



Executive summary

Improved fatigue spectrum derivation and crack-growth analysis for pressurised spacecraft structures

Recent developments of the ESACRACK software

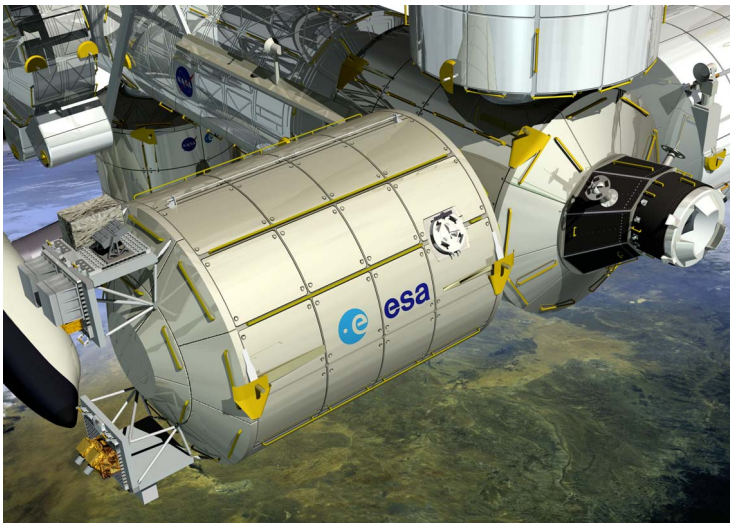


Photo: ESA

Problem area

Fracture control of spacecraft, launchers and their payloads and experiments is of paramount importance for the safety and reliability of manned and unmanned space operations. Fatigue, fracture and safety critical spacecraft structural components like pressurised structures are to be verified on the basis of damage tolerance requirements to prove their reliability under service loading conditions.

The damage tolerant design method for spacecraft structures and pressurised systems implies that the structure is designed such that

stable crack growth will not become unstable (or cause e.g. leakage) during the service life (safe-life), or redundancy is introduced in the structure such that a part of the load path may fail without loss of overall structural integrity (fail-safe).

Damage tolerance for structures can be verified either by test or by analysis. The numerical tools ESA developed and distributed since the mid-eighties to verify the damage tolerance of spacecraft structures are available as the software package ESACRACK. ESACRACK can be used for fatigue spectrum generation and fracture control analysis.

Report no.

NLR-TP-2009-177

Author(s)

R.P.G. Veul
F.P. Grooteman

Report classification

UNCLASSIFIED

Date

April 2009

Knowledge area(s)

Descriptor(s)

load spectra
fatigue
crack growth

This report is based on a presentation held at the Workshop on Fracture Control of Spacecraft, Launchers and their Payloads and Experiments, ESA/ESTEC, Noordwijk, The Netherlands, 9-10 February 2009.

Improved fatigue spectrum derivation and crack-growth analysis for pressurised spacecraft structures

Recent developments of the ESACRACK software

ESACRACK consists of the modules:

- ESALOAD (Ref. 1): for fatigue spectrum derivation;
- NASGRO[®] (Ref. 2): a crack-growth analysis software originally developed by NASA, (with significant contributions made by ESA; its development is currently supported by the NASGRO Consortium);
- ESAFATIG (Ref. 3): for fatigue analysis of metallic structures.

This paper describes the recent developments to the ESACRACK 4 package performed within ESA contract “Structural Integrity of Pressurised Structures” (contract number 14923/00/NL/PA).

Description of work

A number of new developments to this package is made addressing various aspects of structural integrity and fracture control of pressurised structures (this includes hardware under pressure, but also payloads inside pressurised modules of the Space Shuttle and International Space Station).

These new developments, as listed in this paper, can be found in the field of

- New modules and capabilities for analysis of pressurised structures, like the rotational acceleration and interface displacements, residual stresses from welding, new Stress Intensity Factor solutions, and a

- simple more conservative low-cycle fatigue behaviour;
- Development of advanced user interfaces to increase the program user-friendliness;
- Modernisation and improvements to the data storage and handling capabilities, like improved integration of NASGRO[®].

The values of the coefficients of the NASGRO2 crack growth equation strongly depends on the opening factor, which itself depends on the crack growth model used.

Therefore, in a curve fit the same opening model should be applied to calculate the opening factor as used in the crack growth analysis.

Otherwise large deviations are possible due to mismatch of the applied model and coefficients. This is true for any crack growth equation which depends on the opening factor.

Results and conclusions

Improvements to fatigue spectrum derivation and crack growth analysis algorithms for pressurized spacecraft structures are implemented, tested and described. These improvements are available to the ESACRACK users community.

Applicability

The ESACRACK software package can be used for damage tolerance analysis of spaceflight vehicles and payloads as well as ground support equipment.



NLR-TP-2009-177

Improved fatigue spectrum derivation and crack-growth analysis for pressurised spacecraft structures

Recent developments of the ESACRACK software

R.P.G. Veul and F.P. Grooteman

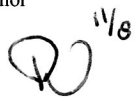

This report is based on a presentation held at the Workshop on Fracture Control of Spacecraft, Launchers and their Payloads and Experiments, ESA/ESTEC, Noordwijk, The Netherlands, 9-10 February 2009.

The contents of this report may be cited on condition that full credit is given to NLR and the authors.

This publication has been refereed by the Advisory Committee AEROSPACE VEHICLES.

Customer	ESA
Contract number	14923/00/NL/PA CCN11
Owner	NLR
Division NLR	Aerospace Vehicles
Distribution	Unlimited
Classification of title	Unclassified
	July 2009

Approved by:

Author  11/8	Reviewer  11/8	Managing department ① 12/8/2009
--	--	------------------------------------

Summary

The European Space Agency (ESA) has developed and distributed since the mid-eighties the software package ESACRACK. A number of new developments to this package is made in the framework of an ESA study addressing various aspects of structural integrity and fracture control of pressurised structures (this includes hardware under pressure, but also payloads inside pressurised modules of the Space Shuttle and International Space Station). Significant improvements in terms of user-friendliness and technical capabilities have been achieved. In addition to the modifications of the software, additional guidelines and examples are developed, e.g. for application of the strip-yield crack-growth retardation model and development of fatigue spectra for Space Shuttle payloads.

Contents

1	Introduction	7
2	ESALOAD	7
2.1	Load curve enhancements	8
2.2	Unit stresses per event	10
2.3	New stress spectrum generation process	10
2.4	Command file option	12
2.5	Extending user friendliness	13
3	ESAFATIG	14
3.1	Low-cycle fatigue behaviour	14
3.2	Database update	15
4	NASGRO®	16
4.1	Residual stresses from welding	16
4.1.1	Residual stress distribution	16
4.1.2	Effects on the Stress Intensity Factor K	18
4.1.3	Effects on crack growth	18
4.1.4	Relaxation of residual stresses by application of overloads	18
4.2	New stress intensity factors	19
4.2.1	Case TC20: Edge crack loaded by a pin load	19
4.2.2	Case TC21: Crack loaded by a variable series of pin loads	22
4.2.3	T-section beam	24
4.3	Crack growth law curve fit	26
5	Conclusions	30
	References	31

Abbreviations

A	area
A, B, C, P	ESAFATIG S-N curve fit constants
B	distance from hole centre to plate edge
c	crack length
C, n, p, q	material parameters in NASGRO2 crack growth equation
CA	constant amplitude
D	hole diameter
ESA	European Space Agency
ESACRACK	ESA fatigue and damage tolerance software package
ESAFATIG	ESA fatigue analysis software
ESALOAD	ESA fatigue spectrum derivation software
f	opening factor K_{op}/K_{max} ,
F_o^c	correction for the presence of free surfaces, such as finite width
H	pitch
i	index to linear or rotational accelerations, pressure, or temperature
I	moment of inertia
ISPR	International Standard Payload Rack
ISS	International Space Station
K	stress intensity factor
\tilde{K}	K factor solution for the infinite width configuration.
K_c	fracture toughness
K_{max}	maximum stress intensity factor
M	moment
N	mean load
NASGRO®	crack-growth analysis software
P	pin load
R	stress ratio of the minimum to the maximum stress, i.e. $R = S_{min}/S_{max}$
RL	distance from centre where residual stress is zero
Rx, Ry, Rz	rotational accelerations
S_0	tension/compression stress
S_1	bending stress in thickness direction
S_2	bending stress in width direction
S_3	pin load stress on the crack surface
S_a	amplitude stress
S_m	mean stress

