JSC-22267B
March 2002

FATIGUE CRACK GROWTH COMPUTER PROGRAM "NASGRO" VERSION 3.0

Reference Manual

ENGINEERING DIRECTORATE

March 2002

National Aeronautics and Space Administration
Lyndon B. Johnson Space Center
2101 NASA Road 1
Houston, Texas 77058-3696

NOTICE

TABLE OF CONTENTS

REVISIONS xiii

SYMBOLS xv

ABBREVIATIONS xvii

SUMMARY 1

1.0 Introduction 3
2.0 Fatigue Crack Growth Analysis 5
2.1 Theoretical Background 5
2.1.1 Crack Growth Relationship 5
2.1.2 Fatigue Crack Closure 8
2.1.3 Threshold Stress Intensity Factor Range 11
2.1.4 Fracture Toughness 13
2.1.5 Criteria for Failure 15
2.1.6 Yielding Checks 16
2.1.7 Crack Growth under Load Interaction 17
2.1.7.1 Generalized Willenborg Model 17
2.1.7.2 Modified Generalized Willenborg Model 19
2.1.7.3 Walker-Chang Willenborg Model 20
2.1.7.4 Strip Yield Model 21
2.1.7.5 Constant Closure Model 30
2.1.7.6 Notes on Using the Load Interaction Models 30
2.2 Details of Crack Growth Analysis 37
2.2.1 Choosing the Crack Geometry 38
2.2.2 Transition of Crack Geometry 38
2.2.3 Entering the Initial Flaw Size 39
2.2.4 Selecting the Material Properties 55
2.2.4.1 Entering Material Properties Data from File 57
2.2.4.2 Entering Material Properties Data Manually 58
2.2.5 Building the Load Schedule 59
2.2.5.1 Defining the Block Cases 59
2.2.5.2 Setting the K_{eac} Check 64
2.2.5.3 Scaling the Stresses 65
2.2.5.4 Setting the Reference Stress Checks 66
2.2.5.5 Combining the Block Cases to Form a Schedule 66
3.0 Critical Crack Size Calculations 69
3.1 Theoretical Background 69
3.2 How to Run a Critical Crack Size Calculation 70

4.0 Stress Intensity Factor Calculations 71
4.1 Theoretical Background 71
4.2 How to Run a Stress Intensity Factor Calculation 71
4.2.1 SIF Calculations Using Linear Stresses 72
4.2.2 SIF Calculations Using Nonlinear Stresses 73
5.0 Sustained Stress (da/dt) Analysis of Glass 75
5.1 Theoretical Background 75
5.2 How to Run a Sustained Stress Analysis 75
5.2.1 Material Properties 77
5.2.2 Entering the Sustained Load Schedule 78
6.0 Crack Growth Data Analysis 79
6.1 Entering a vs. N or da/dN vs. ΔK Data for Curve Fitting 79
6.2 Obtaining the Constants 79
7.0 Boundary Element Method Analysis 83
7.1 The Data Input Interface 83
7.1.1 The Input Philosophy, or: How to build the model 84
7.1.1.1 Points 84
7.1.1.2 Boundary segments 85
7.1.1.3 Boundaries 86
7.1.1.4 Zones 86
7.1.1.5 Crack segments and cracks 87
7.1.1.6 Boundary conditions and concentrated (or point) loads 87
7.1.1.7 Materials 89
7.1.1.8 Special case geometries 89
7.1.1.9 Stress analysis 90
7.1.2 The Menu Tree 90
7.1.2.1 Top level menu 90
7.1.2.2 General hints and notes on using the menu tree and input 91
7.2 Numerical Limits on Input Items; or, How big can the model be? 92
7.3 Stress Analysis 92
7.3.1 Stresses in the Interior 93
7.3.2 Stresses on the Boundary 94
7.3.3 Stresses in the Boundary Layer 95
7.4 Program Output and Files Used 95
7.5 Future Enhancements 96

ACKNOWLEDGMENTS 97

REFERENCES 99

Appendix A – Flow Diagrams A-1
Appendix B – Net-section Stress Computations B-1
Appendix C – Stress Intensity Factor Equations and Tables C-1
Appendix D – Instructions for Adding Crack Cases D-1
D.1 Changes to Subroutines D-1
Appendix E – User-Defined Files E-1
Appendix F – Interpolation Routines F-1
Appendix G1 – NASGRO 3.0 Materials Constants, U.S. Units G1-1
Appendix G2 – NASGRO 3.0 Materials Constants, SI (ESA) Units G2-1
Appendix H – Loading Spectra for Acceptance Vibration Tests H-1
H.1 Sinusoidal Sweep Vibration Test H-1
H.1.1 Narrowband Response – Methodology and Results H-1
H.1.2 Wideband Response – Methodology and Results H-3
H.2 Sinusoidal Dwell Vibration Test H-4
H.3 Random Vibration and Acoustic Tests H-5
H.3.1 Methodology H-5
H.3.2 Results H-6
H.4 References H-6
Appendix I – System Requirements, Software Requirements, and Program Installation I-1
I.1 PC Version of NASGRO I-1
I.1.1 PC Installation Information I-2
I.1.2 PC Execution I-2
I.2 Macintosh Version of NASGRO I-2
I.2.1 Macintosh Installation and Execution I-3
I.3 NASGRO for Computers other than PC and Macintosh I-3
I.3.1 Source Code I-4
Appendix J – Unit Conversion factors J-1
Appendix K – Stress Quantities in Fatigue Load Spectra K-1
Appendix L1 – NASGRO 3.0 Materials Constants for Walker Equation - US units L1-1
Appendix L2 – NASGRO 3.0 Materials Constants for Walker Equation - SI units L2-1

LIST OF FIGURES

Figure 1a - Curve fit to Eq. 2.1 for BT14 Titanium (Russian) 6
Figure 1b - Curve fit to Eq. 2.1 for A357-T6 Cast Aluminum 7
Figure 2 - Crack opening function vs. Stress ratio 10
Figure 3 - Normalized threshold vs. crack depth for steel after Lindley [9] 12
Figure 4 - Crack growth threshold vs. Stress Ratio 12
Figure 5 - Threshold at R=0 vs. yield strength for steels... 13
Figure 6 - Curve fit of Eq. 2.12 for Be-Cu CDA172... 14
Figure 7 - Kc vs. thickness showing R-curve effect for Aluminum 2219-T87 15
Figure 8 - Surface crack fracture toughness curve fit... 16
Figure 9 - Crack tip plastic zones 18
Figure 10 - Schematic of analytical crack-closure model under cyclic loading 23
Figure 11 - Schematic of the Strip Yield model.. 24
Figure 12 - Dugdale model: Superposition of two elastic solutions 25
Figure 13 - Flat to slant crack growth transition 26
Figure 14 - Illustration of constraint transition region... 27
Figure 15 - Constraint factors in tension and compression.. 28
Figure 16 - Through crack cases 1 & 2.. 41
Figure 17 - Through crack cases 3 & 4.. 41
Figure 18 - Through crack cases 5 & 6.. 42
Figure 19 - Through crack cases 7 & 8.. 42
Figure 20 - Through crack cases 9 & 10.. 43
Figure 21 - Embedded crack case 1 43
Figure 22 - Corner crack case 1... 44
Figure 23 - Corner crack cases 2 & 3 44
Figure 24 - Corner crack case 4... 45
Figure 25 - Surface crack cases 1 & 2 45
Figure 26 - Surface crack cases 3 & 4 46
Figure 27 - Surface crack cases 5 & 6 46
Figure 28 - Surface crack cases 7 & 8 47
Figure 29 - Surface crack cases 9 & 10 47
Figure 30 - Surface crack cases 11 & 12 48
Figure 31 - Surface crack cases 13 & 14 49
Figure 32 - Standard specimen crack cases 1 & 2.. 50
Figure 33 - Standard specimen crack cases 3 & 4.. 50
Figure 34 - Standard specimen crack cases 5 & 6.. 51
Figure 35 - Standard specimen crack cases 7 & 8.. 51
Figure 36 - Standard specimen crack cases 9 & 10.. 51
Figure 37 - Crack plane orientations.. 56
Figure 38 - Load schedule example.. 60
Figure 39 - Sustained stress data with curve fits 76
Figure 40 - NASBEM menu tree... 84
Figure 41 - Element intersection types 96

Figure F1 – Comparison of interpolation methods F-2
Figure F2 – Cubic spline fit of ΔK to R F-3
Figure H1 – Frequency response for Sine Sweep vibration test –
Narrowband response...H-2
Figure H2 – Frequency response for Sine Sweep vibration test –
Wideband response H-4

LIST OF TABLES

Table 1 – Effect of R_{so} on Life Prediction for Generalized Willenborg model 33
Table 2 – Effect of ϕ_0 on Life Prediction for Modified Generalized Willenborg model........ 34
Table 3 – Life Prediction for Zhang's Spectra.. 35
Table 4 – Life Prediction for ASTM Spectra... 36
Table 5a – Description of Crack Cases.. 39
Table 5b – Transition of Crack Cases.. 40
Table 6 – Standard NDE Flaw Sizes for STS Payloads 54
Table 7 – Standard NDE Flaw Sizes for STS Payloads 55
Table 8 – Two-dimensional Interpolation Table.. 58
Table 9 – Launch and Landing Spectrum for STS Payloads.. 62
Table 10 – K_{eac} Values for Several Metals... 66
Table 11 – Sustained Stress Constants (English Units).. 78
Table 12 – Sustained Stress Constants (SI Units) 78
Table C1 .... C-3
Table C2 .... C-4
Table C3 .... C-6
Table C4 .... C-7
Table C5 .... C-10
Table C6 .... C-11
Table C7 .... C-12
Table C8 .... C-18
Table C9 .... C-19
Table C10 .... C-21
Table C11 .... C-22
Table C12 .... C-24
Table C13 .... C-25
Table C14 .... C-27
Table H1 – Equivalent Number at Maximum Stress Level..H-3
Table H2 – Values of $A_0(\alpha,n)$ for Various Values of α and n...H-3
Table H3 – Equivalent Number at Maximum Stress Level..H-6

Revision and History

Revision	Description	Date
	Initial Release: VAX Version of NASA/FLAGRO with Reference Manual JSC-22267	**Aug 1986**
A	PC DOS Version of NASA/FLAGRO 2.03 with Reference Manual JSC-22267A	**May 1994**
B	Windows 95, 98, NT & X-Windows Version 3.0 with Reference Manual JSC-22267B <u>Major updates from Rev. A:</u> 1. Graphical User Interfaces under Windows. 2. New NASMAT Module for material data processing 3. Five new load interaction material models for crack growth. 4. Revised equation for threshold stress intensity computation. 5. New crack cases SC11,12,13,14 and SS08,SS09 6. Improved K solutions for TC03, CC02 and SC08 7. Improved da/dN curve fits for the following materials: • A533-B Cl1&Cl2 SMA Weld • D6AC 220-240 UTS Plt&Forg Nom Kic(70);HHA, SW>0.1Hz • HY-180(10Ni) Plt&Forg; SW>0.1 Hz • HY 130SMA Weld • AISI 304/304L Ann Plt&Sht. Cast; 550F Air • 2024-T861 Plt&Sht; L-T; 300F to 400F Air • 2090-T8E41 Plt&Sht; L-T • 7050-T73651 Plt&Sht;T-L; LA&HHA • Inconel 625 Plt&Sht; 600F • Inconel 718; ST(1700-1850F)+A(1325F/8h+1150F/10h) Forg 300F air, >0.3Hz • Inconel 718; ST(1700-1850F)+A(1325F/8h+1150F/10h) GTA Weld STA; 600F Air>0.6Hz	**Sep 1999**
B1 & B2	Release of NASGRO 3.01 & 3.02 Versions with improvements in NASMAT for plotting data and code corrections for sustained load analysis in NASFLA.	**Oct 1999**
B3	Release of 3.03 version. Crack cases SC01 to SC05 now have lower a/c limit of 0.1. Order of input of segment data is changed to be left to right (in order of increasing Delta K) for Walker equation. Also a bug fix in computing c in sc07	**Nov 1999**
B4	Version 3.0.4: Minor bug fixes for SS08,SS09 and revised K solution for TC03, CC02.	**Mar 2000**

B5	Version 3.0.5: Minor bug fixes for SC08, SC13, SC14 and revised K solution for CC02 and CC04. Minor improvement to NASFLA GUI.	**April 2000**
B6	Version 3.0.6: Minor bug fix for crack case SC11. NASGLS and NASSIF GUI's: File selection now always uses GUI's directory as default. NASGLS GUI: Material parameter input boxes now editable only for manual input. Switching between database and manual entry for material parameters now tracked properly. NASFLA GUI and subroutine PLTAVN: Plotting improvements for postscript file. CRITICAL CRACK SIZE GUI: Correction made to input for model SC07. Correction made to code for loading an existing file. Appendix C document updated.	**April 2000**
B7	Version 3.0.7: Inconsequential changes to some routines to avoid compiler error on Compaq. Bug fixes: Crack case SC06 - Plot option for stress disabled. Crack case SC10 had an extra line in batch file.	**May 2000**
B8	Version 3.0.8: Bug fixes: FLAGUI Material routine - Not handling editing of material parameters when more than one material was involved. FLAGUI Schedule routine - Editing Reference stresses for multiple blocks was not working correctly. Indexing for plots for DT02, DT03 was not right Row and Col. numbers were interchanged in plotting tables for DT02, DT03 Reading Kc vs thickness was in wrong order Cth- and Rcl were not being saved to user material file Modified to put Kc value into KIc for KCOPT = 1 or 3	**July 2000**
B9	Version 3.0.9: Bug fixes: FLAGUI: Material routine - Change for USER material file option. Note: Material files for NASGRO ver. 2 do not have threshold values Cth+ and Cth- as required for ver 3. Change made to properly handle Modified Willenborg materal file editing. SIFGUI: "Geometry out of bounds" check corrected.	**Aug 2000**

B10	Version 3.0.10: Bug fixes: FLAGUI: Spectrum routine - Change to format and order of input for long blocks. Change to handle more than one material and combination of blocks. Material routine - Change to handle material files which have lines longer than 80 characters. Canvas routine - Change to display of Problem Title and Batch file name. K Solution: Correction for SC06 for certain crack sizes. Addition: FLAGUI & FORTRAN: Keac check for long blocks. Documentation: Change to reference manual and appendix B	**Sep 2000**
B11	Revision B11 October 2000 -------------------------- Version 3.0.11: Bug fixes: FLAGUI: Spectrum routine - Change to handle long blocks and model CC02. When using manual input of spectrum, blank cells are defaulted to zero. Material routine - Change made so that using non interaction and Walker equations together does not use RSO. NASFLA: Correction to crack case CC01 - Transition was not consistent. NASMAT: Updated to latest version of the graphical user interface, NASMAT.DLL and THRESH.DLL..	**Oct 2000**
B12	Version 3.0.12: Bug fixes: FLAGUI: Material routine - Correction for missing RSO text box. When entering da/dN and Delta K values, if there are more than 17 entries, put a blank line to simulate a pause before displaying next page. NASFLA: Walker model - change of format for printing constants. CMPT1X & CMPT2X - Underload effects incorporated with Walker-Chang model. SISC11 & SISC12 - Bug fix in K solution.	**Dec 2000**

B13	Version 3.0.13 Cosmetic changes: FLAGUI: In Material subroutine the choice of "Forman crack growth rate eqn. constants" was changed to "NASGRO crack growth rate eqn. constants" and "NASGRO" material file was changed to "NASA" material file. In the Specturm subroutine a text box for entry of step numbers in Kmax < Keac check was disabled when not being used.	**Feb 2001**
B14	Version 3.0.14 Replaced old versions of FLAGUI and CCSGUI which had inadvertently been included in version 3.0.13. Added updated versions of SIFGUI and GLSGUI along with NASFLA4.DLL required to run them. NASMAT: Updated to handle input of data in 5 different units (2 US and 3 SI) and to properly use US and ESA SI units for curve fitting.	**May 2001**
B15	Version 3.0.15 Bug Fixes: FLAGUI: Corrected two problems with keac check. One in the manual entry and one in the generate spectrum blocks TWIST & MINITWIST. MATGUI: Made a correction to entering units when new data is being entered. NASFLA: Transition from TC02 to TC03 Corrected Yield Warning in Critical Crack Size Module.	**Jul 2001**
B16	Version 3.0.16 Bug Fixes: NASFLA: Crack cases CC02 & CC04; K-solution for bending and pin loading changed.	**Aug 2001**

B17	Version 3.0.17 Bug Fixes: FLAGUI: Made Dko editable in manual material input. Correction to allow input of alpha when entering 1-D table using strip yield. CCSGUI: Correction to SC08 model to allow input of pitch. Corrected a problem with text boxes being too small. Corrected editing problem in SC09. Critical Crack Size: Corrected warning about yielding. NASFLA: Changed message for transition from TC03 to TC02. For SC03 internal pressure was not used correctly in net section stress calc. TC05: Solutions reverted to earlier series form. Material files: Corrected material O3FC50AB1 which was O3FB50AB1	**Nov 2001**
B18	FLAGUI: Modified schedule manual input so that blank cells in the grid are interpreted to be zero. Also corrected input for SC03 which uses stress quantities S0, S1, and S4. MATGUI: Corrected a grid problem with input of toughness data. It was filtering out everything except numbers.	**Jan 2002**
B19	FLAGUI: Bug fix for tabular input of da/dN data. Batch file was incorrect for 2 or more materials Bug fix – PS01 geometry input had no continue button.	**Jan 2002**

SYMBOLS

a .... Crack depth in thickness or diametral direction
a_0 .... Intrinsic crack length [8-9]
a_g .... Initial guess for critical crack size
A .... Fit parameter in Eq 5.2, constant in Eq 2.28
A_0, A_1, A_2, A_3 .... Coefficients in crack opening function, Eq 2.4-2.7
A_0 .... Parameter given in Appendix H
A_n .... Area of net section
A_k, B_k .... Fit parameters in Eq 2.12
B .... Fit parameter in Eq 2.23
c .... Crack length or half-crack length in width or peripheral direction
C .... Crack growth rate constant in Eq 2.1
Cth .... Coefficient in threshold Eq. 2.11
C_k .... Fit parameter in Eq 2.14
f .... Crack opening function, Eq 2.3
f .... Frequency
fn .... Resonant frequency
f, f_c, f_i, f_w, f_x, f_0, f_1, f_ϕ .... Functions given in Appendix C
F_0, F_1, F_2, F_3, F_4 .... Stress intensity magnification factors
g, g_p, g_w, g_3, g_4 .... Functions given in Appendix C
G, G_L, G_w, G_0, G_1 .... Functions given in Appendix C
h, k .... Functions given in Appendix C
H_c, H_1, H_2 .... Functions given in Appendix C
I, J .... Functions given in Appendix C
I_n .... Moment of inertia of net section
J_{Ic} .... Critical J-integral value (Mode I)
K .... Stress intensity factor (Mode I)
$K_{(a)}$ .... Stress intensity factor at the a-tip
$K_{(c)}$ .... Stress intensity factor at the c-tip
K_c .... Critical stress intensity as used in Eq 2.1
K_{cr} .... Critical stress intensity factor for fracture
K_{max}, K_{min} .... Maximum and minimum stress intensity factors in a load cycle
K_{open} .... Opening stress intensity factor, above which the crack is open
K_{Ic} .... Plane strain fracture toughness (Mode I)
K_{Ie} .... Effective fracture toughness for part-through (surface/corner) crack
$K_{Ie(a)}$ .... K_{Ie} value at the a-tip
$K_{Ie(c)}$ .... K_{Ie} value at the c-tip
K_{eac} .... Environmentally assisted cracking threshold
ΔK .... Stress intensity factor range (K_{max} - K_{min})
ΔK_{eff} .... Effective stress intensity factor range (K_{max} - K_{open})
ΔK_{th} .... Threshold stress intensity factor range

ΔK_0 .... Threshold stress intensity factor range at R = 0
M .... Resultant moment in Eq 2.15
M_0, M_1, M_2, M_3 .... Functions given in Appendix C
m .... Exponent in Walker Eq 2.25
n, p, q .... Exponents in NASGRO Eq 2.1
n, q .... Exponents in Chang Eq 2.26
n .... Fit parameter in Eq 2.24
N .... Number of applied fatigue cycles
P .... Resultant force in Eq 2.15
Q .... Amplification factor for a sine sweep vibration test
R .... Stress ratio (K_{min}/K_{max})
R_{cut^+} .... Positive cutoff stress ratio [21]
R_{cut^-} .... Negative cutoff stress ratio [21]
R_{so} .... Shutoff overload stress ratio [17, 18]
R_u .... Underload stress ratio used in Eq 2.24
S_0, S_1, S_2, S_3, S_4 .... Nominally applied stresses
S_{max} .... Maximum applied stress
S_n .... Nominal net section stress, Eq 2.15
t .... Thickness of plate, sheet, extrusion, or forging
t_0 .... Thickness to meet plane strain condition, Eq 2.13
V_0 .... Fit parameter in Eq 5.1
w .... Specimen width
u, *v*, *w*, *x*, *y*, *z* .... Functions given in Appendix C
Y .... Function given in Appendix C
α .... Plane stress/strain constraint factor
α, β, δ, λ, ζ .... Functions given in Appendix C
β_R .... Crack closure factor correction for free surfaces, Eq 2.10
ϕ .... Parametric angle of the ellipse
ϕ .... Reduction factor in Willenborg model Eq 2.22
ϕo .... Material constant in Modified Willenborg model Eq 2.24
λ .... Sweep rate of a sine sweep vibration test
η .... Exponent in polynomial equations for F_0, F_1, F_2, F_3
ν .... Poisson's ratio
ρ .... Crack tip plastic zone size, Eq 2.17
σ_0 .... Flow stress
σ_{ys} .... Tensile yield strength

ABBREVIATIONS

1-D .... One dimensional
2-D .... Two dimensional
3-D .... Three dimensional
Ann....................................... Annealed
ASW..................................... Aircraft sump water environment
ASTM................................... American Society for Testing and Materials
A(T)...................................... Arc-shaped tension specimen
BA .... Beta annealed
Cl.. Class
CR .... Cold-rolled
CRT...................................... Cold-rolled and tempered
C(T)...................................... Compact tension specimen
CW .... Cold-worked
DA.. Dry air environment
DC(T) .... Disc-shaped compact tension specimen
DW....................................... Distilled water environment
Dyn....................................... Dynamic
EB... Electron beam
ELI .... Extra-low interstitial
Eq .... Equation (number)
ESA .... European Space Agency
Extr....................................... Extrusion
Forg .... Forging
Grd .... Grade
GMA..................................... Gas metal arc
GN2 .... Gaseous nitrogen environment
GSFC.................................... Goddard Space Flight Center
GTA...................................... Gas tungsten arc
HAZ...................................... Heat affected zone
HHA .... High humidity air environment
HPD...................................... Hole preparation defect
JP-4....................................... JP-4 jet fuel environment
LA .... Laboratory air environment
LH2 .... Liquid hydrogen environment
LHe....................................... Liquid helium environment
LN2 .... Liquid nitrogen environment
MA .... Mill annealed
M(T) .... Middle (center)-cracked tension specimen
NASA .... National Aeronautics and Space Administration
NDE...................................... Nondestructive evaluation
NLR...................................... National Lucht-En Ruimtevaartlaboratorium
Nom...................................... Nominal

OA..Over-aged
PA ....Peak-aged
PAR......................................Parallel to weldline
Plt...Plate
RA..Recrystallization annealed
Rc...Rockwell table C hardness
Rnd.......................................Round rod
RTN......................................Return
SA ....Submerged arc
SCT ....Sub-cooled and tempered
SD ....Standard deviation
SE(B)....................................Single edge cracked three-point bend specimen
SE(T)....................................Single edge cracked tension specimen
Sht ....Sheet
SIF..Stress intensity factor
SMA......................................Shielded metal arc
SR...Stress relieved
SSF.......................................Stress scaling factor
ST...Solution treated
STA ....Solution treated and aged
STS.......................................Space transportation system
SW ....Seawater environment
Thk ....Thickness
UTS ....Ultimate tensile strength
VAR ....Vacuum arc remelted
YS ....Tensile yield strength

SUMMARY

This document is a comprehensive reference on the theory and operation of NASGRO version 3.0. It covers mainly the theoretical background of fatigue crack propagation behavior and details of the formulation used in the software. Other sections cover description of the crack geometries, the material database and the boundary element method. Major aspects of design and operation of the program are described. Several appendices are included to provide theoretical details of net-section stresses, stress intensity factors, material properties etc. More is covered here than is usually done in a user's guide; rather it is intended as a comprehensive reference on the methods that form the basis of the software. Because of the interactive nature of the graphical user interface, operation of the program is intended to be self-evident. On-line help is provided on selected topics and example problems are provided in the software. However, time spent in reading this manual will be well rewarded, especially for a beginning user.

NASGRO 3.0 is a suite of computer programs comprised of three modules named NASFLA, NASBEM and NASMAT. The NASFLA program is based on fracture mechanics principles that can be used to calculate stress intensity factors, compute critical crack sizes, or conduct fatigue and sustained load crack growth analyses. These analyses may be done in a fully interactive mode using graphical user interfaces for each sub-module. Material properties for crack growth can be picked up from a large database supplied with the program by selection from a menu of choices. Crack growth properties may also be entered either as a 1-D table or a 2-D table. The fatigue loading spectra can be input easily from a standard file or individual files. User-defined materials properties and fatigue spectra may also be supplied manually and saved for future use. The option to conduct sustained stress analyses is used for glass structures. The second module NASBEM incorporates the boundary element method for solving complex geometries with or without cracks to obtain stress intensity factors and stresses. The third module NASMAT is new in version 3.0 and is used to enter, edit and curve-fit fracture toughness and fatigue crack growth data obtained in a laboratory. It is designed to conveniently store and retrieve data and obtain curve fits to the NASGRO equation.

A major enhancement to NASFLA in this version is the modeling of crack growth under load interaction. Five models, the Generalized Willenborg, the Modified Generalized Willenborg, the Walker-Chang Willenborg, the Strip Yield and the Constant Closure model have been implemented. Some of these models account for the crack growth retardation only and the others account for both acceleration and retardation resulting from overloads and underloads in fatigue loading spectra.

A substantial number of other improvements have been incorporated into version 3.0. The stress intensity factor solutions were expanded to include some new crack cases. Consideration was given to efficiency, so that computations would be executed as quickly as possible. In addition, more options for defining blocks of fatigue loading and combining them to form load schedules have been included in the program. The blocks can be of any size and can be stored in separate files. The format for fatigue loads can be of standard

NASGRO type or of sequential type common in aircraft application. The sequential input is interpreted as being of cycle-by-cycle type. Thus the user is provided with greater flexibility in the definition and application of fatigue spectra.

New graphical user interfaces (GUI) have been built for each of the three modules to guide the users in a Windows environment to create input files and to facilitate easy input of fatigue crack growth data. In the case of NASFLA module, a number of pull-down menus along with timely presentation of the figures of crack geometries makes the input very intuitive. Under windows 95, the user can directly proceed to execute, once the necessary data have been entered. The results of analysis in text and graphic form can also be viewed by using menu choices. A new and unique feature in NASGRO 3.0 is the facility to input a user-specified crack growth equation into the windows front end which will then create a two-dimensional table of crack growth rate for certain ranges of stress intensity factor and stress ratios. This is one of the choices under material property input by the user. Another new option to input 2-D tables with different values of da/dN for each R value has been incorporated. Thus the user has complete control on how the material crack growth behavior can be modeled.

The boundary element module NASBEM is used to analyze two-dimensional geometries with holes and cracks. Stress analysis as well as computation of stress intensity factors at the tips of straight or curved cracks can be done using NASBEM. A new graphical user interface has been built for this module as well using the same wxWindows class library used for the NASFLA module. The geometric parameters such as points, lines and surfaces can be input easily using various pull-down menus and dialog boxes. The geometry can also be plotted immediately to check for errors. Once all the problem data have been entered, the data can be checked and processed to obtain the solution. Necessary changes to data can be made quickly and the problem can be reanalyzed. Output can also be viewed using the pull-down menu. The stress analysis portion of the software has now been separated so that once the solutions for displacements and tractions is obtained, the user can input the locations of interest for stress computation. The stress analysis module can then be executed using a command from the GUI. The stresses along a specified line segment can also be plotted.

The NASMAT module is used to enter, edit and curve fit the crack growth data and has so far been a DOS application used in-house. In the NASGRO 3.0 release, this module has been added and it also features a graphical front end for easy interaction. The data has been organized using an identification code (ID) and the header information for each set of data. The header consists of specimen type, environment, heat treatment and other relevant information. Specification of curve fit options are done using the various windows style input dialog boxes and menus. Fracture toughness data and fatigue crack growth data are the two types of data stored. Data sets are retrieved by specifying the ID in a fast and easy manner. The program allows the user to build the ID codes step by step and provides a number of options to retrieve the desired data. Once a set of data is retrieved, it can be edited, plotted and curve-fitted. This module thus provides a means to organize and process the fracture mechanics properties generated in a laboratory. As more data becomes available, the database will be continually updated. Some of the data in the database has not been fitted so far. This module is expected to be used predominantly in the laboratory and for research.

1.0 Introduction

The NASGRO computer software was developed to provide an automated procedure for fracture control analysis of NASA space flight hardware and launch support facilities. In addition, it is applicable to stress and fracture mechanics analysis of aircraft and non aerospace structures or hardware and may be used as a learning and research tool in fracture mechanics. The primary capability of the program is to calculate fatigue life and crack instability of cyclically or statically loaded structures that contain initial crack-like defects. The original version of NASGRO (NASA/FLAGRO) was completed in August 1986 and was revised in March 1989. General distribution of the program was initiated in 1990 by COSMIC the agency that used to distribute NASA-developed computer software. Version 2.0 was distributed in May 1994 followed by a revised version 2.03 in March 1995.

Features of the NASFLA module of the NASGRO software that are new for version 3.0 include:

New and improved stress intensity factor solutions

Improved threshold equation and associated material constants.

Load interaction (retardation/acceleration) models such as Generalized and Modified Generalized Willenborg models, Walker-Chang Willenborg, Strip Yield and Constant Closure model

Ability to read arbitrary-sized blocks of a fatigue spectrum from files

Ability to read the block mix or schedule information from a file

Ability to create and execute input files using a graphical user interface under PC Windows.

The software has been designed in a modular fashion, in order to allow for systematic revision and portability to various computer systems including personal computers. Appendix A provides the flow diagrams to describe how it is structured. In addition to the source code and executable files for the three modules (NASFLA, NASMAT, and NASBEM), NASGRO 3.0 provides a load spectrum blocks file (BLOCKS), a load step description (comments) file (BLKCOM), material properties files (NASMFC, NASMFM, NASTBC, NASTBM, NASMXC, NASMXM, USRMFC, USRMFM, USRTBC, USRTBM, USRMXC and USRMXM), and data files for sustained stress analysis (NASGLC and NASGLM). More information on these user-defined files is given in Appendix E.

When NASGRO3 (the top-level menu program) is executed (typically by clicking on an icon under windows applications) three radio box choices are presented. The first one brings up the fully integrated Windows 95 version of NASFLA, the second and third options will bring up the Windows 95 versions of NASMAT and NASBEM. Alternately, the individual modules can be directly executed by locating the appropriate executables and double-clicking on them. Once the graphical interface for any of the three modules is activated as mentioned, a set of pull-down menus guides the user to input the problem data, analyze the data and view the results

Using the NASFLA module for crack growth analysis, critical crack size calculation, and stress intensity factor computation will be covered in sections 2 through 4. The sustained stress analysis option will be described in section 5. Discussion of the module NASMAT for processing and storing of fracture mechanics data is contained in section 6 and the boundary element module NASBEM is covered in section 7. Input files are provided for about ten examples. These examples serve to illustrate the use of various options and help verify the installation of the software. In addition, a few examples of the input process are illustrated by means of bit maps of screen captures. This is in the form of HTML files which can be viewed using the default browser on the user's system. On-line help is also available in the form of HTML files that are also viewed using the browser.

Several appendices at the end of the manual are provided in which detailed information on topics such as net-section stress computations (Appendix B), the stress intensity factor solutions (Appendix C), material properties for NASGRO equation (Appendices G1, G2) etc., is given. Also, the information on computer system requirements, including hardware, software, and program installation procedures are given in Appendix I. Information on conversion of units is given in Appendix J. The relation between stress quantities and fatigue loads is described in Appendix K. The material constants for use with the Walker equation are given in appendices L1 and L2.

2.0 Fatigue Crack Growth Analysis

2.1 Theoretical Background

This section provides the theoretical background for the equations used in NASGRO 3.0. It is generally assumed that the user has a basic understanding of the principles of fracture mechanics and fatigue crack growth. Comprehensive treatment of fracture mechanics applications and the underlying theory may be found in references [1 - 4].

2.1.1 Crack Growth Relationship

Crack growth rate calculations in NASGRO 3.0 use a relationship called the NASGRO equation. Different elements of this equation were developed by Forman and Newman of NASA, Shivakumar of Lockheed Martin, de Koning of NLR and Henriksen of ESA and was first published by Forman and Mettu [5]. It is given by:

$$\frac{da}{dN} = C\left[\left(\frac{1-f}{1-R}\right)\Delta K\right]^n \frac{\left(1-\frac{\Delta K_{th}}{\Delta K}\right)^p}{\left(1-\frac{K_{\max}}{K_c}\right)^q} \tag{2.1}$$

where N is the number of applied fatigue cycles, a is the crack length, R is the stress ratio, ΔK is the stress intensity factor range, and C, n, p, and q are empirically derived constants. Explanations of the crack opening function, f, the threshold stress intensity factor, ΔK_{th} and the critical stress intensity factor, K_c are presented in sections 2.1.2 through 2.1.4. Equation 2.1 produces da/dN-ΔK curves that are similar to those obtained from the equation used in the initial (1989) version of NASGRO, but it provides a more direct formulation of the stress-ratio effect. Also, with Eq 2.1, variations in K_c and ΔK_{th} values can have a reduced effect on the linear region of the curve (by a suitable choice of $K_{\max}$), which produces a better fit to data. Figure 1a shows crack growth data (da/dN-ΔK) for the BT14 Titanium alloy (Russian material) plotted together with a curve fit to Eq 2.1. Figure 1b shows the curve fit for the A357 Cast Aluminum alloy along with the crack growth data. This material exhibits a rough fatigue surface unlike most materials. It was not possible to satisfactorily fit this data with the usual values of q and a much larger value of 2.0 was needed to enhance the effect of $K_{\max}$ on the crack growth rate thus achieving a reasonable fit.

