has not been found or the UTS is out of the allowed range.
Mean stress has exceeded fatigue strength coefficient (Sf'). Check your load case scaling.

Check scaling of inputs for realistic stress-strain values.

Error in fatigue analysis module during input phase#

Check jobname.msg file.

Error in processing during fatigue analysis - see .msg file#

Check jobname.msg file.

User abort of fatigue analysis - Job terminated.#

Check jobname.msg file.

This file is of an unknown analysis type. If you use the program FEFAT and examine the output file, the "Fatigue code" value in the header must be set to 3 for crack growth analysis. Any other value is invalid.
Nodal strain data is not allowed for crack growth analysis. Stress data must be used for this analysis.
Program FEFAT terminated by CNTRL-C, CNTRL-Y or abnormal exit.
Error in preprocessor during input phase#

Check jobname.msg file.

Error during preprocessing - see message file for details#

Check jobname.msg file.

Unable to load crack growth analyser PCRACK#

Check jobname.msg file.

User abort of preprocessing - Job terminated.#

Check jobname.msg file.

The specified environment was not found. The program will use the data for air then ask further questions which will enable the simulation of the change in environment.
This input file does not contain any tension stress cycles. It can therefore not be processed.

Cracks cannot grow if they are always experiencing crack closure.

The divisor in the KSN file is invalid.
No keyword found.
No value found.
Value is not a real number.
Error reading local material database.
Error reading central material database.
Material exists in the database, but there are no LEFM data sets associated with it.
No material entered. Program exiting.
The specified environment was not found. The program will use the data for air then ask further questions which will enable the simulation of the change in environment.
Unable to create toggle file correctly. Please contact nCode International Ltd / MacNeal-Schwendler Corporation.
Notch depth and initial crack size cannot both be zero.
Notch depth must not exceed specimen thickness.
The material identity you have entered either does not exist in the materials data bank or there are no fracture mechanics data associated with this material name.
To obtain a list of materials which have fracture mechanics data stored in the data bank, use the F3/List option. To define new fracture mechanics data for this analysis only, select any valid material and edit the parameters on the next page. To store fracture mechanics data in the materials databank permanently, use the materials data manager, PFMAT.
The material selected could not be found on the database !!
Possible causes are: database corrupted, database changed while PCRACK was accessing it
The environment entered is not applicable to the material selected
An error occurred while Writing a Toggle File

Report to MSC.

An error occurred while Setting the Toggle File line length

Report to MSC.

Error in input phase of PCRACK (time history + k solution) #

Check jobname.msg file.

Error in input phase of PCRACK (materials) #

Check jobname.msg file.

Error in input phase of PCRACK (crack lengths) #

Check jobname.msg file.

Error in input phase of PCRACK (notch details) #

Check jobname.msg file.

Error in input phase of PCRACK (environment) #

Check jobname.msg file.

Error in input phase of PCRACK (history effects) #

Check jobname.msg file.

Error opening input files for PCRACK #

Check jobname.msg file.

Error opening output files for PCRACK #

Check jobname.msg file.

Error in calculation of crack length #

Check jobname.msg file.

Error closing input files for PCRACK #

Check jobname.msg file.

Error closing output files for PCRACK #

Check jobname.msg file.

Error closing output files for PCRACK #

Check jobname.msg file.

Error writing session file in PCRACK #

Check jobname.msg file.

Analyzer FEVIB

#P6150 NUMERICAL EXCEPTION ERROR!

A zero divide error has been encountered.

All the NUMERICAL EXECPTION ERROR errors occur during the fatigue calculations from the PSD. They typically occur if the PSD is full of zeros.

#P6160 NUMERICAL EXCEPTION ERROR!

A numerical overflow error has been encountered.

Possible causes are :

Incorrect units defined on input PSD.
Incorrect calibration factor on 'Other' units.

#P6165 NUMERICAL EXCEPTION ERROR!

A negative argument was passed to a square root function.

#P6167 NUMERICAL EXCEPTION ERROR!

A zero or negative argument was passed to a logarithm function.

#P6168 NUMERICAL EXCEPTION ERROR!

A negative value was raised to a real power.

#P20001 Not enough memory to run this FES file #

This message appears if a large FES file is used and there is not very much free memory available. FEVIB might be able to run the FES file if more memory is made available, either by adding memory or increasing the swap file size.

#P20002 Possibly not enough memory to run this FES file #
#P30002 Syntax error in file #

This occurs if the PMX file does not have a number on the first line.

#P30003 Incorrect number of load cases in #

The number of load cases in the PMX file is not equal to the number of load cases in the FES file.

#P30004 Number of loads out of range #

The number of load cases is either less than 1 or greater than 20.

#P30005 Keyword NONE cannot appear on diagonal of PMX matrix #

The PMX File can only have the word NONE on non-diagonal members. An example PMX file is:

```
3
s01.dac NONE              NONE
NONE                       s02.dac NONE
NONE                       NONE s03.dac
```

#P30009 Cannot allocate enough memory #

The PSD file contains too many points. A smaller PSD file should be used. (PSD files are usually less than 10,000 points).

#P30010 Negative value in PSD file #

Values in the PSD must be greater than zero.

20 MSC Fatigue Information

- About MSC Fatigue 1488
- Using MSC Fatigue 1488
- Common MSC Fatigue Issues 1488
- MSC Fatigue and a Corporate Durability Management System 1489
- More Information about nCode International 1489

About MSC Fatigue

MSC Fatigue is an advanced fatigue life estimation software package for use with finite element analysis results. It provides state-of-the-art fatigue design tools which can be used to optimize the life of a product before the prototyping stage.