S_{\max}	maximum stress
t	plate thickness
t_1	thickness of flange of T-beam
t_2	thickness of web of T-beam
W	plate width
w_1	width of flange of T-beam
w_2	width of web of T-beam
x, y, z	linear accelerations
ΔK	stress intensity range
ΔK_0	threshold stress intensity for $R=0$
ΔK_{th}	the threshold stress intensity range
α_{new}	constraint factor
ε	fraction of the uncracked ligament from outer boundary at which bending stresses are calculated
$\sigma_{\text{res}}(x)$	residual stress
σ	unit stress.
σ_{add}	mean additional stress
σ_c	constant stress due to preload
$\sigma_{\text{exp,m}}$	maximum experimental stress applied in the tests
σ_m	mean stress
σ_o	maximum residual stress at the centre of the weld seam ($x=0$)
σ_{ult}	ultimate stress
σ_{yield}	yield stress

1 Introduction

Nowadays, fatigue, fracture and safety critical components like pressurised structures are to be verified on the basis of damage tolerance requirements to prove their reliability under service loading conditions.

The damage tolerant design method for spacecraft structures and pressurised systems implies that the structure is designed such that stable crack growth will not become unstable (or cause e.g. leakage) during the service life (safe-life), or redundancy is introduced in the structure such that a part of the load path may fail without loss of overall structural integrity (fail-safe).

Damage tolerance for structures can be verified either by test or by analysis. The numerical tools ESA developed and distributed since the mid-eighties to verify the damage tolerance of spacecraft structures are available as the software package ESACRACK. ESACRACK can be used for fatigue spectrum generation and fracture control analysis

ESACRACK consists of the modules:

- ESALOAD (Ref. 1): for fatigue spectrum derivation;
- NASGRO[®] (Ref. 2): a crack-growth analysis software originally developed by NASA, (with significant contributions made by ESA; its development is currently supported by the NASGRO Consortium);
- ESAFATIG (Ref. 3): for fatigue analysis of metallic structures.

This paper describes the recent developments to the ESACRACK 4 package performed within ESA contract “Structural Integrity of Pressurised Structures” (contract number 14923/00/NL/PA).

2 ESALOAD

ESALOAD is a software program that has been developed for the European Space Agency (ESA) to create a representative stress spectrum at the location of interest on a specific part of a space vehicle structure. Derived stress spectra can then subsequently be used in crack growth and/or fatigue analyses (using NASGRO[®] and/or ESAFATIG).

Figure 1 gives an overview of the load and stress spectrum derivation in ESALOAD. On the basis of a selected event history, indicating the total number of handling, transport, test and flight events, a fatigue load spectrum can be generated. Predefined load curves (available in data

bases), or calculated load curves, based on knowledge of the structural transmissibility and load input, can be used for the individual events.

The resulting load spectrum, defining all loads seen by a system or subsystem during its lifetime, can be transformed into a stress spectrum using unit stresses, i.e. stresses derived from knowledge of the structural response to unit loads.

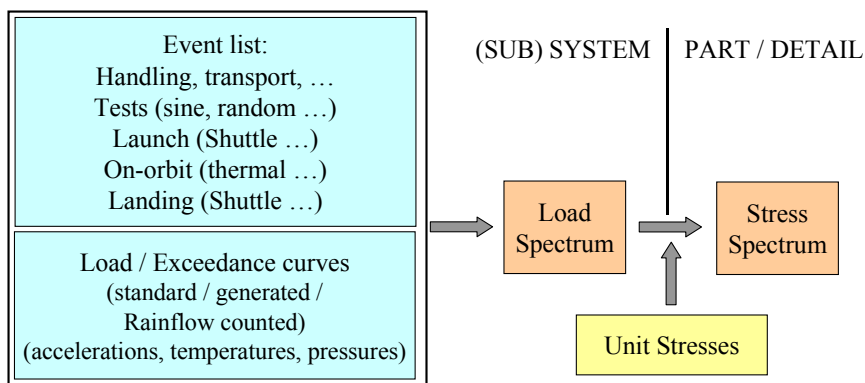


Fig. 1 Overview of ESALOAD load and stress spectrum derivation

In the frame of the ESA contract “Structural Integrity of Pressurised Structures”, various improvements have been made to the ESALOAD program. The major modifications concern:

- event generation and modification;
- load curve enhancements;
- transmissibility and load input definitions;
- unit stresses; and
- stress spectra generation.

Some minor modifications to the ESALOAD program are performed as well, like the possibility to define the maximum acceleration value used in the random load curve generation process, a reply of the effective (program used) Rainflow filter size, and the possibility to select multiple items. In all modifications, compatibility with previous versions is preserved.

Some of the major improvements are described in the following sections.

2.1 Load curve enhancements

The load curve consists of mean and alternating values of linear accelerations, pressure and temperature. This content has been extended to include rotational accelerations and a stress factor.

The possibility to include rotational accelerations was deemed necessary, because they are in many cases defined as part of the loading environment for larger payloads.

The stress factor on additional stresses (see also section 2.2) was introduced for representation of prescribed interface displacements (but can be used to serve other purposes as well). For example: a standard way to store payload is to use the International Standard Payload Rack (ISPR), mounted inside a module of the International Space Station (ISS) (see figure 2). The mechanical connection between this module and the ISPR has a degree of redundancy of two. This means that forces and stresses during launch and landing are not only related to the induced accelerations at the ISPR centre of gravity, but also to the displacements (or deformation) at the interface points between the module and the ISPR due to the coupled stiffnesses. Analysis has shown that these forces can contribute up to about 50% of the total interface force. Similar phenomena may be significant at other interfaces, e.g. internal interfaces of the integrated ISPR.

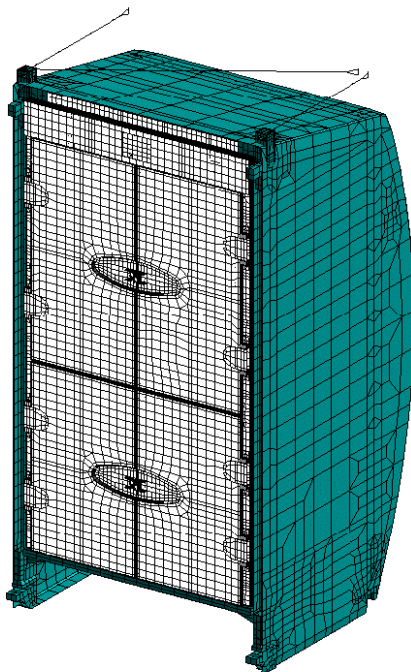


Fig. 2 Example of International Standard Payload Rack.

One possible way to overcome this, is to include enforced displacements at the interface points, which would require the definition of unit stresses due to applied unit interface displacements. However, special care has to be taken to combine the dynamic loads due to these displacements with the centre of gravity accelerations. Another way to overcome this, and this is implemented in ESALOAD, is by means of location dependent additional stresses, caused by the enforced

interface displacements. The stress factor, defined for each line of the load curve, is applied on the additional stress, which is part of the unit stress definition (see section 2.3).

2.2 Unit stresses per event

Unit stresses are defined as stresses (tension, bending, pin bearing, etc.) at the location of interest due to a unit load on the specific part for which load curves are derived. In ESALOAD, one set of unit stresses was applicable to all load events, whether transport, test, shuttle ascent, etc.