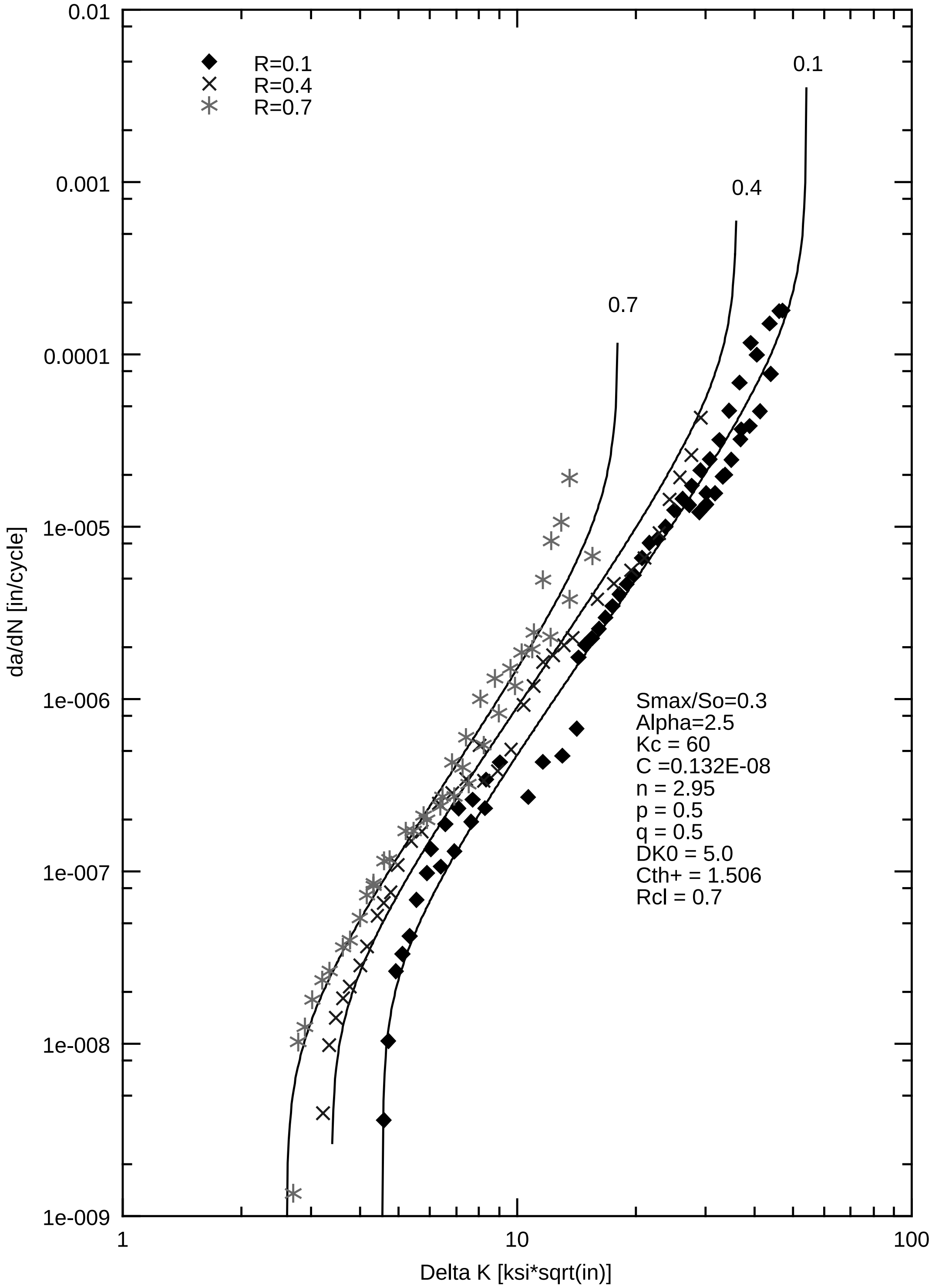


Figure 1a – Curve fit to Eq. 2.1 for BT14 Titanium (Russian)

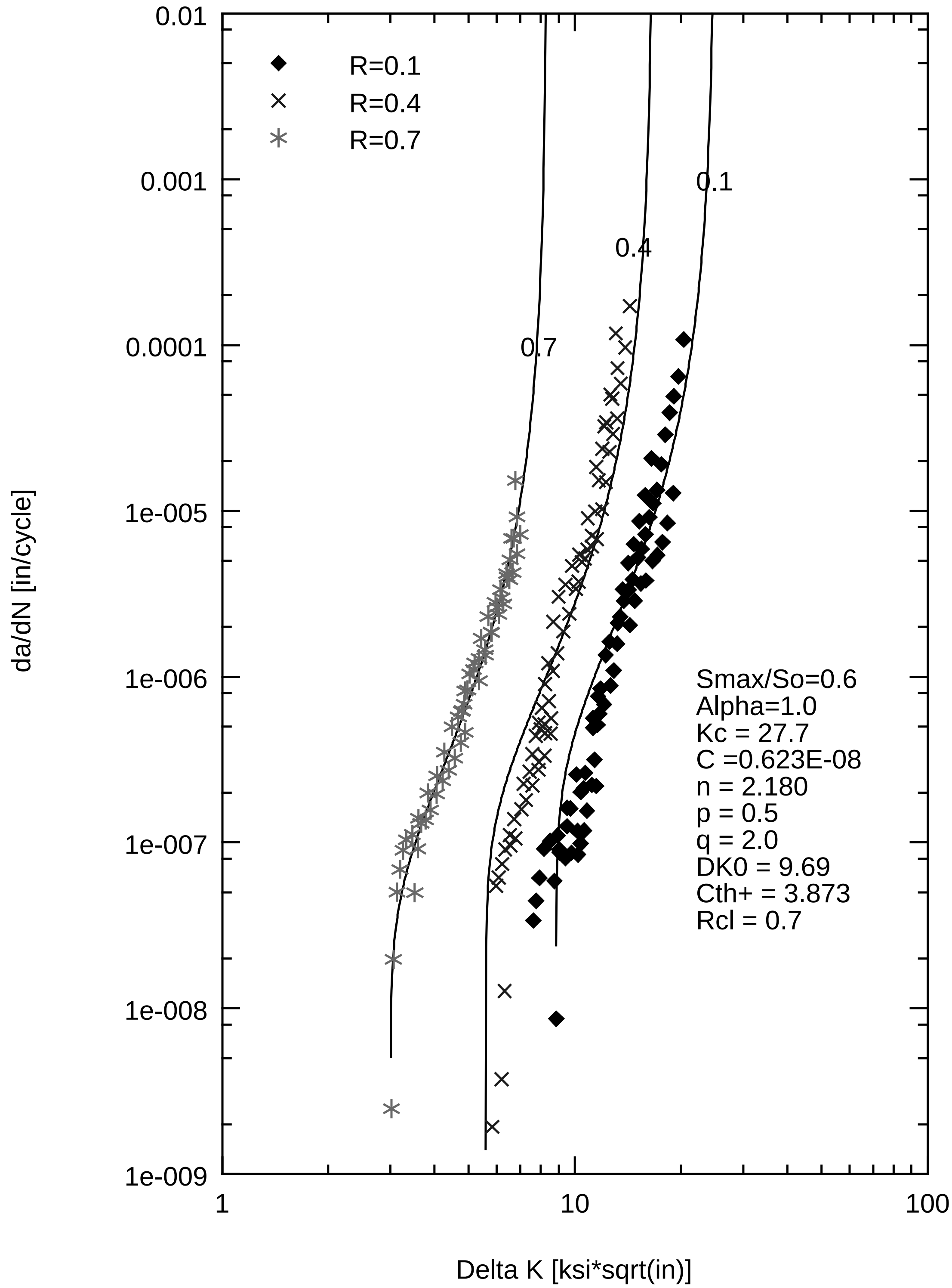


Figure 1b – Curve fit to Eq. 2.1 for A357 Cast Aluminum

To analyze cracked bodies under combined loading, the stress intensity factor is expressed as:

$$K = [S_0F_0 + S_1F_1 + S_2F_2 + S_3F_3 + S_4F_4]\sqrt{\pi a} \tag{2.2}$$

The stress quantities S_0, S_1, S_2, and S_3 are the applied tension/compression, bending in the thickness and width directions, and pin bearing pressures. For the crack case TC05, which has biaxial tension or compression loading, the term S_4 is used for the stress in the lateral direction. The F_i values are geometric correction factors applicable to each type of applied stress and derived specifically for each crack case.

The program incorporates fatigue crack closure analysis for calculating the effect of the stress ratio on crack growth rate under constant amplitude loading. The crack opening function, *f*, for plasticity-induced crack closure has been defined by Newman [6] as:

$$f = \frac{K_{op}}{K_{max}} = \begin{cases} \max\left(R,\, A_0 + A_1R + A_2R^2 + A_3R^3\right) & R \geq 0 \\ A_0 + A_1R & -2 \leq R < 0 \end{cases} \tag{2.3}$$

and the coefficients are given by:

$$A_0 = \left(0.825 - 0.34\alpha + 0.05\alpha^2\right)\left[\cos\left(\tfrac{\pi}{2} S_{max} / \sigma_0\right)\right]^{1/\alpha} \tag{2.4}$$

$$A_1 = (0.415 - 0.071\alpha) S_{max} / \sigma_0 \tag{2.5}$$

$$A_2 = 1 - A_0 - A_1 - A_3 \tag{2.6}$$

$$A_3 = 2A_0 + A_1 - 1. \tag{2.7}$$

In these equations, α is a plane stress/strain constraint factor, and S_{max} / σ_0 is the ratio of the maximum applied stress to the flow stress. Selection of values for these parameters will be discussed in the next section.

2.1.2 Fatigue Crack Closure

The plane stress/strain constraint factor, α, has been treated as a constant for the purposes of curve fitting the crack growth data for each particular material system. Values range from 1, which corresponds to a plane stress condition, to 3, which corresponds to a condition of plane strain. Materials, such as high-strength steels, for which the K_{Ic}/σ_{ys} ratio is fairly low, are assigned relatively high α values (2.5 or higher), while materials with higher K_{Ic}/σ_{ys} ratios usually have α values ranging from 1.5 to 2.0. While better correlation with experimental results may be obtained by allowing α to vary with K_{max} [6], reasonable agreement has been obtained by using it strictly as a fitting parameter.

In addition, S_{max}/σ_0, the ratio of the maximum applied stress to the flow stress, is assumed to be constant. Using this parameter as a constant has been shown to produce acceptable results for positive stress ratios, where the effect of S_{max}/σ_0 on the crack opening function is relatively small [5]. Most materials that were curve fit for NASGRO 3.0 use a value of $S_{max}/\sigma_0 = 0.3$, which was chosen because it is close to an average value obtained from fatigue crack growth tests using various specimen types.

Some materials, however, exhibit only a very small stress ratio effect, and therefore may be evaluated without considering the effects of crack closure. In these special cases, a curve-fitting option that allowed the crack opening function to be bypassed was chosen. The parameters for this bypass option are $\alpha = 5.845$, $S_{max}/\sigma_0 = 1.0$. These values were selected in order that f in Eq 2.3 would be equal to zero for negative stress ratios and would be equal to R ($K_{op} = K_{min}$) for $0 \le R < 1$. Thus, for positive stress ratios, the Eq 2.1 crack growth relationship reduces to:

$$\frac{da}{dN} = \frac{C\Delta K^n \left(1 - \frac{\Delta K_{th}}{\Delta K}\right)^p}{\left(1 - \frac{K_{max}}{K_c}\right)^q} \tag{2.8}$$

where the entire ΔK range contributes to crack propagation, and, for negative stress ratios, reduces to:

$$\frac{da}{dN} = \frac{C\, K_{max}^n \left(1 - \frac{\Delta K_{th}}{\Delta K}\right)^p}{\left(1 - \frac{K_{max}}{K_c}\right)^q} \tag{2.9}$$

since $\Delta K/(1-R) = K_{max}$. Figure 2 shows the opening function, f, as a function of the stress ratio for $S_{max}/\sigma_0 = 0.3$ and $\alpha = 2.5$, typical for many steels, together with the $S_{max}/\sigma_0 = 1.0$, $\alpha = 5.845$ closure bypass option. Note that f, which reflects the amount of plasticity-induced crack closure that is present in a material, approaches the closure bypass option at the higher stress ratios (around R = 0.7). However, for negative stress ratios, the S_{max}/σ_0 ratio greatly influences the amount of closure present in the material. Therefore, unconservative life predictions may be obtained from NASGRO analyses of structures such as overwrapped pressure vessels, which cycle to a relatively high compressive stress, with a negative stress ratio on the order of R = -1 to -2. In such cases, better correlation with test results may be achieved by substituting the absolute value of S_{min} for S_{max} and using this to calculate an appropriate S_{max}/σ_0 value.

It should also be noted that Eq 2.8 (the closure bypass option for R>0 with $\alpha = 5.845$, $S_{max}/\sigma_0 = 1.0$) may be further reduced to the Paris equation ($da/dN = C\Delta K^n$) by setting the parameters p and q equal to zero. Similarly, Eq 2.1 may be reduced to a closure-corrected

Paris equation by setting *p* and *q* equal to zero. In either case, the threshold (ΔK_{th}) and critical crack growth (K_c) asymptotes are retained as cut-off values.

For constant amplitude fatigue loading, Newman and Raju [7] have shown that multiplying ΔK by a crack-closure factor, β_R, produces more accurate crack growth predictions for semi-elliptical surface cracks and quarter-elliptical corner cracks. This β_R factor is only applied at points where the crack front intersects a free surface, and it is a function of the stress ratio. For R>0, β_R is given by:

$$\beta_R = 0.9 + 0.2R^2 - 0.1R^4 \tag{2.10}$$

and for R ≤ = 0, β_R is assumed to have a value of 0.9. In NASGRO 3.0, ΔK is multiplied by β_R for all corner crack models and for the surface crack in a plate (SC01 and SC02), the surface crack in a sphere (SC03) and the surface crack in a hollow cylinder (SC04 and SC05) cases.

As a final note on crack closure in NASGRO 3.0, it should be remembered that the crack opening function was derived from an analysis of center-cracked panels, subjected to a

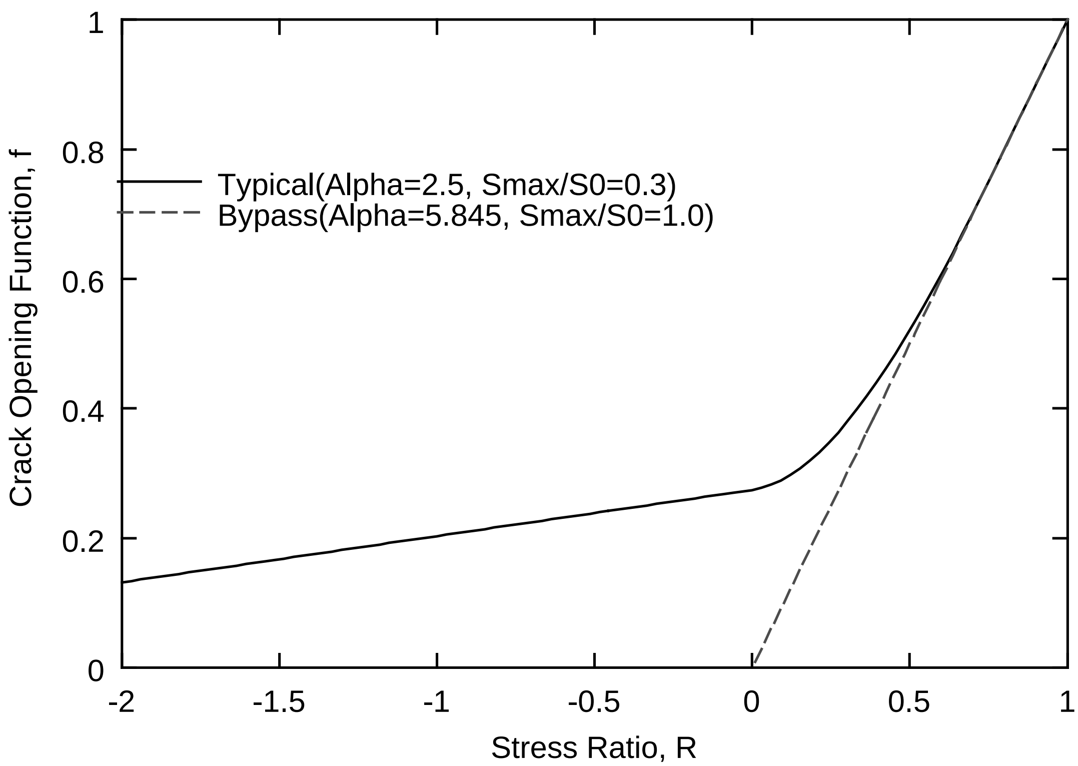


Figure 2 – Crack opening function vs. stress ratio

constant load amplitude condition, in which the crack front advances through a zone of plastically deformed material. This means that the effect of bending was not included in the analysis and that the crack opening equation (Eq 2.3) accounts for plasticity-induced crack closure, but not necessarily other effects, such as extensive oxide-induced or roughness-induced crack closure.

2.1.3 Threshold Stress Intensity Factor Range

The threshold stress intensity factor range in Eq 2.1, ΔK_{th}, is approximated by the following empirical equation:

$$\Delta K_{th} = \Delta K_0 \left[\frac{a}{a + a_0} \right]^{1/2} \bigg/ \left[\frac{1 - f}{(1 - A_0)(1 - R)} \right]^{(1 + C_{th} R)} \tag{2.11}$$

where R is the stress ratio, f is the Newman closure function, A_0 is a constant(Eq 2.4) used in f, ΔK_0 is the threshold stress intensity factor range at $R = 0$, C_{th} is an empirical constant, a is the crack length, and a_0 is an intrinsic crack length. This equation is a modification of a previous one [5], using the arctan function, that takes into account the small crack effect demonstrated by Tanaka, et al. [8]. The present form of the equation was chosen over the ΔK_{th} formulation used in the NASGRO 2.0 version because it provides a better fit to experimental crack growth data. The spread for various R ratios can be controlled much better using the parameter C_{th}. Values of C_{th} for positive and negative values of R, and ΔK_0 are stored as constants in the NASGRO materials files. The intrinsic crack size a_0 has been assigned a fixed value of 0.0015 in. (0.0381 mm).

Figure 3 shows the influence of crack size on the normalized threshold behavior of various steel alloys. It shows a plot of $\Delta K_{th}/\Delta K_{th(LC)}$ versus crack depth data, which were compiled by Lindley [9], where $\Delta K_{th(LC)}$ represents the "long crack" fatigue threshold. While some researchers have proposed models for a_0 based on grain size, others have not assigned any physical significance to the a_0 parameter [8 - 9]. As shown in Figure 3, using a constant value of 0.0015 in. (0.0381 mm) for a_0 provides a good fit to the data points. Further, this value agrees with an asymptotic merging of short crack with long crack behavior at around 0.025 in. (0.635 mm), the demarcation coded into the NASGRO March 1989 version.

The parameter R_{cl} is the cutoff stress ratio above which the threshold is assumed constant. To increase the flexibility with which data can be fitted, provision has been made to use independent value of C_{th} for negative and positive R values. Fig 4 shows a fit of data for 2219-T6 Aluminum alloy. The values of constants chosen to obtain this fit are: $C_{th}^{+} = 2.8$, $C_{th}^{-} = 0.4$, $R_{cl} = 0.62$, $\alpha = 2.0$, $S_{\max} / \sigma_0 = 0.3$. Analogous to the use of R_{cl} for large positive R values, a value of R $(= R_p)$ at which the threshold stress intensity factor reaches a maximum value is computed, for all values of R less than R_p, the threshold is assumed to be constant.

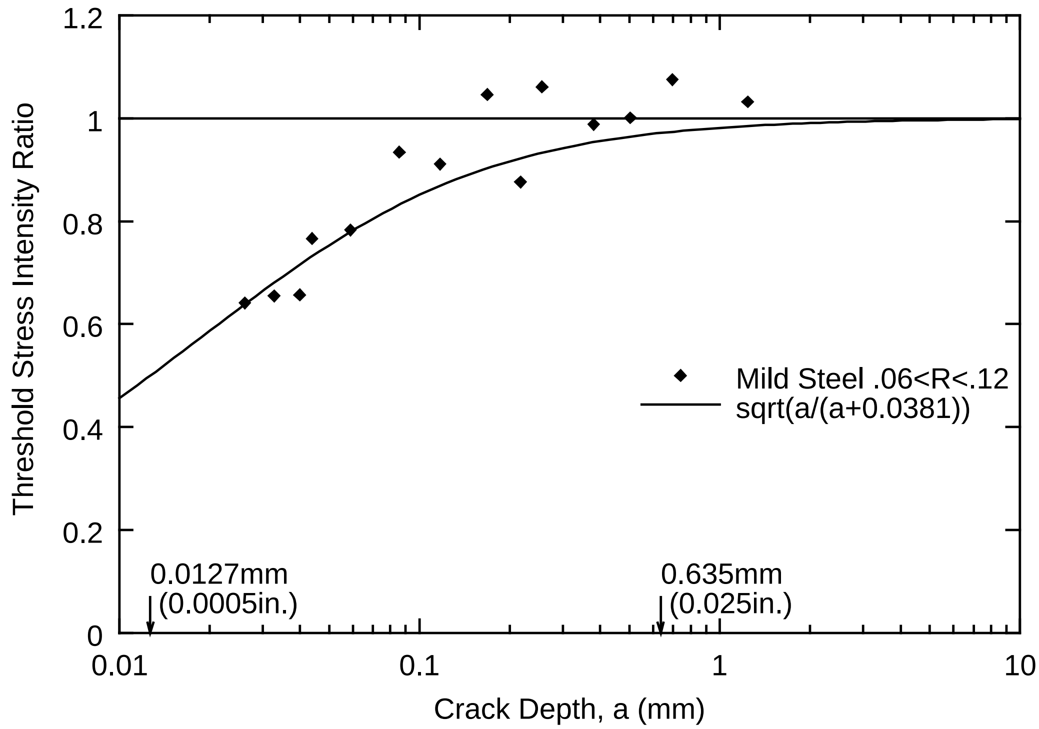


Figure 3 – Normalized threshold vs. crack depth for steel after Lindley [9]

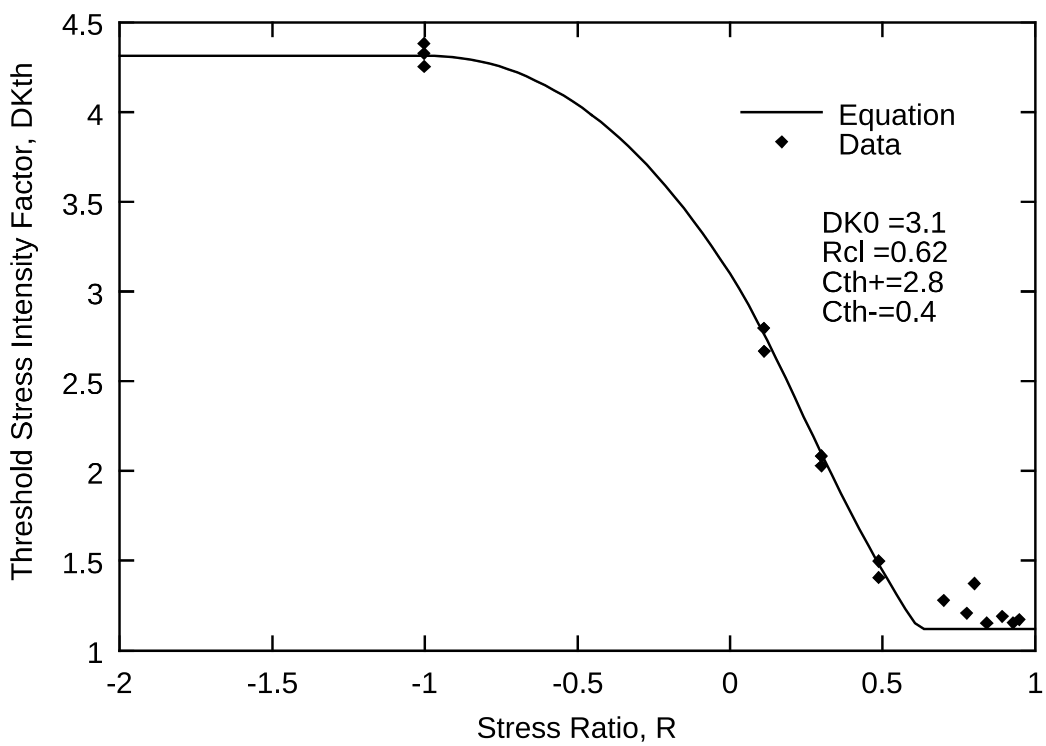


Figure 4 – Crack growth threshold vs. R for aluminum

In some cases, where limited threshold data were available for a particular material, ΔK_0 was estimated from data for a similar material. For example, Figure 5 shows a plot of $\Delta K_0/\Delta K_{th}$ versus σ_{ys} data, which was used to estimate ΔK_0 for various steel alloys when threshold data were limited or unavailable. Many researchers have recognized the correlation between yield strength and ΔK_{th}. For example, Yoder, et al. [11], formulated an equation for ΔK_{th} that is related to yield strength and the square root of grain size. However, where data are available, Eq 2.11 provides a better fit to threshold data at both positive and negative stress ratios.

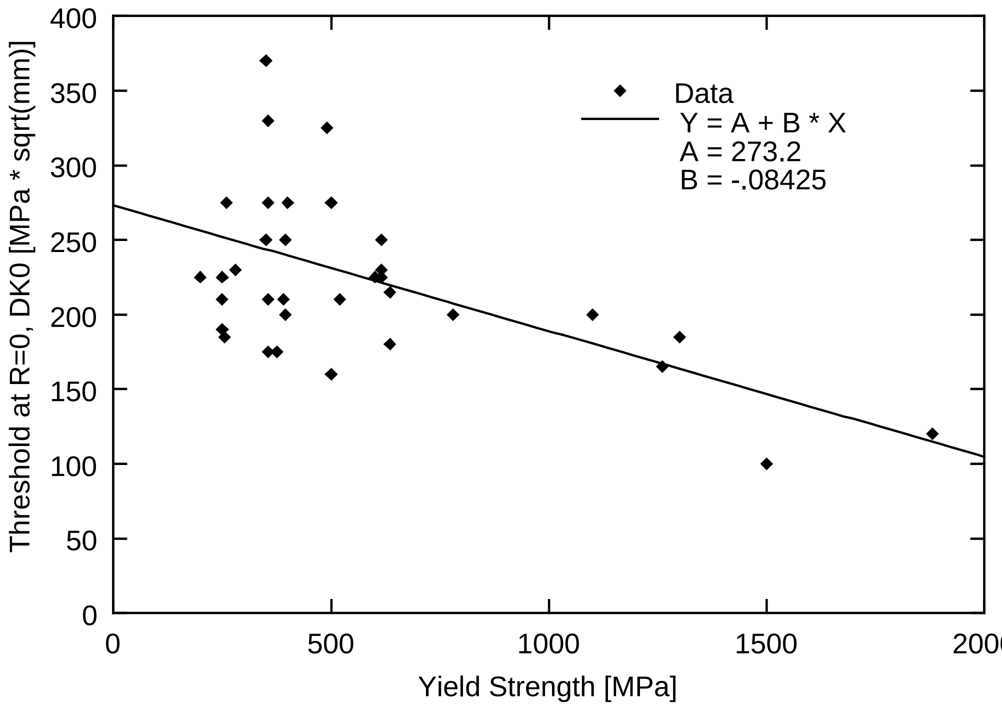


Figure 5 – Threshold at R=0 vs. yield strength for steels

2.1.4 Fracture Toughness

Fracture toughness properties of a material are essential for reliable crack growth analyses. These include plane strain fracture toughness (K_{Ic}) values, part-through fracture toughness (K_{Ie}) values, and any other available fracture toughness (K_c) values as a function of thickness. K_{Ie} values are especially important because flaws in real structures are often part-through (surface or corner) cracks. The main reason that fracture toughness depends on thickness is that differences in constraint produce changes in the stress state in the material. Thin structures, where the constraint is small, experience a plane stress condition, whereas thicker structures that are in plane strain have more constraint.

The following relationship has been adopted, for through crack problems, to describe the K_c –vs.-thickness behavior for various materials:

$$K_c / K_{Ic} = 1 + B_k e^{-\left(A_k \, t/t_0\right)^2} \qquad (2.12)$$

where

$$t_0 = 2.5\left(K_{Ic} / \sigma_{ys}\right)^2 . \qquad (2.13)$$

This is a generalization of the relationship proposed by Vroman [12], which can be obtained from Eq 2.12 and 2.13 by letting A_k = 5 and B_k = 1. These equations are used by NASGRO 3.0 to calculate a K_c value to substitute into Eq 2.1 for all through crack geometries (TCxx and SSxx crack cases). The one-dimensional surface crack cases (SC06, SC09, and SC10) also use Eq 2.12 and 2.13 for K_c determination. Figure 6 shows a plot of this curve for beryllium-copper alloy CDA172 [13]. When valid plane strain fracture toughness data were not available for a curve fit, K_{Ic} was estimated, based either on K_{Ic} data for a similar material or J_{Ic} data whenever possible, or derived from K_c values that were estimated from the asymptotes of the da/dN-ΔK curves.

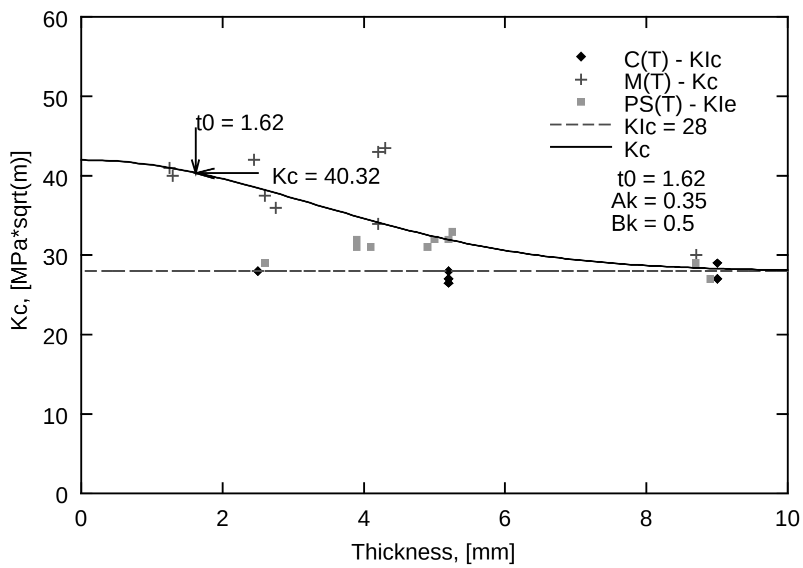


Figure 6 – Curve fit of Eq. 2.12 for Be-Cu CDA172

In addition to being a function of thickness, fracture toughness depends on crack length, or more directly, on stress level. This commonly known R-curve effect is especially prominent in thin structures. Figure 7 shows K_c data, obtained from 2219-T87 aluminum specimens, plotted as a function of thickness for different crack lengths. It should be noted that the K_c -vs.-thickness curve fit for a given material in the NASGRO 3.0 database currently represents an average of the data points and does not account for the effect of stress level/crack size on fracture toughness. Therefore, the user should be cautioned that a crack growth analysis of a particularly high stress, low cycle fatigue condition could produce unconservative results at shorter crack lengths. Fracture toughness data obtained from part-through cracked PS(T) specimens can also show a variation in toughness with crack size and stress level, but demonstrate little dependence on thickness. For the part-through crack geometries (CCxx and two-dimensional SCxx crack cases), K_c in Eq 2.1 is set equal to a constant value of K_{Ie}, taken from the NASGRO material properties files. During the curve-fitting process, K_{Ie} for a given material/ environment combination was selected, based on available fracture toughness data from PS(T) or other surface-cracked specimens whenever possible. Otherwise, K_{Ie} was approximated by the following equation [14]:

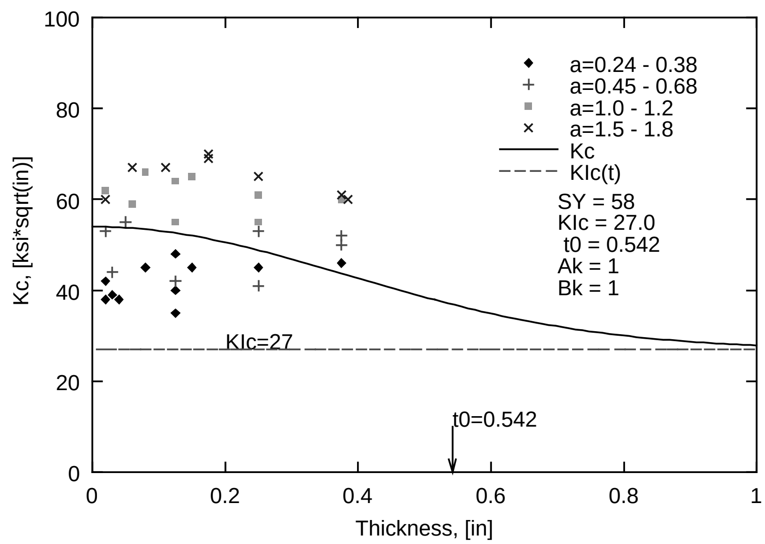


Figure 7 – Kc vs. thickness showing R-curve effect for Aluminum 2219-T87

$$K_{Ie} = K_{Ic}\left(1 + C_k K_{Ic} / \sigma_{ys}\right) \tag{2.14}$$

where C_k is an empirical constant with units of length$^{-1/2}$. The value of C_k is 1 for US units and 6.275 for SI units. This relationship holds reasonably well for a variety of materials, as shown in Figure 8. For materials which have a very high K_{Ic}/σ_{ys} ratio, K_{Ie} values calculated by this equation are very large. In these cases, to be conservative, the K_{Ie} values entered into the NASGRO materials files have been limited to 1.4 times the K_{Ic} values.

2.1.5 Criteria for Failure

In NASGRO 3.0, crack instability is usually assumed to occur if K_{max} exceeds the fracture toughness (K_c) of a material. For through crack geometries, K_{max} is compared with K_c calculated from Eq 2.12 and 2.13. For most of the part-through crack geometries, K_{max} at both the a-tip and c-tip are compared with K_{Ie}. However, for the part-through crack cases that have free surfaces, K_{max} at the c-tip (SC01-SC05, SC11, SC12) or K_{max} at the corner points (CC01-CC04) is compared with 1.1 times K_{Ie}. Failure is also assumed to occur if the net section stress exceeds the flow stress of the specified material. The flow stress is assumed to be the average of the yield and ultimate strengths.