MSC Fatigue has been developed by nCode International Ltd. in conjunction with MSC Software Corporation. nCode is a recognized world leader in durability management, providing fatigue modeling systems for measurement, test, and analysis engineers. nCode has been working in this specialist area for over 12 years and their consultant engineers and materials scientists have over 200 years of experience in this field between them.

Using MSC Fatigue

There are two basic rules for any fatigue analysis:

1. The accuracy of the inputs is critical to the acceptability of the answers.
2. Fatigue life estimates are always approximate, i.e., two results within a factor of two (2) of each other are essentially the same result.

Remember, to set up a fatigue job you need three (3) inputs:

1. Load-time histories for each concurrent time varying load case.
2. Individual FEA results for each independently time varying load case.
3. Appropriate materials properties.

The job setup requires the association of the first two inputs and the assignment of the third to the selected areas of the finite element model (nodes/elements) using Patran group facility. You will also need to define some analysis specific parameters such as a mean stress correction method and the stress/strain parameter.

Common MSC Fatigue Issues

Loading

Realistic loading time histories are often difficult to obtain. A measurement of these is required during typical and extreme operating conditions. This is normally carried out by a dedicated measurement team using transducers and data acquisition equipment. MSC Fatigue time history files are automatically generated using the nCode data acquisition products (see More Information about nCode International, 1489). This type of integration contributes to the overall efficiency of product life assessment.

Materials

Cyclic materials data are required for modeling fatigue. These are normally obtained by testing smooth specimens in a laboratory using constant amplitude loading. The raw materials test data must be processed to obtain the cyclic properties used in MSC Fatigue. The nCode materials test analysis products provide the additional tools required to capture and process the raw data (see More Information about nCode International, 1489).

Geometry

The quality of the FEA results and hence, fatigue life estimates is critically linked to the FEA mesh quality. It is important to represent the geometry accurately in the critical areas (not necessarily over the whole model). Good representation of the geometry will be aided by the use of quadratic elements which can better represent curved shapes. Quadratic elements are also more accurate when analyzing complex stress fields in the regions around stress concentrations. Joints between elements of widely differing sizes, especially where stress gradients are high, will give erroneous results. The refinement of the mesh until no significant change in local stress occurs (h-convergence) is strongly recommended since correct global stiffness is a necessary but not sufficient condition for good fatigue analysis.

Fatigue Test

Any fatigue analysis should ultimately be checked out by testing. Fatigue testing often takes a long time and is expensive to set up and run. However, these drawbacks are not a reason to avoid testing, but it is important that the usefulness of the test is maximized. In practical terms, this means monitoring the test and correlating it with the MSC Fatigue analysis. MSC Fatigue allows the user to output a local elastic strain-time history which could be compared with local strain measurements under test conditions.

It is often possible to shorten variable amplitude tests, using fatigue editors which remove the non-damaging portions of the time history. The benefit in reducing test times and hence costs are very attractive, especially if the product can be put on the market earlier. Such editors are part of the associated software systems from nCode (see More Information about nCode International, 1489).

MSC Fatigue and a Corporate Durability Management System

MSC Fatigue on its own cannot shorten the time to market for a new or changed product. It must be part of a corporate approach to durability and in particular fatigue design. For example, realistic product life estimates can only be obtained if accurate materials data are available and the loading sequences are well defined or at least typical. This means that a strategy must be adopted for obtaining both of these items. Materials data could be obtained from test by external specialists if necessary (including nCode/MSC Software Corporation).

Loading information usually requires a measurement team to instrument and collect the required load time histories under realistic conditions. It is also normal to test a product in fatigue prior to releasing to market. MSC Fatigue helps to eliminate unnecessary tests by allowing the engineer to check out fatigue performance analytically and to optimize the selection of material, manufacturing process, and geometry, within the constraints of total cost and loading environment. Carrying out this optimization analytically, rather than by testing, can shorten the time to market and reduce costs significantly.

More Information about nCode International

nCode International Ltd. is based in the heart of England in a city called Sheffield, which is famous for its specialist steel production and cutlery businesses. nCode has been providing software and consultancy covering all aspects of durability management, including:

- real time measurement of loading data

- engineering measurement and test data pre/postprocessing
- fatigue test systems
- fatigue and crack growth modelling software
- materials test data analysis

nCode's products for integrated durability management in the measurement, test and analysis fields include:

Measurement

- Ruggidized data acquisition systems for in-field measurements
- A/D systems for 386/486/Pentium PCs with automatic links to Linux workstations
- Preferences to many commercial data acquisition hardware systems which carry out on-line rainflow analysis

Test

- Materials testing services to obtain cyclic properties for use in fatigue modelling
- **TESTLAB** - a raw materials test data analysis tool producing cyclic properties for use in fatigue analysis
- Multi-channel test rig drive signal preparation and export tools
- Fatigue editors which can reduce fatigue test times by over 90%