In reality the unit stresses may be different for different events, due to e.g. different configurations (boundary conditions) during transport, test, launch/landing and/or on-orbit, or the fact that a local mode is important for the response of the structure due to random loads. Also, the preload and the newly introduced additional stresses may differ between launch, landing or other phases of the life of a structure (where they may be e.g. zero). Therefore, one major modification was to change ESALOAD with the capability to define unit stresses per event.

As mentioned in section 2.1, angular accelerations are included in the load curve. Therefore it is also necessary to extend the unit stress definitions with these three rotations.

Location dependent additional stress input is added to represent e.g. enforced interface deformation due to coupling with adjacent structures. A maximum and minimum stress can be defined in the unit stress definition menu, see figure 3. The additional stress variation, multiplied by the stress factor (see section 2.3), which is generally different for each line of a load curve, is considered as alternating stress in the stress spectrum derivation. The preload in the last line of figure 3 can be used to define a constant average stress. The additional stress can be used if one wants to define mean and alternate stresses that vary with the stress factor.

2.3 New stress spectrum generation process

The process that is used to create a stress spectrum from the combination of a load spectrum and unit stresses in ESALOAD had to be changed considerably due to the significant number of newly added load components in the load spectrum.

Unit Stress Association File Management

Selected Unit Stress File :

Unit stresses [MPa] due to :

	Tension	Bending1	Bending2	Point	Biaxial
X-acceleration	<input type="text" value="5.1"/>	<input type="text" value="2.4"/>	<input type="text" value="0.0000"/>	<input type="text" value="0.0000"/>	<input type="text" value="0.0000"/>
Y-acceleration	<input type="text" value="3.1"/>	<input type="text" value="1.5"/>	<input type="text" value="0.0000"/>	<input type="text" value="0.0000"/>	<input type="text" value="0.0000"/>
Z-acceleration	<input type="text" value="0.8"/>	<input type="text" value="1.2"/>	<input type="text" value="0.0000"/>	<input type="text" value="0.0000"/>	<input type="text" value="0.0000"/>
Pressure	<input type="text" value="290."/>	<input type="text" value="33."/>	<input type="text" value="0.0000"/>	<input type="text" value="0.0000"/>	<input type="text" value="0.0000"/>
Temperature	<input type="text" value="0.0000"/>	<input type="text" value="0.0000"/>	<input type="text" value="0.0000"/>	<input type="text" value="0.0000"/>	<input type="text" value="0.0000"/>
X-acc. rotation	<input type="text" value="0.5"/>	<input type="text" value="0.25"/>	<input type="text" value="0.0000"/>	<input type="text" value="0.0000"/>	<input type="text" value="0.0000"/>
Y-acc. rotation	<input type="text" value="0.5"/>	<input type="text" value="0.25"/>	<input type="text" value="0.0000"/>	<input type="text" value="0.0000"/>	<input type="text" value="0.0000"/>
Z-acc. rotation	<input type="text" value="0.5"/>	<input type="text" value="0.25"/>	<input type="text" value="0.0000"/>	<input type="text" value="0.0000"/>	<input type="text" value="0.0000"/>
Min-add. stress	<input type="text" value="0.0000"/>	<input type="text" value="0.0000"/>	<input type="text" value="0.0000"/>	<input type="text" value="0.0000"/>	<input type="text" value="0.0000"/>
Max-add. stress	<input type="text" value="0.0000"/>	<input type="text" value="0.0000"/>	<input type="text" value="0.0000"/>	<input type="text" value="0.0000"/>	<input type="text" value="0.0000"/>
Preload	<input type="text" value="0.0000"/>	<input type="text" value="0.0000"/>	<input type="text" value="0.0000"/>	<input type="text" value="0.0000"/>	<input type="text" value="0.0000"/>

Fig. 3 Part of unit stress definition window.

The unit stresses are multiplied by the load factors of the load spectrum. However, it is clear that multiple loads, i.e. loads from different directions, may occur at the same time, but no information about their relative phase exists. As an assumption, stress components due to acceleration components (both linear and rotational) are evaluated in all possible combinations. This is not done for stresses due to pressure and temperature. Therefore only alternating acceleration components should normally be combined in one load line (mean loads of all types can be combined). This should not be a problem since they are normally related to different time constants.

Because of the inclusion of angular accelerations, a new stress generation procedure had to be developed and implemented in ESALOAD. Dependent on the load curve input, a stress spectrum generation processes will be performed. On a line-by-line basis the load curves are evaluated. When only linear accelerations and/or temperature and/or pressure are defined, i.e. rotational accelerations and stress factor are zero or undefined, the already existing stress spectrum generation process is used. When rotational accelerations and/or a stress factor are also defined, the “new stress spectrum generation process” is performed. This latter process is new to ESALOAD.

The load curve information consists of mean and alternating loads. By multiplying this with the associated unit stresses, mean and alternating stresses are determined.

The mean stresses are defined by:

$$\sigma_m = \sum_i [N_i \sigma_i] + \sigma_{add} + \sigma_c \quad (1)$$

In which N is the mean load and the σ unit stress. The i represents linear or rotational accelerations, pressure, or temperature. σ_c is the constant stress due to preload and σ_{add} is the mean additional stress that is determined by the mean of the maximum and minimum additional stress defined in the unit stress definitions, multiplied by the stress factor. This stress factor is defined in percentages (80 means 80%) and in ESALOAD the additional stress is multiplied by 80/100.

The alternating stresses are defined by:

$$\Delta\sigma_j = \sum_i [\pm \Delta N_i \Delta\sigma_i] \pm \Delta\sigma_{add} \quad (2)$$

in which the \pm sign indicates that the term $N_i \sigma_i$ is either added to or subtracted from the existing terms. Because there are seven independent variables (6 accelerations and the additional stress), this results in a maximum of 128 alternating stresses (so $j = 1, \dots, 128$). The alternating additional stress is determined by multiplying the amplitude of the alternating additional stress with the stress factor from the load curve.

The introduction of the rotational accelerations and the stress factor increases the possible sign combinations from 8 (x, y and z) to 128 (x, y, z, Rx, Ry, Rz , additional stress) combinations for the alternating stress. This could cause an undesired increase in lines of the stress spectrum. In the next step, these 128 combinations are reduced to 4 lines using the approach already available in ESALOAD: find the maximum and minimum absolute values of these combinations, and divide this range into three equal ranges to come to 4 different alternating stress levels in the stress spectrum. The number of cycles assigned to each stress level is equally distributed over these stress levels, starting from the maximum stress.

2.4 Command file option

A command file option has been implemented in ESALOAD. This command file makes it possible to reduce the time spent on opening database and user files, automate the spectrum generation process, and it provides better traceability of how the spectra are derived.

The command file is an ASCII file that contains the list of actions for ESALOAD to open all required files and the actions to perform a load and stress spectrum generation. This file allows

to automatically open a predefined list of ESALOAD input files (i.e. unit stresses, event file, load curve databases or transmissibility files), to select output files (i.e. load or stress spectrum) and to execute a combination of runs (load and/or stress spectrum generation with or without normalisation and compression) following a given order.

ESALOAD allows to create, edit and execute a command file thanks to a specific user interface, which facilitates the selection of files and definition of commands.

The import time domain data, the load curve generation and the event generation commands cannot be performed during the command file execution.

2.5 Extending user friendliness

The load curve modification and generation via the Graphical User Interface has been greatly improved in ESALOAD. For creating a new load curve (for instance for pressure or thermal tests) it was necessary to introduce load curves by hand in a text file or directly in the user load curve database file or project load curve database file. This is now possible from within the ESALOAD program. Additionally, adding the capability to modify load curves not only by column, but also by row or single element, and the possibility to add and delete rows increases the load curve data modification possibilities.

ESALOAD has the built-in capability to calculate load curves based on structural behaviour (transmissibility), test load input and flight load input. These transmissibility, test load input definition or flight load input definition are stored in files. The ESALOAD program has been changed to handle user-defined filenames, instead of using a predefined filename, and to change the contents of this file. Therefore it is also possible to use a different file for each analysis (project).