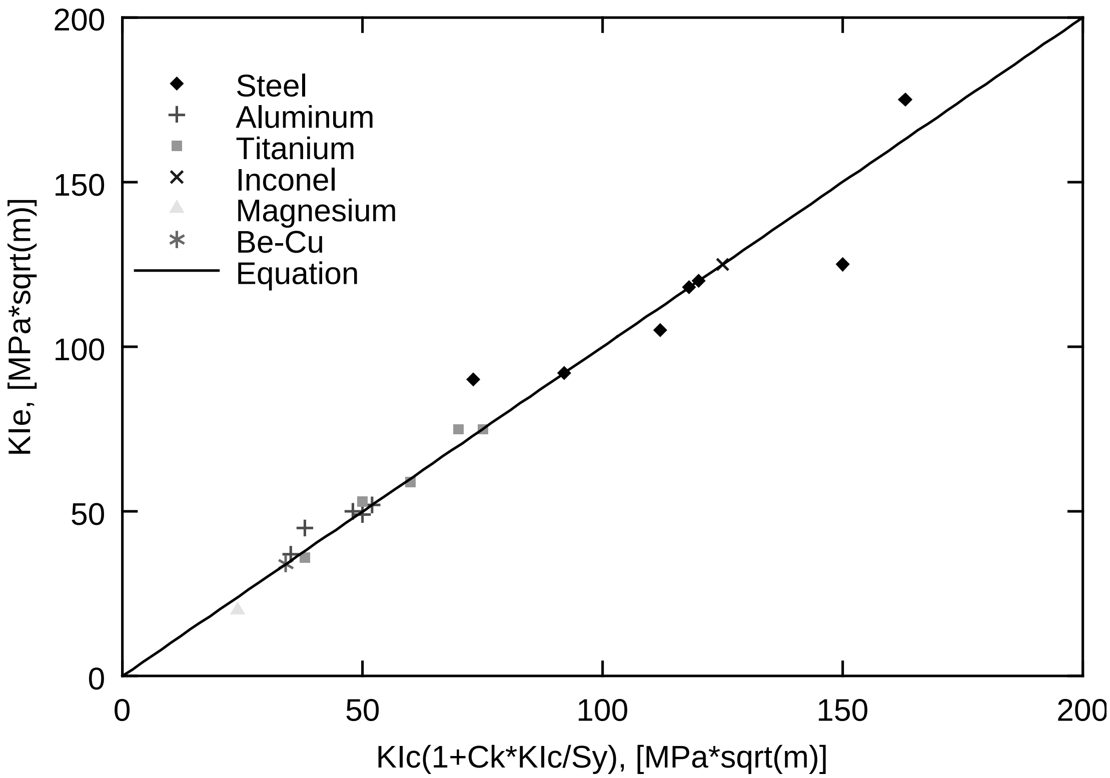


Figure 8 – Surface crack fracture toughness curve fit

In addition, there are two optional conditions for failure which may be specified by the user. The first is that the user may select an option of checking K_{max} with the threshold for environmentally-assisted (stress corrosion) crack growth (K_{eac}) for some or all of the blocks entered in a schedule. If K_{max} exceeds a user-specified value of K_{eac}, fracture is assumed and the crack growth analysis is terminated. More details about setting the K_{eac} check may be found in section 2.2.4.2. The second failure condition involves the use of a "reference stress", which can be specified during each block case entry. This procedure is described in section 2.2.4.4. Here, failure is assumed to occur if the value of K calculated using the "reference stress" (e.g., stress value at limit load) exceeds K_c for the specific material used. The K_c value selected depends on whether the crack is through or part-through, and is based on the criteria outlined in the previous paragraph.

2.1.6 Yielding Checks

A warning is given when the net-section stress exceeds the yield strength of the material, where the net-section stress is expressed as:

$$S_n = P/A_n + Mc/I_n \quad . \tag{2.15}$$

Here, A_n is the net area, P is the resultant force, M is the resultant moment, c is the distance to the outer fibers, and I_n is the moment of inertia in the net section. Appendix B contains the net section yielding equations that are applicable to each of the crack cases. Crack growth calculations continue after the net section warning is issued, but the user should realize that the results may be nonconservative. One reason for this is that K_c is reduced as the material exceeds yield. Therefore, in situations where a part must be analyzed above net section yield, further life may be obtained, but a lower toughness value should be used.

For most of the part-through crack cases, both K_{Ic} and K_{Ie} data are required because checks are made for failure as a through crack when net ligament yielding occurs [15]. The ligament yielding check is made using the following criterion:

$$a + \rho \geq t \tag{2.16}$$

where the plastic zone size, ρ, is given by:

$$\rho = \frac{1}{2\pi}\left(\frac{K_{\max}}{\sigma_{ys}}\right)^2 . \tag{2.17}$$

This check is made, but no warning is given. Before the ligament yielding criterion is met, K_{max} is compared to K_{Ie}, and after it is met, K_{max} for the appropriate through crack is also compared with K_c calculated from Eq 2.12 and 2.13. For surface cracks, the program also checks to see if the crack depth has exceeded the thickness of the part. When this occurs, the crack will transition to the appropriate through crack solution, if available. A list of crack cases showing transition relationship is shown in Table 5b.

2.1.7 Crack Growth under Load Interaction

2.1.7.1 Generalized Willenborg Model

The Generalized Willenborg model, based on Gallagher's [17] generalization of Willenborg's [16] original development, was incorporated into NASGRO 3.0. This model deals with crack retardation effects only and the formulation is as follows.

The effect of current loading on crack growth is known to be influenced by the load history; the term "load interaction" describes the interplay of these influences. The Generalized Willenborg model, utilizes a residual stress intensity, K_R, which determines the effective stress ratio due to a load interaction as follows:

$$R_{eff} = \frac{K_{\min} - K_R}{K_{\max} - K_R} = \frac{K_{\min,eff}}{K_{\max,eff}} \tag{2.18}$$

This value of R_{eff} is used instead of the actual stress ratio within the crack growth equation and has the effect of retarding the crack growth. Since K_R depends on threshold which in turn depends on R_{eff}, an iterative scheme has been developed and used.

The retardation for a given applied cycle of loading depends on the loading and the extent of crack growth into the overload plastic zone. Gallagher [17, 18] expressed the Willenborg residual stress-intensity factor as

$$K_R{}^W = K_{\max}{}^{OL}\left(1 - \frac{\Delta a}{Z_{OL}}\right)^{\frac{1}{2}} - K_{\max} \tag{2.19}$$

where $K_{\max}{}^{OL}$ is the maximum stress intensity for the overload cycle, and Δ a is the crack growth between the overload cycle and the current cycle as shown in Figure 9. The overload plastic zone size is given by

$$Z_{OL} = \frac{\pi}{8}\left(\frac{K_{\max}}{\alpha_g \sigma_{ys}}\right)^2 \tag{2.20}$$

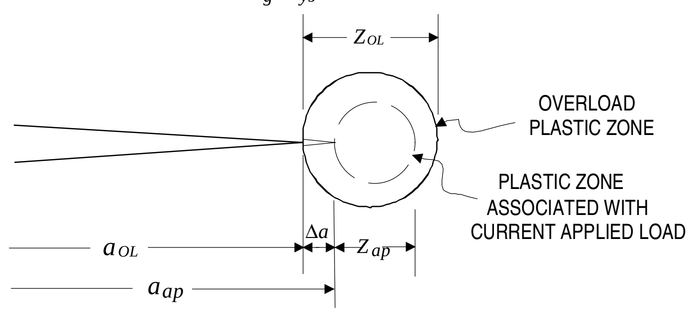


Figure 9 - Crack tip plastic zones

The constraint factor α_g is taken from a fit developed by Newman [19] and is given by

$$\alpha_g = 1.15 + 1.4\, e^{-0.95\left(\frac{K_{\max}}{\sigma_{ys}\sqrt{t}}\right)^{1.5}} \tag{2.21}$$

This expression is used for one-dimensional crack models; for two-dimensional cases, limit values of 1.15 or 2.55 are used for α_g depending on whether the crack tip under consideration emerges through the free surface (plane stress) or is buried (plane strain).

$K_R{}^W$ represents the difference between the stress intensity required to produce a plastic zone equal to $Z_{OL} - \Delta a$ and the current maximum applied stress intensity $K_{\max}$. In the original development, retardation is considered to occur if $K_R{}^W > 0$. In the Generalized Willenborg model, a modified residual stress-intensity K_R is used, related to $K_R{}^W$ by

$$K_R = \phi\, K_R{}^W \tag{2.22}$$

where

$$\phi = \frac{1 - \dfrac{\Delta K_{th}}{\Delta K}}{\left(R_{SO} - 1\right)} \tag{2.23}$$

R_{SO} is the shut-off value of the stress ratio, $K_{\max}{}^{OL} / K_{\max}$. When this value is exceeded, $K_{\max,eff}$ is set equal to $\Delta K_{th} / (1 - R)$ and crack growth is arrested. No special consideration is given to multiple overloads and their effect is taken to be the same as that for a single overload.

2.1.7.2 Modified Generalized Willenborg Model

A load interaction model termed the Modified Generalized Willenborg (MGW) model was developed by T. R. Brussat of Lockheed Martin. Based on this formulation, a computer model was developed and incorporated into NASGRO 3.0. The following description is based on his private communication and lecture notes [20].

The MGW model extends the Generalized Willenborg load interaction model [17] by taking into account the reduction of retardation effects due to underloads. The MGW model (like the Generalized Willenborg), utilizes a residual stress intensity, K_R, which determines the effective maximum and minimum stress due to a load interaction. The equations are:

$$K_{\max}^{eff} = K_{\max} - K_R$$

$$K_{\min}^{eff} = Max\{(K_{\min} - K_R), 0\}, \quad for\ K_{\min} > 0$$

$$= K_{\min} \qquad for\ K_{\min} \le 0$$

These effective stress intensity factors are used instead of the actual $K_{\max}, K_{\min}$ within the crack growth equation and have the effect of retarding the crack growth. In addition, an underload (i.e., a compressive or tensile load that is lower than the previous minimum load subsequent to the last overload cycle) can reduce such retardation. The stress ratio R_U given by $S_{UL} / S_{\max}{}^{OL}$ (the ratio of current underload stress* to overload stress) is used to adjust the factor ϕ. This reduction is achieved by means of Eqn. (2.22). The factor ϕ in that equation is now given by

* Proper rainflow cycle counting will pair the underload minimum with the overload maximum as part of the reordering process that precedes input of the spectra to NASGRO

$$\begin{aligned} \phi &= 2.523\phi_0 / (1.0 + 3.5(.25 - R_U)^{.6}), \quad R_U < 0.25 \\ &= 1.0 \qquad\qquad\qquad\qquad\qquad\quad , \quad R_U \geq 0.25 \end{aligned} \tag{2.24}$$

The parameter ϕ_0 is the value of ϕ for $R_U = 0$. Parameter ϕ_0 is a material dependent parameter that can be determined, ideally, by conducting a series of typical aircraft spectrum tests. The value of ϕ_0 ranges typically from 0.2 to 0.8.

2.1.7.3 Walker-Chang Willenborg Model

Another load interaction model implemented into NASGRO 3.0 is the Walker-Chang model developed at Rockwell. Chang and Engle [21] developed a version of the Generalized Willenborg model which takes into account the acceleration due to negative loads. The formulation was computerized into a code named CRKGRO at Rockwell under contract from US Air Force. They use the Walker equation for positive stress ratios and an equation developed by Chang for negative stress ratios. The retardation effects are modeled as in the case of Generalized Willenborg. The following set of equations defines the basic Walker-Chang model.

For $\Delta K > \Delta K_{th}$, $R \geq 0$

$$\begin{aligned} &da / dN = C\left[\Delta K / (1 - \bar{R})^{1-m}\right]^n \\ &R < R_{cut}^+ ,\ \bar{R} = R \\ &R > R_{cut}^+ ,\ \bar{R} = R_{cut}^+ \end{aligned} \tag{2.25}$$

For $\Delta K > \Delta K_{th}$, $R < 0$

$$\begin{aligned} &da / dN = C\left[(1 + \bar{R}^2)^q K_{\max}\right]^n \\ &R \geq R_{cut}^- ,\ \bar{R} = R \\ &R < R_{cut}^- ,\ \bar{R} = R_{cut}^- \end{aligned} \tag{2.26}$$

For $\Delta K < \Delta K_{th}$,

$$da / dN = 0 \tag{2.27}$$

In the above equations, R_{cut}^+, R_{cut}^- are the cutoff values for positive and negative stress ratios. The threshold stress intensity factor range for this model is determined using

$$\Delta K_{th} = (1 - A\,R)\Delta K_o \tag{2.28}$$

To account for the reduction of the overload retardation effect, caused by compressive spike loads following tensile overload, the overload plastic zone size is modified as follows:

$$(Z_{ol})_{eff} = (1+\overline{R})Z_{ol} \tag{2.29}$$

where $\overline{R}$ is as defined in Eq. 2.26 and Z_{ol} is the plastic zone size corresponding to maximum stress intensity factor.

Whenever the Willenborg load interaction model is invoked, the effective stress ratio R_{eff} is computed and used in the above equations(2.25, 2.26 2.28 and 2.29). Otherwise the above equations are used for crack growth in the non interaction mode.

2.1.7.4 Strip Yield Model

This section describes the Strip Yield Model, one of the fatigue crack growth load interaction models in NASGRO 3.0. This model was developed by the European Space Agency (ESA) and the National Aerospace Laboratory (NLR) in the Netherlands in cooperation with the NASA Langley Research Center and the NASA Johnson Space Center. Reference [22] contains the details of the model and its implementation into NASGRO. Strip Yield is a mechanical model based on the assumption that a growing fatigue crack will propagate through the crack tip plastic region, and that this plastic deformation left in the wake of the crack will contribute to stress interaction effects such as stress-level dependence and crack growth rate acceleration and retardation. This section will first present an historical foundation for plasticity-induced fatigue crack closure and then give some insight into using this component of NASGRO 3.0.

Fatigue cracks can grow only when they are open, and historically cracks were assumed to open when the applied load increased from its minimum value, or passed from compression into tension in the case of reversed loading. The underlying assumption is that the crack tip plastic zone moves with and remains ahead of the tip as the crack grows. Extending linear elastic fracture mechanics principles to fatigue, the driving force for the crack growth rate was taken to be the stress-intensity factor range

$$\Delta K = K_{\max} - K_{\min} \tag{2.30}$$

Plots of experimental data of crack growth rate da/dN versus ΔK on log-log scales show a “sigmoidal” shape. The large central linear region suggests a power-law relationship, and the simplest such form was given by the Paris [23] equation

$$\frac{da}{dN} = C_1(\Delta K)^{n_1} \tag{2.31}$$

where C_1 and n_1 are material constants. Since this equation is restricted to modeling the linear portion of such sigmoidal plots, in time, extensions were proposed to model the threshold and instability regions, for example equation (2.1) which is described in detail in section 2.1.1.

In the mid-sixties, Elber [24] discovered that fatigue cracks can remain closed during the loading step until a load substantially higher than minimum load (or zero load in the case of reversed loading), and close early during the unloading step before reaching minimum load (or zero load in the case of reversed loading). This was attributed to the permanent deformation that was left on the crack flanks as the crack propagated through the crack tip plastic zone, a phenomenon termed *plasticity-induced fatigue crack closure*. For such cases, Elber proposed modifying the basic Paris growth law by using an *effective* stress-intensity factor range

$$\frac{da}{dN} = C_2\left(\Delta K_{eff}\right)^{n_2} \tag{2.32}$$

where C_2 and n_2 are material constants,

$$\Delta K_{eff} = K_{\max} - K_{open} \tag{2.33}$$

and K_{open} is the stress-intensity factor value at which the crack opens. In general the stress-intensity factor values at which the crack opens and closes are not identical, but the difference is small. For the case when K_{open} is less than $K_{\min}$, Eq. (2.33) reverts to Eq. (2.30)

The benefit of accounting for crack closure is that events such as spike overloads lead to considerable permanent deformation on the crack flanks and thus to an elevated K_{open} and a reduced (or retarded) crack growth rate.

The stress interaction occurs as follows. Overloads cause large plastic zones and previously virgin material yields; this is called primary plasticity. As the crack grows the large amount of permanent deformation will be on the crack flanks, causing a rise in the opening stress and thus a decrease in the crack growth rate. This lower rate is sustained if subsequent cycles have smaller maximum loads than the overload, and the new plastic zones will be wholly contained within the overload plastic zone. This is termed secondary plasticity. Eventually the crack grows through the primary plastic zone, the region of large plastic deformation is well back in the crack wake. At this point, the material returns to primary plasticity and the crack growth rate increases, perhaps to stabilize at its original pre-overload value.

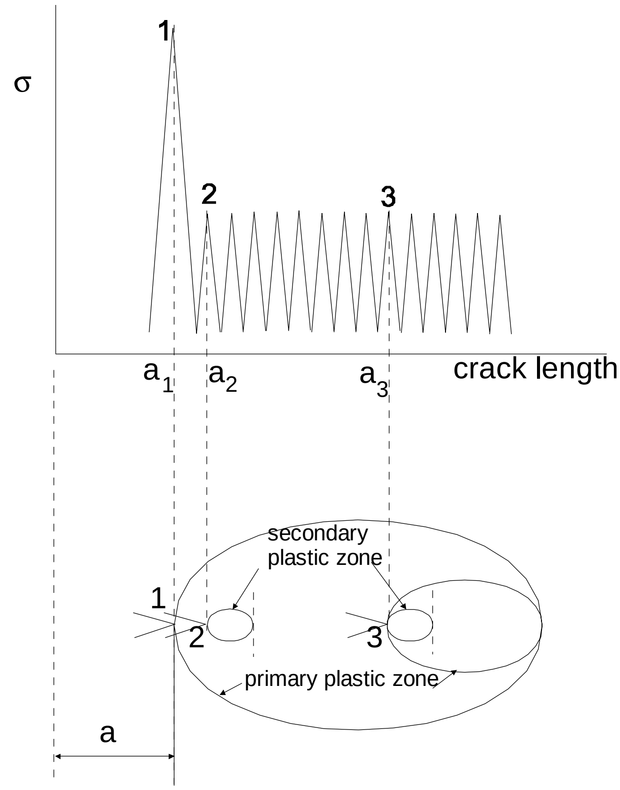


Figure 10 - Schematic of analytical crack-closure model under cyclic loading

It remains to discuss how to calculate K_{open}, the stress at which the crack opens and begins to grow. There exist empirical curve-fits in which the opening stress ratio $K_{open}/K_{\max}$ is cast as a function of applied stress ratio R, the maximum-stress-to-flow-stress ratio $S_{\max}/\sigma_o$, and constraint factor α. One such fit is used for the function f in the NASGRO equation (2.1). These fits are derived from experimental data or finite-element predictions for constant-amplitude fatigue tests. As such they can account for stress level effects in constant amplitude fatigue data by collapsing da/dN - ΔK curves for multiple R values, but they cannot account for stress-interaction effects such as growth rate retardation or acceleration after overloads or underloads.

The Strip Yield model in NASGRO 3.0 calculates a value for K_{open} by using a crack-opening model based on the Dugdale strip-yield model [25] but modified to leave plastically deformed material in the crack wake. In this strip-yield model it is assumed that all plastic deformation is contained within an infinitesimally thin strip located along the crack line in an infinite thin sheet. The material within the strip is represented by a series of finite-width rigid-perfectly plastic bar elements. These bar elements are either intact (in the plastic zone ahead of the crack tip, region 2 in figure 11) or broken (in the crack wake, region 3 in Figure 11). Elements in the plastic zone can carry tensile and compressive stresses, while the crack wake elements in contact can only carry compressive stresses. Outside this strip, in the elastic continuum(region 1 in Figure 11), the material is perfectly elastic.

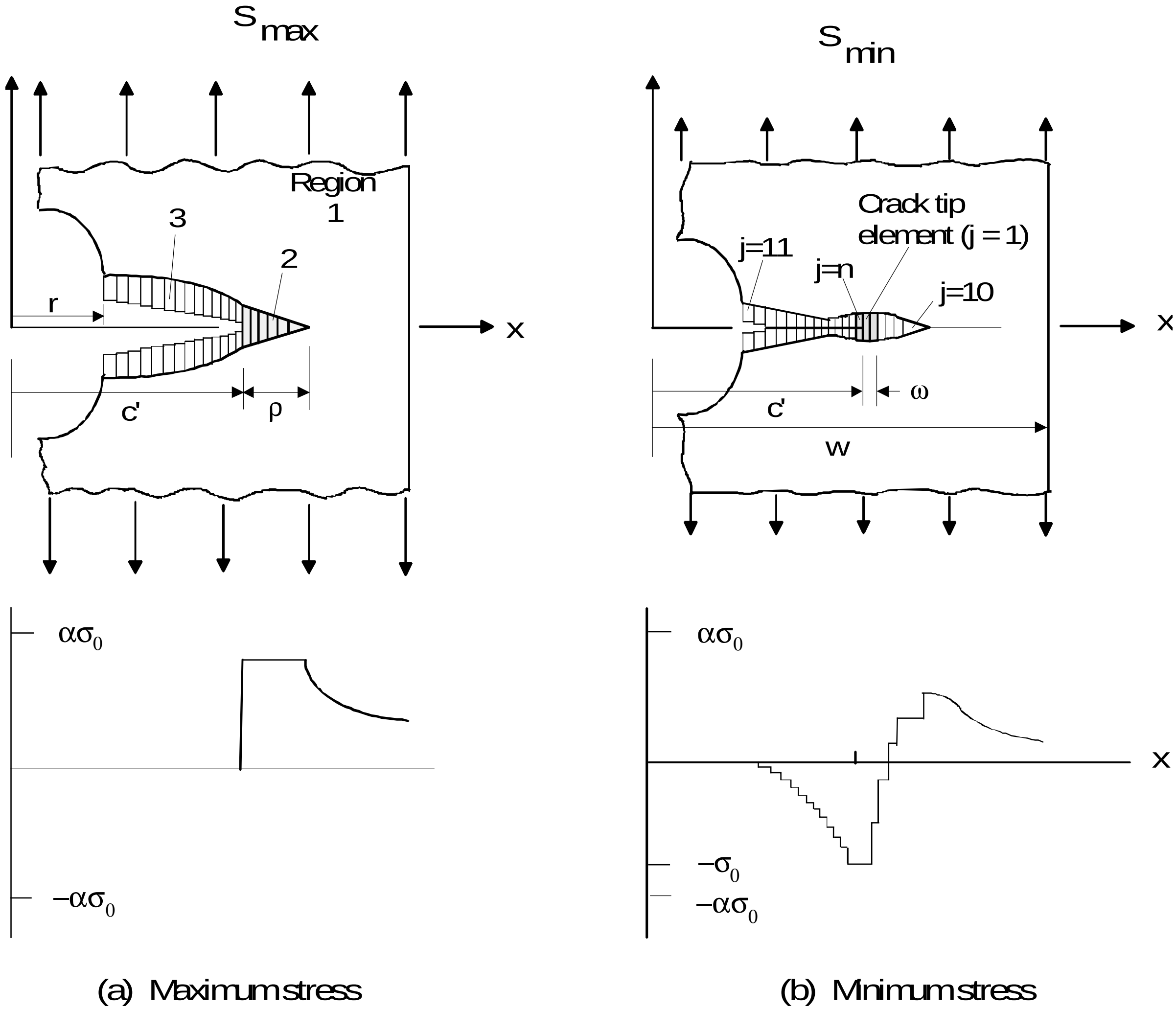


Figure 11 - Schematic of the Strip Yield Model

In simple terms, the strip-yield computations generally proceed as follows: a crack growth increment size is determined based on factors such as load step size and previous crack growth rate. Over this increment the opening stress is assumed to be held constant at the value from the previous increment. At maximum applied load the plastic deformation in elements ahead of the crack tip is calculated and the crack is grown this increment by releasing an appropriate number of elements. At minimum load the contact stress is calculated for elements now in contact due to the recent plastic deformation at maximum load. This in turn gives rise to a new opening stress and crack growth rate for the next increment, and the number of load cycles expended in the current increment.

The advantage of using a strip yield model is that the stress and deformation solution can be obtained by superposition of two elastic solutions: a crack in a plate subjected to remote uniform stress and a crack in a plate subjected to uniform stress acting along a portion of the crack surface, as shown in Figure 12 below.

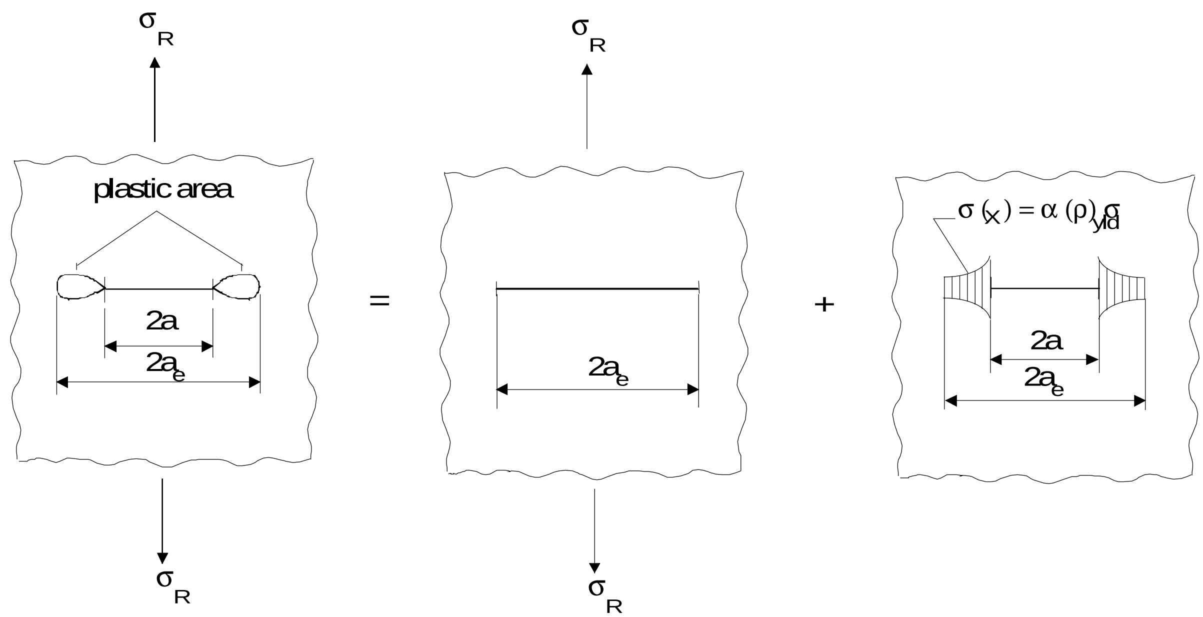


Figure 12 - Dugdale Model: Superposition of two elastic solutions

Dugdale's original strip-yield model was defined only for thin sheets, i.e., under plane stress conditions. To accommodate a more general state of stress in modern strip-yield models, the local yield stress is elevated by a constraint factor α, where α=1 for plane stress and α=3 for plane strain. In practice, though, it is difficult to find fully plane strain or plane stress conditions, and α ranges between 1.15 and 2.5. In compression the constraint factor is taken to be unity.

2.1.7.4.a Strip Yield Model - Constant constraint-loss option

NASGRO 3.0 contains two distinct implementations of the Strip Yield model. In one the constraint distribution ahead of the crack tip is spatially constant, while the other features a spatially parabolically decaying constraint; both feature a constraint-loss mechanism by which the current state of stress, ranging from plane strain to plane stress, can be accounted for.

The first model, used predominantly by NASA, FAA, their contractors, and airframe manufacturers, is the so-called constant constraint-loss option. In this option, the tensile constraint factor α is constant along the elements of the plastic zone, but its value depends on the state of stress, ranging from plane strain to plane stress. This constraint loss is based on the observation that cracks which start initially with a flat face eventually grow in a slant face mode, as shown in Figure 13 below.

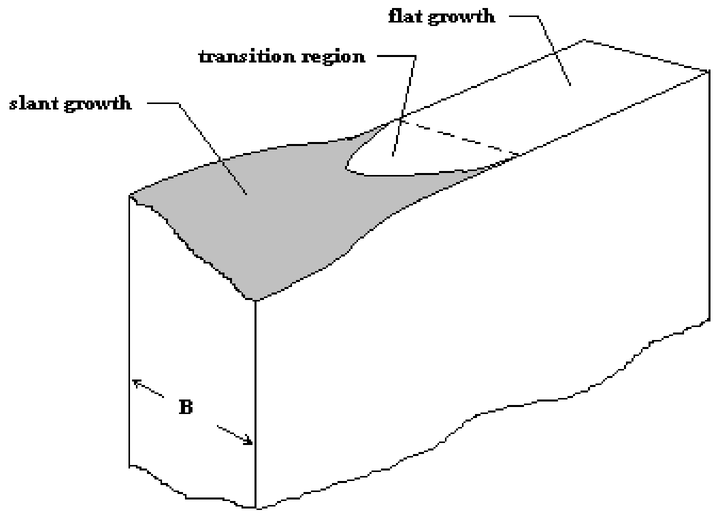


Figure 13 - Flat to slant crack growth transition

This transition is a manifestation of changing constraint: at low crack growth rates the flat crack growth indicates plane strain constraint, while at high rates the slant crack growth indicates plane stress constraint. In many materials the transition from flat to slant crack growth appears to end at the same crack growth rate and is independent of applied stress ratio. Because the crack closure concept can be used to collapse crack growth rate data to a single da/dN - ΔK_{eff} relation, the effective stress intensity factor can be used to control the transition from flat to slant growth, i.e. determine which constraint value is appropriate. Newman proposed that the transition occurs when the cyclic plastic zone size (calculated from ΔK_{eff}) reaches a percentage of the specimen thickness:

$$(\Delta K_{eff})_T = \mu \quad \sigma_0 \sqrt{B} \tag{2.34}$$

where μ is the proportionality coefficient, σ_0 is the flow stress (average of yield and ultimate), B is the specimen thickness, and $(\Delta K_{eff})_T$ is the effective stress intensity factor at transition. He found that a value of 0.5 for μ was suitable for a range of materials within a ± 20% scatter band for thin sheet; values tend to be lower for larger thickness values and higher for smaller thickness values. The constraint value does not change abruptly when the effective SIF crosses its transition value. Rather, there is a region about this transition value in which the constraint varies linearly from its plane strain value to its plane stress value. The extent of this transition region is not well-understood, however, it has been estimated conservatively at 1.5 decades of rate, as shown in Figure 14 below.

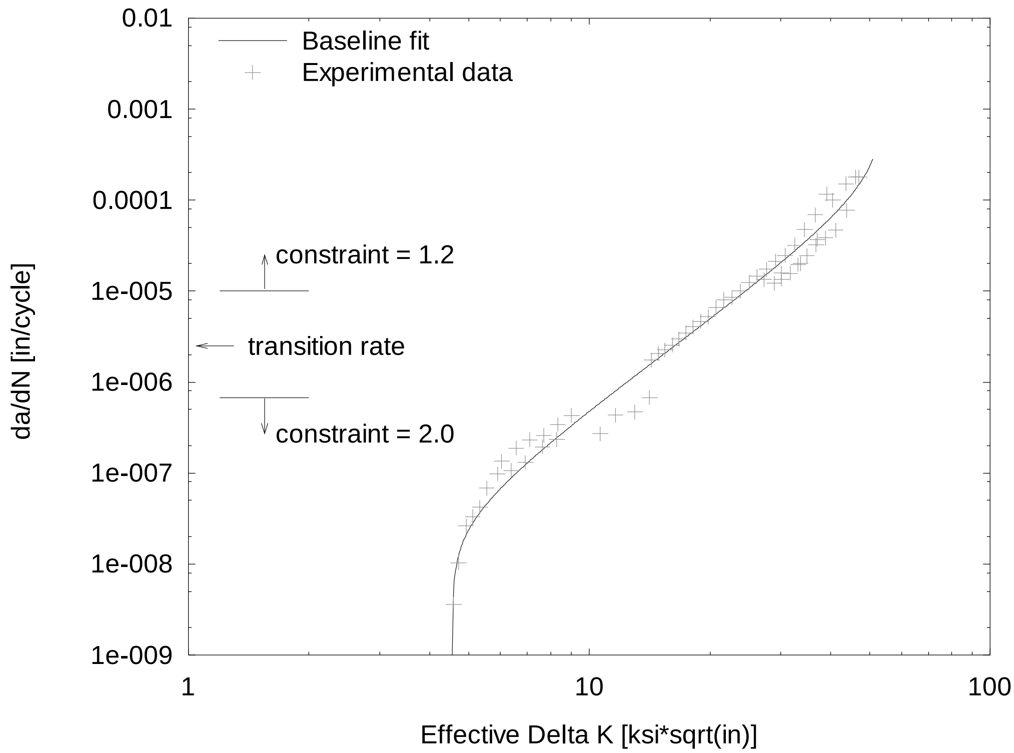


Figure 14 - Illustration of constraint transition region

Operationally the procedure is as follows. The Strip-Yield model initially calculates the transition value of the effective SIF from Eq 2.34 above, and uses it to determine the transition crack growth rate. Then it calculates the crack growth rate corresponding to the constraint value and opening stress for the current increment of crack growth. If this value of rate is greater than the upper limit of the transition band, then the constraint factor for the next crack growth increment is set to the appropriate plane stress value. If the rate is below the transition band, the constraint factor for the next crack growth increment is set to the appropriate plane strain value. This implementation is very similar to Newman's FASTRAN fatigue crack closure model.

This model is available only for the table look-up option of crack growth rate description, and tensile constraint factor values are chosen by material category. For example, plane strain values for aluminum alloys are generally taken to be 1.9, for steel alloys it is 2.1, and for titanium alloys it is 2.5. Plane stress values are assumed to be 1.2 for all materials, while compressive constraint factors both in the plastic zone and in the crack wake are assumed to be unity for all materials. Future research may refine these assumptions.

The NASGRO materials database contains such look-up tables for a number of materials, and they can be augmented by user-supplied data. To set the value of the tensile constraint factor, the user would run a number of constant-amplitude crack-growth rate predictions with this model and fix the value of α to give the best fit.

2.1.7.4.b Strip Yield Model - Variable constraint-loss option

The second option, used predominantly by the European Space Agency (ESA) and their contractors, is the so-called variable constraint-loss option. In this option, the tensile constraint factor α varies along the elements of the plastic zone according to a parabolic expression derived from finite-element analyses. The constraint decays spatially from its value at the crack tip (α_{tip}) to a plane stress value of 1.15 at the forward end of the plastic zone. Constraint loss is also built into this option, but in contrast to the model above, the plane strain or plane stress tensile value of α_{tip} is calculated from the ratio of plastic zone size to specimen thickness. This relates the constraint loss to K_{max}, whereas the model described in the previous section relates it to $\Delta K_{eff.}$. Furthermore, as seen in Figure 15, the compressive constraint factors in the plastic zone and in the crack wake are spatially constant, and their values are given by $\alpha_{tip}/\alpha_{new}$ and $1/\alpha_{new}$, respectively, where the material parameter α_{new} characterizes the ratio of tensile tip constraint to compressive constraint.

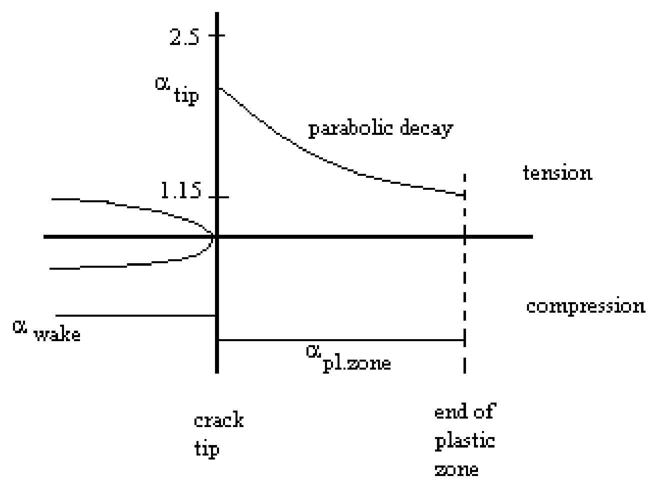


Figure 15 - Constraint factors in tension and compression

Operationally the procedure is as follows. The Strip-Yield model calculates the plastic zone size to thickness ratio for the current increment of crack growth; for small values of this ratio (less than 0.1), α_{tip} is set to its plane strain value of 2.35. For large values (greater than 1.5), α_{tip} is set to its plane stress value of 1.15. Intermediate values are determined from piece-wise linear interpolation. The compressive constraint factors are then calculated from α_{tip} and α_{new}.