Analysis

- **nSoft-E** - an advanced engineering software system
  This system is available under VMS/DOS/Linux for handling measurement and test data with integrated tools for graphical data presentation, manipulation, editing, filtering, classification, statistical and spectral analysis. Powerful user-focused features increase analysis efficiency and provide the user with flexible data presentation tools.
- **FATIMAS / KRAKEN** - A set of fatigue analysis tools for measurement engineers working from strain-time histories obtained from measurements made on prototypes or in-service components. These systems include special features for measurement engineers including a Kt library and strain gauge rosette preprocessors (FATIMAS-SN for stress based fatigue analysis, FATIMAS-EN for strain based crack initiation analysis and KRAKEN for crack growth modeling). Fatigue editors are also available based on range, cycle, and damage editing methodologies.
- **SIGNATURE** - Signature analysis for use in analyzing speed related frequency responses.
- **RAINTIME** - Time history regeneration from Rainflow cycle histograms for use in test rig control and fatigue editing.
- For more information about nCode products and how to reap the benefits of an integrated durability management system contact:

World-wide	**USA**	
nCode International Ltd 230 Woodbourn Road Sheffield, S9 3LQ England	nCode Inc. 26877 Northwestern Hwy., Suite 220 Southfield, MI 48037 USA	
Tel: 44 (0)114 2755 292 Fax: 44 (0)114 2758 272	Tel: (248) 350-8300	

Index

MSC Fatigue User's Guide

A

abort, 31, 86
abort job, 35, 93, 666
accuracy, 327, 349, 690, 1432
addition, 965, 1028
aerospace
    spectrum, 286
alpha solution
    see compliance or PKSOL, 535
amplitude distribution analysis, 1034
analysis, 48
    biaxiality, 53, 56
    crack growth, 59, 556
    crack initiation, 54
    design optimization, 6
    factor of safety, 54, 58
    frequency response, 656
    full, 30, 31, 85, 86, 663
    global multi-location, 5
    manager, 96
    partial, 30, 85, 89, 663
    random vibration, 659
    S-N (total life), 53
    static, 78
    total life, 4
    transient, 81
    types, 48
Analysis Manager, 96
analysis manager, 96, 667
angular variation of principal stresses, 764
arithmetic commands, 995
arithmetic manipulation (MART), 963
ASCII FES file format, 361
ASCII to Binary convertor (MATD), 1180
auto spectral density, 260, 960, 1035
autocovariance, 733
averaging, 48
averaging methods, 267

B

back calculation, 335, 349, 680, 690
band filters, 1040, 1046
Bannantine, 435
Bannantine-Socie, 493
bar elements, 773
Basquin, 1407, 1413
Basquin slope, 1357
batch mode
    PFMAT, 173
    PTIME, 219
Bauschinger, 1396
Bauschinger effect, 1394
beta solution
    see compliance or PKSOL, 535
biaxiality, 53, 56, 321, 505
biaxiality analysis, 5
biaxiality correction, 57, 324, 343
biaxiality corrections, 514
biaxiality ratio, 350, 422, 505, 513, 926
bin sizes, 321, 674, 678, 795
Binary to ASCII convertor (MDTA), 1178
broad band, 735
Brown, 435
Brown-Miller, 491
    parameter, 491
BS5400 Pt.10, welded, 1379
buffer overlap, 268
buffer size, 269
Butterworth filtration (MBFL), 1043

C

calculate normals, 31, 37, 38, 86, 94
calibration, 335, 680
carburizing, 1372
cast iron
    grey, 190
        characterization, 191
        properties, 191

Chadhury and Dover Method, 750
changing colors, 1177
Coffin-Manson, 1407
coherence function, 738
cold rolled
see surface finish/treatment, 374
cold rolling, 1371
colors, 1177
combination method, 354, 673, 678, 692
combined stress states, 489
command line operations, 1454
command line options, 1458
command syntax, 986
command templates, 1011
complex Fourier series, 719
compliance, 60, 535
user defined, 550
compliance function, 599
component size, 1365
concatenation, 1022
Configuration
Analysis Manager, 38, 40, 96
configuration
analysis
manager, 38, 40, 96
control
load
file, 287
coordinate systems, 773
corrections
biaxiality, 57
elastic-plastic, 56
plane stress, 558
correlated random loading, 757
correlation with test, 909
covariance, 737
crack closure, 4, 556, 597, 602
crack extension, 613
crack growth, 4, 531, 556, 593
cycle-by-cycle approach, 600
graphical results, 576
material response, 599
see PCRACK, 532
solution parameters, 59
crack growth data analysis, 580, 1132
crack initiation, 4, 1384
application in MSC.Fatigue, 1413
multiaxial analysis, 433
solution parameters, 54
crack parameters, 60
crack propagation rectangle, 596
crack size, 594
final, 561
initial, 561
crack tip, 612
create
material, 154
material database, 158
time history, 213
waves, 227
critical plane analysis, 439
critical plane approaches, 486, 490
cross platform compatibility, 1213
cross plots, 252
cross-covariance, 736
curve
cyclic stress-strain, 1398
strain life, 1406
curves
final a-n, 575
final growth, 576
material, 164
S-N, 1354
stress-strain, 1388
cutoff, 1407
cycle counting
multiaxial, 499
cycle file lister (MCYL), 1116
cycles
sequence, 608
stress, 1352
to failure, 1342
to initiate crack, 1342
to propagate crack, 1342
cycles and damage analysis (MCDA), 1111

cyclic
hardening, 1397
material properties, 1432
softening, 1397
stress-strain, 1396
stress-strain curve, 1398
yield, 1401
cyclic plasticity, 439, 518
cyclic plasticity modeling, 440