The ESALOAD user interface has been improved by increasing robustness but also providing new facilities as the multiple items selection throughout all ESALOAD lists.

User interface of flight data generation to event file has been improved when selecting/deselecting curves.

A facility to view and edit resulting load and stress spectrum files is now possible throughout ESALOAD.

In addition to the modifications of the software, additional guidelines and examples are developed for generation of fatigue spectra for Space Shuttle payloads.

3 ESAFATIG

The computer program ESAFATIG (Ref. 3) can be used to calculate the fatigue life of metallic structural components, based on a linear damage accumulation method. Stress spectra created with the ESALOAD module of ESACRACK can be used for the analysis. The material database consists mainly of data from MIL-HDBK-5F. Additional data for various threaded fasteners have been included.

3.1 Low-cycle fatigue behaviour

The ESAFATIG program is based on S-N curves that are fitted through experimental data. For high stress levels (related to fatigue life smaller than approximately 10^4 cycles) usually no experimental data is available and therefore the calculated number of cycles is obtained via extrapolation.

This extrapolation leads in general to non-conservative results. Therefore, it was decided to implement a simple more conservative low-cycle fatigue behaviour in ESAFATIG, as follows (see figure 4):

- If S_{\max} exceeds or is equal to the ultimate strength, then the number of cycles is set to 1
- If S_{\max} exceeds the yield strength, then a warning is issued.
- If S_{\max} exceeds the maximum experimental stress applied in the test, then a warning is issued.

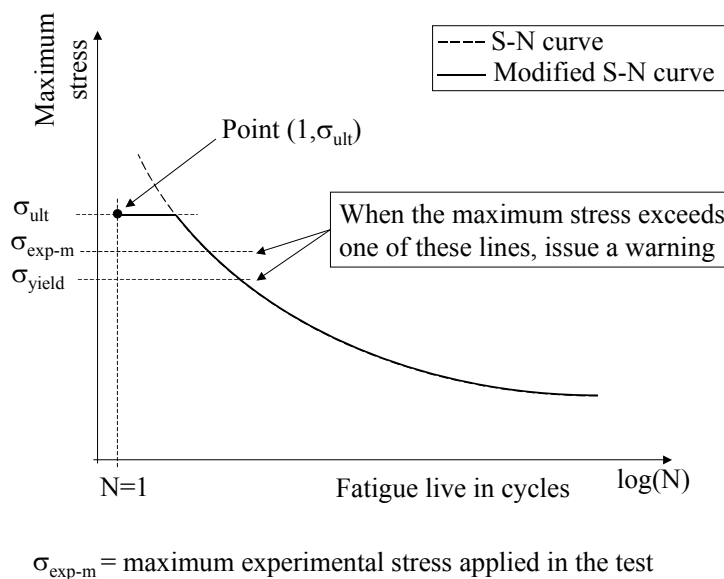


Fig. 4 Implemented low-cycle fatigue behaviour in ESAFATIG

This implementation is in line with the discussion in reference 4. Schijve mentions that a second horizontal asymptote occurs at the upper side of the S-N curve. The material will fail in the first cycle when the maximum stress equals or exceeds the ultimate stress, but a slightly smaller stress level can survive a number of cycles in the order of 100 or more. This is due to strain hardening.

The modifications made to ESAFATIG are not intended to make it reliable for low cycle fatigue analysis. It makes however the user more aware of its limitations: when stress levels are applied that exceed the test range used in fitting the S-N curve or when the yield stress is exceeded. In the output file this information is provided for every stress level. Whether the results are sufficiently accurate, despite the warnings, has to be judged by the user. This depends on the margin obtained, whether only few cycles experience plasticity, whether the situation is strain-controlled (e.g. at notch), whether 'shake-down' is expected, etc.

In the case where reversed plastic deformation is expected for a significant part of the life, it is recommended to perform a proper low cycle fatigue analysis. A low cycle fatigue analysis module is included in NASGRO[®] 4.

3.2 Database update

The S-N database has been updated to include yield and ultimate stress and the maximum stress that is applied in the test (i.e. the stress parameters related to the smallest life obtained during the tests). Additionally, the S-N database is also updated to reflect MIL-HDBK-5J. For some materials the curve fit constants are updated, new material data are included, and ESAFATIG is modified such that it can handle both equivalent stress formulations used in MIL-HDBK-5J.

These formulations are:

$$\log N_f = A - B \log(S_{\max}(1 - R)^P - C) \quad (3)$$

$$\log N_f = A - B \log(S_a + PS_m - C) \quad (4)$$

where A , B , C and P are the material curve fit constants, R is the stress ratio of the minimum to the maximum stress, i.e. $R = S_{\min}/S_{\max}$, S_a is the amplitude stress, and S_m is the mean stress.

4 NASGRO®

NASGRO® (Ref. 2) is a suite of computer programs that can be used to:

- calculate stress intensity factors, compute critical crack sizes, or conduct fatigue and sustained load crack growth analyses based on fracture mechanics principles. Material properties for crack growth can be selected from a large database supplied with the program;
- solve complex two-dimensional geometries with or without cracks to obtain stress intensity factors and stresses via the boundary element method;
- enter, edit and curve-fit fracture toughness and fatigue crack growth data obtained in a laboratory.

Improvements to the NASGRO module include residual stresses from welding and three new stress intensity factors. In addition to the modifications of the software, additional guidelines are developed, e.g. for application of the strip-yield crack-growth retardation model.

4.1 Residual stresses from welding

As a result of the arc welding process and the subsequent cooling cycle(s) residual stresses are introduced near the welds. The formation of these residual stress fields is complicated and, most certainly, has to be added to other residual stresses resulting from machining, rolling, etc.

In a previous version of ESACRACK (Ref. 5) a residual stress option was implemented. It has not been available in the Windows version until now. It will be introduced again, with some small modifications:

- relaxation of the residual stress is performed assuming ideal plastic material properties, using either the yield strength from the materials data base, or a user-defined value (previously, the Ramberg-Osgood formulation was used, which requires an additional parameter);
- residual stresses are implemented in more cracked models than previously (TC06 to 08, SS01, SC02 to 05 are added to SC01 and TC01).

The main features of the residual stress implementation are briefly described below.

4.1.1 Residual stress distribution

Residual stresses are considered on a macroscopic level (also called residual stresses of the first kind). It has been assumed that these stresses are oriented in weld line direction. Residual stresses perpendicular to the weld line are assumed to be one order of magnitude smaller and, therefore, can be safely ignored.

In figure 5 a welded coupon is shown. The coordinate system and the relevant residual stress component are indicated in the section A-A'. The residual stresses $\sigma_{res}(x)$ are assumed to be of the following type

$$\sigma_{res}(x) = \sigma_o \left[1 - (x/RL)^2 \right] e^{-0,5 (x/RL)^2} \tag{5}$$

The maximum $\sigma_{res}(x)$ is assumed to be at the centre of the weld seam ($x=0$): σ_o . The zero crossing is at $x = \pm RL$. Eq. 3 is in principle valid for an infinite width W ($\sigma_{res}(x)$ tends to zero for infinite x ; equilibrium is not fully satisfied when W is finite). $\sigma_{res}(x)$ is constant in thickness direction. RL and σ_o are user-defined. If σ_o is not exactly known, it may be conservatively set equal to the yield strength: in case of high applied loads the residual stress will be reduced. In general, the approach is considered to be on the conservative side, especially since the beneficial effect on the stress intensity factor of compressive residual stresses, when the crack length exceeds $2RL$, is neglected.