This model is available for both the equation and table look-up options of crack growth rate description. The equation description is available for the traditional NASGRO equation

(equation 2.1) as well as an enhanced version of this equation, developed by NLR (Dutch National Aerospace Laboratory), in which growth due to spike overloads is treated via a tearing analogy. The NASGRO materials database contains equation parameters for a number of materials provided by ESA and NLR, and they can be augmented by user-supplied data. To set the value of the tensile-to-compressive constraint ratio α_{new}, two or three different α_{new} values are chosen for Strip-Yield comparisons to selected constant-amplitude and spike load sequences. Engineering judgment and interpolation then gives the best fit for α_{new}. This final value is then used to set the remaining equation constants if they are needed.

2.1.7.4.c Strip Yield Model - Usage

Use of either implementation of the Strip Yield module is virtually transparent to the user. After selecting the strip-yield model from the menu of interaction models and choosing the appropriate constraint model option ("spatially constant constraint variation" or "spatially variable constraint variation"), the user is presented with a choice of computation speed modes: full or fast. In the "full" mode, the strip-yield model is used to calculate the opening stress throughout the entire analysis. The "fast" mode makes use of the fact that for some load spectra the opening stress may stabilize and oscillate about some average value at some point in the load spectrum; the model is used only until a cycle-averaged value of opening stress is stabilized (within a user-specified tolerance, generally about 1%) within a load schedule. Subsequent load schedules then bypass the strip-yield calculations and use this stabilized value directly in crack growth equation (2.1). At this point the Strip Yield model mimics the non-interaction model, albeit with a computationally derived closure level.

In the Strip Yield model, the crack growth increments are chosen on the basis of load step size and crack growth rate from the previous increment. The first cycle of any load step is always analyzed separately and a minimum number of five increments to span the rest of the load step is guaranteed. For the case of aircraft load spectra or variable-amplitude load spectra, where load steps tend to contain very few cycles or frequently only one cycle, this basis of choosing the increment size would quickly force a cycle-by-cycle analysis, an accurate analysis but at a great expense of computer time. If the user wishes to analyze aircraft or variable-amplitude spectra but avoid cycle-by-cycle analyses, then there are two additional parameters to specify. These parameters govern an algorithm in which only the most severe load cycles are considered for the computations, where "severe" is effectively determined by the user input for these parameters. The first parameter is the difference in applied stress, $\Delta S_{\max}$, from one significant maximum load (or minimum load) to the next that is required to trigger a calculation. This ensures that significant overloads or underloads are captured by the algorithm and that the increment size is set appropriately. The second parameter is the maximum number of cycles $\Delta N_{\max}$ that can be expended in an increment before another calculation is triggered. This ensures that spectra containing long constant-amplitude steps use enough increments to give good results.

The user then chooses the initial defect type: closed crack, such as in the case of fatigue cracks, or open crack, such as in the case of sawcuts.

Diagnostic messages are printed at the end of the analysis giving information on constraint type and value, computation speed mode, initial defect type, and cycle-averaged opening stress values (these are used in the “fast” computation mode but may be of general statistical interest for the “full” mode as well.)

2.1.7.5 Constant Closure Model

This crack growth model was originally developed at Northrop and was used on some of their classified programs. It is a simplified closure model based on the observation that for some load spectra the closure stress does not deviate substantially from a certain stabilized value. This stabilized value is determined by assuming that the spectrum has a “controlling overload” and a “controlling underload” that occur often enough to keep the residual stresses in the crack wake constant, and thus the closure level constant.

The closure level can be determined in three ways. Firstly, it can be calculated from a function that is an empirical fit of test data. The fit usually has three segments of which two are straight line segments joined in the middle by a relation using the Walker crack growth law. The user provides the fitting constants. Secondly, the closure level can be entered directly. Thirdly, Newman’s closure function may also be used to calculate the closure level.

This model is available for the NASGRO equation or a 1-D or a 2-D table look-up.

2.1.7.6 Notes on using the Load Interaction Models

This section is meant to indicate to users, based on experience here at JSC, when to use the load interaction models. In general, caution should be exercised when these models are used because they can be unconservative compared to the non interaction model. This is so because the dominant effect modelled is retardation, even if accelerated growth is predicted in a few cases. Before applying these models for life predictions, it is recommended that the user gain sufficient experience and fine tune the various model parameters based on comparisons with test data for the kind of spectra relevant to the usage. Certain special structural features might cause the predictions from NASGRO 3.0 to be in disagreement with test data. An example of this would be a crack growing out of a hole which has been subjected to considerable yielding. Such a crack would exhibit a lot of retardation in experiments but none of the NASGRO models accounts for the presence of such yielding.

In the case of the Generalized Willenborg model, the only parameter that can be varied is the overload shutoff ratio R_{so}. Table 1 shows the effect of this parameter on life prediction for Zhang’s spectra. Table 2 shows the effect of ϕ_0 on life prediction using the modified generalized Willenborg model. These are the spectra with single or multiple overloads applied after several constant amplitude cycles. A trend of decreasing values of predicted life with increasing values of R_{so} was found. For the Modified Generalized Willenborg model, because ϕ_0 appears in the denominator, the opposite trend, i.e., increased life with increasing values of ϕ_0 was found. Table 3 summarizes life predictions using various growth models for Zhang's spectra. Table 4 shows life predictions using the various models for ASTM round robin spectra that are typical aircraft spectra.

Generalized Willenborg Predictions (N_{test} / N_{pred}) for Zhang's Spectra
(Material :7475-T7351)

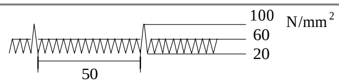
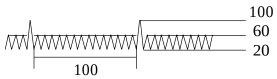
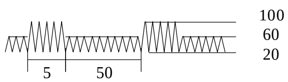
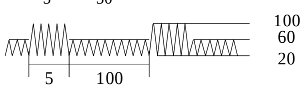
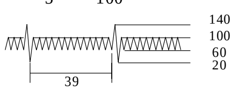
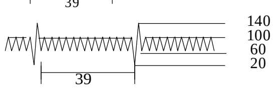
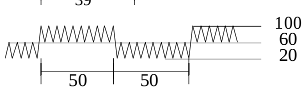
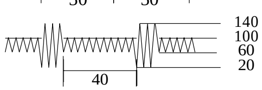
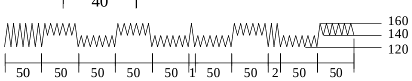
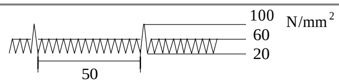
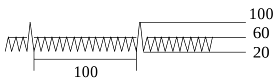
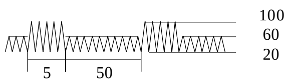
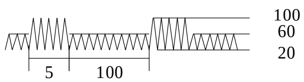
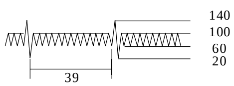
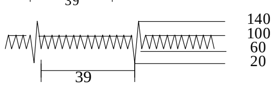
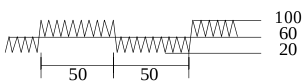
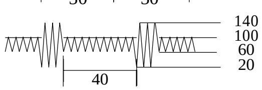
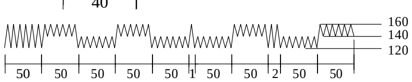
Zhang's Test	Loading History	Test Life (Cycles)	Non-Interaction	Generalized Willenborg Model (R_{SO} =2.3)	Generalized Willenborg Model (R_{SO} =3.0)	Generalized Willenborg Model (R_{SO} = 4.0)
1		**474,240**	**1.33**	**0.77**	**0.94**	**1.13**
2		**637,730**	**1.69**	**0.93**	**1.17**	**1.42**
3		**251,210**	**0.98**	**0.71**	**0.79**	**0.89**
4		**409,620**	**1.33**	**0.86**	**1.00**	**1.16**
5		**179,320**	**0.91**	**0.78**	**0.83**	**0.86**
6		**251,050**	**1.38**	**1.25**	**1.29**	**1.34**
7		**253,840**	**0.83**	**0.68**	**0.73**	**0.78**
8		**149,890**	**1.20**	**1.13**	**1.15**	**1.18**
10		**57,680**	**1.67**	**1.67**	**1.67**	**1.67**

Table 1 Effect of Rso on life prediction for Generalized Willenborg

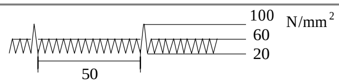
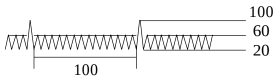
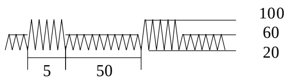
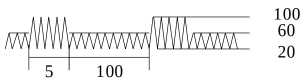
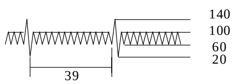
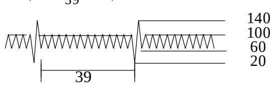
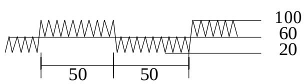
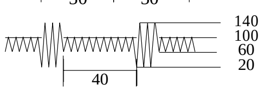
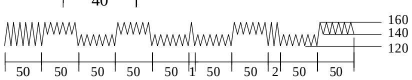
Modified Generalized Willenborg Predictions (N_{test} / N_{pred}) for Zhang's Spectra
(Material :7475-T7351)

Zhang's Test	Loading History	Test Life (Cycles)	Non-Interaction	Modified Generalized Willenborg Retardation Model (Φ_0* = 0.2)	(Φ_0 = 0.4)	(Φ_0 = 0.6)
1		474,240	1.33	0.98	0.55	0.26
2		637,730	1.69	1.22	0.64	0.24
3		251,210	0.98	0.81	0.60	0.46
4		409,620	1.33	1.03	0.68	0.44
5		179,320	0.91	0.86	0.80	0.72
6		251,050	1.38	1.32	1.27	1.18
7		253,840	0.83	0.74	0.62	0.54
8		149,890	1.20	1.17	1.14	1.09
10		57,680	1.67	1.67	1.67	1.67

***Φ_0 = Retardation parameter**

Table 2 -Effect of ϕ_0 on Life Prediction for Modified Generalized Willenborg

NASGRO Predictions (N_{test} / N_{pred}) for Zhang's Spectra
(Material :7475-T7351 Al, Case TC01)

Zhang's Test	Loading History	Test Life (Cycles)	Non-Interaction	Interaction Models: Generalized Willenborg *(modified)* (Φ_0 = 0.2)	Generalized Willenborg *(original)* (R_{SO} *=3.0)	Constant Closure(.5)	Strip Yield
1		**474,240**	**1.33**	**0.98**	**0.94**	**0.97**	**0.49**
2		**637,730**	**1.69**	**1.22**	**1.17**	**1.25**	**0.53**
3		**251,210**	**0.98**	**0.81**	**0.79**	**0.66**	**0.54**
4		**409,620**	**1.33**	**1.03**	**1.00**	**0.93**	**0.58**
5		**179,320**	**0.91**	**0.86**	**0.83**	**1.84**	**0.84**
6		**251,050**	**1.38**	**1.32**	**1.29**	**2.58**	**1.16**
7		**253,840**	**0.83**	**0.74**	**0.73**	**1.47**	**0.65**
8		**149,890**	**1.20**	**1.17**	**1.15**	**1.68**	**1.09**
10		**57,680**	**1.67**	**1.67**	**1.67**		**1.38**

Φ_0 = retardation parameter for R_U(underload-to-overload ratio)=0 , *R_{SO} = shut-off overload ratio*

Table 3 Life predictions for Zhang's Spectra

(N_{test} / N_{pred}) for ASTM Round Robin Spectra
(Material: 2219-T851, L-T AL)

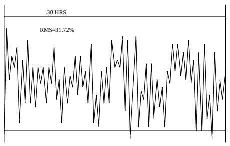
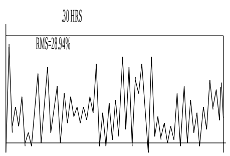
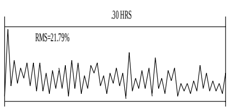
Spec. No.	Loading Spectrum	Stress ksi	Test Cycles	Non Int. NASGRO	Non Int. Walker	Willenborg Walker Rso= 3.0	Willenborg Generalized Rso= 3.0	Willenborg Modified Phi0=0.4	Strip Yield	Constant Closure Cfspec=0.5
M-81	Air-Air	20	115700	0.78	0.71	0.49	0.54	0.54	0.62	0.70
M-82		30	58585	1.27	0.92	0.62	0.87	0.92	1.27	1.15
M-83		40	18612	1.28	0.79	0.54	0.88	0.94	1.71	1.16
M-84	Air-Grn	20	268908	0.86	0.77	0.51	0.57	0.55	0.52	0.60
M-85		30	95642	1.25	0.98	0.63	0.81	0.82	0.98	0.87
M-86		40	36397	1.54	1.04	0.67	0.99	1.04	1.64	1.06
M-88	Ins-Nav	30	380443	1.36	1.19	0.56	0.66	0.59	0.51	1.23
M-89		40	164738	1.73	1.37	0.63	0.80	0.79	0.92	1.56
M-90	Fighter	20	218151	0.91	0.78	0.50	0.58	0.57	0.60	0.76
M-91	Comp.	30	65627	1.23	0.90	0.57	0.76	0.80	1.13	1.03
M-92		40	22187	1.30	0.83	0.52	0.82	0.87	1.59	1.09

Table 4 Life predictions for ASTM round robin spectra

One of the choices the user makes while using Strip Yield is the "full" mode which has better accuracy at the expense of computation time, or the "fast" mode which provides faster results at the expense of accuracy. It is advised that if this "fast" option is chosen, several analyses be done at different tolerances to gauge their effect and to ensure convergence. It may be noted that the "fast" option is useful in the case of load spectra where the opening stress stabilizes. This usually occurs in spectra where the mean stress stays relatively constant for many load steps. The fast option is not suitable for spectra that will produce opening stresses which fluctuate greatly rather than stabilize.

Additional user input to Strip Yield are the two parameters used to prevent Strip Yield from analyzing aircraft spectra by default on a cycle-by-cycle basis. As mentioned in the previous section, the first parameter $\Delta S_{\max}$ is the change in applied stress from one significant load to the next before triggering a K_{open} calculation. The second parameter $\Delta N_{\max}$ is the maximum number of cycles expended between K_{open} calculations. Larger values chosen for both parameters mean a larger crack increment size and fewer K_{open} calculations, possibly resulting in lower accuracy. However, the effect on speed of computations is of opposite nature, i.e., larger values result in faster execution. For example, for single-spike overloads $\Delta N_{\max}$ would be chosen small enough to ensure that each spike is captured in a separate K_{open} calculation. An analogy to $\Delta S_{\max}$ is the stress level clipping used in cycle counting of sequential loading. For example, for random loading $\Delta S_{\max}$ would be chosen small enough to ensure that large deviations in loading trigger separate K_{open} calculations. These two parameters are highly load spectrum dependent, and only experience in using the program will allow one to make reasonable "educated" assessments.

2.2 Details of Crack Growth Analysis

When the crack growth module is selected from the top level menu, the GUI for the NASFLA module is activated. The following types of analyses are possible:

1) Direct Life Prediction Analysis
(Given initial flaw size, find Life and final flaw size)

2) Indirect Life Prediction Analysis - Type 1
(Given Life, find the scale factor multiplier)

3) Indirect Life Prediction Analysis - Type 2
(Given Life, find initial flaw size)

If option 1 is selected, you will be prompted for the crack case, the initial flaw size, the material properties, and the load schedule. The program will then apply the load schedule to the crack case until the schedule has been repeated the desired number of times or until a failure condition is met, whichever occurs first. At that time, both life and final flaw size will be reported. If option 2 is selected, you will be prompted for the crack case, the material properties, life, and the load schedule. The program will then iteratively calculate the scale factor multiplier needed to cause failure by the time the specified life is reached. If option 3 is selected, you will be prompted for the crack case, the material properties, life, and the load schedule. The program will then iteratively calculate the initial flaw size needed to cause failure by the time the specified life is reached.

2.2.1 Choosing the Crack Geometry

The user selects a suitable crack geometry from among the many configurations built into NASGRO. Version 3.0 features several improvements to the stress intensity factor solutions that existed in the previous releases. These include the use of finite element analyses to verify earlier approximate solutions and extension of the valid range of dimensions for several crack cases. In addition, several surface-crack cases have been extended to include general loading. Up to four nonlinear stress distributions may now be combined and applied to the SC02, SC04, and SC06 crack cases. An example of how to apply a nonlinear stress distribution to crack case SC04 may be found in example 2. In order to provide a consistent approach, S_0 is reserved for tension and compression, S_1 and S_2 are bending stresses in through-the-thickness(out of plane) and width (in plane) directions, S_3 is the bearing stress, and S_4 is reserved for the second tension/compression stress for cases that have biaxial loading.

The stress intensity factor solutions that have been incorporated into NASGRO 3.0 are listed in Table 5a, and the geometries for the crack cases are shown in Figures 16 through 34 (beginning on page 39). Through crack (TC) geometries are shown in Figures 16 through 20 and the embedded crack (EC) case is shown in Figure 21. The corner crack (CC) geometries are depicted in Figures 22, 23 and 24, the surface-crack (SC) cases are presented in Figures 25 through 30, and ASTM standard specimen (SS) geometries are shown in Figures 31 through 34. Three data table (DT) cases and one polynomial solution (PS01) have also been included. The menu for selecting the crack geometry is as follows: first, the general class such as through cracks, corner cracks, surface cracks etc., is chosen and then the particular geometry within that class. After selecting the appropriate crack case, the figure is displayed in a graphical window and the user will be prompted for dimensional information such as width, thickness, diameter, etc. in an adjacent window. Additional information regarding the stress intensity factor solutions may be found in Section 4 and Appendices C and D.

2.2.2 Transition of Crack Geometry

Crack growth analysis is usually conducted on part-through cracks, such as surface or corner crack in plate. As the crack grows, the depth of the crack may exceed the thickness before the crack becomes unstable. In such instances, growth is continued using the corresponding through crack and then the crack will grow some more before becoming critical. Table 5b shows the transition relation between crack cases.

Table 5a – Description of Crack Cases

Through Cracks:	
TC01:	Through crack at center of plate
TC02:	Through crack at edge of plate
TC03:	Through crack from an offset hole in a plate
TC04:	Through crack from hole in a lug
TC05:	Through crack from hole in a plate with a row of holes
TC06:	Through crack in a sphere
TC07:	Through crack in a cylinder (longitudinal direction)
TC08:	Through crack in a cylinder (circumferential direction)
TC09:	Through crack from hole in a plate under combined loading
TC10:	Through crack from hole in a cylinder (circumferential direction)
Embedded Cracks:	
EC01:	Embedded crack in a plate
Corner Cracks:	
CC01:	Corner crack in a rectangular plate
CC02:	Corner crack from hole in a plate
CC03	Corner crack from hole in a lug
CC04	Corner crack from hole in a plate (one or two cracks)
Surface Cracks:	
SC01:	Surface crack in a rectangular plate – tension and/or bending
SC02:	Surface crack in a rectangular plate – nonlinear stress
SC03:	Surface crack in a spherical pressure vessel
SC04:	Longitudinal surface crack in a hollow cylinder – nonlinear stress
SC05:	Thumbnail crack in a hollow cylinder
SC06:	Circumferential crack in a hollow cylinder – nonlinear stress
SC07:	Thumbnail crack in a solid cylinder
SC08:	Thumbnail crack in a threaded, solid cylinder
SC09:	Circumferential crack at thread root in a cylinder
SC10:	Circumferential crack in a threaded pipe – nonlinear stress
SC11	Surface crack from hole in a plate (one or two cracks)
SC12	Surface crack from hole in a lug (one or two cracks)
SC13	Surface crack in bolt head fillet - Shear bolt
SC14	Surface crack in bolt head fillet - Tension bolt
Standard Specimens:	
SS01:	Center-cracked tension specimen M(T)
SS02:	Compact tension specimen C(T)
SS03:	Disc-shaped compact tension specimen DC(T)
SS04:	Arc-shaped tension specimen A(T)
SS05:	Three-point bend specimen SE(B)
SS06:	Edge cracked tension specimen SE(T) – constrained ends
SS07:	Notched round bar specimen R-bar(T) – circumferential crack
SS08:	Notched plate with a surface crack
SS09:	Notched plate with a corner crack
SS10:	Notched plate with a through crack
Data Tables:	
DT01:	One-dimensional data table for through cracks
DT02:	Two-dimensional data table for through cracks
DT03:	Two-dimensional data table for part-through cracks
Polynomial Series:	
PS01:	$F_0 = C_0 + C_1 u + C_2 u^2 + \ldots + C_5 u^5$ where $u = \left(a/D \right)^m$

Table 5b – Transition Relationship between Crack Cases

From	To	Condition/Comment
CC01	TC02	
CC02	TC03	S_1 should be zero
CC03	TC04	
CC04	TC03	S_1 should be zero, no. of cracks=1
SC01, SC02	TC01	For SC02, uses equivalent stresses
SC03	TC06	S_1 should be zero
SC04	TC07	Uses equivalent stresses
SC05	TC08	
SC11	TC03	Occurs only if number of cracks=1
SC12	TC04	Occurs only if number of cracks=1
TC03	TC02	Assumes C= B +D/2
SS08	SS10	
SS09	SS10	

2.2.3 Entering the Initial Flaw Size

In order to simplify the process of performing a crack growth analysis, input of the initial crack size has been automated for NASA space flight hardware. The automated crack size input is based on a "standard" nondestructive evaluation (NDE) crack size which has a high probability of detection when inspections are performed in accordance with the proper specifications. Standard NDE crack sizes for the different crack models are listed in Table 6 (English units) and Table 7 (SI units). For a surface crack in a solid circular section, case SC07, the crack depth, *a*, is a function of *c* and *D* and can be calculated from:

$$a = r\left(1 + \tan\theta - \sec\theta\right), \quad \theta = \frac{c}{r} = \frac{2c}{D} \tag{2.35}$$

where D=2 r is the major diameter. Alternately, the surface crack length, c, may be calculated from the following expression:

$$c = r \tan^{-1} \frac{a\left(2r - a\right)}{2r\left(r - a\right)} \quad . \tag{2.36}$$

The initial flaw sizes listed in Tables 6 and 7 are acceptable for payloads that require fracture control approval by the NASA Lyndon B. Johnson Space Center.

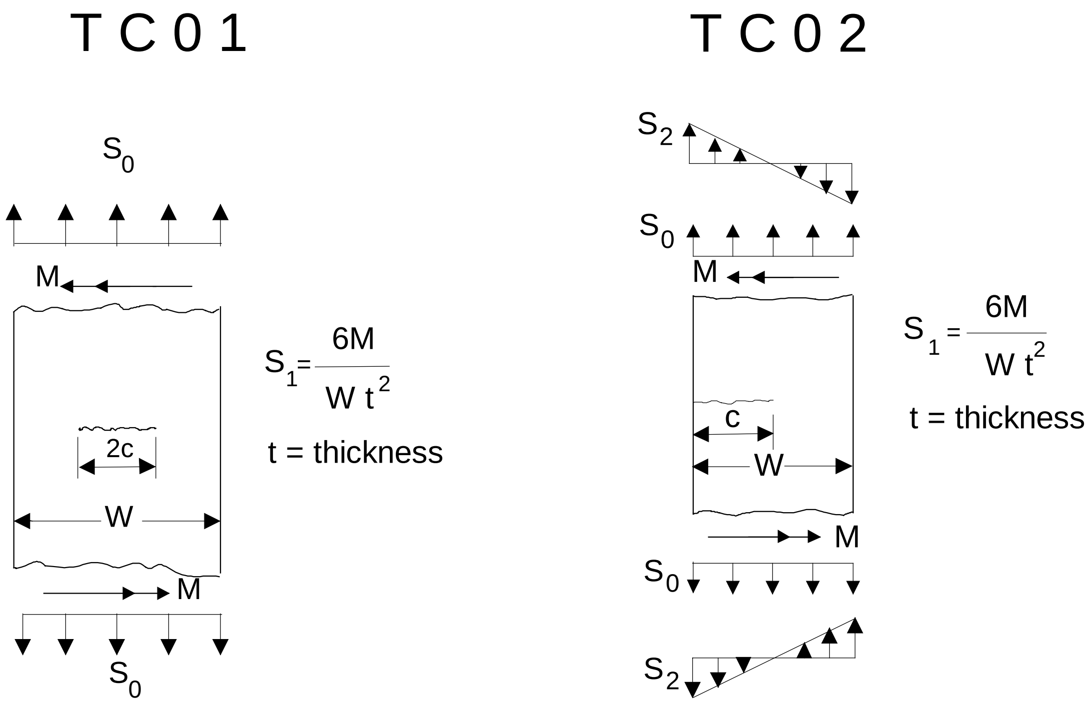


Figure 16 – Through crack cases 1 and 2

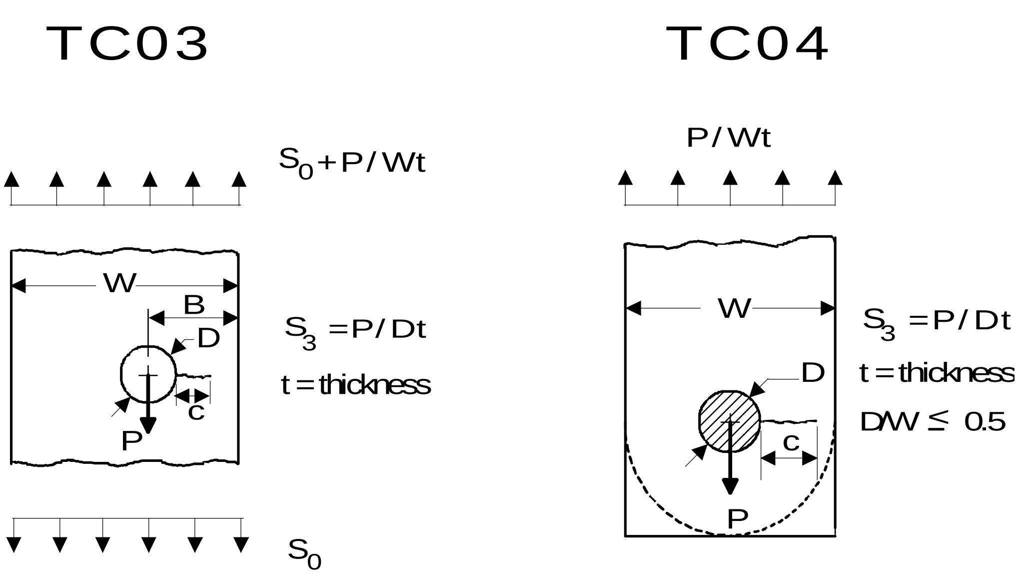
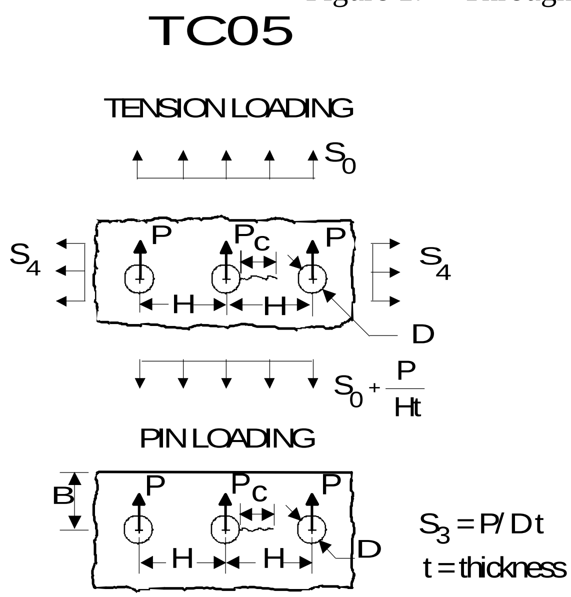


Figure 17 – Through crack cases 3 and 4



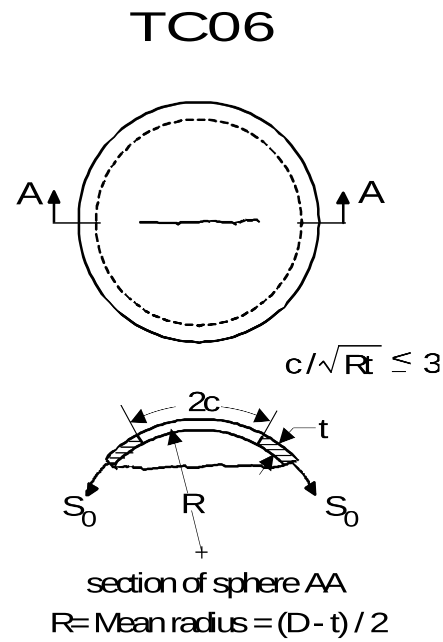
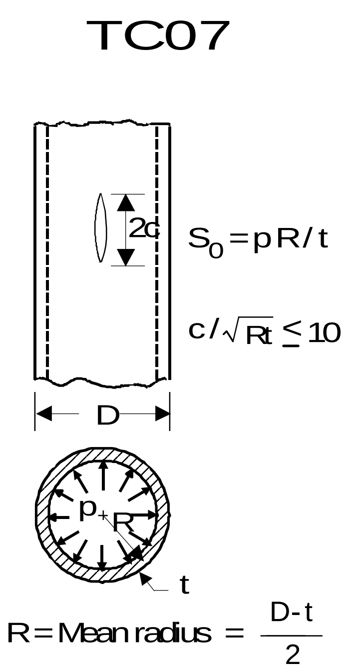


Figure 18 – Through crack cases 5 and 6



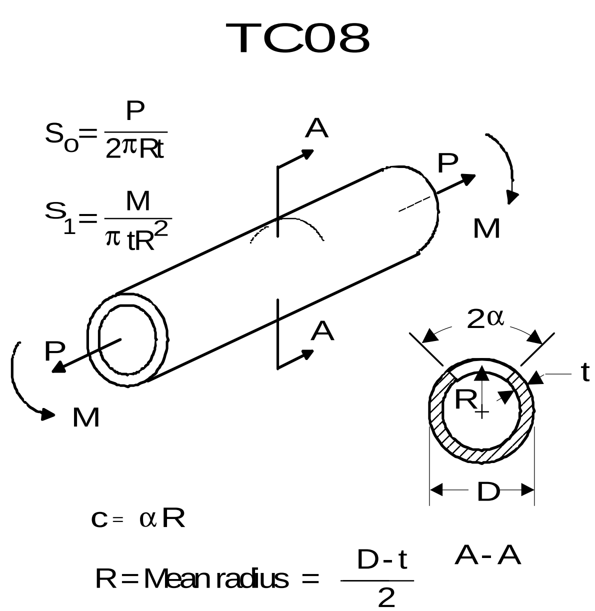


Figure 19 – Through crack cases 7 and 8

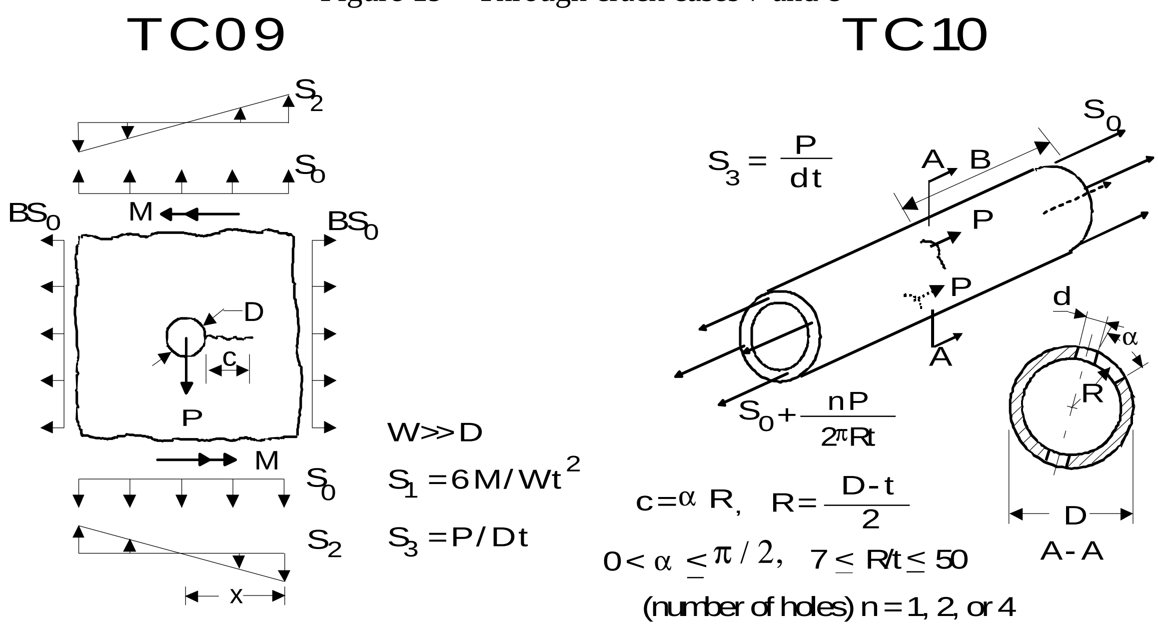


Figure 20 – Through crack cases 9 and 10

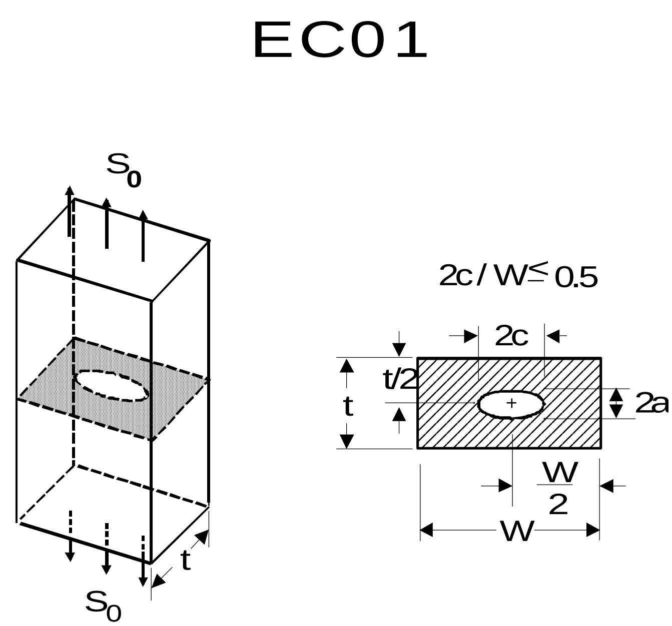


Figure 21 – Embedded crack case 1

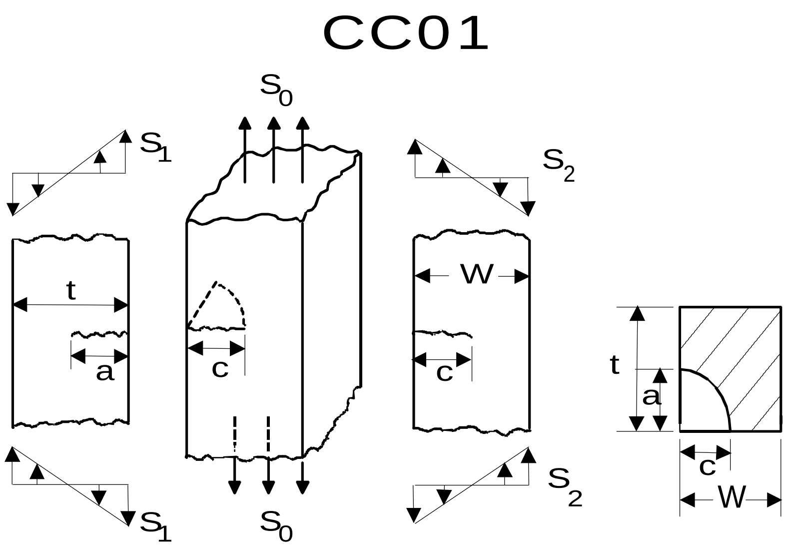


Figure 22 – Corner crack case 1

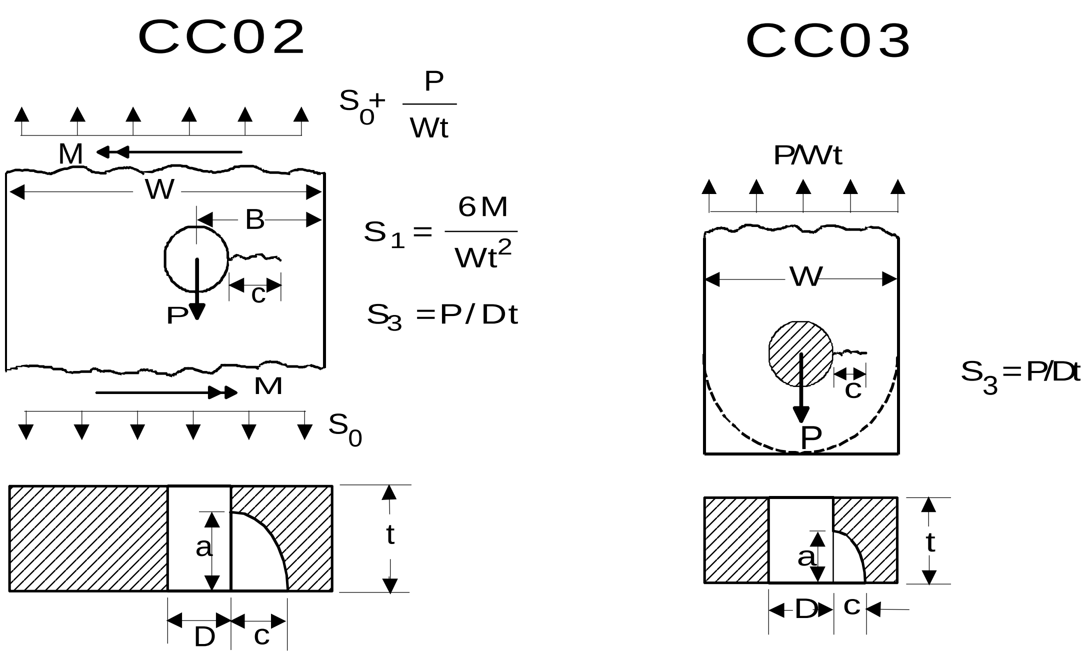


Figure 23 – Corner crack cases 2 and 3

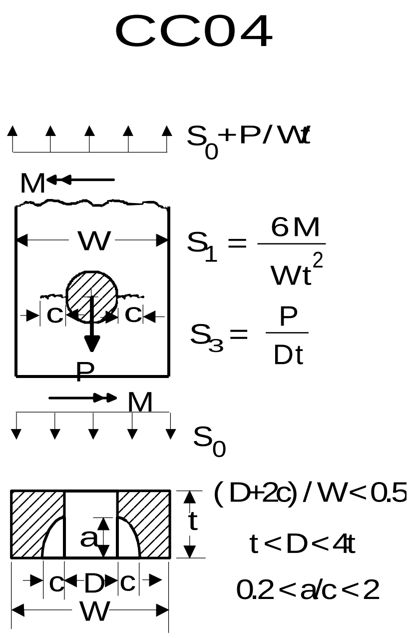
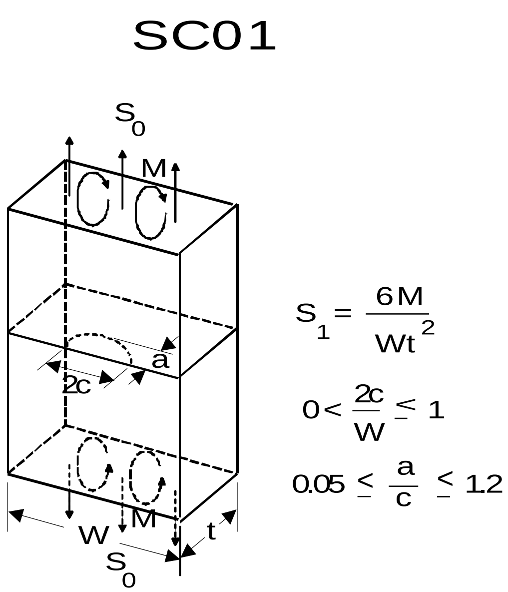