D

DAC to RPC file conversion, 1202
damage, 318, 357
low
removal, 956
removal, low, 956
damage calculation, 501
damage calculation, multiaxial, 444
damage summation, 484, 1380, 1420
Dang Van Criterion, 527
Dang Van Method, 430
database
materials, 5, 63, 137
MSC.Patran, 5, 83
PSDs, 655
results, 70, 654
time history, 5, 71, 208
validate, time history, 249
delete, 36, 93, 666
files, 30, 85, 663
job, 30, 85, 663
materials, 160
time history, 248
delta rosette, 912, 913
design criterion, 54, 58, 323, 336, 337, 338, 341, 652, 681, 682, 683, 686
design life, 54, 58, 327, 336, 559, 681
specifying, 332, 677
design optimization, 4, 106, 330, 669, 797
Dirlik, 652
Dirlik method, 751
distortion-energy theory, 487
division, 965, 1028
duty cycle
event, 949
introduction, 936, 939, 945
results, 952
Sequence, 949
setup, 947

E

edit
material, 149
time history, 225, 236
elastic line, 166
elastic strain, 1393
elastic to elastic plastic conversion, 927
elastic to elastic-plastic conversion, 917
elastic-plastic correction, 4, 324, 342, 908
elastic-plastic corrections, 1415, 1416, 1419
elastic-plastic lines, 166
elastic-plastic to elastic conversion, 916, 931
elastic-plastic to elastic-plastic conversion, 917, 931
endurance limit, 1356, 1372
ensemble averaging, 263
environment, 69, 556, 597, 615
MSC.Fatigue, 1463
error messages, 1470
errors, 35, 93, 790, 882, 1003
estimating material properties, 154, 622
example problems, 1222
comparison to another code, 1276
keyhole, 1223
pressure vessel, 1262
spot weld, 1305
ten simple notched geometries, 1286
transient results, 1296
vibration fatigue, 1311
examples
formula processor (MFRM), 1011
EXCEL, 1120
executable, 292, 533, 627
extra details (footer), 1076
extract matrix, 356
Extract PSD, 670
extract PSD, 690
extract time histories, 353
extracting files, 1017, 1019

extracting results, 901
extrusions, 1385

F

factor of safety, 4, 54, 58, 107, 1381
factor of safety analysis, 325, 429
    multiaxial, 524
factors, 1412
fast analysis, 87, 102, 368
fast Fourier filtering, 1036
fast fourier transform, 261, 711
Fatemi-Socie, 435
Fatemi-Socie Parameter, 467
fatigue analysis, 321
Fatigue input file (FES), 361
fatigue limit, 166, 1356, 1401
fatigue strength, 344, 687
fatigue strength reduction, 1133
fatigue units, 216
FATTRANS, 295
fattrans
    program, 294
FEFAT, 106, 107, 533
    batch operation, 417, 418
    change parameters, 340
    design optimization summary page, 333
    graphical options, 346, 359
    life-based factor of safety, 326
    material optimization, 338
    new job, 348
    new node/element, 348
    parameter optimization, 335
    recalculate, 350
    reset original parameters, 349
    results display, 344
    sensitivity analysis, 337
    stage 1 design optimization, 330
    stage 2 design optimization, 334
    stress-based factor of safety, 327
    utilities, 360
    utility batch operation, 419
FEFTRN, 109
FEMLF, 424, 427
    batch operation, 438
    crack initiation analysis, 433
    safety factor analysis, 429
FEVIB, 669, 670
    batch operation, 693
    change parameters, 685
    material optimization, 684
    new job, 688
    new node/element, 688
    parameter optimization, 680
    recalculate, 690
    reset original parameters, 690
    results display, 687
    sensitivity analysis, 681
    stage 1 design optimization, 675
    stage 2 design optimization, 680
    summary page, 679
file
    DAC format, 274
    extract time history, 225
    insert time history, 225
file conversion, 1178, 1202, 1213, 1216
file dump, 1072
file list functionality, 1451
files, 31, 86
    ASCII format, 401
    compliance function, 553
    crack growth, 559
    damage histogram, 333
    delete, 36, 93, 666
    description of, 369
    fatigue input, 401
    fatigue results file, jobname.fef/fos, 414
    gauge definition, 893
    job information, jobname.fin, 369
    jobname.asc, jobname.fes, 401
    jobname.fpp, rainflow file, 410
    material database, 63
    MSC.Patran external results, 77
    necessary for crack growth, 293, 534
    necessary for spot weld, 781, 874
    needed for global analysis, 293
    needed for vibration fatigue, 629
    rainflow cycle counted, 333, 679
    spot weld, 817
    squeeze results, 577
    surface finish/treatment, 65, 67
fill down, 81
filter results, 82, 659

filter type, 1040
filter types, 1046
finite element, 1350
    results, 5, 48, 74
FLAGR0, 589
fonts, 1455, 1459
Forman's law, 600
forms
    extract results, 902
    gauge tool, 892, 897
    job control, 30, 85, 663, 788, 880
    loading, 70, 654
    loading information, 787
    main, 47, 649
    material, 61, 653
    material information, 784, 876
    results, 790, 882
    results display, 102
    solutions parameters, 782, 875
    spectrums, 105
formula processor (MFRM), 984
Fourier analysis, 723
Fourier density coefficient, 721
Fourier series, 717
Fourier transform pair, 722
fracture mechanics
    applications, 621
    principles of, 594
    see crack growth, 556
    triangle, 535, 595
frequency domain, 710, 747
    life approaches, 749
frequency fatigue life, 695
frequency response analysis, 211
frequency response analysis (MFRA), 1049
frequency response results, 656
fringe plots, 102