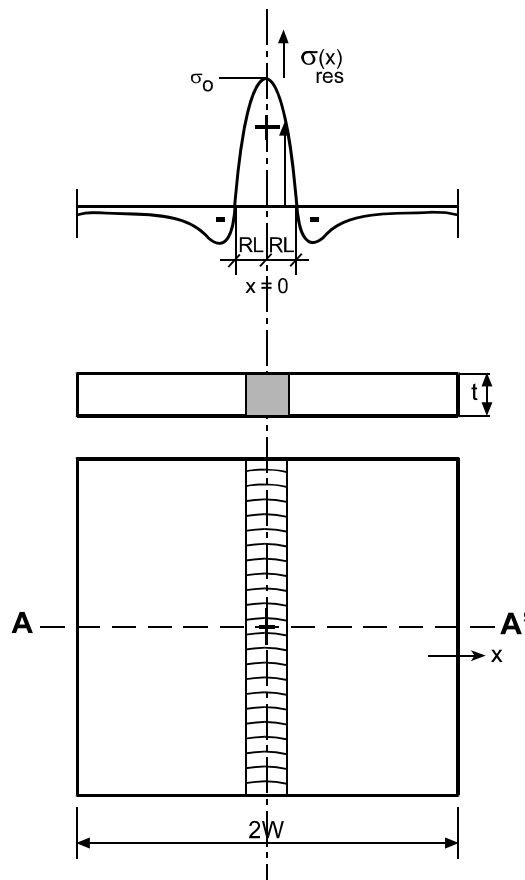


Fig. 5 Welded coupon with coordinate system and residual stress distribution

4.1.2 Effects on the Stress Intensity Factor K

It is assumed that linear elastic (fracture) mechanics apply, therefore the Stress Intensity Factor induced by the residual stresses (K_{res}) can be simply added to the Stress Intensity Factor induced by other loading systems.

In a first step K_{res} is evaluated for a plate of infinite width. The effect of the residual stress distribution is determined by numerical integration over the crack surface.

To this infinite width solution, a finite width correction is applied in the following way:

$$\begin{aligned} K &= F_o^c S_o \sqrt{\pi c} + \text{the effect of other load systems} \\ &= F_o^c K^{\sim} + \text{the effect of other load systems} \end{aligned} \quad (6)$$

where F_o^c is a correction for the presence of free surfaces, such as finite width. K^{\sim} is the K factor solution for the infinite width configuration.

4.1.3 Effects on crack growth

The presence of residual stresses has an effect on K_{max} and the load ratio R , but not on $\Delta K = K_{max} - K_{min}$. The crack growth laws of NASGRO generally include R to account for crack closure.

In view of the complicated effect in an application of the Strip-Yield model it is not allowed for the user to include residual stresses in combination with this model.

4.1.4 Relaxation of residual stresses by application of overloads

The introduction of the yield strength implies that yielding and residual stress relaxation occurs if high loads are applied. In the present implementation of the residual stresses it is assumed that the yielding behaviour of the welded material is ideal plastic. In the original model the Ramberg-Osgood stress-strain curve was used. However, the corresponding material parameters are not present in the material databases. Therefore, it was decided to switch to ideal plastic behaviour and give the user the choice to use the yield strength given in the materials files as typical values (for the base or weld material) or a user specified value. Note that the use of a too low value of the yield limit may lead to non-conservative results.

With this assumption yielding occurs when

$$\sigma_o + S_{\max}^{(1)} \geq \sigma \quad (7)$$

In that case the parameter σ_o in the description of the residual stresses is set to the value σ_o^{new} according to

$$\sigma_o^{new} = \sigma_y - S_{max}^{(1)} \quad (8)$$

In Eq. 5 and 6, $S_{max}^{(1)}$ is the first component in the loading system, which represents remote tensile loading (bending will not give relaxation through the complete thickness and is therefore not considered).

Due to this relaxation of the residual stress when high loads are applied, it is not necessary to consider this stress in the net section stress calculation to determine the residual (ductile) strength.

4.2 New stress intensity factors

Three additional stress intensity factor (SIF) solutions were implemented in the NASGRO module of ESACRACK. The configurations are¹⁾:

- TC20 represents a single edge through-the-thickness crack, loaded in tension/compression (S_0), bending in thickness direction (S_1), bending in width direction (S_2), and pin load on the crack surface (S_3). See figure 6.
- TC21 simulates a through-the-thickness crack growing through a riveted or bolted lap joint, starting from a hole and growing in one direction, loaded in tension/compression and pin loads. See figure 9.
- TC22 represents a simple T-section beam, loaded in tension/compression and bending, where a single through-the-thickness crack starts in the top of the web and grows into both flanges (base part). See figure 12.

4.2.1 Case TC20: Edge crack loaded by a pin load

The configuration for TC20 represents a through-the-thickness edge crack loaded by a pin load and is shown in figure 6. It is identical to a through-the-thickness crack at the edge of a plate (TC02), except that a pin load is added.

The stress intensity factor K for this model equals the stress intensity factor of TC02 (edge crack), superimposed by the K for a single edge through crack under a point load. This K is derived in reference 6, page 2.29. Additionally the SIF for a correction moment to balance the distributed load is applied.

¹⁾ These new stress intensity factors are currently only available in NASGRO 3.

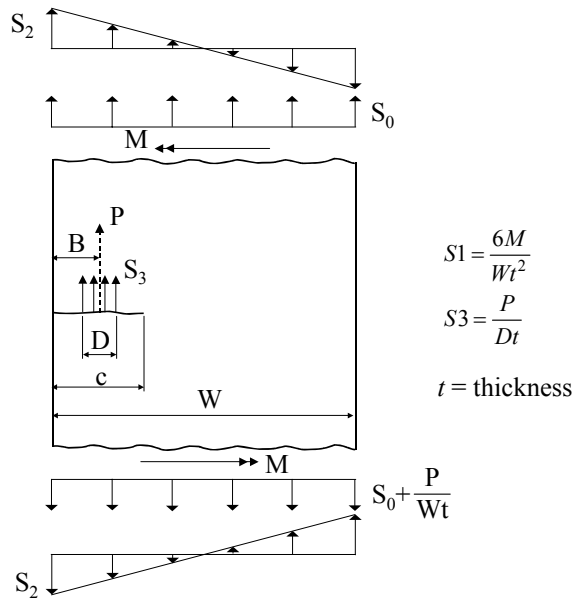


Fig. 6 Pin load acting on an edge crack in a finite width sheet specimen (TC20)

The net section stress definition for TC20 consists of the net section stresses of TC02 (see Appendix B of Ref. 2), superimposed with the pin load S_3 to arrive at the net section stress of TC20. This additional terms σ_{a3} and σ_{b3} are the stress caused by S_3 due to a reduction of the cross-sectional area and bending stress caused by S_3 due to asymmetry of the crack location, and are defined as:

$$\sigma_{a3} = S_3 \frac{D}{W - c}$$

$$\sigma_{b3} = S_3 \frac{6D}{(W - c)^2} \left(\frac{W + c}{2} - B \right) (1 - 2\varepsilon) \tag{9}$$

in which ε is the fraction of the uncracked ligament from outer boundary at which bending stresses are calculated (set to 0.13, see Appendix B of Ref. 2).

Under the assumption that the effect of the hole on the crack growing out of the hole, while the other side is already cracked, can be neglected, this SIF solution allows continuing the crack growth calculation of a through crack from an offset hole in a plate (TC03), for spectra containing pin loads (S_3), see figure 7.