Figure 24 – Corner crack case 4



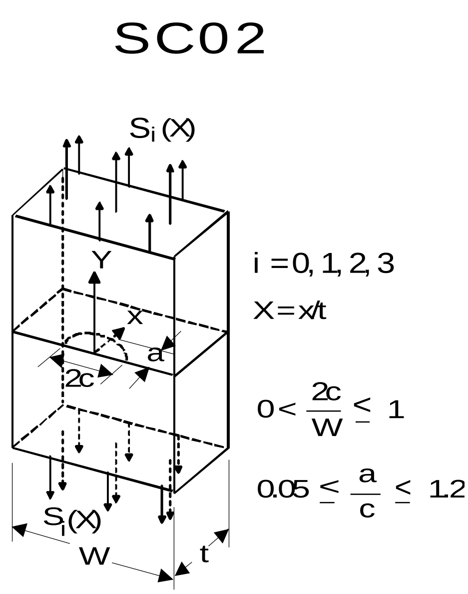
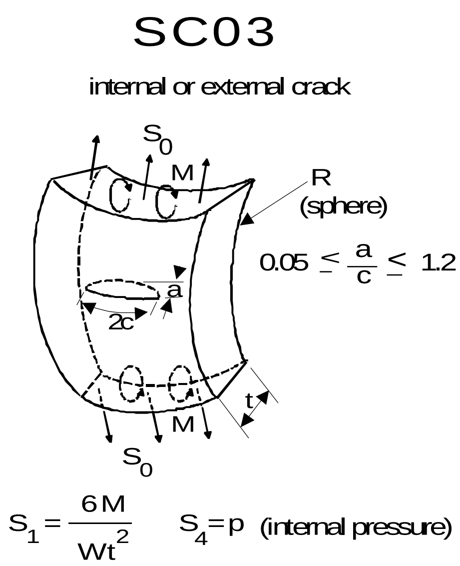


Figure 25 – Surface crack cases 1 and 2



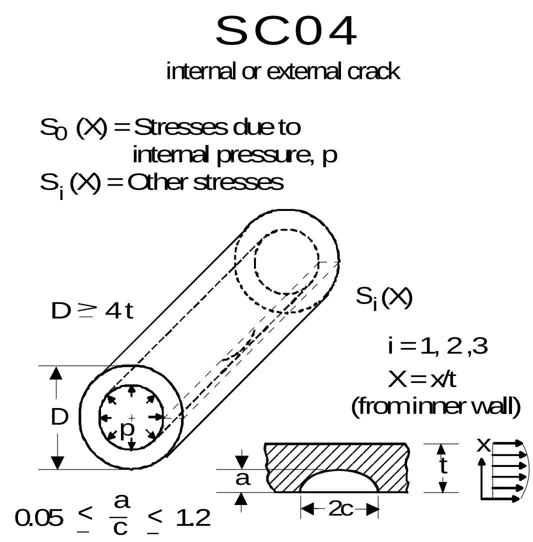
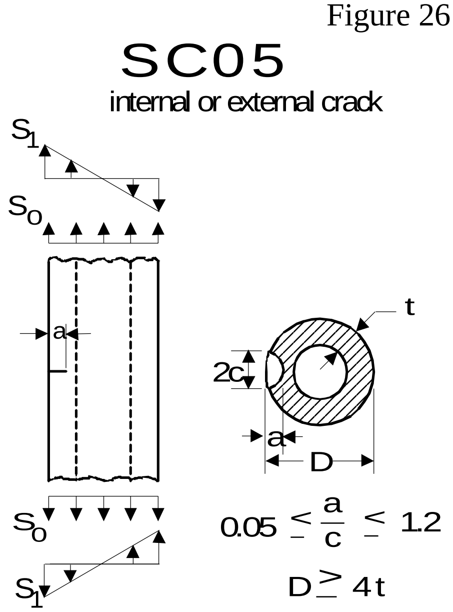


Figure 26 – Surface crack cases 3 and 4



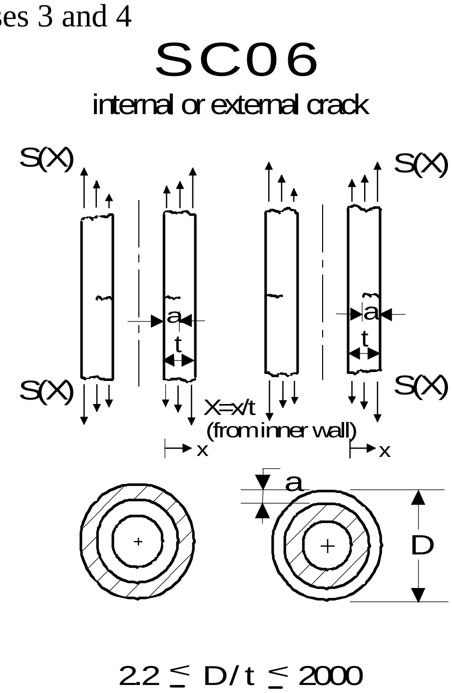


Figure 27 – Surface crack cases 5 and 6

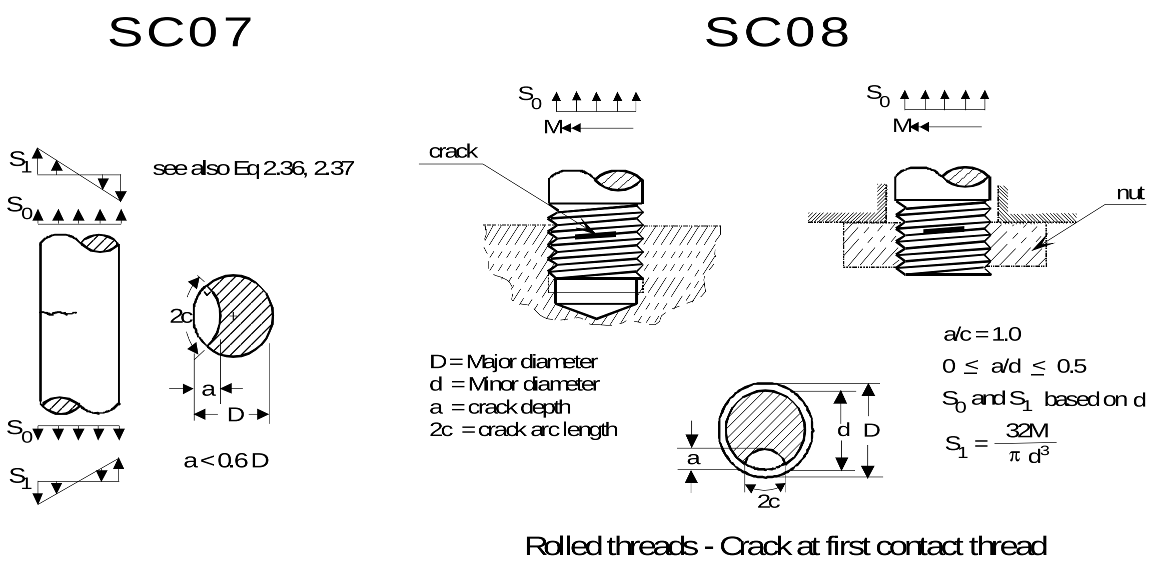


Figure 28 – Surface crack cases 7 and 8

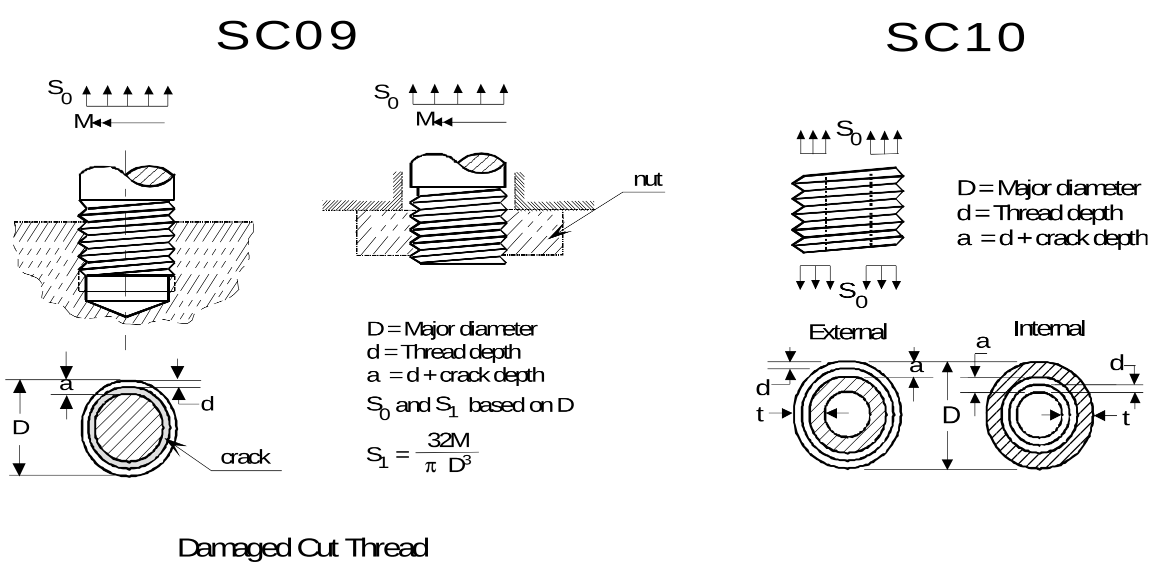


Figure 29 – Surface crack cases 9 and 10

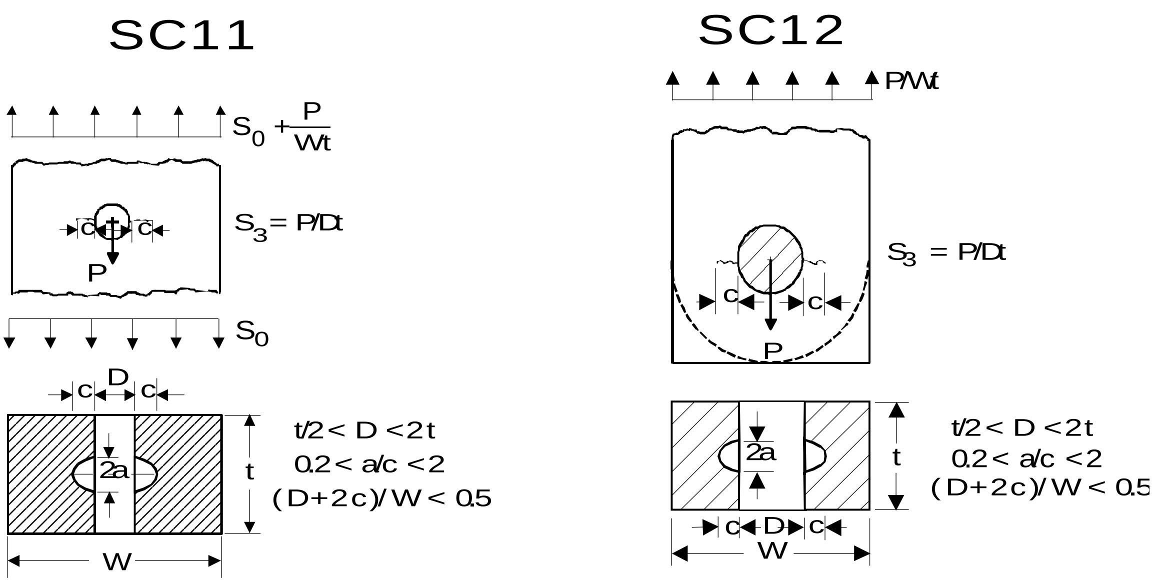
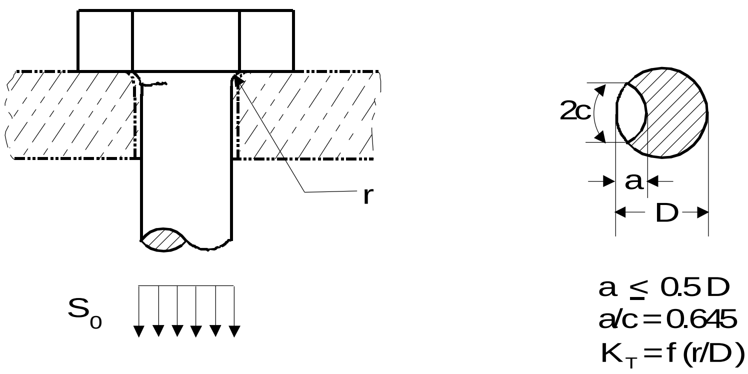


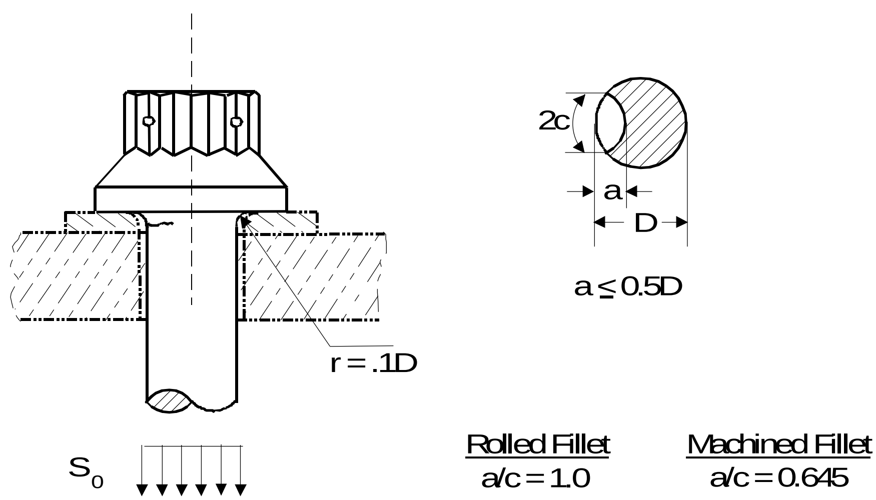
Fig. 30 - Surface Crack Cases 11 and 12

SC13



Shear or Machine Bolt - Machined Fillet

SC14



Tension Bolt - Crack in Bolt Head Fillet

Figure 31 – Surface crack cases 13 and 14

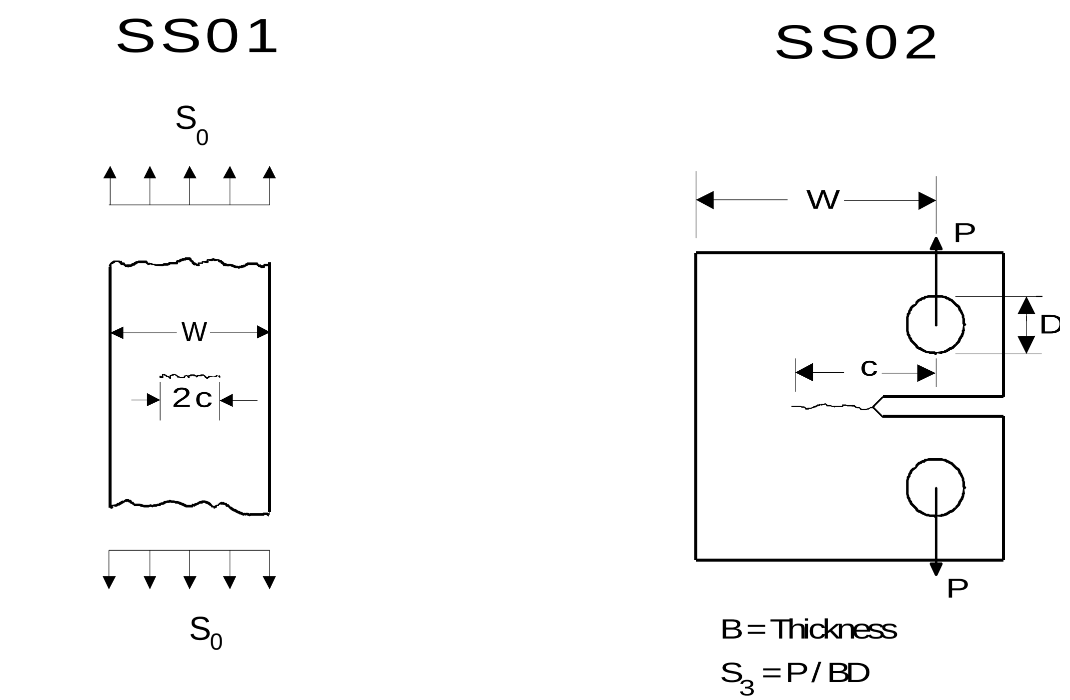


Figure 32 – Standard specimen crack cases 1 and 2

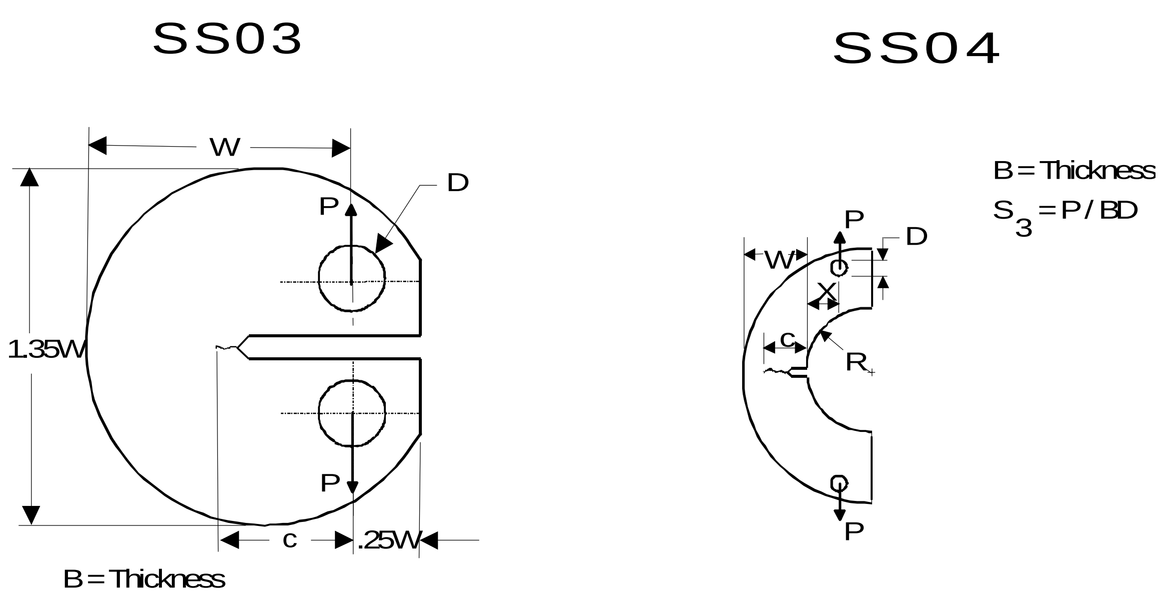


Figure 33 – Standard specimen crack cases 3 and 4

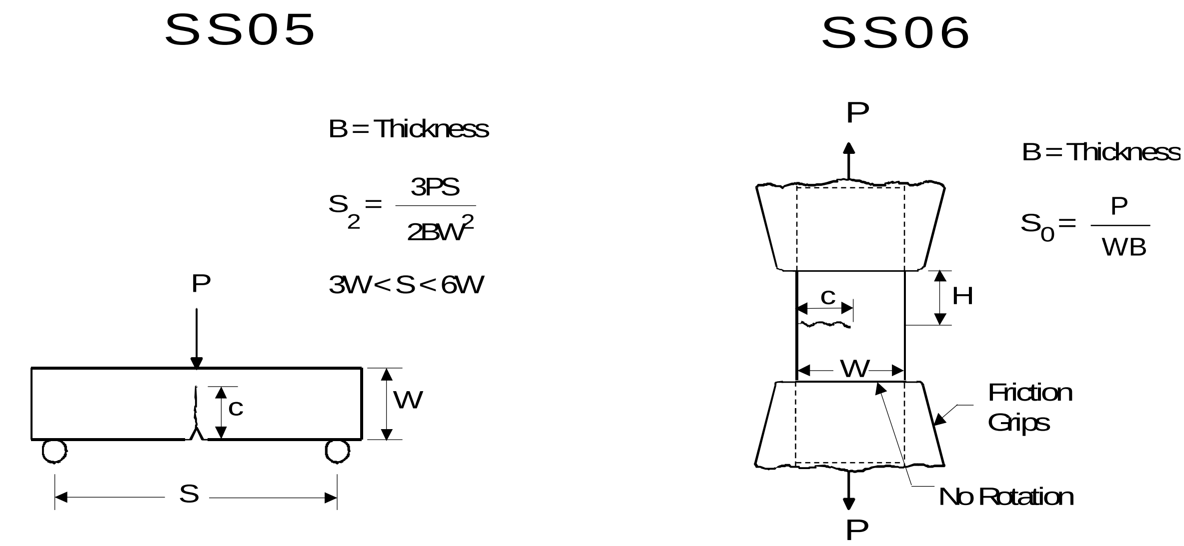


Figure 34 – Standard specimen crack cases 5 and 6

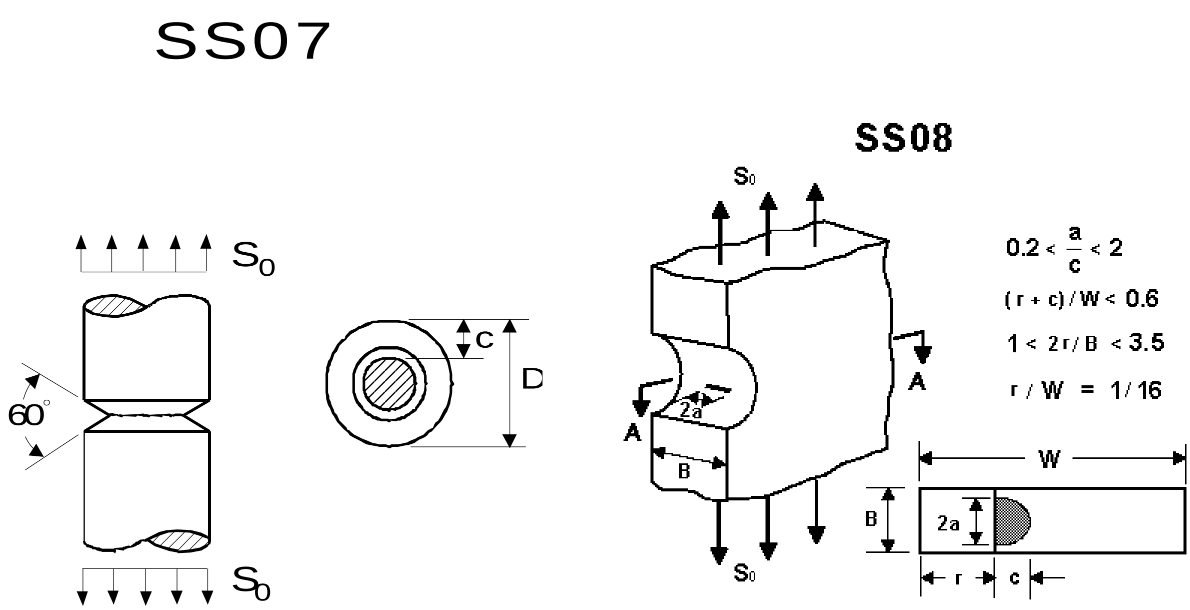


Figure 35 – Standard specimen crack cases 7 and 8

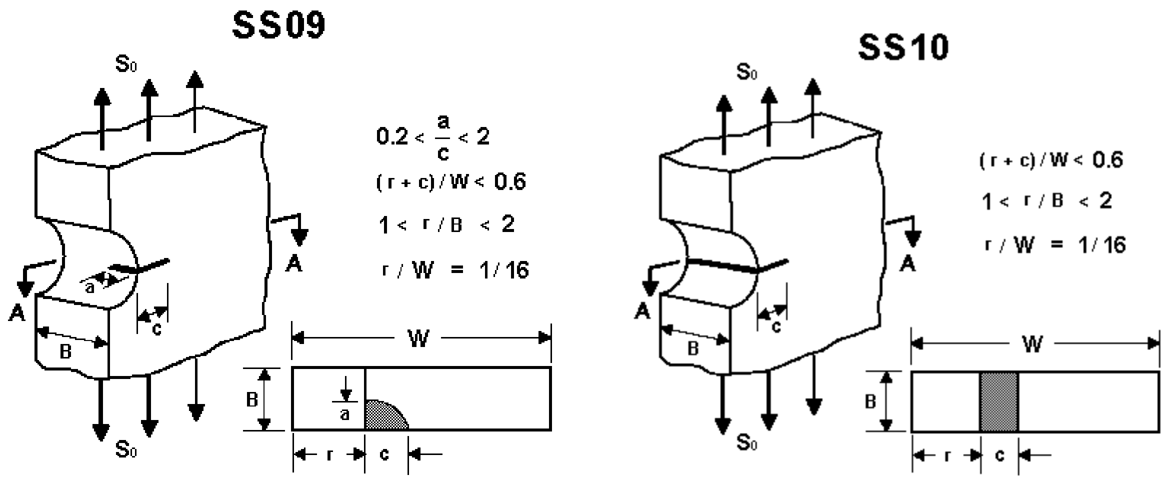


Figure 36 – Standard specimen crack cases 9 and10

If user-defined flaw size is selected, one enters the initial crack depth *c* for a through crack or the initial *a* and *a/c* ratio for a part-through crack. If the indirect life prediction option is chosen, the program will prompt for the NDE inspection technique, if applicable. For surface crack cases SC01 through SC05, you will also be asked whether you want to use the maximum or minimum crack depth for the initial flaw size. The NDE limits that are used for these crack sizes are shown in Tables 6 and 7.

Table 6 – Standard NDE Flaw Sizes for STS Payloads
US Customary Units

Crack Case	NDE Inspection Technique or Flaw Size Criterion	Thickness Range (in.)	***Crack Size (in.) a	c
TC01, TC06, TC07, TC08 (open surface)	EC	t ≤ 0.050	–	0.050
	P	t ≤ 0.050	–	0.100
	P	0.050 < t ≤ 0.075	–	0.15-t
	MP	t ≤ 0.075	–	0.125
TC02 (edge)	EC	t ≤ 0.075	–	0.100
	P	t ≤ 0.100	–	0.100
	MP	t ≤ 0.075	–	0.250
TC03, TC04, TC05, TC09 (hole)	EC	t ≤ 0.075	–	0.100
TC10	P	t ≤ 0.100	–	0.100
	MP	t ≤ 0.075	–	0.250
	HPD – driven rivet	any thickness	–	0.005
	HPD – other holes	t ≤ 0.050	–	0.050
EC01	R	0.025 ≤ t ≤ 0.107	0.35t	0.075
	R	t > 0.107	0.35t	0.7t
	U	t ≥ 0.300	0.065	0.065
CC01 (edge), CC05	EC	t > 0.075	0.075	0.075
	P	t > 0.100	0.100	0.100
	MP	t > 0.075	0.075	0.25
	U	t > 0.100	0.100	0.100
CC02, CC03 (hole), CC04	EC	t > 0.075	0.075	0.075
	P	t > 0.100	0.100	0.100
	MP	t > 0.075	0.075	0.25
	U	t > 0.100	0.100	0.100
	HPD – not driven rivet	t > 0.050	0.050	0.050
SC01, SC02. SC03 (open surface)	EC	t > 0.050	0.020	0.100*
SC11, SC12			0.050	0.050**
	P	t > 0.075	0.025	0.125*
			0.075	0.075**
	MP	t > 0.075	0.038	0.188*
			0.075	0.125**
	R	0.025 ≤ t ≤ 0.107	0.7t	0.075
		t > 0.107	0.7t	0.7t
	U	t ≥ 0.100	0.030	0.150*
			0.065	0.065**
SC04, SC05	EC (ext and int)	t > 0.050	0.020	0.100*
			0.050	0.050**
	P (ext)	t > 0.075	0.025	0.125*
			0.075	0.075**
	MP (ext)	t > 0.075	0.038	0.188*
			0.075	0.125**
	R (ext and int)	0.025 ≤ t ≤ 0.107	0.7t	0.075
		t > 0.107	0.7t	0.7t
	U (ext and int)	t ≥ 0.100	0.030	0.150*
			0.065	0.065**
SC06	EC (ext and int)	t > 0.050	0.020	–
	P (ext)	t > 0.075	0.025	–
	MP (ext)	t > 0.075	0.038	–
	R (ext and int)	0.025 ≤ t ≤ 0.107	0.7t	–
	U (ext and int)	t ≥ 0.100	0.030	–
SC07	EC	–	Eq 2.36, 2.37	0.050
	P	–	Eq 2.36, 2.37	0.075
	MP	–	Eq 2.36, 2.37	0.125
SC08 (rolled threads), SC13(machined fillet), SC14(machined or rolled fillet)	P	–		0.075
SC09, SC10 (machined threads)	max machining defect size	–	thd depth +0.005	–

Notes:

EC = eddy current | R = radiographic | MP = magnetic particle
P = dye penetrant | U = ultrasonic | HPD = hole preparation defect (max)
*minimum crack depth | **maximum crack depth | ***1 in. = 25.4 mm

Table 7 – Standard NDE Flaw Sizes for STS Payloads
SI Units

Crack Case	NDE Inspection Technique or Flaw Size Criterion	Thickness Range (mm)	***Crack Size (mm) a	c
TC01, TC06, TC07, TC08 (open surface)	EC	t ≤ 1.270	–	1.270
	P	t ≤ 1.270	–	2.540
	P	1.270 < t ≤ 1.905	–	3.81-t
	MP	t ≤ 1.905	–	3.175
TC02 (edge)	EC	t ≤ 1.905	–	2.540
	P	t ≤ 2.540	–	2.540
	MP	t ≤ 1.905	–	6.350
TC03, TC04, TC05, TC09 (hole)	EC	t ≤ 1.905	–	2.540
TC10	P	t ≤ 2.540	–	2.540
	MP	t ≤ 1.905	–	6.350
	HPD – driven rivet	any thickness	–	0.127
	HPD – other holes	t ≤ 1.270	–	1.270
EC01	R	0.635 ≤ t ≤ 2.718	0.35t	1.905
	R	t ≥ 2.718	0.35t	0.7t
	U	t ≥ 7.620	1.651	1.651
CC01 (edge), CC05	EC	t > 1.905	1.905	1.905
	P	t > 2.540	2.540	2.540
	MP	t > 1.905	1.905	6.35
	U	t > 2.540	2.540	2.540
CC02, CC03 (hole), CC04	EC	t > 1.905	1.905	1.905
	P	t > 2.540	2.540	2.540
	MP	t > 1.905	1.905	6.35
	U	t > 2.540	2.540	2.540
	HPD – not driven rivet	t > 1.270	1.270	1.270
SC01, SC02. SC03 (open surface)	EC	t > 1.270	0.508	2.540*
SC11, SC12			1.270	1.270**
	P	t > 1.905	0.635	3.175*
			1.905	1.905**
	MP	t > 1.905	0.965	4.755*
			1.905	3.175**
	R	0.635 ≤ t ≤ 2.718	0.7t	1.905
		t > 2.718	0.7t	0.7t
	U	t ≥ 2.540	0.762	1.270*
			1.651	1.651**
SC04, SC05	EC (ext and int)	t > 1.270	0.508	2.540*
			1.270	1.270**
	P (ext)	t > 1.905	0.635	3.175*
			1.905	1.905**
	MP (ext)	t > 1.905	0.965	4.755*
			1.905	3.175**
	R (ext and int)	0.635 ≤ t ≤ 2.718	0.7t	1.905
		t > 2.718	0.7t	0.7t
	U (ext and int)	t ≥ 2.540	0.762	1.270*
			1.651	1.651**
SC06	EC (ext and int)	t > 1.270	0.508	–
	P (ext)	t > 1.905	0.635	–
	MP (ext)	t > 1.905	0.965	–
	R (ext and int)	0.635 ≤ t ≤ 2.718	0.7t	–
	U (ext and int)	t ≥ 2.540	0.762	–
SC07	EC	–	Eq 2.36, 2.37	1.270
	P	–	Eq 2.36, 2.37	1.905
	MP	–	Eq 2.36, 2.37	3.175
SC08 (rolled threads), SC13(machined fillet), SC14(machined or rolled fillet)	P	–		1.905
SC09, SC10 (machined threads)	max machining defect size	–	thd depth +0.127	–

Notes:
EC = eddy current R = radiographic MP = magnetic particle
P = dye penetrant U = ultrasonic HPD = hole preparation defect (max)
*minimum crack depth **maximum crack depth ***1 in. = 25.4 mm

2.2.4 Selecting the Material Properties

The fracture mechanics data which have been curve fit for this release of NASGRO are contained in a database that includes approximately 6000 sets of fracture toughness data and about 3000 sets of crack propagation data [26]. References for the fracture mechanics data included the 1978 and 1982 editions of Hudson's Compendium [27, 28] and the Damage Tolerant Design Handbook [29]. The remaining references were taken from miscellaneous published reports and journal articles. Curve fit constants to Eq 2.1 were generated for over 300 different material-environment conditions and have been entered into the NASGRO material files (NASMFC and NASMFM). For a complete description of the curve fitting methodology and a comparison of the curve fits with the crack growth data, see reference [30].*

The curve fit constants for the materials that are entered into the NASMFC and NASMFM materials files are listed in Table G1 (English units) and Table G2 (SI units) in Appendices G1 and G2. *These constants are normally typical or least squares statistical fits to data, i.e. not minimum or design allowable properties*. Also, in many cases, the tensile, threshold, and toughness properties are estimated or averaged over a fairly wide range of values. Factors of safety on life (e.g. 2 or 4) are then used to achieve adequate conservatism in analysis. Many of the curve fits are for a laboratory air (LA) condition, which is an acceptable assumption for most space hardware, which are not exposed to wet environments. In some cases, when environmental effects are minimal, data for other environments, such as dry air, high humidity air, or saltwater, have been included in the curve fit. In other cases, a separate curve fit has been generated for a specific environment. Some material/environment combinations are susceptible to frequency effects, and the frequency range for which data were available are listed with the curve fits.