G

Garud, 440
gate, 88
gauge definition file, 893
gauge names, 901, 908
gauge Poison's ratio, 920
gauge tool, 892
gauge types, 908
generate material properties, 154
geometry, 12, 44, 705
    crack, 539
    crack at holes, 542
    cracks at corners, 545
    cracks in hollow cylinders, 547
    cracks in solid cylinders, 546
    elliptical, semi-elliptical, 543
    spot welds, 548
    standard specimens, 541
    tubular welds, 548
    user defined, 550
    welds, 548
Gerber, 53, 652, 1361, 1379, 1381
get results, 82, 659
global variables, 85, 661
Goodman, 53, 652, 1361, 1379, 1381
gradients
    stress, 957
graphic colors, 1177
graphical display
    histograms, 357
graphical operations, 1453
graphical options, 1457
graphics
    create time history, 223
    material data, 164
grid point stresses, 110
groups, 66, 68, 69

H

Haigh diagram, 1360
Hancock method, 750
hardening rule, 520
header and footer manipulation (MFILMNP), 1071
high cycle fatigue
    multiaxial, 524
high pass filter, 1040, 1046
high-cycle fatigue, 1342, 1384
histogram, 345, 357, 688
histogram plots, 808
histograms, 437
history effects, 558, 597
history of fatigue, 1341

Hoffmann-Seeger, 57, 515, 908, 909
hysteresis, 1396, 1403

I

identify location, 108, 670
initial crack, 600
interactive, 95, 667
interpolation, 569
intrusions, 1385
inverse Fourier transform, 711
iron
  grey
    cast, 190
irregularity factor, 669
irregularity factor reconstruction, 1198

J

J integral, 607
job control, 30, 85, 663
job setup, 19, 23, 25, 27, 47
jobname, 49, 50
  details, 366

K

K solution, 535
keypad commands, 1448
keywords, 1467
kinematic hardening, 522
Kitagawa diagram, 601, 603
Kt evaluation (MKTAN), 1133
Kt library, 1139

L

Large Models
  Handling, 295
LEFM, 4
  see crack growth, 556
life
  interpolation, 577
limitations, 1470
linear static analysis, 210
linear systems, 732, 738
Linux based plotting utility (MQPLOT), 1149
list a file, 1117
list cycles matrix, 1119
list functionality, 1451
load case IDs, 79, 83
load magnitude, 80
loading, 12, 44, 283, 1346, 1350, 1366, 1412
  conditions, 1353
loads
  type of, 1351
local strain, 4
logarithmic functions, 966
lookup files, 1137
lookup table functions, 1000
low cycle fatigue, 503
low pass filter, 1040, 1046
low-cycle fatigue, 1342, 1384

M

MacDiarmid Method, 433
Manager
  Analysis, 96
Manson-Coffin, 1341
marker plots, 791
Markov transition matrix reconstruction, 1197
MART, 963
  batch operation, 969
  module operation, 964
mask mode, 1446
Massing's hypothesis, 1395, 1403
MATD, 1178
material
  checking, 571
  crack growth, 565
  cutoff, 54, 58
  database manager, 137
  environment, 564
  listing, 177
Materials
  Temperature Dependent Poperties, 192

materials, 12, 44, 1343
changing Young's modulus, 171
crack growth, 63, 69
crack growth data, 153, 157
crack initiation, 61, 67
crack initiation data, 152, 156
data values, 143
database, 5
database manager, 63
edit parameters, 338, 684
estimate, 154
full listing, 140
generic data, 152, 156
LEFM estimation, 622
multiple mean S-N Curves, 195
optimization, 338, 684
search and list, 141
searching for better, 339, 684
searching for worse, 338, 684
S-N (total life), 62
S-N component, 65, 169
S-N materials, 65, 169
spot weld, 833
surface values, 143, 146
temperature dependent, 192
total life data, 153, 157
vibration fatigue, 653
maximum factor, 58
maximum safety factor, 54
maximum shear strain, 914
maximum shear stress, 914
maximum shear stress theory, 488
MBFL, 1043
module operation, 1044, 1048
MCDA, 1111
batch operation, 1115
module operation, 1112
McDiarmid criterion, 525
MCLF, 1092
batch operation, 1108
keywords, 1104, 1107
module operation, 1093
MCOE, 970
module operation, 971
preferences, 979
MCONFIL, 1213
batch operation, 1215
module operation, 1210, 1213
MCYL, 1116
batch operation, 1120
module operation, 1117
MDACREM, 1202
batch operation, 1208
module operation, 1203
MDTA, 1178
batch operation, 1183
module operation, 1181, 1183
mean stress, 328, 337, 342, 682, 686, 1359, 1378, 1410
correction, 4, 53, 54, 323, 338, 652, 683
Gerber, 1362
Goodman, 1362
method, 56
Morrow, 1411
Smith-Watson-Topper, 1412
mechanical treatments, 1371
MENM, 1446, 1463
Mertens-Dittmann, 56, 908, 1416
MFCG, 580
batch operation, 589
module operation, 584
MFFF, 1036
batch operation, 1043
module operation, 1038
MFILMNP, 1071
batch operation, 1081
module operation, 1073
MFLF, 695
batch operation, 709
module operation, 696
postprocessing, 707
MFRA, 1049
batch operation, 1010, 1059
command templates, 1011
module operation, 1050
MFRM, 984
module operation, 1007
MIL Handbook, 1360
Miller, 1341
Miner's Rule, 464