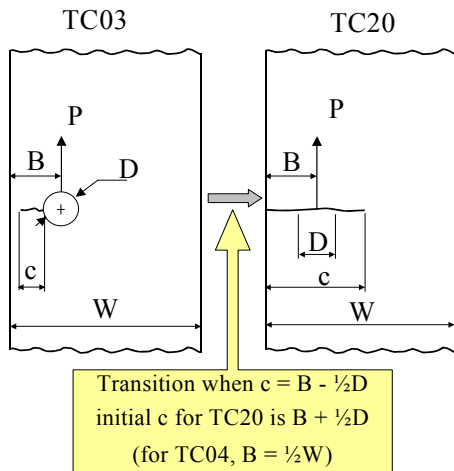


Fig. 7 Continuation of TC03 to TC20

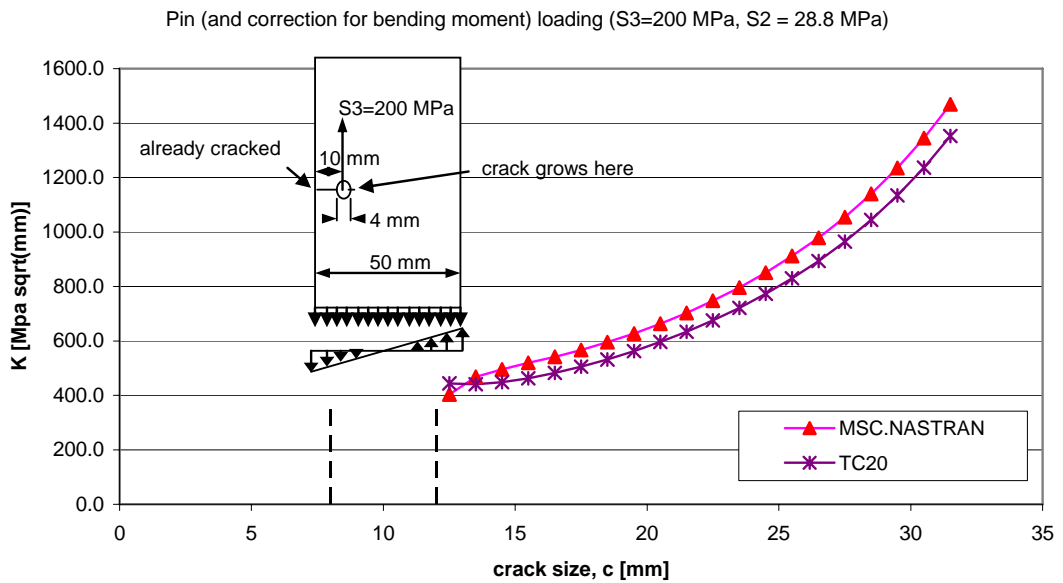


Fig. 8 SIF comparison for MSC.Nastran and TC20 results - pin loading

A finite element model was created of a plate with a width of 50 mm, thickness of 1 mm, hole diameter of 4 mm at a distance of 10 mm from the edge. The MSC.Nastran stress intensity factor results for the pin loading case are compared with the TC20 results in figure 8. The small deviation between the MSC.NASTRAN results and the TC20 result is due to the modelling of the crack tip element in NASTRAN discovered recently, causing the NASTRAN result to be conservative.

4.2.2 Case TC21: Crack loaded by a variable series of pin loads

The TC21 configuration represents a through the thickness crack loaded by a variable series of point loads (i.e. extending over a number of fastener holes), see figure 9. In the implementation of the SIF solution it is assumed that the crack tip on the one side stays at the edge of the rivet hole (no growth at this tip). For the crack tip on the other side it is assumed that the crack jumps over the diameter of the rivet when the crack tip enters a rivet hole. For simplicity, it is also assumed that the holes have no effect on the crack growth.

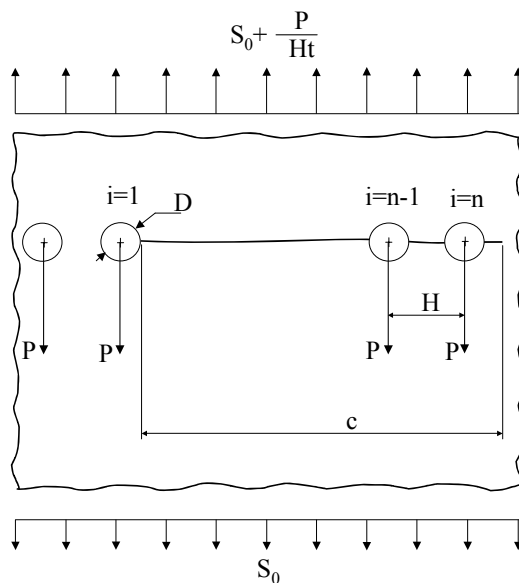


Fig. 9 Crack growth through an area where pin loads are acting (TC21)

The stress intensity factor for this geometry is the superposition of the effect of remote tension loading S_0 via the existing SIF solution of a through-the-thickness crack at the centre of a plate (TC01), and the SIF of the effect of one or more rivet (point) loads acting on the crack surface. The latter is obtained from the solution of a point load on upper surface of a crack (Ref. 7, case 4.4). A correction with the TC05 solution for a crack growing between the first two holes is applied to the TC21 solution.

Taking into consideration the above assumptions, this SIF solution allows continuing the crack growth calculation of a through crack from a hole in a plate with a row of holes (TC05) beyond a fastener hole, see figure 10.

A finite element model of a plate with holes was created to validate the TC21 implementation. The results are shown in figure 11 for both remote and pin loading. This figure shows that the TC21 result is identical to TC05 result between the 1st and 2nd hole, due to the applied TC05 correction, accurate between 2nd and 3rd hole and conservative beyond the 3rd hole.