Each particular curve fit in Tables G1 and G2 applies only for the orientation(s) specified. If no crack orientation is specified, the curve fit should be assumed to apply to any orientation except S-T, S-L, C-R, C-L, and R-L. Figure 37 shows the nomenclature of the crack plane configurations for both rectangular and round product forms [31].

After selecting the total number of materials, the program prompts for the crack growth interaction model to be used. This may be any one of the five models: non interaction, Generalized Willenborg, Modified Willenborg, Walker-Chang, Strip Yield or Constant Closure model. Once the model is selected, the type of crack growth properties to be entered for that model needs to be selected. The most common choice is input from NASGRO materials files, in which case properties are picked up from appropriate files automatically.

---

* Copies of JSC 26254 are available by phoning R. G. Forman at (281) 483-8926 or by sending a FAX request to (281) 244-2319.

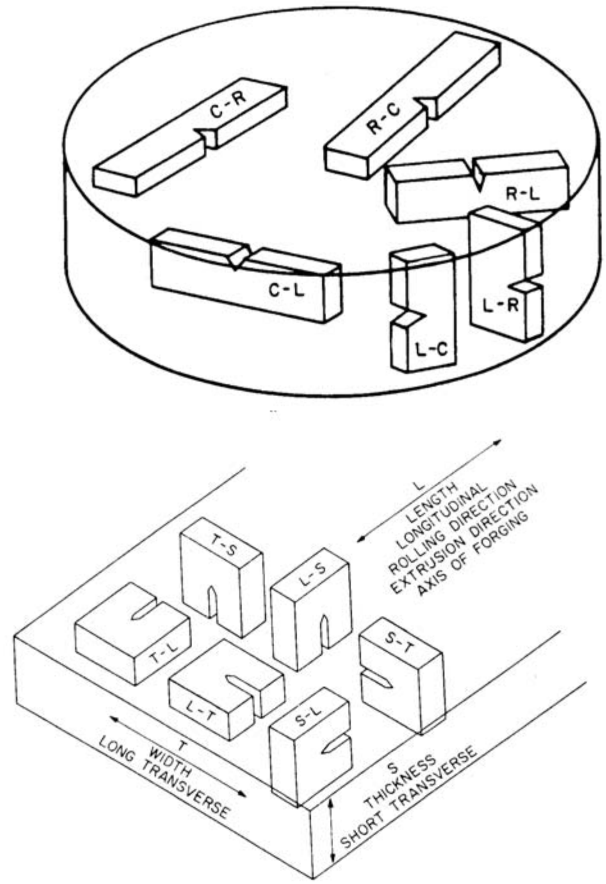


Figure 37 – Crack plane orientations

Input of material properties can also be accomplished manually (option 3) by entering 1-D or 2-D tables of da/dN-ΔK data or by supplying the constants to Eq 2.1. If a 1-D or 2-D table is specified, NASGRO uses an interpolation routine to compute da/dN values for a given crack growth analysis. Additional information regarding the interpolation routines that are used by NASGRO 3.0 may be found in Appendix F.

Two-dimensional tables are especially applicable to problems which include fatigue spectra with multiple stress ratios and environmentally accelerated crack growth that cannot be analyzed by Eq 2.1. The format for the tabular input is shown in Table 8. Here, the $\Delta K_c(R)$ values are the critical stress intensity factor ranges that cause fracture. For computational purposes, these values are assumed to correspond to data points having a growth rate of 25.4E-1 mm/cycle (1.0E-1 in/cycle), or any other higher growth rate value entered in the first row of the table. In a similar manner, the last row of data should contain the $\Delta K_{th}(R)$ values for no growth correspond to points with a growth rate of 25.4E-8 mm/cycle (1.0E-8 in/cycle), or less. The example problem 5 demonstrates entry of a 2-D table of da/dN-ΔK material properties using option 3.

In the graphical user interface, there is an option to enter a user-defined equation for crack growth as a function of ΔK and R in symbolic form. Any algebraic expression including standard functions such as Sin, Cos etc., are allowed. The GUI then computes the 2D table of ΔK values for selected R ratios. Once the table is filled, it can be saved for future use just as in the case of manually entered table of data.

Properties that are entered manually may be saved in the user-defined material files (USRMFC USRMFM, etc.,) and may be accessed in future crack growth analyses using option 4. Option 6 may be used to delete data from the user-defined material files. Note that these deletions are unrelated to the crack growth analysis in progress and will be retained even if the crack growth calculation is not completed.

Table 8 – Two-dimensional Interpolation Table

da/dN mm/cycle (in/cycle)			ΔK		
	R	…	…	…	…
25.4E-1 (1E-1)	$K_c(R)$	…	…	…	
…	…	…	…	…	
			4 to 25 columns (all elements must be filled)		
…	…	…	…	…	4 to 25 rows
…	…	…	…	…	
25.4E-8 (1E-8)	$\Delta K_{th}(R)$	…	…	…	

Another method of entering a 2-D table has also been added recently. In this method, the da/dN values may be different for each of the R ratios. Here too, the first da/dN value should be at least 0.1 in/cycle. If the first entry is less than 0.1 in/cycle, the first entry will be set to 0.1 in/cycle and your entry will be placed in the next row. The last value of da/dN will be used as the threshold value. Once the input is complete, the table is internally converted to the form shown in Table 8, and then computations proceed as usual.

2.2.4.1 Entering Material Properties Data from File

To use the properties from the NASGRO materials files (NASMFC and NASMFM) in your crack growth analysis, choose option 1. The curve fit constants for Eq 2.1 will be selected for the material that you specify using the codes shown in Tables G1 and G2. The basic format for the code follows:

- **Letter** to identify material category
- **Number** to identify alloy group
- **Two letters** to identify alloy and heat treatment
- **Two numbers** to identify product form and crack orientation OR
  Letter and number for weld type and crack orientation
- **Two letters** to identify the environment
- **Two numbers** to identify the temperature.

The program proceeds step by step through the category, alloy group etc. until the final selection is made. Choose option to select 1-D or 2-D da/dN-ΔK tables or Eq 2.1 curve fit constants that have been previously saved to the user-defined materials files. A menu system similar to that used for the NASGRO materials files has been enabled for the user-defined files also. After selecting the appropriate code, specify whether the data is in the form of Eq 2.1 curve fit constants, a 1-D table, or a 2-D table.

2.2.4.2 Entering Material Properties Data Manually

To enter material properties manually, choose option 3. The program prompts for a material header information page and along with some descriptive information, σ_{ult}, σ_{ys}, K_{Ie}, K_{Ic}, A_k, and B_k are to be provided After values for these constants have been entered
the following menu appears:

Enter da/dN analysis option:
W Closure model (NASGRO equation)
x Walker crack growth rate equation
1 1-D Table - Delta K vs. da/dN
2 2-D " - Delta K (R) vs. da/dN.

Choose the first option if you have curve fit the data using the procedure described in section 6.2 and want to use the constants for the crack growth analysis and/or save them in a user-defined materials file. The program prompts for C, n, p, q, ΔK_0, R_{cl}, α, and S_{max}/σ_0. Similarly, if **x** is chosen, the constants for Walker equation will be prompted for. C, n, m, q etc., as listed in Appendix L will be requested.

To enter a one-dimensional table of da/dN-ΔK values for a single stress ratio, select option 1 and the following message will be printed:

Enter da/dN values in descending order for at least 4 data points.

After the entry of a 1-D or 2-D table has been completed, the option to plot are presented.

If plotting is selected, you are given an opportunity to:

produce DeltaK vs R plot
produce da/dN vs DeltaK plot
terminate plotting

in the case of the two-dimensional table. Since 1-D tables have data at only one stress ratio, the ΔK vs. R plot option is not available.

If you select a plot you will be prompted for maximum and minimum x and y values for the axis and the plotting device.

2.2.5 Building the Load Schedule

The concept of the “load schedule” was introduced in version 2.0 of NASGRO. The load schedule concept provides a lot of flexibility by allowing the user to repeat blocks and/or combine different blocks together. This is especially useful for analyses of parts that are subjected to both pre-flight testing and flight loads. In addition, the load schedule method provides a means for entering larger spectra.
Figure 38 shows an example of a load schedule and the associated terms. A load schedule is created by filling up to 9999 blocks with different block cases. Up to 20 different block cases can be combined, ordered, and/or repeated within these available blocks. Each distinct block case can contain up to 200 load steps when stored in the BLOCKS file, where a load step is defined as any number of cycles (up to 999,999,999.99) of stress alternating between two specified limits. New options to input unlimited sized blocks such as those occurring in aircraft applications have been added to NASGRO 3.0. In such a case, each block is stored in a separate file either in standard NASGRO format or in sequential form. In the example shown in Figure 38, blocks 1 through 4 of the load schedule contain block case 1, which is a single cycle. Block case 2, which is constructed of 1 load step that has 5 cycles, has been put in blocks 5 and 7. Finally, block 6 of the load schedule contains block case 3, a more complicated case that has 7 load steps.

To begin entering the load schedule, enter a title or heading for the schedule. You may also indicate if you would like to give a description (up to 20 characters) for each step in the block. Then the program prompts for the number of times to apply the load schedule (in direct mode) or for the life in schedules (in indirect mode). Enter the number of schedule repetitions (up to 9,999,999) desired. Next, you are asked to select the number of block cases that will be defined. You may enter up to 20 distinct block cases to be combined to form the load schedule.

2.2.5.1 Defining the Block Cases

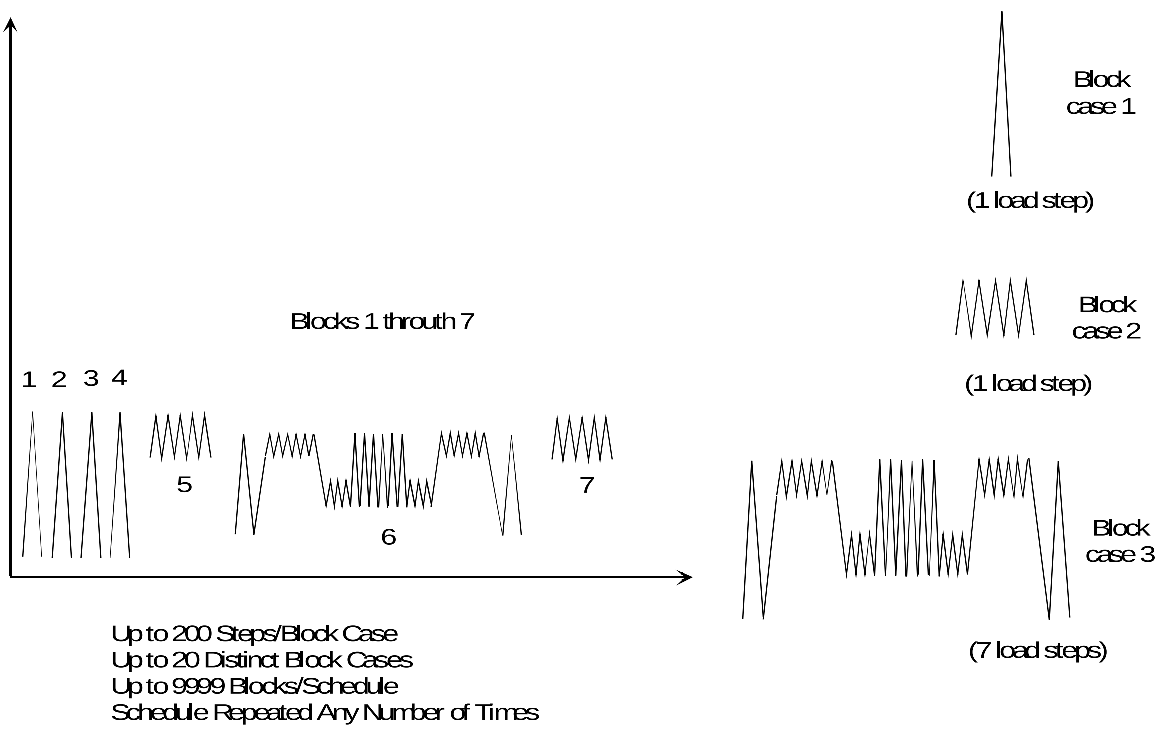


Figure 38 – Load schedule example

First, you are prompted for the number of load steps for the first block case and then the following block menu is printed:

Display Menu of Blocks on File
Enter/Edit a Block from File
Enter/Edit a Block manually
Delete a Block from file
Generate a Block for Acceptance Vibration Testing
Enter File Names of Long Aircraft-type Blocks
Enter more steps or Next Block or to End Input

To enter a block that has been saved in file BLOCKS, choose second option. The loading spectrum for payloads, which experience stresses associated with the launch and landing of the Space Shuttle, is given in Table 9. The number of cycles at each stress level for the combined launch/ascent and landing/ descent condition are tabulated in the column labeled "Total". This spectrum was developed at the Goddard Space Flight Center (GSFC) and is reported in reference [32]. It is appropriate for analyzing*:

– primary, load-carrying payload structure in the Space Shuttle Orbiter payload bay
– payloads with a fundamental (first mode) frequency below 50 Hz.

Payloads that have a fundamental frequency above 50 Hz may be analyzed by applying the following multiplication factors to the *number of cycles* of the GSFC flight spectrum:

*This spectrum and its suggested application are provided in NASGRO as an aid for analysis of shuttle payloads. Inclusion of the application here does not constitute a requirement. Application of the GSFC flight spectrum for a specific payload should be in compliance with the appropriate requirement documents, or be approved by the proper authority.

Table 9 – Launch and Landing Spectrum for STS Payloads

Load Step	Cycles/flight			Cyclic Stress (% limit value)	
Number	Launch	Landing	Total	Minimum	Maximum
1	1	1	2	-100	100
2	3	1	4	-90	90
3	5	3	8	-80	80
4	12	3	15	-70	70
5	46	3	49	-60	60
6	78	3	81	-50	50
7	165	13	178	-40	40
8	493	148	641	-30	30
9	2229	891	3120	-20	20
10	2132	1273	3405	-10	10
11	2920	2099	5019	-7	7
12	22272	6581	28853	-5	5
13	82954	8701	91655	-3	3

fundamental frequency	multiplication factor
0-50 Hz	1
50-100 Hz	2
100-200 Hz	4
200-300 Hz	6

The combined launch and landing spectrum has been saved as a block on file. To use it as one of the blocks in a schedule, specify 13 load steps, select option to enter a block from file, and type “GSFC” for the block name. After you enter the name of the block, the program will ask for the material number (if you have selected more than one material) and then for example, if you are using the GSFC spectrum and you specify 5, the load steps 1 through 9 in Table 9 will be written to load steps 5 through 13 of the block case, and load steps 1 through 4 will be left blank. This may be useful if you want to add one or more load steps before or after a block that has been saved on file. However, you must remember to enter the correct number of load steps so that the block that has been saved will not be truncated in a way that is not desired. Data may be added to blank load steps or existing data for a particular load step may be overwritten by editing the block case manually. NASGRO 3.0 also includes an option to change the mean stress of block cases that have been saved on file.

After defining the block case from data that have been stored previously in a file, the program prints a dialog box:

> Do you want to change the mean value for this block ? (Yes/No)
> (by adding a constant value to (t1) and (t2)).

If you enter "Yes", you are prompted for the constant value that needs to be added to increase or decrease the mean stress of each of the stress quantities. Note that the value entered should have the same units as those quantities stored in the file. The example problem 3 demonstrates the use of this option in the crack growth analysis of a threaded fastener with preload.

To enter a block case manually, select option 3 and the program will ask for the load step number and the material number. You will also be prompted for the number of cycles and the appropriate stress quantities (S_0, S_1, S_2, S_3 and/or S_4) at times (t1) and (t2) for that particular load step. If the crack case has more than one stress quantity, you will be asked if you want to use the same maximum and minimum stresses for the next stress quantity:

> Do you want to use the same Max., Min., for the stress quantity S*x* ?

Indicating "yes" will assign values for that stress quantity at (t1) and (t2) that are equal to those you have just entered for the previous stress quantity. Remember that if you want to zero out one or more of the stresses (e.g. no tension), you will still be allowed to scale the stresses later (see section 2.2.4.3) and you can set scale factor = 0 for the appropriate stress quantities. As soon as all of the stress quantities (S_0, S_1, S_2, S_3 and/or S_4) at (t1) and (t2) have been assigned for the first load step, you will be prompted for the next load step number. The process should be repeated until all load steps have been entered. When entry is complete, type 0 and the block menu will reappear.

NASGRO 3.0 has extended the spectrum entry to include automated block generation from vibration test data. It should be noted that each block case should contain only one vibration test type. To enter a vibration test block, specify one load step and select option 5. The following menu will appear:

> Narrow Band Sine Sweep Test Spectrum
> Wide Band Sine Sweep Test Spectrum
> Sine Dwell Test Spectrum
> Random Vibration/Acoustic Test Spectrum.

When generating a block case for a vibration test block, the program will first prompt for the data needed to calculate the number of equivalent cycles, and then will prompt for the stress quantities appropriate for the current crack model. Information regarding how the number of equivalent cycles for each of these vibration test types is calculated may be found in Appendix H. If you choose option 1, you are prompted for the sine sweep rate, the amplification factor, the resonant frequency, and the notch factor. Also, you must choose whether to apply an up sweep only or both up and down sweeps. If option 2 is selected, the program prompts for the sine sweep rate, the two frequencies that define the wide band, and whether to apply an up sweep only or both up and down sweeps. For option 3, the sine dwell test, you are only asked for the time duration and the frequency of the test. Finally, option 4 is used for either random vibration or acoustic vibration tests, since the analysis method is the same. If both tests are performed, this option must be chosen twice so that one block case will be defined for the random vibration test and one block case will be defined for the acoustic vibration test. When selecting this option, you must enter the time duration of the test and the resonant frequency. After both the vibration test information and the stresses have been entered, the block menu will reappear. An illustration of this option to define blocks based on vibration test data may be found in example1.

A new option has been added to the menu that gives users various choices of defining the distinct block cases. Option 6 in the menu (see p. 56) is meant to input long aircraft type of spectra in three different formats. Usually at the input prompt for number of steps in any block case, one needs to enter the actual number of load steps while entering manually or picking up blocks from the BLOCKS file. To use the new option of reading the blocks from individual files (such as long aircraft spectra generated by means of other software etc.,) simply enter 1 for the number of steps. The actual number of steps will be determined by the program after reading the files. The following options are available for reading the blocks from files.

1. Standard NASGRO format as in BLOCKS file, i.e.,
A title line with the first eight characters for a block name and the remaining for a title
As many lines as the number of load steps with each line having the following nine numbers:
Cycles S0(t1) S0(t2) S1(t1) S1(t2) S2(t1) S2(t2) S3(t1) S3(t2)
Since all stress quantities are present in this format, only one file name is required per block.

2. Loads sequence (peak-valley) in Northrop format (as furnished by SWRI)
The load values to be input in this format are 10 times the psi values in
integer form. Alternately, they may be thought of as percentages expressed
in integer form, because NASGRO uses ksi as basic unit.
A title line
Flight no. Number of values in this flight (up to 1000) loads in free format, i.e., any number of integers per line with as many lines as needed. Example:
1 102 1728 395 922 435 1216 395 1106 395 1261 339 798 395 1728 79

Any number of flights can be input in one file. One file name is needed for each stress quantity.

3. Loads sequence (peak-valley) in percentages
A title line
Flight no. No of values in this flight(up to 1000) (integers in free format)
Loads in free format, i.e., any number of real values per line with as many lines as needed.
Example:
10.5 11.9 6.2 12.1 15.2 18.2 30.2 11.0 23.0 11.2

Any number of flights can be input in one file. One file name is needed for each stress quantity.

In the case of format choices 2 and 3, each consecutive pair of loads is interpreted as a single cycle and the crack growth proceeds as a cycle-by-cycle analysis. Also, as indicated above, the applied load spectra for each stress quantity, tension, bending, pin load etc., can be different. The user will be prompted to enter as many file names as the number of stress quantities for the crack case being analyzed. These files are consolidated into a new file named xxx.NAS that will be created where xxx is the first portion (upto eight characters) in the name of the first block file. This file is written in the standard NASGRO format for future use. The scale factors can, of course, be different for different stress quantities.

Additionally, version 3.0 now includes an option to generate spectra commonly used in the aircraft industry. Currently there are three spectra available: TWIST, MINI-TWIST, and FALSTAFF. The generated spectra are automatically stored in the *NASGRO* format in the files *FULTWIST*, *MINTWIST*, and *FALSTAFF*, respectively, but beware the file size can be quite large. The spectrum peak stress is automatically scaled to a value of 100; for the TWIST spectra this gives a mean stress of 38.46. This can be adjusted to the actual requirements by means of the scale factor, which is specified in later input. Finally, the TWIST spectra also have an option to clip cycles at values relative to the mean stress.

When entry of the first block case (using any or a combination of options 2, 3, and 5) is complete, choose 0 from the block menu to continue. An error message will be printed if the block entry has not been completed (i.e., if there are blank load steps). Otherwise a table of the number of cycles and unscaled stresses (including the effect of any increase in mean values) at each load step will be printed to the screen.

2.2.5.2 Setting the K_{eac} Check

For some crack growth analyses, it will be necessary to enter the appropriate value for the critical stress intensity for environmental crack growth, K_{eac}. The purpose of this input is to ensure that K_{max} does not exceed K_{eac} for material and environmental combinations that could cause sustained stress or environmentally-assisted crack propagation. This is applicable to pressure vessels or other metallic components that are exposed to propellant fluids, gaseous hydrogen, high temperature air, or other severe fluid-material combinations. Since many parts are exposed to environments that have low K_{eac} values for only a portion of the entire spectrum, the K_{eac} check is designated at the end of the input of each block type, and $K_{max} < K_{eac}$ is only checked for the load steps that you specify. If you want to set the K_{eac} check, click the “Yes” box. The program will ask which load steps you want to check, and what the appropriate K_{eac} value is.

A materials file for K_{eac} data has not been developed because K_{eac} is often variable with temperature and environment. Table 10 lists K_{eac} data for several material-environment conditions which were obtained from experimental programs associated with the development of the Apollo and Space Shuttle vehicles, and that may be useful in the analysis of other space flight hardware. The main assumption in using K_{eac} is that the fatigue crack growth rate below K_{eac} is the same as it is in laboratory air. This is not always the case, but has been considered to be acceptable for the material/environment combinations listed in Table 10. An example of how to set the K_{eac} check on the analysis of a Ti-6Al-4V pressure vessel subjected to MMH is given in example 4.

Table 10 – K_{eac} Values for Several Metals

Alloy	**Environment**	**Temp**		**Keac**	
		°F	**(°C)**	**ksi-in$^{1/2}$**	**(MPa-mm$^{1/2}$)**
Ti-6A1-4V (STA, forg.) (base metal, weld, HAZ)	Nitrogen tetroxide (n204)	75	(24)	40	(1390)
	N204	100	(37)	36	(1251)
	N204	125	(52)	31	(1077)
	N204	150	(66)	25	(869)
	Monomethyl hydrazine (MMH)	75	(24)	40	(1390)
	MMH	100	(37)	38	(1320)
	MMH	125	(52)	34	(1181)
	MMH	150	(66)	30	(1042)
	Aerozine-50 (A50)	75	(24)	40	(1390)
	A50	100	(37)	38	(1320)
	Distilled water (DW)	75	(24)	42	(1459)
	DW	100	(37)	37	(1286)
	Isopropyl alcohol	75	(24)	40	(1390)
Ti-6A1-4V (STA, forg.) (base metal)	Trichlorotrifluoroethane	75	(24)	30	(1042)
	″	100	(37)	27	(938)
Ti-6A1-4V (STA, forg.) (weldline)	Trichlorotrifluoroethane	75	(24)	22	(764)
	″	100	(37)	19	(660)
Inconel 718 (STA) (base metal and weldline)	Air	850	(454)	50	(1737)
	Air	1000	(538)	13	(452)
	Air	1250	(677)	10	(348)
	500-5000 psi (3.45-34.5 MPa) Gaseous Hydrogen (GH2)	-100	(-73)	24	(834)
	″	75	(24)	20	(695)
A286 steel (STA)	″	75	(24)	85	(2954)
	Air	1000	(538)	25	(869)
Inconel 706 (STA)	Air	1200	(649)	10	(348)

2.2.5.3 Scaling the Stresses

After setting the K_{eac} check, the program will continue by prompting you for the information needed for definition of the next block case, and this process will continue until all block cases have been completed.

When all block cases have been defined, the program will prompt for the scale factors. As many scale factors as the number of stress quantities will be asked. If the first stress quantity is tensile; otherwise, whichever of the other 3 stress quantities is appropriate for the crack case will be substituted for S_0. When the crack case includes more than one stress quantity, you will be prompted for all of the necessary scaling factors. If you want to eliminate one of the stress quantities (e.g., there is no bending), set the corresponding scale factor to zero. After all the scaling factors for block case 1 have been set, a table of the scaled stresses and the number of cycles for each load step in the block case will be printed to the screen. The program continues by giving an option to set the reference stresses (see section 2.2.4.4) for the same block case before allowing you to set the scaling factors for the next block case. This process should be continued until scaling factors for each of the block cases have been entered.

2.2.5.4 Setting the Reference Stress Checks

For some crack growth analyses, a check is required to ensure that the residual strength of the structure is not exceeded by a seldom-occurring, severe load, such as a limit load, which is greater than the maximum load in the fatigue spectrum. This load corresponds to a stress called the "reference stress" by NASGRO 3.0. Since this load may occur at any time within a block, failure due to the reference stress is checked at every increment of crack growth. As described in section 2.1.5, failure is assumed to occur if the stress intensity factor corresponding to the reference stress exceeds K_c for the specified material.

After entering the scaling factors for each of the block cases, you will be prompted about whether you want to enter reference stresses for the current block case:

> Do you want to input Reference Stresses for this block ?
> (to check for crack instability at reference or limit stress).

If you enter "no", the program will continue on to prompt for the scaling factor(s) for the next block case. If you enter "yes", you will be asked for the reference stresses for each of the stress quantities appropriate for the crack case. After entering each reference stress, you are also asked to specify whether it should be checked at time (t1) or (t2) during the load cycle. This will depend on how the solution to the problem is constructed. However, if the stresses are fully reversed about zero and cycles from a negative stress at (t1) to a positive stress at (t2) (as in the Goddard spectrum), you will probably want to select (t2). An illustration of how to check the reference stresses is given in example 7.

2.2.5.5 Combining the Block Cases to Form Schedule

After all of the scaling factors and reference stresses have been entered, the dialog box to input the load schedule will be printed. Information such as starting block, ending block, and block case numbers can then be input.

Examples of different ways of combining block cases to form a load schedule may be found in example 1. An additional method of building the schedule has been added to the current version 3.0. This option asks if the schedule will be input from a file. If the answer is "No", the manual input described earlier will be used. If the answer is "Yes", the name of the file that contains the block mix is asked. After supplying the file name, if format type 1 is chosen, the block repetition information is same as in the manual input shown. The input file contents for a simple case are shown below:

Testing 1 - file for format type 1.
1 1
2 1
3 1
4 3

In each line, the first number refers to the block case number and the second number is the number of times that block should be repeated.

If format type 2 is chosen, the block repetition is done by giving the actual block numbers as they occur. The same example as above will now look like:

Testing 2 - file for format type 2
6
1 2 3 4 4 4

The first number is the total number of blocks and the remaining are block case numbers as they occur in the schedule. This format is useful when the file is automatically generated.

At this point, entry of the load schedule has been completed. If you are running the program interactively in the indirect mode, the iterations for initial crack size can begin by using the execute button. If you are running in direct mode, use the pull down menu entitled "print/plot options". This opens a dialog box with radio box choices so that output can be selected based on schedule/block/step basis or based on a certain crack growth increment. If the crack growth increment basis is chosen, you need to enter the size of the interval. If the schedule/block/step basis is chosen, you would enter those numbers. If you enter 0, 0, 0 the program will print only the final results, either when the entire schedule has been repeated the desired number of times or if the crack goes critical sooner. The final results specify whether the crack has reached the critical size or not, the crack size, and the current cycle, load step, block, and schedule numbers. In addition to the final results, you may specify a block interval for printing a short line summary, which includes the current crack size and stress intensity values. For part-through cracks, *a*, *c*, K_{max} at the a-tip, and K_{max} at the c-tip are printed. As an example, if you enter 1, 10, 0 the program will print a line of results at the end of every tenth block case in every schedule repetition. Similarly, if you want the program to print the results only at the end of every fourth schedule, you should enter 4, number of blocks in schedule, and 0. If you need more detailed results for specific block cases, you can specify a block interval for printing the results for every load step. For example, if you enter 2, 5, and 3 the program will print a short summary at every fifth block of every other schedule and a full printout of the results at each load step for every third block of every other schedule.

In the same dialog box, you will be prompted for information about plotting options.

3.0 Critical Crack Size Calculations

NASGRO 3.0 provides the capability of calculating the critical crack size for a specified stress level. The critical crack size computation, which also includes a check for net section yielding, is useful for application to proof test flaw screening. This calculation could also be used to screen a part to determine if a complete crack growth analysis is required. For many spectra, a full crack growth analysis may not be required if the critical crack size exceeds the NDE crack size by a large order of magnitude.

3.1 Theoretical Background

In applying the expression for the stress intensity factor (Eq 2.2) to a critical crack size calculation, the S_0, S_1, S_2, S_3, and or S_4 terms are the applied stresses for proof pressure, limit load, or some other specified condition and F0, F1, F2, F3, and/or F4 are the corresponding magnification factors. Therefore, the critical crack size, a_{cr}, may be defined as:

$$a_{cr} = \frac{1}{\pi}\left[K_{\max} \Big/ \sum_n S_n F_n \right]^2 \tag{3.1}$$

where the F_n terms are functions of critical crack size and shape. For an iterative solution to Eq 3.1, the root(or solution) of the following equation must be found:

$$G = K_{cr} - K_i = 0 \tag{3.2}$$

where K_i is the stress intensity corresponding to a trial flaw size, a_i, and a defined flaw shape, a/c. Newton's method is used, and the iterative process is started by entering an initial guess, a_g, which is within the geometric bounds of the crack case. The crack sizes for successive steps are then calculated by:

$$a_{i+1} = a_i + da_i \tag{3.3}$$

where

$$da_i = \frac{-G(a_i)\Delta a}{G(a_i + \Delta a) - G(a_i)} \tag{3.4}$$

with $\Delta a = 0.005 a_g$ = constant. The iteration continues until $|G| \leq 10^{-6}$ and the corresponding crack length converges to the critical crack size a_{cr}.

The iteration is stopped if the stresses are such that a valid solution is not obtained within the geometric bounds of the chosen model. The next recourse is to seek a solution using a more suitable combination of stresses. Occasionally, though, even if the stresses provide a valid solution for a_{cr}, the iteration falls outside the geometric bounds because a poor starting value was chosen. Then, convergence is usually achieved by trying a higher or lower starting value, depending on the trend of the failed iteration.

3.2 How to Run a Critical Crack Size Calculation

When you select the option to perform a critical crack size calculation, the GUI can be used to input geometry and dimensions of the crack case to be used. Then, if it is a part-through crack geometry, it will bring up a radio box choice with the following choices:

critical crack length is to be based on K at a-tip
critical crack length is to be based on K at c-tip
critical crack length is to be based on max K.