Miner's rule, 1420
Miner's sum, 1381
Miner's summation constant, 349, 462, 690
MKTAN
batch operation, 1141
graphical mode operation, 1141
module operation, 1133
MLEN, 1016
batch operation, 1023
module operation, 1017
MMFM, 1024
batch operation, 1031
keywords, 1031
module operation, 1027
MMLF, 438
batch operation, 479
module operation, 453
modify
time history, 235
modulus of elasticity, 1392
Mohr's circles, 483
monitor, 30, 85, 663
monitor job, 34, 91, 665
monotonic behavior, 1388
Morrow, 1406, 1411
motif mode, 1455
mouse operations, 1453
MPLTSYS, 1159
module operation, 1163
MPOD, 820
batch operation, 827
keywords, 827
module operation, 820
polar plot manipulation, 823
MQPLOT, 1149
batch operation, 1152
module operation, 1149
MREGEN, 1195
batch operation, 1202
module operation, 1198
MREMDAC, 1202
batch operation, 1208
module operation, 1203
Mroz, 440
MRSTATS, 1061
batch operation, 1070
module operation, 1062
MSC.Fatigue
accessing, 14, 45
architecture, 5
features, 3
five-box trick, 13, 45
general setup parameters, 47
jobname, 50
MSC.Patran environment, 47, 649
organization of manual, 6
overview, 2
preparing an analyses, 12, 44
MSC.Nastran, 109
MSLF, 1083
batch operation, 1089
module operation, 1085
MSSA, 911
batch operation, 932
graphical operation, 921
module operation, 917
strain gauge rosette analysis, 918
MTCD, 1121
batch operation, 1126
module operation, 1122
multiaxial fatigue, 5, 421
multiaxial fatigue analysis, 438, 961, 1130
multiaxial fatigue life, 424, 427
multiaxial methods
Fatemi-Socie, 435, 446, 463
Normal Strain, 434, 445, 462
Shear Strain, 435, 446, 463
STW-Bannantine, 435, 446
SWT-Bannantine, 463
Wang-Brown, 435, 450, 463
multiaxial stress, 422
multiaxial stress and crack growth, 623
multiaxial stress-strain, 481
multiaxiality, 350, 481, 759
multichannel creator/editor -MCOE, 970
multi-file display, 251
multiple degrees of freedom systems, 758
multiple file manipulation (MMFM), 1024
multiple mean S-N Curves, 195
multiple random processes, 736

multiplication, 965, 1028
MWFLCRE, 1216

N

narrow band, 735
narrow band method, 749
NASA/FLAGRO, 5, 589
Neuber, 56, 909
Neuber's rule, 1415
nitrided
    see surface finish/treatment, 374
nitriding, 1372
node
    worst case, 332, 677
noise floor, 269
non-proportional loading, 423, 909
nonproportional loading, 484, 493
normalization, 269
notch correction
    multiaxial, 524
notch effects, 556
notches, 597, 605, 1364
    specification of, 561
nugget calculation, 796

O

one degree of freedom systems, 755
optimization, 330
    see FEFAT, 335
    see FEVIB, 680
overlay plots, 252
overloads, 609

P

Palmgren-Miner, 1380, 1420
Palmgren-Miner linear damage summation, 4
parameter modification, 57, 517
parameters
    crack growth, 59
    crack length, 59
    factor of safety, 54, 58
    general, 47
    general results, 72, 655
    material, 61, 653
    S-N (total life), 53
Paris, Paul, 595
Paris' law, 600
partially correlated random process, 732
PAT3FAT, 294, 629
PCPOST
    introduction, 571
    operation, 572
    options, 573
PCRACK, 106, 533
    batch operation, 578
    environment, 563
    general information, 557
    introduction, 532, 556
    local geometry, 560
    material, 563
    material editing, 565
    module operation, 556
    output parameters, 559
    postprocessing, 567
    summary page, 567
peak valley extraction, 88, 254
percentage gate, 88

PFMAT, 63, 114, 116, 117, 121, 128, 137
create material, 154
delete, 161
edit material, 149
graphical display, 164
graphical options, 165
introduction, 114, 116, 117, 121, 128, 137
load material, 147
material listing, 177
menu options, 139
menu structure, 139
merge entries, 159
new database, 159
preferences, 167
unload material, 149
weld classifier, 162
PFPOST, 363, 436
introduction, 363
operation, 363
options, 364
parameters, 364
PFSETFONT, 1455
PKSOL, 60, 533
batch operation, 578
geometric definitions, 539
graphics, 566
introduction, 535
module operation, 536
plot options, 552
references, 554
tabular listing, 553
planar tee rosette, 914
plane stress, 558
plane stress correction, 60
plastic strain, 1393
plastic zone, 612
plasticity correction, 56
plating, 1371
plots
crack size, 551, 566
da/dN-Delta K, 1345
damage, 345
final a-n, 575
final growth, 576
histogram, cycle, 345, 688
material, 164
sensitivity, 107, 345
stress intensity, 551
y function, 551
plotting utilities, 1149
poisson's ratio, 457
polar plots, 438, 476, 808, 815, 820
Poncelet, 1340
postprocessing, 38, 40, 100, 667
power spectral density function, 725
power spectrum, 654, 711
power spectrum database manager, 655
preferences
material checking, 168
material database, 167
material units, 167
multiaxial, 477
S-N axis, 167
spot weld, 812
pre-processing, 318
principal stresses, 760, 913
printer/plotter definitions (MPLTSYS), 1159
probability, 1422
probability density function, 700, 746
probability density function (PDF), 748
product information
nCode International, 2
program
fattrans, 294
program environment, 1463