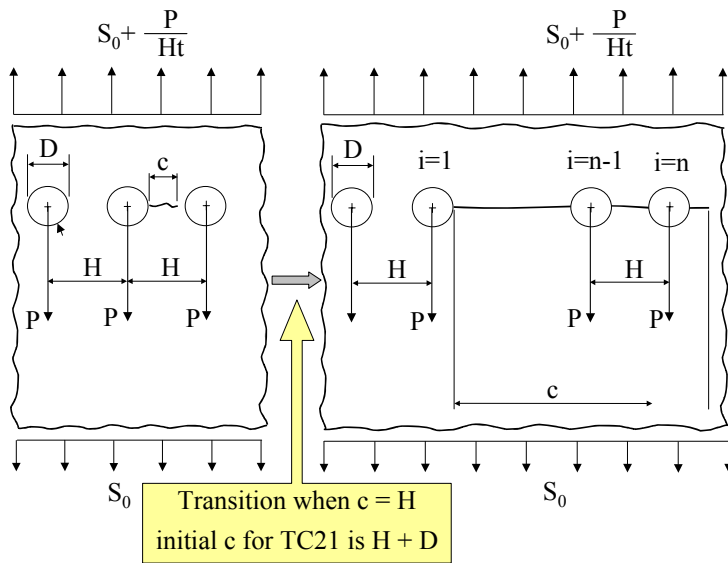


Fig. 10 Continuation of TC05 to TC21

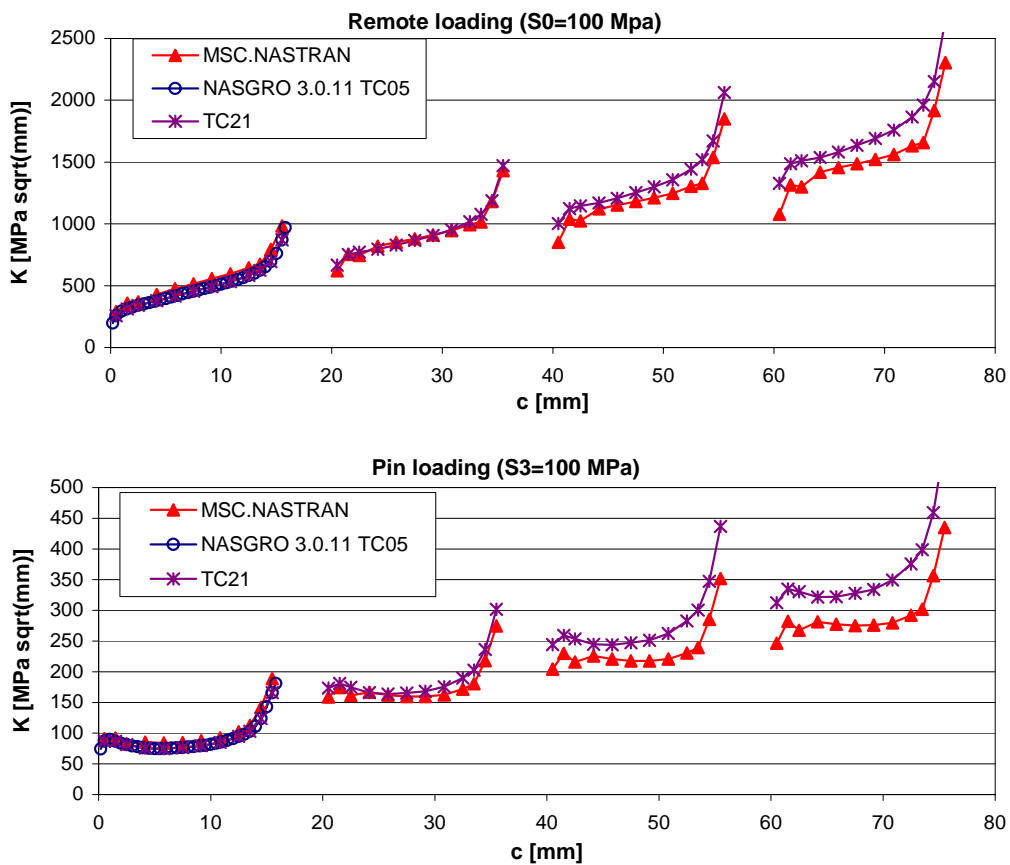


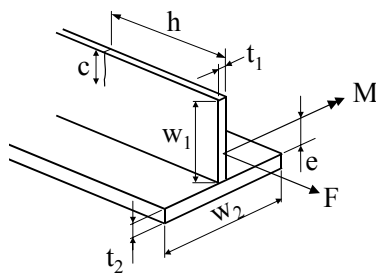
Fig. 11 SIF comparison for MSC.Nastran, TC05, and TC21 results for remote loading (top) and pin loading (bottom)

4.2.3 T-section beam

For this model it is assumed that the crack starts in the single leg of the T-section (further called the web). When the web is completely cracked, the crack continues in the top part of the T, growing in both directions (further called flange). With this SIF solution the fracture mechanics calculation does not have to be performed for an isolated flange (by means of e.g. TC02), which may be very conservative.

The derivation of the stress intensity factor is split in two parts (figure 12). The first part covers the edge crack in the web; the second part covers the centre crack in the flange.

Part 1: edge crack in web



Part 2: cracked web and crack in flange

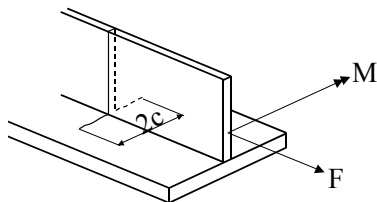


Fig. 12 Crack growth in a T-shaped beam accounting for load redistribution in flange and base parts (TC22)

Part 1:

The stress intensity factor for the first part is identical to TC02 with the stress components S_0 and S_2 as input parameters. An additional bending restriction correction of the flange is added since the TC02 solution implemented in NASGRO is based on a plate for which bending is *not* restrained. These stress components are defined by

$$\begin{aligned}
 S_0 &= \frac{F}{A} + \frac{M}{I} \left(\frac{w_1}{2} - e \right) \\
 S_2 &= \frac{Mw_1}{2I}
 \end{aligned}
 \tag{10}$$

Part 2:

It is assumed that after transition to the centre crack in the flange configuration the initial crack size c is equal to $\frac{1}{2}t_1$. Further, it is assumed that the two parts of the cracked web are uniformly loaded in tension and that the load that is introduced in the cracked web is completely redistributed towards the flange at some distance from the crack. This means that no load is transferred at the mid crack area. Under these conditions, the stress intensity factor solution of a through the thickness centre crack due to tension and bending in thickness direction is applicable for this configuration.

The stress intensity factor solution is validated by finite element analyses for 10 different configurations of web and flange geometry. The FEM calculated values (solid lines) of the geometry factor due a tensile load is compared to the values predicted by TC22 (symbols) in figure 13. The left figure shows the results for a crack in the web and the right figure for a crack in the flange. The accuracy for a crack in the web is better than 7% for any configuration. The accuracy for a crack in the flange is better than 8% for all configurations as long as $t_1/w_2 < 0.5$. For (rather unrealistic) configurations with $t_1/w_2 > 0.5$ the solution is (much) more inaccurate and should not be used.

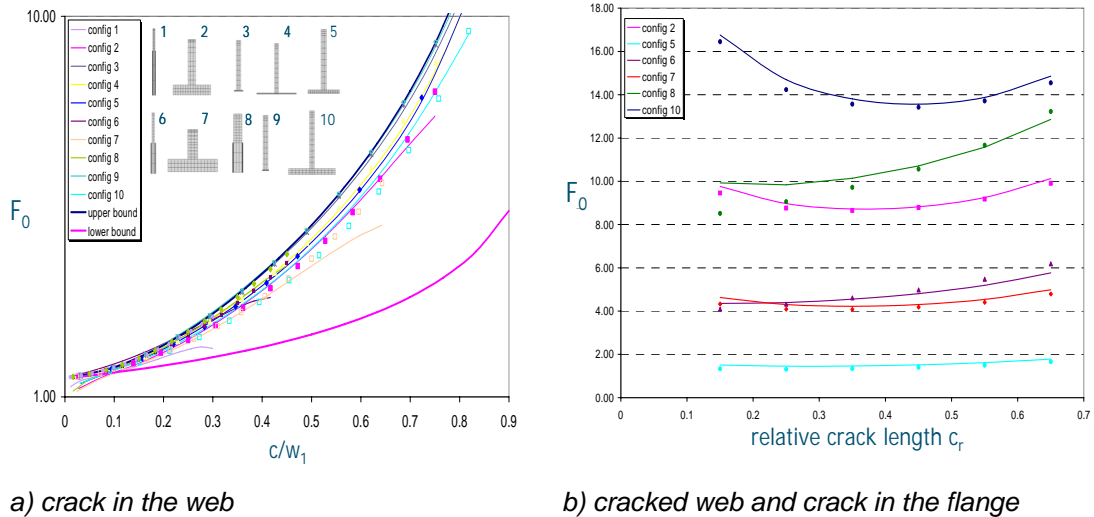


Fig. 13. FEM calculated geometry factors F_0 (solid lines) for different T-beam configurations compared to NASGRO predictions (symbols) due to a tensile load

4.3 Crack growth law curve fit

ESACRACK contains several crack growth models and a number of crack growth laws. The most important one is the NASGRO2 crack growth equation:

$$\frac{da}{dn} = C \left(\frac{1-f}{1-R} \Delta K \right)^n \frac{\left(1 - \frac{\Delta K_{th}}{\Delta K} \right)^p}{\left(1 - \frac{K_{max}}{K_c} \right)^q} \quad (11)$$

where C , n , p and q are assumed to be material parameters, R the stress ratio, f the opening factor K_{op}/K_{max} , ΔK the stress intensity range, K_{max} the maximum stress intensity, K_c the fracture toughness and ΔK_{th} the threshold stress intensity range given by:

$$\Delta K_{th} = \frac{4}{\pi} \Delta K_0 \arctan(1 - R) \quad (12)$$

where ΔK_0 is the threshold stress intensity for $R=0$.

The unknowns in the crack growth law are: C , n , p , q , f , ΔK_0 and K_c . They are determined by fitting the above relation to a set of experimental crack growth data. In general, the crack growth data is based on constant amplitude loading (CA) on a standardised specimen. From the raw experimental a versus n data the da/dn and ΔK can be derived. Often this is done by means of the secant method.

NASGRO comes with a number of databases containing the values for C , n , p , q , ΔK_0 and K_c for various material types depending on condition and environment. However, a major shortcoming is that the NASGRO material databases lack the original raw experimental data (a versus n) on which the fit is based. The consequence is that a refit of the coefficients is impossible, which might be necessary as will be shown in here. A material database facility for structural engineering has been developed for ESA called FRAMES-2 (Ref. 8). FRAMES-2 stores test data as well as derived data and allows the user to manipulate these data.