Select the desired option and enter K_{cr}. For a part-through crack, enter K_{Ie} from table G1 or G2, and for a through crack, enter the K_c value as calculated from Eq 2.11 and 2.12. Then enter the material's yield strength (see tables G1 and G2 in Appendices G1 and G2). The input also requires a trial value for the iteration process:

For part-through cracks, you will be asked to enter the crack shape ratio, a/c also. If the crack size will not converge, you may try running the routine with a different a/c ratio. Finally, the program needs the stresses S_0, S_1, S_2, S_3 and S_4 (whichever are applicable) in the part. An example of a critical crack size calculation of a Ti-6Al-4V pressure vessel membrane is given in the software.

4.0 Stress Intensity Factor Calculations

4.1 Theoretical Background

The stress intensity factor (SIF) is a measure of the severity of a crack in an elastic solid and is closely related to the stress field in the vicinity of the crack tip. There is a direct relationship between the SIF and the energy release rate which governs the criticality of a crack. Since the range of SIF (ΔK) during a fatigue loading cycle governs the crack growth rate, knowledge of the SIF for a given crack geometry is essential in any fatigue crack growth computation.

For the crack configurations described in section 2.2.1, and shown in Figures 16 through 36, a compilation of the equations or tables used in computing the SIF is given in Appendix C. The appropriate references which document the details of the solutions are also provided. Unique nonlinear interpolation routines were developed for accurate and efficient table lookup of the tabular solutions. Since most tables are multi-dimensional (e.g., variables of a/c, a/t, 2c/w, as in SCO2 shown in Figure 25), preprocessing is performed after entry of geometric dimensions to derive a two-dimensional table for a specific problem. Spline coefficients are calculated for this reduced table and reordered into a one-dimensional array for use in the crack growth analysis. This special preprocessing and dimensional array reduction results in computer run times approximately one-twentieth of those required for direct multi-dimensional and nonlinear interpolation procedures.

In order to analyze crack cases that are not currently contained in the stress intensity subroutines, SIF solutions may be added to NASGRO 3.0 by following the directions in Appendix D. Also, stress intensity factor solutions for other geometries may be obtained by running a boundary element analysis using the NASBEM module (see Chapter 7).

4.2 How to Run a Stress Intensity Factor Calculation

To compute stress intensity factors, choose the relevant option from the NASFLA main menu to run the K solution module. The GUI has pull down menus to input the crack case to be used and the necessary geometric dimensions, usually the thickness and width, radius, or diameter. Under the computating options tab, display of the following radio box choices will appear:

compute Stress Intensity Factors
compute Correction Factors

The first option is used for calculating the stress intensity factor(s) for a specific crack case or for plotting the variation in stress intensity factors with respect to geometry. The second option is used to calculate whichever correction factors F_0, F_1, F_2, F_3, and F_4 apply to a particular crack case. The stress and crack size information can also be entered from a text box. An example of how to solve for the correction factors in order to determine the proof pressure required for a glass window is given in the software.

4.2.1 SIF Calculations Using Linear Stresses

If you select option to compute the stress intensity factors, the program will prompt you for the material's yield strength (see tables G1 and G2 in Appendices G1 and G2) and the appropriate applied stresses. Then the following choices will be presented:

Tabulate solutions
Plot solutions.

If you choose tabulation, you will be prompted for crack lengths for the K calculations. For one-dimensional cracks, enter *a* or *c* values only; for two-dimensional cracks enter both the *a* and *c* values for which you want K(a) and K(c) calculations. If you choose to plot, the program will prompt you for the plotting device:

screen output if running X-Windows version
screen output if running plain Unix version (requires Tek4014 device)
screen output if running PC version
Postscript file NASPLT.XXX

and then display the following menu of options for choosing the x-axis and curve-defining variables:

x-axis	Curve defining variable variable	(= constant on each curve)
1	c	a/c
2	a	a/c
3	a/c	c
4	a/c	a
5	c	a
6	a	c.

The plotting routine allows you to obtain plots of K(a) and K(c) vs. a variable along the x-axis (choose from the first column) for several curves of a constant variable (choose from the second column). The plots may then be further non-dimensionalized (e.g. with respect to thickness t or width w). For options 1 and 2, you are given the choice to non-dimensionalize a or c by the following prompt:

Enter NAME of x-axis non-dimensionalizing variable

For options 3 and 4, you are given the choice to non-dimensionalize the curve defining variable, c or a, by the following prompt:

Enter NAME of non-dimensionalizer for curve defining variable

For options 5 and 6, you are given both choices. You can non-dimensionalize a and/or c by a, c, t, or w.

Next the program prompts you for the maximum and minimum limits for the x-axis variable and asks for the number of curves desired. Up to 5 curves of the constant variable are permitted. Finally, the program needs to know the maximum and minimum limits for the curve-defining variable. If more than 2 curves were selected, they are evenly distributed between the maximum and minimum values specified. If you have selected screen mode as the plotting device, the plot will be displayed on your screen. If you have selected postscript as the plotting device, you will need to send the resultant NASPLT file to the printer, using an appropriate command. An example of how the plotting routine works is shown in the software.

4.2.2 SIF Calculations Using Nonlinear Stresses

Three of the stress intensity factor solutions (SC02, SC04, and SC06) allow nonlinear stresses to be applied. After entering the yield strength of the material, the program prompts for the number of stress distributions which you want to include in the analysis. Up to four stress distributions are permitted for each crack case. For the SC04 crack case, the program allows you to enter the internal pressure and automatically generates stresses through the thickness of the pressure vessel. You may indicate that you want this option by clicking on the radio box showing

Generate stresses due to Unit (1 ksi or 1 MPa) Internal Pressure (Yes or No)

For crack cases SC02, SC06, SC10, and if you choose not to generate stresses for the SC04 case, the program will prompt you for the nonlinear stresses:

Enter values of Nondimensional Position (x/t) and Stress
(For linear case, 2 locations (e.g. x/t = 0 and 1) are sufficient).

Actual stress values or scaled stresses may be entered for up to 50 different non-dimensional positions through the thickness of the plate or cylinder. There is also an option to plot the nonlinear stresses you have just entered as a function of the non-dimensional thickness. If yes, the program will ask for the graphics device number, x-axis and y-axis labels, and a title for the plot. Then you are permitted to choose whether the axes should be drawn linear-linear or log-log. Note that if you are plotting to a hard copy device, the program creates a file called NASPLT.XXX, which may later be sent to your printer using an appropriate command. Next, the following dialog box is displayed:

Tabulate solutions
Plot solutions.

If you choose tabulation, you will be prompted for both the *a* and *c* values that you want to use for the K(a) and K(c) calculations.

If you previously entered no to indicate that you did not want to plot the stress distribution as a function of x/t, you are only given the option to tabulate the K(a) and K(c) values. S_n and S_n/σ_{ys} values for each crack size are also listed in the table, if they are available. Otherwise these columns will contain zeros. After the table is completed, you will be given an opportunity to flag the table for printing:

5.0 Sustained Stress (da/dt) Analysis of Glass

The strength of a glass surface is governed by the distribution of cracks present and the growth of these cracks under sustained stresses [34]. The amount of crack growth depends on crack size and geometry, the amount of sustained stress, and the environment (mainly humidity and temperature) to which the glass structure is exposed. NASGRO 3.0 provides the capability for performing sustained stress analyses for glass structures, such as optical lenses or windows, which contain cracks that may be accidentally introduced during handling. These analyses are carried out in a manner similar to the crack growth analyses of metals (see section 2.2), but the crack growth is a function of time instead of cycles.

It is expected that NASGRO 4.0 will include the ability to model crack propagation simultaneously as a function of time and the number of fatigue cycles. This capability is necessary for an accurate modeling of environmental crack growth in metallic materials.

5.1 Theoretical Background

In performing sustained stress analyses, the governing variable is time and therefore, da/dt is the operative variable for crack growth rather than da/dN. NASGRO 3.0 provides two models for calculating crack growth. One formulation is the following exponential equation proposed by Wiederhorn [34]:

$$da/dt = V_0 e^{BK} \tag{5.1}$$

and the other is a Paris-type model that is expressed by:

$$da/dt = AK^n \tag{5.2}$$

At crack growth rates below approximately 1 μin/h (25 nm/h), the exponential equation provides a more conservative approach than Eq 5.2 and should be used in all sustained stress analyses for NASA space hardware. However, although verification tests are needed, it is believed that the Paris-type equation will provide more accurate predictions in this region. At intermediate crack growth rates, the two equations are comparable, and the exponential equation produces only slightly less conservative values than the Paris-type model. Figure 39 shows a comparison of sustained stress data for borosilicate glass UBK7 in a high humidity air condition with curve fits to Eq 5.1 and 5.2.

5.2 How to Run a Sustained Stress Analysis

An example of how to run a sustained stress analysis of a glass window may be found in the software. Begin by selecting sustained load analysis option from the NASGRO radio box choices. As with the other options, you will be prompted for the units and for an output file name. Next, enter a title for the calculation and the crack case that you want to evaluate.

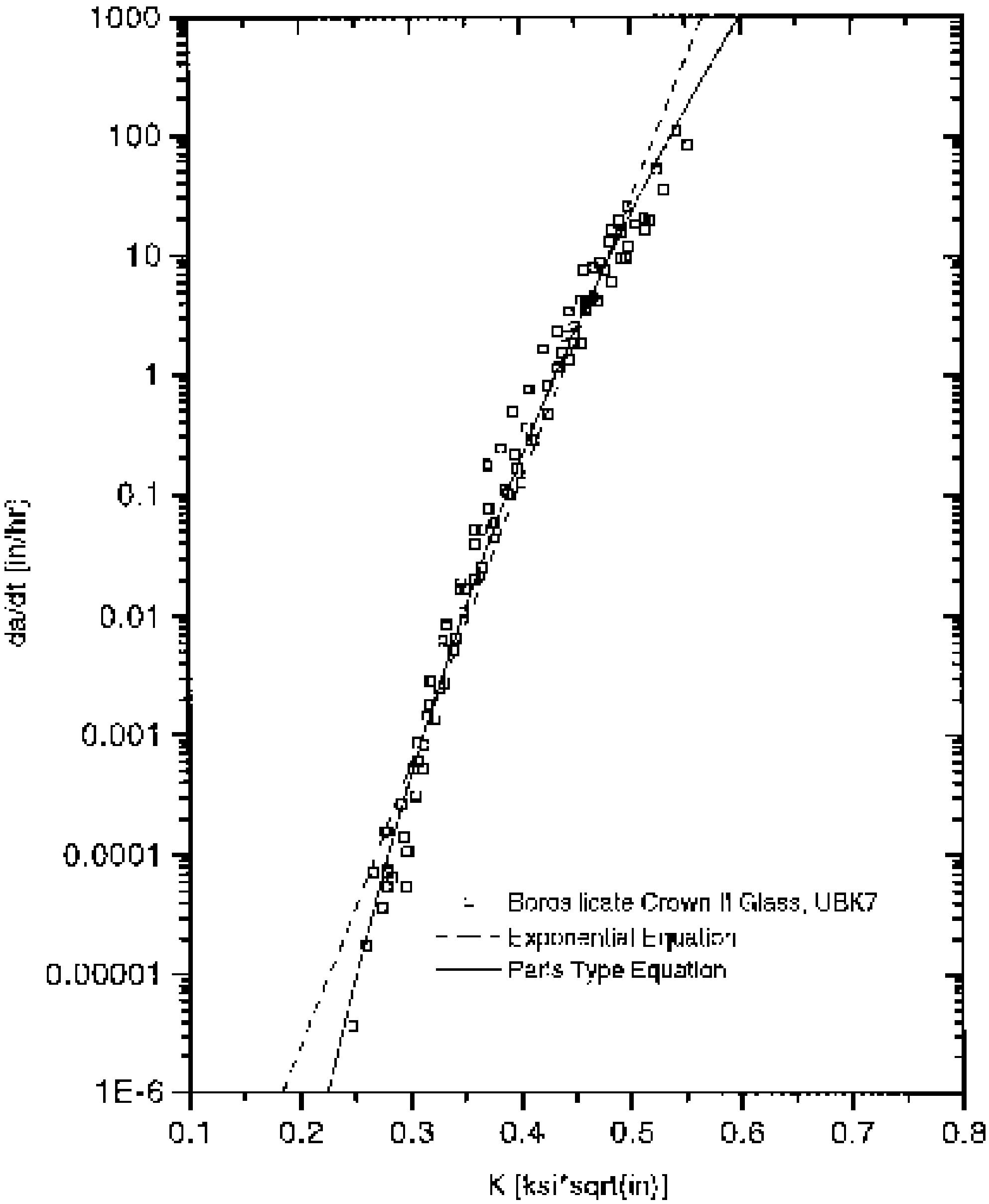


Figure 39 – Sustained stress data with curve fits

5.2.1 Material Properties

Sustained stress data for various glass materials were obtained from references [33-36] and have been curve fit to Eq 5.1 and 5.2 . The V_0, B, A, and n constants for these materials have been entered into the NASGLC (English units) and NASGLM (SI units) files, and are listed in Tables 11 and 12. Most of the curve fits are for a high humidity air (HHA) condition, which is a conservative assumption for most space systems glass hardware. The identification codes for the glass data have followed the nomenclature outlined in section 2.2.3, but are not needed for input to the program.

Table 11 – Sustained Stress Constants (English Units)

Material	Code	A	n	V_0	B	K_{Ic}	SD
Soda-Lime Glass, LA	W1AA10AB1A1	0.132E07	20.05	0.128E-10	50.47	0.670	0.034
Borosilicate Glass, C7740, HHA	W1AB10AD1A1	0.135E11	28.54	0.430E-14	75.04	0.675	0.024
Borosilicate Crown I Glass, BK7, HHA	W1AB10AD1A2	0.158E08	20.93	0.530E-10	51.99	0.643	0.022
Borosilicate Crown II Glass, UBK7, HHA	W1AB10AD1A3	0.497E08	21.13	0.471E-10	54.27	0.656	0.009
Lead Glass, SF1, HHA	W1AC10AD1A1	0.874E10	21.83	0.246E-10	68.41	0.775	0.021
Aluminosilicate Glass, C1723, HHA	W1AD10AD1A1	0.288E08	27.54	0.161E-13	59.37	0.697	0.016
96 Silica Glass, C7900, HHA	W1AE10AD1A1	0.193E11	29.16	0.295E-14	75.38	0.798	0.015
96 Silica Glass, C7913, HHA	W1AE10AD1A2	0.128E12	32.84	0.908E-16	80.50	0.810	0.015
Fused Silica, C7940, HHA	W1AF10AD1A1	0.492E11	34.44	0.113E-16	79.91	0.574	0.018
Fused Silica, DW	W1AF10WA1A1	0.510E14	35.89	0.341E-17	96.79	0.574	0.018

Table 12 – Sustained Stress Constants (SI Units)

Material	Code	A	n	V_0	B	K_{Ic}	SD
Soda-Lime Glass, LA	W1AA10AB1A1	0.426E-23	20.05	0.326E-09	1.45	23.29	1.18
Borosilicate Glass, C7740, HHA	W1AB10AD1A1	0.363E-32	28.54	0.109E-12	2.16	23.46	0.82
Borosilicate Crown I Glass, BK7, HHA	W1AB10AD1A2	0.224-23	20.93	0.135E-08	1.50	22.33	0.76
Borosilicate Crown II Glass, UBK7, HHA	W1AB10AD1A3	0.343E-23	21.13	0.120E-08	1.56	22.80	0.32
Lead Glass, SF1, HHA	W1AC10AD1A1	0.495E-22	21.84	0.672E-09	1.97	26.91	0.73
Aluminosilicate Glass, C1723, HHA	W1AD10AD1A1	0.268E-33	27.54	0.408E-12	1.71	24.22	0.54
96 Silica Glass, C7900, HHA	W1AE10AD1A1	0.580E-33	29.16	0.748E-13	2.17	27.73	0.51
96 Silica Glass, C7913, HHA	W1AE10AD1A2	0.806E-38	32.84	0.231E-14	2.32	28.14	0.51
Fused Silica, C7940, HHA	W1AF10AD1A1	0.107E-40	34.44	0.287E-15	2.30	19.95	0.28
Fused Silica, DW	W1AF10WA1A1	0.649E-40	35.89	0.865E-16	2.79	19.95	0.28

After entering the number of sets of glass properties to be used for input in the analysis, the program will show radio box choices to select from the following glass properties input options:

Manual input
Input from NASGRO glass properties file

NASGRO 3.0 requires input of material properties in the form of constants for Eq 5.1 and 5.2. This may be accomplished manually (option 1) or automatically by using the constants in the glass properties file (option 2).

To enter the constants manually, select option 1. The program will prompt you for the constants, and after they are entered, the glass properties menu will reappear if more materials are to be entered. Otherwise the program will continue.

To enter the constants from the NASGRO materials file, select option 2. Curve fit constants for ten glass/environment combinations are available. There is a text box to input the required K_{Ic} value. K_{Ic} values and their associated standard deviations are listed in Tables 11 and 12 for a variety of different glass materials. Then you should input the type of fit for the da/dt data. Choose the appropriate radio box choice for Paris-type equation (Eq 5.1), or the Exponential equation (Eq 5.2).

5.2.2 Entering the Sustained Load Schedule

The sustained stress data are entered in a manner similar to that in which the load schedule for a crack growth analysis (see section 2.2.4) was entered. However, since most sustained loads are fairly simple, a repeatable schedule is not actually built by combining blocks, and there are no options for saving the sustained stress blocks to a file.

The number of load steps in a block and the total number of blocks will be prompted for. This is followed by the stress scaling factors. If you use SSF = 1, S0 will be the actual tensile stress. You will then be prompted for scaling factors S1, S2, and S3 (whichever are applicable for the chosen crack case). These scaling factors are applied to the entire block.

To enter a block manually, the program will prompt for load step number, material number, duration of sustained stress in hours, and the stress(es). This input process will be repeated until all load steps in the block are defined. The block menu will reappear. Choose option 1 to define more blocks or option 0 to continue. The graphical user interface provides tabs for geometry, material, spectrum and print options as well as one for execution. For the most part, the input process is fairly self-explanatory.

6.0 Crack Growth Data Analysis

This section describes the software module NASMAT used to store and curve fit the crack growth rate data and fracture toughness data. Sections 6.1 and 6.2 describe the process of entering either a vs. N or da/dN vs. ΔK data and obtaining the constants for the NASGRO fatigue crack growth equation (Eq 2.1). The graphical user interface is self-explanatory and hence the description given here is brief. The first screen provides four choices: NASA database or user database for crack growth data and NASA or user database for fracture toughness. There are two files (NASADATA.DAT, NASAHEAD.DAT) supplied with the software that contain the NASA database for crack growth data and one file (NASAKCDTA.DAT) that contains the NASA toughness database. These files are encripted so that users cannot alter them. When new crack growth data are entered, they are saved in a files named USERHEAD.DAT that stores the header information, and USERDATA.DAT. that contains the data. These files constitute a small scale database, and are updated when subsequent changes, additions, or deletions are made by the user. Similarly, the new toughness data is stored in the file USERKCDTA.DAT. Section 6.3 describes the entry and processing of fracture toughness data.

6.1 Entering a vs. N or da/dN vs. ΔK Data for Curve Fitting

If you have fatigue crack growth data at several stress ratios that needs to be curve fit to Eq 2.1, the first step is to enter the data. To do this, follow the choice and dialog boxes on the screen. It is not necessary to define an identification code that is as complicated as those listed in tables G1 and G2. However, make sure that the last character in the ID code is reserved for the number of stress ratios for which you have data and the total number of characters is 11 or less. The program will then check to be sure that there is not already data with the same ID code. If none exists, you are asked if you want to create an entry for it.

When entering new data, you are first asked for information to be included in a header file. The header information consists of: the alloy name, heat treatment, product form, test environment, specimen type, crack orientation, specimen thickness and width, yield strength, ultimate strength, number of stress ratios, R values, test frequency, and the reference number. Next, you will be prompted for the units that will be used with the data for the first stress ratio to be entered. If you are using English units, you are permitted to enter da/dN data in micro-in/cycle or in/cycle. If you are entering SI data, three options for units are given. It should be noted, however, that the data are always stored in either ksi-in$^{1/2}$ and in/cycle for English or in MPa-mm$^{1/2}$ and mm/cycle for SI. Next choice is given weather you want to enter a vs. N data or da/dN-ΔK data.. The da/DN data can be entered either directly or in the form of a vs N data which will be converted to da/dN. If the later option is chosen, the specimen type and the dimensions will be prompted for. The data entry radio box allows the user to choose the method of entry depending on the type of data to be entered, and the appropriate radio box buttons will be displayed. The choices for da/DN data are: from keyboard, NASA lab file, text file and digitizer. For keyboard input, data pairs are entered in cells on a grid. Once the data is entered it can be saved and curve-fitted.

6.2 Obtaining the Constants

To obtain the constants for the crack growth relationship, Eq 2.1, first select source of the data set that you want to curve fit. The data may be from NASA database or user database. The data ID can be partially entered or built from a series of menus. The user then clicks on the "show data" button to display available data. Each of the available data sets that match the ID entered are displayed. You then click on the ID of interest and click the "load data" button. Available R values are shown and check boxes are provided to choose data for plotting only(P) or for plotting and fitting(F). Many of the curve fits in the NASGRO materials file were generated using the data at R = 0.1 as the base data set. A good first guess would be to use the data set that has the least scatter and widest range of data points. Once the data sets are choosen for fitting, click on the "plot/curve fit" button and the next screen provides all the necessary choices and text boxes for entry of parameters. Curve fitting crack growth data is an iterative process which consists of entering guesses for some of the constants, specifying a data set for a least squares fit if desired, and plotting the data at varying R values with the curve fit at each stress ratio. The process is continued by changing the values of the constants slightly until the best fit to the experimental data is obtained.

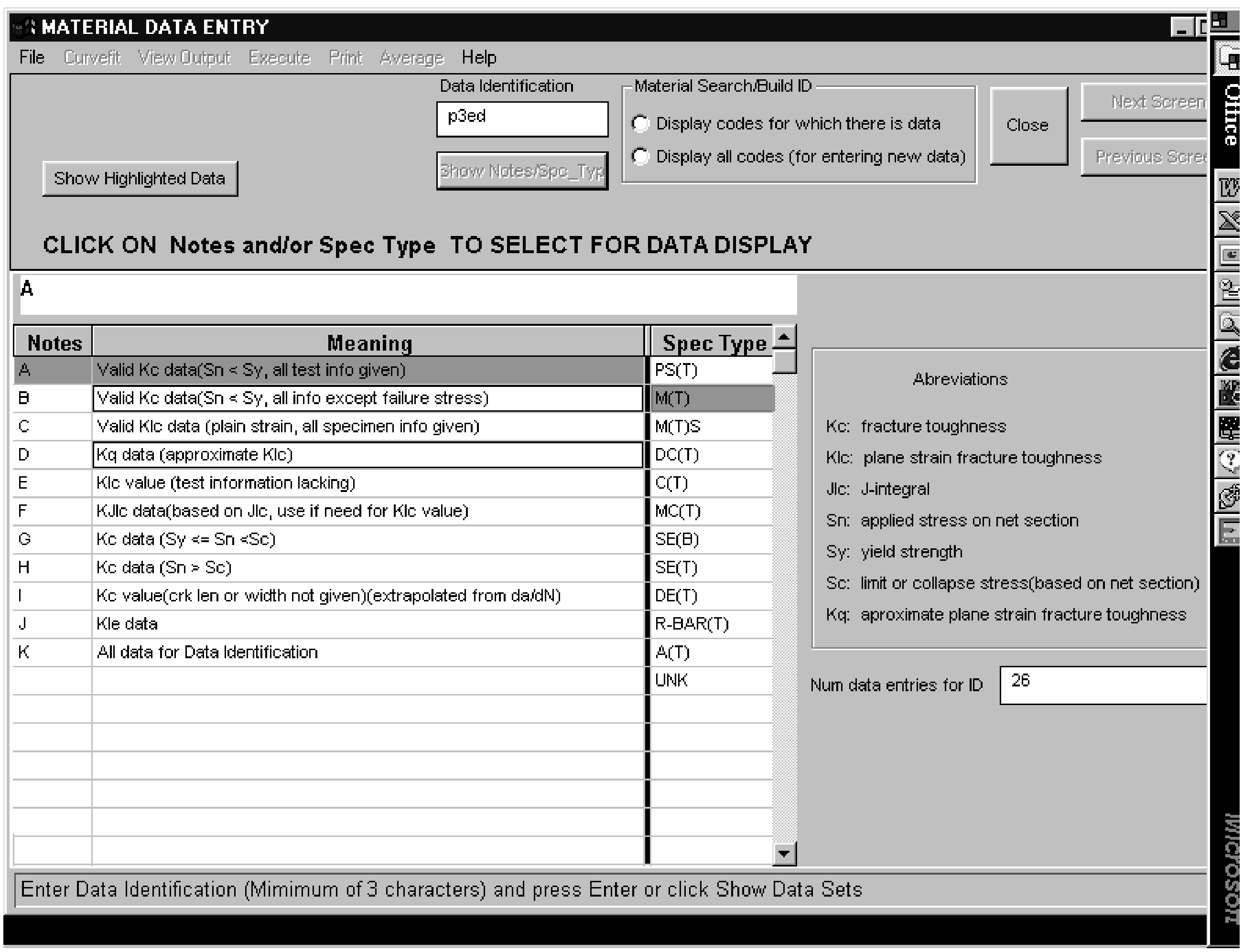
Some of the salient parameters needed to curve fit are: S_{max}/σ_0 ratio, toughness value, constraint parameter α and threshold information. For the S_{max}/σ_0 ratio, a value of 0.3 is commonly used. Additional information about the use of this parameter may be found in section 2.1.2. Discussion on the constraint factor α may be found in sections 2.1.1 and 2.1.2. Since the least squares routine uses all of the data points in the base data set, sometimes it does not provide desirable values of *C* and *n* that work for all the stress ratios. In this case, you will want to input values for *C* and *n* manually. You can also choose to plot upto 9 curves for various R values in addition to the base curve that you want to draw on the plot. For example, if you have data at three different stress ratios and have used the R = 0.1 data as the base set, enter two R values to indicate that you want curves drawn at the other two stress ratios so that you can compare the fit to the data. The stress ratio values for which you want curves drawn are input into text boxes.

You also need to enter the fracture toughness value K_c. You may either enter K_c directly (e.g., if the data are from part-through specimens and K_{Ie} is known) or K_c can be calculated based on K_{Ic}, σ_{ys}, A_k, B_k, and the specimen thickness. Additional theoretical information on fracture toughness may be found in section 2.1.4. There are also text boxes provided to enter values for *p*, *q*. The constants *p* and *q* control the shape of the asymptotes in the threshold and critical crack growth regions, respectively. . A separate grid is provided to enter threshold data for positive and negative stress ratio domains. This data is used to compute the threshold. Additional information regarding the threshold parameters ΔK_0 and R_{cl} may be found in section 2.1.3.

If satisfactory curve fit constants have been obtained, you may obtain a hardcopy of the plot. If the curve fit that was plotted is not acceptable, the program allows you to start over with different parameters or constants and refit.

6.3 Entering and processing the fracture toughness data

Once the fracture toughness database is selected from the opening screen, the next screen prompts for a data ID input. For the NASA database, the ID can be built from a sequence of text boxes by selecting the alloy type etc., Then, by clicking on the notes/specs. button, the user can select the desired criteria and the specimen type for which available data will be displayed. Once the data is displayed, curve fitting to determine the constants in Eq. 2.12 can be done. The fit and the data can also be plotted for comparison purpose. The plot can be redone using different limits on x and y axes. Selected toughness data can also be averaged. The graphical user interface is self-explanatory. It essentially helps to screen the data using various criteria and process it as desired. The screen capture below shows the selection crieteria.



7.0 Boundary Element Method Analysis

This chapter describes the use of the NASBEM (NASA Boundary Element Method) computer program, a tool for fracture mechanics and stress analysis of two-dimensional elastic bodies of arbitrary geometry and loading, with or without cracks, and multiple zones of different materials.

Although transparent to the user, NASBEM consists of three parts: a user-friendly data input interface, the boundary element method (BEM) computational engine which also contains the stress-intensity factor (SIF) calculation code, and a post-processing stress analysis component with optional graphical output. The data input interface and stress analysis code were developed at NASA's Johnson Space Center and their use is described in this chapter. The boundary element computational engine was furnished by researchers at the University of Texas as part of their FADD (Fracture Analysis by Distributed Dislocations) computer code. In this BEM, cracks are modeled by point dislocations and any external and internal boundaries by conventional boundary elements. Please see reference [39] for the theoretical basis.

Most of the code is written in FORTRAN-77, with the only exception being the use of C++ for the graphical user interface. Compilation of the source code requires no special measures; some UNIX systems require a particular compilation flag to enable compiler access to system functions such as DATE( ) and TIME( ). On the HP9000 series workstations, for example, this is the "+e" flag.

7.1 The Data Input Interface

The data input interface for NASBEM is entirely menu-driven. That is, the user is not prompted sequentially for data input; i.e. not asked first to enter the number of materials in the problem, then asked repeatedly for each material to enter the relevant parameters, and so on for each item in the pre-ordained list of input items. Rather, the user chooses from a menu which facet of the problem of interest to enter, e.g. material data, and then enters the relevant parameters one line per material without program prompt. The user can then travel back up the menu tree and choose to enter other facets of the problem of interest. In this fashion the user need enter only those items relevant to the problem of interest in the order he or she desires. One can even quit the program before input of a particular problem's data set is complete (first saving the entered data to file) and resume input at a later time to complete the data set and run the computations. The advantage of this type of input system is mainly to accommodate the many types of geometries, loading types, and crack configurations that can be solved by NASBEM. No two problems may be alike, making a sequential-type input in which one is asked about every possible input item cumbersome.

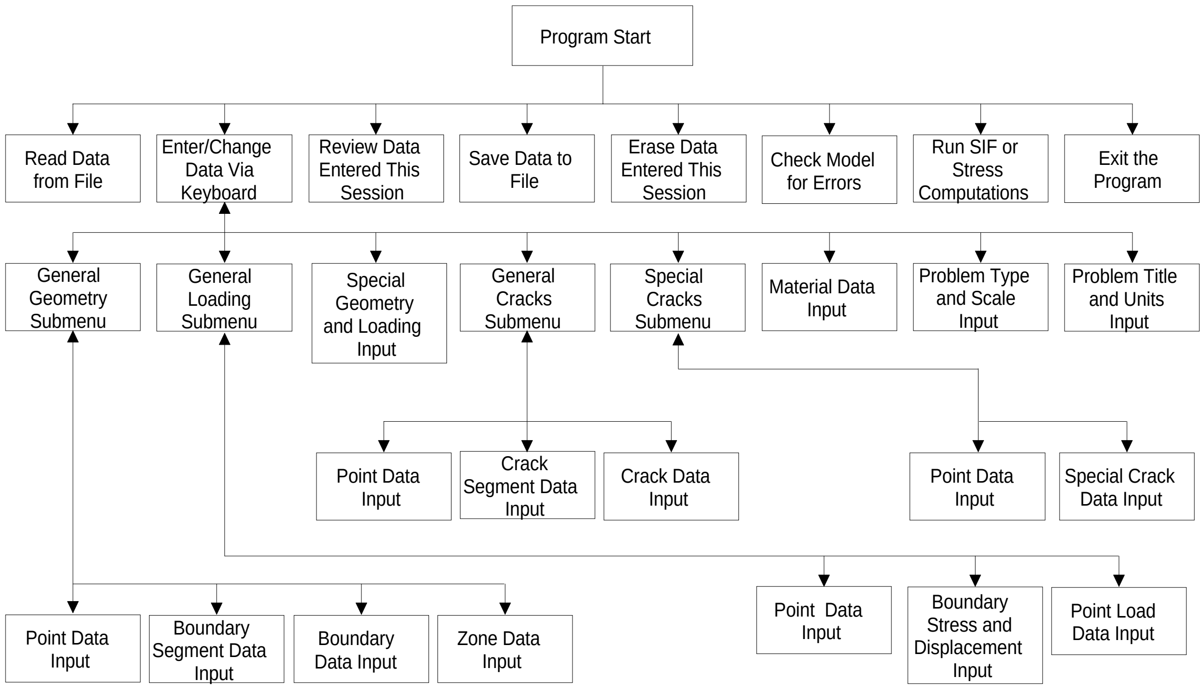


Figure 40 – NASBEM menu tree

The individual menu choices will now be explained in some detail.

7.1.1 The Input Philosophy, or: How to build the model?

Before discussing the menu tree and the individual choices, it would be instructive to describe the input philosophy.

The underlying principle is that of a pyramid. One starts at the bottom by defining the most basic unit, the point, and builds up the geometry from there to define boundary segments from points, boundaries from boundary segments, crack segments from points, cracks from crack segments, and zones from boundaries and cracks. In addition, one attaches materials to zones and boundary conditions (the boundary loads and displacements) to segments. All the aforementioned quantities are assigned id numbers, i.e. numeric labels, by the user to aid in defining which segments are built from which points, which boundaries are built from which segments, etc.

7.1.1.1 Points

Points represent prominent locations in the model:

– boundary segment endpoints and midpoints in the case of linear or curved segments, or endpoints and arc center points in the case of circular arc segments,

- crack segment endpoints (and arc center points in the case of circular arc segments) for the general crack case (see later),
- crack endpoints only for the special crack case (see later),
- concentrated load locations,
- hole centers in the case where hole boundaries are entered as special case boundaries (see later in this section)
- a point within an infinite body (but not on a boundary) where displacements are fixed to eliminate rigid body motion

Note that since points feature in fixing the locations of many input items, point id numbers used in a given input item definition may also be used in another input item. For example, while the definitions of a crack segment and a boundary segment may contain the same point id number, this point id number in both definitions must refer to the same physical point.

7.1.1.2 Boundary segments

Segments connect these points in space to form part of a boundary. A segment is defined as a section of the boundary that shares common or unique geometric and loading features. For example, a linear section of a boundary can be considered a segment, as can be a circular arc section, as long as the boundary conditions along that section of the boundary can be adequately defined by specifying their values at the segment endpoints and midpoint. This means that a linear section of the boundary that has a distributed load acting over only a portion of this section must be split into several linear segments: one spanning the length of the distributed load, and one or more over those stretches without a load. It then follows that discrete boundary conditions (such as pin and roller displacement constraints to eliminate rigid-body solution modes) can be accommodated by defining a short segment, each endpoint (and midpoint) of which is considered subject to the same type of boundary condition. The same principle applies to circular arc sections or partially curved sections.

In the same fashion, boundary slope and boundary discontinuities such as corners/kinks or intersections with edge cracks must fall on segment endpoints.

Available boundary segment types are curve (linear or parabolic) and clockwise/counter-clockwise circular arc; quadratic elements are generated internally with uniform spacing for each segment, where the number of elements per segment governs element size. Some notes about element size and spacing:

- element size near an edge crack intersection should be of the same order of magnitude as those on the crack near the intersection
- if a point is near a boundary, the element size near the point load should be of the same order of magnitude as the distance between the point load and the boundary
- *it is necessary to use denser element spacing on the boundary near stress concentrations and near crack tips, whether the tip is near the crack mouth (i.e. short edge crack) or approaching a boundary.*
- one of the attractive features of this BEM is that it generally requires fewer elements in the mesh than other BEM implementations. However, the user is urged to experiment

with element density and distribution and to perform convergence studies to assure accurate results.