programs
FASTAN, 368
FEFAT, 297, 307, 308, 310, 316, 317
FEFTRN, 109
FEVIB, 670
MART, 963
MASD, 260
MATD, 1178
MBFL, 1043
MCDA, 1111
MCLF, 1092
MCOE, 970
MCONFIL, 1213
MCYL, 1116
MDACREM, 1202
MDTA, 1178
MENM, 1446, 1463
MFCG, 580
MFFF, 1036
MFILMNP, 1071
MFLF, 695
MFRA, 1049
MFRM, 984
MLEN, 1016
MMFD, 251
MMFM, 1024
MMLF, 438
MP3D, 357
MPLTSYS, 1159
MPOD, 820
MPVXMUL, 254
MQPLOT, 1149
MREGEN, 1195
MREMDAC, 1202
MRSTATS, 1061
MSLF, 1083
MSSA, 911
MTCD, 1121
MTPD, 346
MWFLCRE, 1216
PAT3FAT, 294
PCPOST, 571
PCRACK, 556
PFMAT, 114, 116, 117, 121, 128, 137
PFPOST, 363
PFSETFONT, 1455
PKSOL, 535
PTIME, 200, 201, 202, 208
SPOTW, 792
SSG, 906
proportional loading, 423, 909
PSDF from time history, 260
PSDFs from Time Histories, 235
PTIME, 71, 80, 200, 201, 202, 208, 655
ASCII convert and load, 217
block program, 231
central database listing, 273
copy from central database, 220
copy from remote database, 221
DAC file format, 274
delete option, 248
duplicate, 230
edit details, 237
edit x-y points, 238
graphical create, 223
graphical edit, 236
graphics options, 247
introduction, 200, 201, 202, 208
list entries, 242
load files, 214
look-up transform, 239
menu structure, 210
modify a time history, 235
options, 213
polynomial transform, 238
rainflow matrix, 232
sample rate adjust, 241
search and list, 243
time history, 245
units conversion, 240
waveform creation, 227
white noise, 232
x-y points, 221
x-y PSDF points, 222

R

R ratio, 1353
rainflow, 699, 753
rainflow cycle counting, 4, 318, 356, 1347, 1349
multiaxial, 443

rainflow extraction method, 256
rainflow matrix, 51, 71, 80, 232, 655, 782, 874
rainflow reconstruction, 1196
random process theory, 717
random vibration analysis, 213, 659
range extraction method, 255
ranges, 105
read job, 30, 36, 85, 94, 663, 666
reading, 1443
rectangular rosette, 912, 913
reference
    stress, 54, 328
references, 554, 1438, 1443
regenerate
    time history
        from PSD, 1210
regions, 66, 68, 69
residual K, 609
residual stress, 336, 337, 338, 341, 529, 681
results, 48, 363
    biaxiality, 104
    case IDs, 82, 659
    crack growth, 571
    damage, 104
    database, 70, 73, 75, 654, 656
    duty cycle, 952
    finite element, 5, 74
    get/filter, 79, 82, 659
    life, 104
    list, 106, 669
    locations, 48
    log of life, 104
    most damaged nodes/elements, 366
    MSC.Patran external, 70, 73, 77
    MSC.Patran FEA, 70, 73, 76
    multiaxial, 436, 472, 474
    power spectrum, 654
    power spectrum (PSD), 656
    read, 101, 668
    static, 70, 73
    stress factor, 104
    summary page, 574
    surfaces, 74, 76, 77
    tabular, 366, 367
    transfer function, 654, 656
    transformations, 70, 654
    transient, 70, 73
    translator, 109
    types, 78
retardation, 559, 614
reverse file, 1023
rosette
    construction, 457
    definition, 456
    types, 456
rosette configuration, 919
RPC to DAC file conversion, 1202

S

S/N curves, 4
sample rate, 218, 241
save, 30, 85, 663
    job, 33, 90
    preference settings, 168
scale factor, 82, 336, 558

scaling factor, 337, 341
scaling factors, 338
scatter, 166
seam weld, 4
Seeger-Beste, 56, 908, 1419
sensitivity
    notch factor, 329
    size factor, 329
sensitivity analysis, 337, 802
sensitivity plots, 808
sequence effects, 1121
service environment, 598
SGG
    technical details, 909
shape factor, 68, 908
sheet thicknesses, 803
short crack parameter, 563, 601
short cracks, 603
shot peened
    see surface finish/treatment, 374
shot peening, 1371
signal statistics, 745
similitude, 503, 1387
sine waves
    create, 227
    summation, 229
single location analysis, 438, 695
slip bands, 1385
small cycle correction, 324, 342
Smith-Watson-Topper, 1412
smoothing, 1023
S-N curve
    definition, 1376
    general, 1359
    spot weld, 785
    tensile properties, 1358
Socie, 435
software strain gauges, 889
    analysis route, 891
    basic information, 890
    correlation with test, 909
    creating, 892, 897
    deleting, 901
    extracting results, 901
    gauge tool, 892
    input file, 903
    job control, 906
    loading information, 905
    material information, 904
    modifying, 897, 899
    module operation (SSG), 906
    solution parameters, 904
solution parameters
    crack growth, 50
    crack initiation, 50
special commands, 1449
spectrum
    aerospace, 286
    file
        aerospace, 286
spectrums, 105