7.1.1.3 Boundaries

Boundaries are built from boundary segments. In this fashion boundaries of arbitrary shape can be assembled from linear, parabolic, and circular arc segments. In the present context a boundary is considered to be a closed line in space marking the edge of a given material. Thus, there are external boundaries, internal boundaries (i.e. holes, circular or other), and boundaries that coincide along one or more shared segments, as in the case of a body consisting of two different materials.

It is possible to have a geometry in which the physical external boundary is so far away from any other geometric feature of interest (e.g. a crack) that entering the actual boundary coordinates is impractical, and for which the loading along such a boundary is uniform. In such a case one can select the problem scale to be one with an infinite external boundary and supply the uniform remote stress, and dispense with entering any other information about the external boundary. In fact the NASBEM input interface makes that distinction in several places, reminding the user that only finite boundaries need be assigned id numbers and counted in zone definitions.

Boundaries must be defined by giving the constituent boundary segment id numbers in consecutive order (important!) while traveling around the boundary, keeping the body to the left. Thus, external boundaries are generally defined in a counter-clockwise manner, while internal boundaries are generally defined in a clockwise manner, although local sense reversals are possible as in the case of a U- or C-shaped indent or bulge in a boundary.

Finally, some points to note are
- although cracks may intersect a boundary, crack segments are not used in the boundary definition
- please see the subsection “Special case geometries” for simplified input of special boundary cases

7.1.1.4 Zones

Zones are regions within the body enclosing a single material. To this end zones are bounded by boundaries, both external and internal, and cracks, both edge and internal. For example, a body consisting of three distinct regions, where two regions of the same material are separated by a region of a different material, would be entered as three zones, with two zones tagged with the one material and the third zone tagged with the other material. All zone interfaces are assumed to be perfectly bonded and cracks are not allowed to lie along or intersect an interface.

Multi-zone problems are useful for modeling stiffened plates, for example, where the stiffener would be entered as a separate zone with its elastic modulus adjusted for the different stiffener thickness relative to the plate.
Multi-zone problems by definition contain boundaries which coincide along one or more stretches. The boundary segments in these stretches need be defined only once; however, the id numbers of these segments will feature in the definition of each of the boundaries concerned. Note that multi-zone geometries are not allowed for problems with an infinite external boundary.

7.1.1.5 Crack segments and cracks

Cracks are built up in the same manner as boundaries; that is, points define crack segments and crack segments define cracks. Some points to be noted:

- while physical cracks have top and bottom faces, this code makes no such distinction and cracks are modeled by a single line of segments only;
- crack segments are different from boundary segments, and while crack segments may intersect boundaries (and therefore boundary segments), boundaries are built from boundary segments only and do not include crack segments
- edge cracks must intersect boundaries at boundary segment endpoints; that is, the point id number specified for the crack segment endpoint representing the edge point must be the same as an id number specified for a boundary segment endpoint. This does not apply to special case hole boundaries, since all segment generation is done internally and not specified by the user
- edge cracks should be defined by specifying the constituent crack segments from the edge (i.e. crack mouth) to the crack tip
- crack segments cannot intersect or coincide with inter-zonal boundary segments
- crack loading conditions are specified as uniform normal and tangential loads over a crack segment
- for edge cracks the crack element and boundary element sizes near the crack/boundary intersection should be of the same order of magnitude
- please see the subsection “Special case geometries” for input of special crack cases

7.1.1.6 Boundary conditions and concentrated (or point) loads

Boundary conditions are specified by giving the type (load or displacement) and value at a boundary segment’s endpoints and midpoint. Components in the *x*- and *y*-directions are used for linear and parabolic segment types, while circular arc segments require normal-tangential components.

The default boundary condition is one of zero load in both component directions, so that only displacement and non-zero load conditions need be specified. Note however that if a condition exists in which one component is a zero load but the other is a displacement or non-zero load, both components must be specified at that point. Regardless of which point on a segment has a user-specified boundary condition, the option to review entered data always displays the boundary conditions at all three segment points.

It should be noted that a body must be fixed in space in order to eliminate rigid body motion from the solution. For finite bodies this can be accomplished by choosing two points on the boundary, one where both x- and y-displacements are set to zero and another point where either x- or y-displacement is set to zero. For infinite bodies this is accomplished by specifying a point within the body where both displacements are set to zero; the boundary element algorithm assumes that rotations at infinity are zero.

Recall that a boundary segment is defined as a section of the boundary along which the geometry and boundary conditions respectively share common or unique features. A boundary segment in fact serves as a tool for a simple mesh generator within the NASBEM computational engine. To this end, the rules for boundary condition (BC) generation along a segment are the following:

– the BC (load or displacement) assigned to a segment midpoint is assigned to all element nodes on that segment except the beginning node (i.e. segment start point) and the ending node (i.e., segment endpoint)
– the BC types assigned to either segment endpoint are valid only for those two nodes corresponding to the segment endpoints
– the BC values for all nodes on a segment (including those two at the segment endpoints) are interpolated from the values entered by the user at all three segment points (midpoint and two endpoints)

This means that

– the BC at a point are valid as one approaches that point along the segment for which the condition is specified; thus it is possible for a point on the boundary that is a segment endpoint to have two BC type combinations, one for the first segment and another for the second segment
– mixed BC (load in one coordinate direction, displacement in the other), such as those used to fix a body in space in order to eliminate rigid-body solution modes, should be examined carefully before being set
– one solution to such a mixed BC problem is to specify one segment as load-load (or displacement-displacement) along its entirety, and to specify the adjoining segment as mixed, with the same mixture of BC types specified on both endpoints and the midpoint
– in this version of NASBEM, mixed BC can be accommodated only on segments which are parallel to one of the coordinate axes, ie not on the following: inclined straight segments, curved segments, or circular arc segments

Care must be taken in how boundary segments are defined in the geometry input section so that boundary conditions are properly accommodated at the segment endpoints/midpoints and properly generated on the elements along the segment. Since elements in a segment are generated internally to be quadratic, so are boundary conditions on those elements. This means that segments should be defined so that boundary conditions on a segment are

symmetric about the segment midpoint or
uniformly increasing or decreasing between segment endpoints.

Any applied concentrated loads are defined independently of boundary conditions by specifying their location (via point id numbers) and their value. Similar to the way in which boundary id numbers and crack id numbers are attached to zone definitions under the Geometry Input menu, any concentrated loads must be attached to zone id numbers under the Boundary Condition Input menu. For point loads close to a boundary, the boundary element size in the vicinity of the point load should be of the same order of magnitude as the distance between the point load and the boundary. (Recall that element size is governed by the number of elements chosen for a particular segment.) Point moments can be simulated by having two point loads acting in opposite directions and separated by a small distance.

7.1.1.7 Materials

This program assumes that all materials are linearly elastic and homogeneous. To this end, the user must simply provide the elastic modulus and Poisson ratio to fully define a material.

7.1.1.8 Special case geometries

The interface allows input of two special case geometric entities to save the user time in entering commonly occurring geometry components.

The first is input of circular hole boundaries with up to two collinear cracks, optional uniform internal pressure, and optional pin (or bearing) loading.

Recall that boundary segments represent sections of the boundary that have unique geometric/loading features and that edge cracks must intersect a boundary at boundary segment endpoints only. A pin load on a hole is commonly approximated by a cosine load distributed along a 180° arc of the circle, centered about the bearing load resultant. Thus, for a first cut, potential boundary segment endpoints are the load arc endpoints as well as the crack/hole intersection(s). However, depending on the relative location of the bearing load and the crack intersection(s), the hole may have to be divided into additional segments in order to adequately capture the boundary conditions. This job of selecting the segment distribution around a hole can quickly become tedious and prone to errors.

Rather than define such holes manually by calculating segment endpoints and boundary conditions, with this special case option one merely specifies the hole location, direction of the pin load resultant, the average bearing stress on the hole (given as P/2rt, the pin load value per unit thickness, divided by the hole diameter), and id numbers of cracks intersecting the hole. NASBEM will calculate the appropriate segment distribution, boundary conditions, and an element distribution within those segments (dependent on the crack-size-to-hole-

diameter ratio, and relative crack(s) and pin load position). Note that special case hole boundaries and ordinary boundaries are the same creatures and differ only in the manner of input. Thus they share the same id numbering system and there should be no duplication of id numbers between the two.
The second special case is that of SIF analysis of a single straight crack for multiple lengths.

Rather than having to define new crack segment endpoints and new cracks for each length, and then running the code for each length, one merely defines a single crack by its endpoints for the initial length, gives the final-to-initial crack length ratio and the number of lengths to calculate, and then sends the job off to the computational part of the code only once. NASBEM will calculate the appropriate crack configurations and tabulate the output as length versus SIF. These results can then be used easily in other modules of NASGRO. Note that since for this case only crack endpoints are needed to define the crack and the crack is straight, the definitions of crack and crack segment are synonymous and separate specification of crack segments is not necessary (or, in fact, allowed).

7.1.1.9 Stress analysis

Stress analysis is specified during the post-processing phase by giving a number of points, either individually or as falling along one of several available line types, at which to calculate stress values. Note that stress analysis is not available in any zone containing a crack, nor for the special case of a single straight crack with multiple lengths. Thus, for the former, one might consider defining multiple zones made of the same material for cases in which one needs both SIF and stress analysis.

7.1.2 The Menu Tree

This section describes the menu tree in some detail.

7.1.2.1 Top level menu

The opening screen of NASBEM presents a menu with the following choices which can be selected at any time during input by traveling up or down the menu tree:

1) **Open existing file** – select this item to read input data from a file created earlier. This file may have been created by the program in an earlier session and the data it contains may or may not be complete. Alternatively, the file might have been created by using a text editor, adhering to a prescribed format (important!); see choice **(4) Save data to file** for more information.
2) **New File** – select this item to start entering data for a new problem. Then one can get into the menu from which to choose the topic (e.g. geometry) for which you wish to enter data.
3) **Save data to file** - select this item to save any data entered this session (keyboard or file) to a file. The data file is written in the following format: input item sections are written in a prescribed order, where each section begins with three comment lines (each starting

with the character #). The first comment line contains a one- or two-word section description, the second comment line contains a description of what is written on the first data line, and the third comment line contains a description of the order in which data items are written on the second and subsequent data lines. Following the comment lines are the section's data lines, where the first data line generally contains the number of items to read (e.g. points) and subsequent data lines contain the information associated with each item (e.g. point id number and coordinates). Note: for those users who wish to manually create a data file, it is advisable to first choose this menu option without having any actual data to save; NASBEM will write a data file that can serve as a template with all the data sections listed in the proper order (important!).

Under execute menu the following two options are available:

1) **Check input data for errors** – select this item to check input data for errors, omissions, and completeness. While it is impossible to check for all possible pathologic situations, NASBEM makes an effort to catch the most common input errors such as boundaries that are not closed, or boundary segments that are not contiguous.
2) **Run the stress-intensity factor and/or stress computations** – select this item when data input is complete and you wish to run the BEM to compute SIFs and/or stresses. Note that this option will also perform the data check of choice (6) before any computations, but that (6) returns you to this top-level menu immediately, errors or not, while (7) does so only if there are input errors. For no errors, choice (7) returns you to the top menu only after the computations are complete.

7.1.2.2 General hints and notes on using the menus and input.

Finally, some general thoughts about entering information:

– one does not need to completely finish entry of some input section (e.g. points) before moving on to some other input section (e.g. crack segments). One can move up and down the menu tree, entering data for different input items at will in any desired order, although one should of course complete the definition of a particular item (a point, for example) before moving on.

– the numeric labels or id numbers given to input items need not be in numerical order; in some cases it may be advantageous to assign id numbers in distinct sets, e.g. 1-7 for points in a certain part of the geometry, 10-19 for points in a different part of the geometry, etc.

– while an entire session's input can be erased, individual input items cannot. Entered data can be corrected by going to the appropriate input menu and re-typing the offending id number and typing the correct parameters. Individual items can be discarded simply by not including their ID numbers in the definitions of other items.

– this system allows one to define many quantities but specify that only a subset of those be used in the definitions of items higher up in the hierarchical order of the input pyramid mentioned earlier. For example, one could define 50 points but only use 20 in the definition of boundary segments, crack segments, special case hole boundaries, etc., leaving 30 point definitions unused. Similarly, in the same session, one could define 14 boundary segments but only use 9 in defining boundaries. This allows one-time entry

of a large data set but quick problem redefinition by re-typing certain item definitions for quick "what if?" studies.
- this program is written in FORTRAN-77, and as such it accepts input data delimited by commas, spaces, or tabs. Required input information is presented in each data input section in columnar fashion with the columns centered about the tab stops (every eight spaces). Thus the user can enter data delimited by single spaces, or (so that data appears on screen under column headings) by multiple spaces or simply by one or two tabs.

7.2 Numerical Limits on Input Items; or, How big can the model be?

The limits on array sizes (and thus problem sizes) are contained in two "include" files: file JBLIM for the input interface, and file FADLIM for the BEM computational engine. Currently these limits and the corresponding parameters are:

– max no. of zones = 4	(maxzon in JBLIM; L0 in FADLIM)
– max no. of boundaries = 5 (4 for Macintosh version)	(maxbnd in JBLIM; L1 in FADLIM)
– max no. of boundaries per zone = 5	(maxbpz in JBLIM)
– max no. of boundary segments = 60	(maxseg in JBLIM)
– max no. of boundary segments per boundary = 12	(maxspb in JBLIM)
– max no. of elements per boundary = 60	(maxepb in JBLIM; N1 in FADLIM)
– max no. of points = 180	(maxput in JBLIM)
– max no. of cracks = 4	(maxcrk in JBLIM; L2 in FADLIM)
– max no. of crack segments = 20	(maxcs in JBLIM)
– max no. of crack segments per crack = 10	(maxspc in JBLIM)
– max no. of elements per crack = 25	(maxepc in JBLIM; N2 in FADLIM)
– max no. of cracks per zone = 4	(maxcpz in JBLIM)
– max no. of materials = 4	(maxmat in JBLIM; L0 in FADLIM)
– max no. of point loads = 6	(maxpl in JBLIM; N3 in FADLIM)

*These can be adjusted by editing the two "include" files FADLIM and JBLIM and recompiling the code; the parameters with the greatest effect on required array storage space and, on some computer systems, executable file size are the products of maxbnd*maxepb (see JBLIM) and L1*N1 (see FADLIM). Note: usable no. of elements per boundary is limit-1.*

7.3 Stress Analysis

The program may be used to compute stresses in geometries that are free of cracks. In the case of multi-zone geometries, stresses can be computed in any of the zones that are crack-free. Special means have been used to overcome the "boundary layer" effect [40], the decay in accuracy in stress and strain as the boundary is approached that is commonly experienced with boundary element techniques. The stress computing algorithms have been tested on several geometries and found to yield acceptably accurate values at all locations within the bodies up to and including their boundaries.

The program allows the specification of points for computing stresses in one of three ways: (a) point by point, (b) straight (or quadratic) line, and (c) circular arc subtending any angle about any point. The output is presented in tabular form or plotted as a function of distance (or angle) along the line (or arc) of interest.

In developing expressions for stresses that are valid for all locations within a body, there are three regions that have been considered: (a) the interior, (b) the boundary, and (c) a narrow boundary layer adjacent to the boundary elements. Different sets of stress expressions are employed for each of these regions. For a point in the boundary layer, stresses are obtained by interpolation between proximal boundary and interior points.

7.3.1 Stresses in the Interior

In the standard Boundary Integral Equation formulation, stresses in the interior of the body are obtained from the strains. The strains are obtained by taking appropriate partial derivatives of the displacements, given by

$$u_\alpha(p) = \int_\Gamma \left[t_\beta(Q) U_{\alpha\beta}(p,Q) - u_\beta(Q) T_{\alpha\beta}(p,Q) \right] dQ \tag{7.1}$$

Here p is the point of interest (in the interior), and Q is a generic point on the boundary Γ; t_β and u_β are the boundary tractions and displacements; and $U_{\alpha\beta}$ and $T_{\alpha\beta}$ are the displacement and traction kernels. The kernels, (with α= 1,2) are given by

$$U_{\alpha\beta} = \frac{1}{8\pi(1-\bar{\nu})G} \left[(-3+4\bar{\nu})\delta_{\alpha\beta} 1nr + r,_\alpha r,_\beta \right] \tag{7.2}$$

$$T_{\alpha\beta} = -\frac{1-2\bar{\nu}}{4\pi(1-\bar{\nu})r} \left[\left\{ \delta_{\alpha\beta} + \frac{2r,_\alpha r,_\beta}{1-2\bar{\nu}} \right\} \frac{\partial r}{\partial n} - r,_\alpha n_\beta + r,_\beta n_\alpha \right] \tag{7.3}$$

where G is the shear modulus and $\bar{\nu}$ equals ν for plane strain (or $\nu/(1+\nu)$ for plane stress) and ν is Poisson's ratio; r is the distance between the source point p and the field point Q, and a comma beside the r denotes a derivative with respect to the corresponding coordinate of the field point; n_β denotes the components of the unit outward drawn normal (to the boundary) at Q and $\partial r/\partial n$ is the derivative of r with respect to this normal; and $\delta_{\alpha\beta}$ is the kronecker delta.

In order to overcome the problem of diminished accuracy near the boundary, a consequence of the singularity in $T_{\alpha\beta}$ as $r \to 0$, an alternate kernel is developed in [39]. This is expressed as

$$W_{\alpha\beta} = \frac{1}{4\pi(1-\bar{\nu})} \left[2(1-\bar{\nu})\varphi\delta_{\alpha\beta} + \varepsilon_{\beta\gamma} r,_\alpha r,_\gamma + (1-2\bar{\nu})\varepsilon_{\alpha\beta} 1nr \right] \tag{7.4}$$

where φ is the angle made by the join of the source and the field points with the x-axis, and $\varepsilon_{\alpha\beta}$ here equals β-α. Interior displacements are shown in [39] to be related to this new kernel by the expression:

$$u_\alpha(p) - u_\alpha(\hat{P}) = \int_\Gamma \left[t_\beta(Q) U_{\alpha\beta}(p,Q) - \frac{\partial u_\beta(Q)}{\partial s} W_{\alpha\beta}(p,Q) \right] ds \tag{7.5}$$

where $\hat{P}$ is the point on the boundary where a line from *p* parallel to the x-axis intersects it. Equation (7.5) allows a means (through its partial derivatives) of obtaining the stresses and is so used in the program. The reduced singularity in $W_{\alpha\beta}$ (as compared to $T_{\alpha\beta}$ allows accurate determination of stresses close to the boundary; the boundary layer is made thinner but not eliminated by these means. In this program, equation (7.5) is used for points farther than 0.15*l* away from the boundary where *l* is the length of the proximal element.

7.3.2 Stresses on the Boundary

Stresses at points on the boundary are obtained from the computed boundary tractions and displacements. Referring to the normal and tangential directions at the boundary point of interest by ξ and η , the stress components $\sigma_{\xi\xi}$ and $\sigma_{\eta\eta}$ are related to the tractions by

$$\sigma_{\xi\xi} = t_1 \cos(1,\xi) + t_2 \cos(2,\xi) \tag{7.6}$$

$$\sigma_{\xi\eta} = t_1 \cos(1,\eta) + t_2 \cos(2,\eta) \tag{7.7}$$

The tangential stress component is given by

$$\sigma_{\eta\eta} = E'\varepsilon_{\eta\eta} + \nu'\sigma_{\xi\xi} \tag{7.8}$$

where E' equals Young's modulus *E* for plane stress ($E/(1-\nu^2)$ for plane strain) and ν' equals ν for plane stress ($\nu/(1-\nu)$ for plane strain). Finally, the tangential strain $\varepsilon_{\eta\eta}$ is related to the boundary displacements by

$$\varepsilon_{\eta\eta} = \cos(1,\eta)\frac{\partial u_1}{\partial u} + \cos(2,\eta)\frac{\partial u_2}{\partial \eta} \tag{7.9}$$

Stress components in the normal and tangential directions are obtained from equations (7.6), (7.7) and (7.8), from which components in the x- and y- directions are readily obtained.

7.3.3 Stresses in the Boundary Layer

For locations in the boundary layer, defined in the implementation as a layer adjacent to the boundary of thickness 0.15l, where l is element length, a process of interpolation between the proximal boundary and interior points is carried out. Note that as l is not, in general, uniform throughout, the thickness of the boundary layer varies as one travels along the boundary.

The process is straightforward if the point of interest is within the exclusive boundary layer of a single element. Complications arise when the point falls in the inter-elemental region common to adjacent elements.

Figure 41 shows the four types of inter-elemental boundary regions that arise depending upon the angle made by element 2 with respect to element 1 at the point of intersection P_0, elements 1 and 2 being approximately equally proximal to the point of interest. Regions I and II refer to parts of the boundary layer where the interpolation is based on boundary points located on either (but not both) of the two elements. However, if the point of interest is within any of the inter-elemental triangles: $\Delta P_0 P_1 P_m$ or $\Delta P_0 P_m P_2$ (Type 1), $\Delta P_0 P_3 P_a$, $\Delta P_0 P_a P_b$, or $\Delta P_0 P_4 P_b$ (Type 3), or $\Delta P_0 P_3 P_m$ or $\Delta P_0 P_m P_4$ (Type 4), triangular interpolation is carried out to compute the stresses, the vertices of these triangles lying either on the boundary or considered to be in the interior. Note that although straight lines are used in Figure 41 to represent the elements and the boundary layer, they are not, in general, straight but are curvilinear instead. The implementation in the program takes this into account.

7.4 Program Output and Files Used

The input interface creates a temporary file BEMDAT.TMP for use by the BEM computational engine. This file is overwritten by NASBEM during every run and may be erased by the user after program termination.

SIF results (when calculated) are echoed to the screen and written to file BEMDAT.OT1 with appropriate header information. Stress-intensity factors are listed with crack numbers and crack sizes. For internal cracks SIF's are listed for first tip and second tip, where "first" and "second" refer to the crack tips in the order they were specified by the user. For the special case of a single-straight crack with multiple sizes, the SIF's are listed for the crack sizes corresponding to the user-specified number of calculation steps.

Stress calculations are echoed to the screen in tabular form and in plots, and written to file BEMDAT.OT2 with appropriate header information. Also written to this file are the displacements and *x*- *y*-tractions (stresses) at each boundary node.

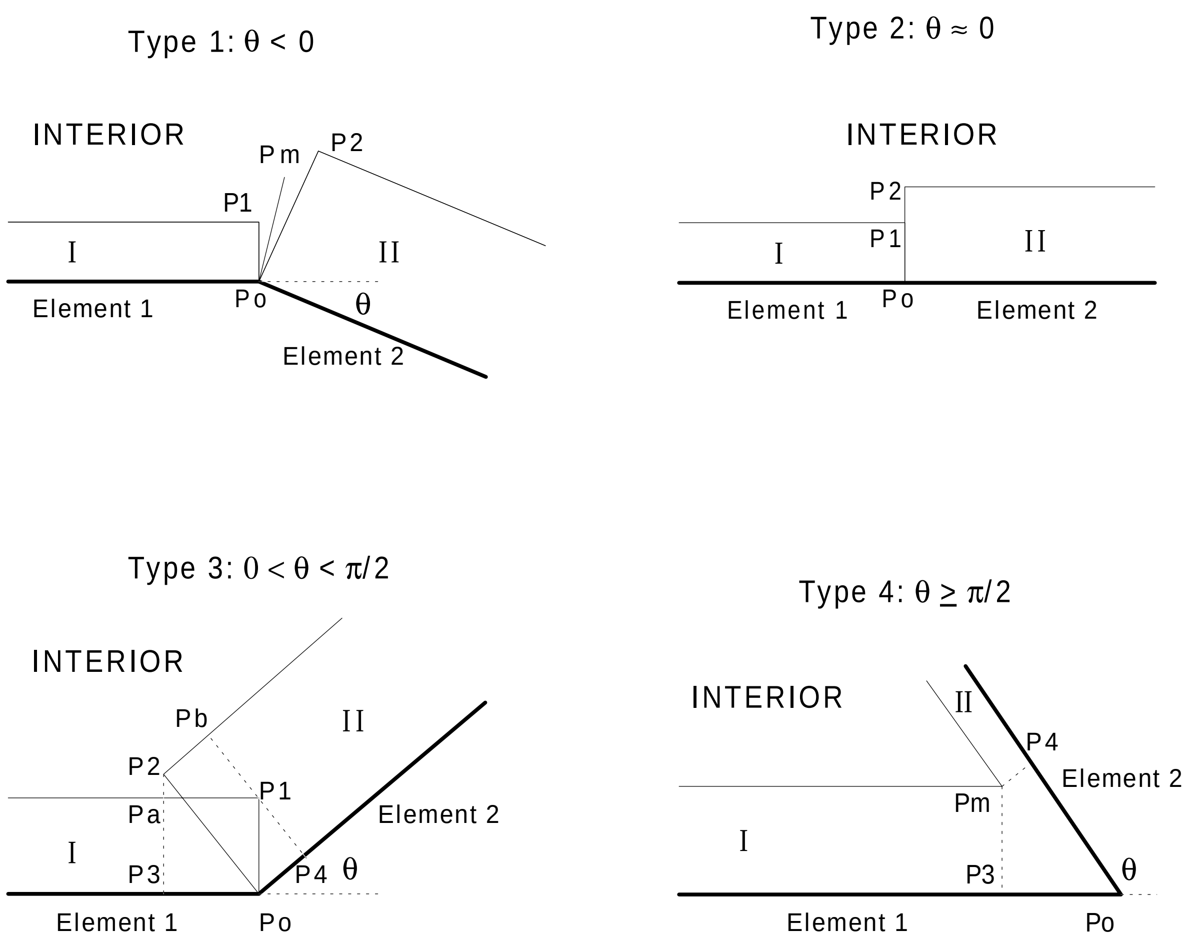


Figure 41 – Element intersection types

7.5 Future Enhancements

Some features to be incorporated into future releases include:

– plotting the geometry or geometry components
– plotting boundary displacements

- allowing stress calculations for the special crack case
- uniting the definitions for crack segments and boundary segments; they are separate at the moment because the boundary segment definition allows for a more general segment type
- allowing edge cracks to intersect interzonal boundary segments
- improving the user-selectability of element spacing for special case boundary holes
- implementing the context-sensitive on-line help system

Acknowledgments

The task of developing NASGRO 3.0 had the participation and cooperation of many individuals, working for NASA, the European Space Agency, the FAA, the USAF, U. S. aerospace contractors, and European aerospace contractors. The development and technical evaluation of this program has been monitored by the NASA Fracture Control Methodology Panel (formed in March 1985) and the Interagency Working Group (formed 1993).

References

[1] Broek, D., *Elementary Engineering Fracture Mechanics*, 4th edition, Nijhoff, 1985.

[2] Broek, D., *The Practical Uses of Fracture Mechanics*, Kluwer Academic Publishers, 1989

[3] Barsom, J. M. and Rolfe, S. T., *Fracture and Fatigue Control in Structures: Applications of Fracture Mechanics*, 2nd edition, Prentice-Hall, Inc., 1987.

[4] Anderson, T. L., *Fracture Mechanics: Fundamentals and Applications*, CRC Press, 1991.

[5] Forman, R. G., and Mettu, S. R., "Behavior of Surface and Corner Cracks Subjected to Tensile and Bending Loads in Ti-6Al-4V Alloy," *Fracture Mechanics: Twenty-second Symposium, Vol. 1, ASTM STP 1131*, H. A. Ernst, A. Saxena, and D. L. McDowell, eds., American Society for Testing and Materials, Philadelphia, 1992, pp. 519-546.

[6] Newman, Jr., J. C., "A Crack Opening Stress Equation for Fatigue Crack Growth," *International Journal of Fracture*, Vol. 24, No. 3, March 1984, pp. R131-R135.

[7] Newman, Jr., J. C. and Raju, I. S., "Prediction of Fatigue Crack-Growth Patterns and Lives in Three-Dimensional Cracked Bodies," *Advances in Fracture Research (Fracture 84)*, Sixth International Conference on Fracture, Vol 3, 1984, pp. 1597-1608.

[8] Tanaka, K., Nakai, Y., and Yamashita, M., "Fatigue Growth Threshold of Small Cracks," *International Journal of Fracture*, Vol. 17, No. 5, October 1981, pp. 519-533.

[9] Lindley, T. C., "Near Threshold Fatigue Crack Growth: Experimental Methods, Mechanisms, and Applications," *Subcritical Crack Growth Due to Fatigue, Stress Corrosion, and Creep*, L. H. Larsson, ed., Elsevier Applied Science Publishers, New York, pp. 167-213, 1985.

[10] Schmidt, R. A. and Paris, P. C., "Threshold for Fatigue Crack Propagation and Effects of Load Ratio and Frequency," *Progress in Flaw Growth and Fracture Toughness Testing*, ASTM STP 536, American Society for Testing and Materials, Philadelphia, 1973, pp. 79-94.

[11] Yoder, G. R., Cooley, L. A., and Crooker, T. W., "A Critical Analysis of Grain-Size and Yield Strength Dependence of Near Threshold Fatigue Crack Growth in Steels," *Fracture Mechanics: Fourteenth Symposium -- Vol. 1: Theory and Analysis, ASTM STP 791*, J. C. Lewis and G. Sines, eds., American Society for Testing and Materials, Philadelphia, 1983, pp. 348-365.

[12] Vroman, G. A., "Material Thickness Effect on Critical Stress Intensity", Monograph #106, TRW Space & Technology Group, February 1983.

[13] Forman, R. G. and Henkener, J. A., "An Evaluation of the Fatigue Crack Growth and Fracture Toughness Properties of Beryllium-Copper Alloy CDA172," NASA-TM-102166, 1990.

[14] Henkener, J. A., Lawrence, V. B., and Forman, R. G., "An Evaluation of Fracture Mechanics Properties of Various Aerospace Materials," *Fracture Mechanics: Twenty-Third Symposium, ASTM STP 1189*, R. Chona, ed., American Society for Testing and Materials, Philadelphia, 1993, pp. 474-497.

[15] Orange, T. W., Sullivan, T. L., and Calfo, F. D., "Fracture of Thin Sections Containing Through and Part-through Cracks," *Fracture Toughness Testing at Cryogenic Temperatures*, ASTM STP 496, American Society for Testing and Materials, Philadelphia, 1970, pp. 61-81.

[16] Willenborg, J., Engle, R.M. and Wood, H.A., "A Crack Growth Retardation Model Using an Effective Stress Concept," AFFDL-TM-71-1-FBR, Wright Patterson Air Force Laboratory, January 1971.

[17] Gallagher, J.P.,., "A Generalized Development of Yield Zone Models," AFFDL-TM-74-28-FBR, Wright Patterson Air Force Laboratory, January 1974.

[18] Gallagher, J.P., Hughes, T.F.., "Influence of Yield Strength on Overload Affected Fatigue Crack Growth Behavior in 4340 Steel," AFFDL-TR-74-27-FBR, Wright Patterson Air Force Laboratory, February 1974.

[19] Newman, J.C. Jr., Private Communication, 1995.

[20] Brussat, T. R., Private communication, May 1997.

[21] Chang, J.B. and Engle, R.M. "Improved Damage-Tolerance Analysis Methodology," Journal of Aircraft, Vol. 21, 1984, pp. 722-730.

[22] Ten Hoeve, H.J. and de Koning, A.U., "Implementation of the Improved Strip Yield Model into NASGRO Software - Architecture and Detailed Design Document," National Aerospace Laboratory (NLR) Report: NLR CR 95312L, 1995.

[23] Paris, P.C., Gomez, M.P., and Anderson, W.P., “ A Rational Analytic Theory of Fatigue,” The Trend in Engineering, Vol. 13, 1961, pp. 9-14.

[24] Elber, W., “The Significance of Fatigue Crack Closure,” Damage Tolerance of Aircraft Structures, ASTM STP-486, 1971, pp. 230-242.

[25] Dugdale, D.S., “Yielding of Steel Sheets Containing Slits,” Journal of Mechanics and Physics of Solids, Vol. 8, 1960, pp. 100-104.

[26] Lawrence, V. B. and Forman, R. G., “Structure and Applications of the NASA Fracture Mechanics Database,” *Computerization and Networking of Materials Databases: Third Volume, ASTM STP 1140*, American Society for Testing and Materials, Philadelphia, 1992.

[27] Hudson, M. C. and Seward, S. K., “Compendium of Sources of Fracture Toughness and Fatigue Crack Growth Data for Metallic Alloys,” *International Journal of Fracture*, Vol. 14, 1978, pp. R151-R184.

[28] Hudson, M. C. and Seward, S. K., “Compendium of Sources of Fracture Toughness and Fatigue Crack Growth Data for Metallic Alloys,” *International Journal of Fracture*, Vol. 20, 1982, pp. R59-R117.

[29] Gallagher, J., *Damage Tolerant Design Handbook*, University of Dayton Research Institute, prepared for Air Force Wright Aeronautical Laboratories, Wright-Patterson Air Force Base, Ohio, December 1983.

[30] Henkener, J. A., Lawrence, V. B., Williams, L. C., and Forman, R. G., “Derivation of Crack Growth Properties of Materials for NASA/FLAGRO 2.0,” JSC 26254, NASA Lyndon B. Johnson Space Center, Houston, Texas, June 1994.

[31] E 616-89, “Standard Terminology Relating to Fracture Testing,” *1993 Annual Book of ASTM Standards*, Vol. 3.01, American Society for Testing and Materials, Philadelphia, 1993, pp. 647-658.

[32] Brodeur, S. J. and Basci, M. I., “Fracture Mechanics Loading Spectra for STS Payloads,” AIAA-83-2655-CP, 1983.

[33] Wiederhorn, S. M. and Roberts, D. E., “Fracture Mechanics Study of Skylab Windows,” National Bureau of Standards Report No. 10892, May 1972.

[34] Wiederhorn, S. M., Evans, A. G., Fuller, E. R., and Johnson, H., “Application of Fracture Mechanics to Space-Shuttle Windows,” American Ceramic Society, July 1974, pp. 319-323.

[35] Wiederhorn, S. M., "Dependence of Lifetime Predictions on the Form of the Crack Propagation Equation," *Fracture 1977*, Vol. 3, 1977, pp. 893-901.

[36] Schwartzberg, F. R., Kiefer, T. F., and Rothermel, L. J., "Strength and Fracture Behavior of MDA Window Materials," MCR-73-27, Final Report for Contract NAS8-24000, Martin Marietta Corporation, Denver, Colorado, January 1973.

[37] Henkener, J. A. and Forman, R. G., "Fatigue Crack Growth and Fracture Toughness Data for Selected Space Systems Structural Alloys," JSC 24976, NASA Lyndon B. Johnson Space Center, Houston, Texas, March 1991.

[38] SSP 30560, "Glass, Window, and Ceramic Structural Design and Verification Requirements," National Aeronautics and Space Administration, Space Station Freedom Program Office, Reston, Virginia, June 1991.

[39] Chungchu Chang, "A Boundary Element Method for Two Dimensional Linear Elastic Fracture Analysis", Ph.D. Dissertation, The University of Texas at Austin, December, 1993.

[40] Ghosh, N., Rajiyah, H., Ghosh, S., and Mukherjee, S., 1986, "A New Boundary Element Method Formulation for Linear Elasticity", *Journal of Applied Mechanics*, Vol. 53, pp. 69-76.