spot weld, 105
analysis route, 780, 873
basic information, 780, 872
change parameters, 805
coordinate system, 773, 829
damage calculation, 834
design optimization, 797
job control, 787
job setup, 774, 777, 871
list results, 814
loading information, 786, 876
material information, 783, 875
material optimization, 804
material properties, 833
parameter optimization, 801
polar plots, 815
postprocessing, 791
preferences, 812
results, 790, 882
results display, 806
schematic, 772
sensitivity analysis, 802
solution parameters, 782, 875
stress calculations, 831
stress factors, 834
submittal schematic, 779, 872
theory, 828
three sheet correction, 816
spot weld analysis, 4
SPOTW, 792
batch operation, 819
global analysis, 794, 884
list results, 814
preferences, 812
stage 1 design optimization, 797
stage 2 design optimization, 800
SSG, 906
batch operation, 910
interactive operation, 906
stacked rectangular rosette, 914
stacked tee rosette, 914
stand alone, 13
state of stress, 423
static analysis results, 78
static failure, 486
static fracture, 556, 597, 607
static offset, 80
stationarity, 669, 692, 763
statistical analysis, 1061
statistics, 1421
status, 34, 91, 789, 881
Steinberg method, 750
strain gauge rosette analysis, 918
strain gauge rosettes, 911
strain gauges, 5, 889
strain life vs. stress life, 1407
strain-life, 1344, 1384
strain-life curve, 1406
stress
gradients, 957
stress combination, 58, 320
stress concentration, 336, 338, 343, 1133
stress intensity, 538, 595
stress state, 423
stress-life, 1344
stress-strain analysis, 911
stress-strain combination, 53, 56, 60, 652
stress-strain curve, 1388
stress-strain relationships, 1392
stress-strain state, 504
stress-strain tracking, 462
subcases, 85, 661
submit, 31, 86
schematic, 291, 779, 872
schematic for vibration fatigue, 627
script, 383
subtraction, 965, 1028
superposition, 1370
surface finish, 466, 702
surface finish/treatment, 4, 65, 67, 329, 334, 337, 343, 682, 687, 1364, 1367, 1412
codes, 374
surface finish/treatment correction, 338, 683
surface resolved stress, 423
surface stress, 94
surface treatment, 467, 702
surfaces, 76

T

tee rosette, 913
tensile strength, 1389
terminal definition, 290, 533, 626, 1446

text reports, 102
theory
  background, 1340
  crack growth, 593
  crack initiation, 1384
  determining S-N curve, 1355
  factor of safety, 1381
  factors influencing life, 1364
  history, 1341
  inputs, 1343
  limits of S-N curve, 1357
  loading, 1346
  materials, 1343
  mean stress, 1359
  multiaxial, 481
  multiaxial fatigue, 485
  rainflow cycle counting, 1347, 1349
  spot weld analysis, 828
  summary, 1342
  total life, 1351
  vibration fatigue, 710
thermal treatments, 1372
thresholds, 602
time correlated damage analysis (MTCD), 1121
time domain, 710, 740
time histories, 70
time history, 107
  add, 213
  DAC file format, 274
  database, 5
  database manager, 71
  from PSD, 1210
  import ASCII file, 217
  load type, 216
  lookup transform, 239
  scale, 238
  search and list, 243
  static, 80
  units, 216
time signal regeneration, 727
time steps, 82
total life, 1351
  application in MSC.Fatigue, 1374
  solution parameters, 53, 652
transfer function analysis (MFRA), 1052
transfer functions, 654
transformations, 74, 656
transient analysis, 81, 211
transition life, 1408
translate, 33, 90
translator, 294, 295, 629
transverse sensitivity, 457, 912, 919
Tresca theory, 488
triaxial stress states, 768
trigonometric functions, 965, 998
true fracture ductility, 1393
true fracture strength, 1393
true stress and strain, 1389
truncation of cycles, 512
Tunna method, 750
two degree of freedom systems, 757

U

ultimate tensile strength, 328, 1389
uniaxial stress, 422
uniaxiality, 483
units, 49, 283
  crack geometry, 537
  damage histogram, 349
  time history, 240
Universal file, 109
Universal file format, 111, 416
utilities, 5
  advanced loading manipulation, 963
  ASCII conversion, 360
  graphical hardcopy, 961

V

Validation Problems, 1221
validation problems
  crack growth, 1243, 1270
  crack initiation, 1235, 1263, 1276, 1286, 1296
  S-N, 1225
  spot weld S-N, 1305
  vibration fatigue S-N, 1311

vibration fatigue, 4, 625
analysis methods, 652, 683, 708, 749
analysis route, 628
basic informations, 627
job control, 663
job setup, 649
theory, 710
utilities, 693
vibration fatigue analysis, 1128
von Mises' theory, 487

W

Wang-Brown, 435, 499
waterfall analysis, 266
waterfall parameters, 271
waterfall plot, 263
waterfall plots, 1216
waterfall slice, 697
Weibull distribution, 1424
weighted average stress range, 597
weld
analysis, 771
weld classification, 66
weld classifier, 4, 162
white noise, 232, 735
wide band, 735
window functions, 262
window size, 1066
window type, 1056, 1065
window types, 268
windows based plotting (WNPLOT), 1152
Wirsching method, 750
Wöhler, August, 1341, 1354, 1432

X

X driver, 1446
X resources, 1462
XDB file, 109
xy plots, 102

Y

y-function, 535
yield function, 518
yielding criteria, 487
Young's modulus
changing, 171