The NASRGO2 equation can be easily rewritten into a linear equation in terms of C , n , p , q , by taking the logarithm. Based on this equation only C , n , p and q can be determined by the curve fit procedure. This means that the material properties ΔK_0 , K_c and f have to be specified up front and cannot be determined by the fit. Ideally, a good estimate should be obtained for both ΔK_0 and K_c from the asymptotes of the tails in a da/dn versus ΔK plot of the experimental data.

However, in general none or only a few experimental results will lie in these tails and therefore require specially designed experiments to obtain proper values. Lack of data in the tails prevents a good estimate of the coefficients p and q as well and, in general, a value should be specified by the user upfront. When no data points are located in the tail(s), the user should select a low value for p (e.g. 0.25) and a high value for q (e.g. 0.75) to be conservative. Both values should lie between 0 and 1.

In general, for the opening factor f , the Newman crack opening function (Ref. 2) is used. However, in case of the Strip-Yield model this often results in a fit which looks accurate in a da/dn versus ΔK plot, but which is incapable to reproduce the original experimental crack growth curve and can lead to large errors in life prediction, as depicted by the red line in figure 14. This is caused by the difference in the crack opening factor as computed by the Strip-Yield model. The fit is very sensitive for small changes in this factor.

The opening factor can change with crack length for the Strip-Yield model (blue-line in figure 15), while this is constant for the Newman equation (red-line in figure 15), although for CA loading the opening factor computed with Strip-Yield is more or less constant as well. For variable amplitude loading the opening factor strongly fluctuates. Figure 16 shows the large fluctuation in opening factor for a spike load sequence, which makes it questionable to use it in a fit. Figure 14 demonstrates that the fitted coefficients cannot be based solely on the da/dn versus ΔK plot, because all the fits look equally well! The da/dn versus ΔK plot is only to guarantee that a proper fit is obtained. The quality of the fit should be examined by an a versus n plot in which the numerical prediction is compared with the original raw experimental data. The computed crack growth curve is already available from the crack growth analysis to determine the opening factor. Because the system is over-determined, it is possible for a given, valid, α_{new} , K_c and ΔK_0 to always find a “good” fit (C , n , p and q values) that results in a da/dn - ΔK and a - N plot that coincides with the experimental data.

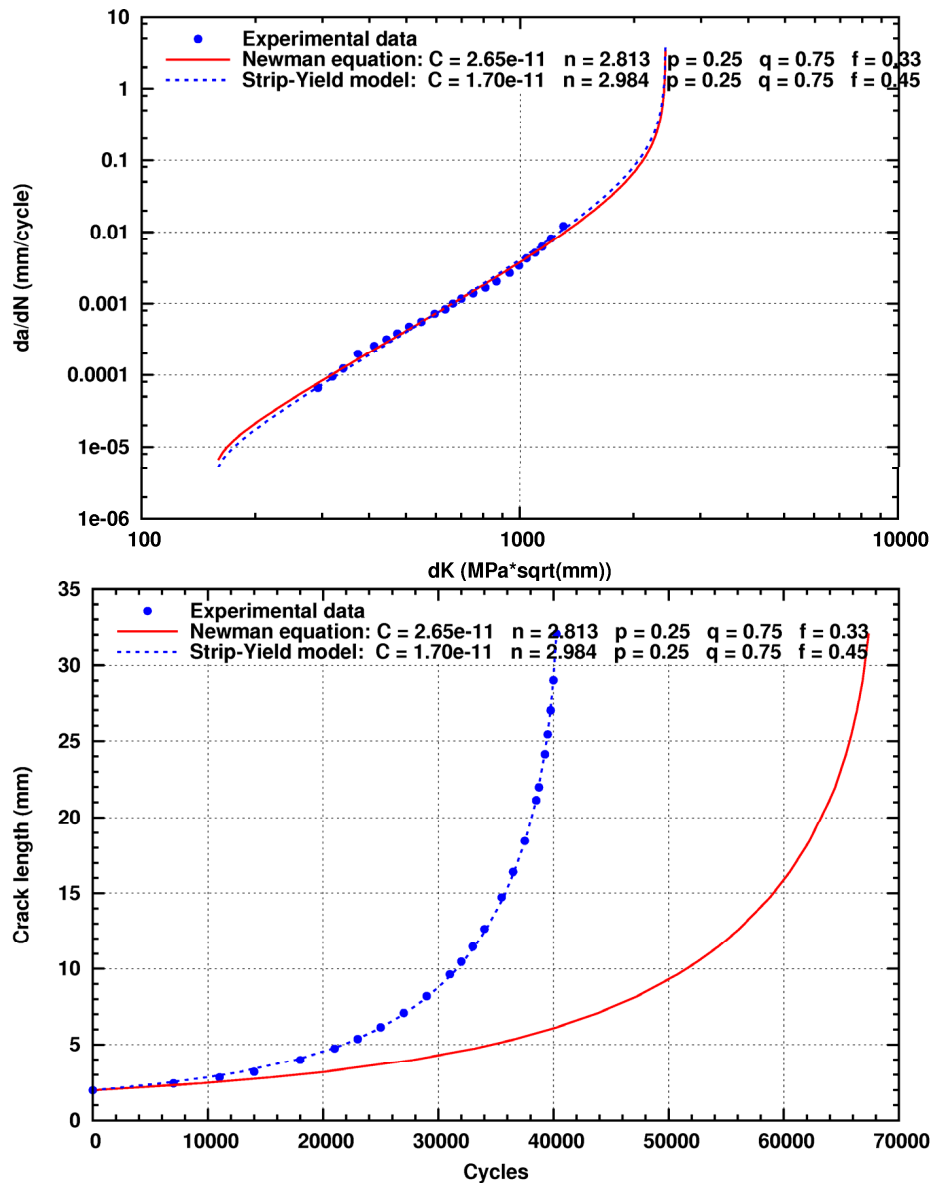


Fig. 14 NASGRO2 equation curve fits using Newman equation (red-line) and Strip-Yield model (blue-line) to compute the opening factor, AL2219-T851

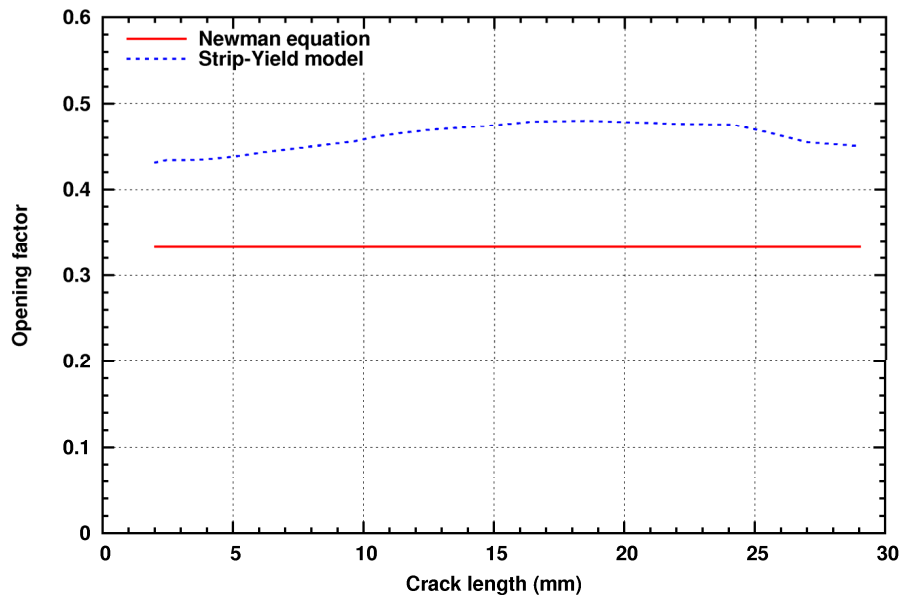


Fig. 15 Opening factor as function of the crack length determined with the Strip-Yield model for a CA load sequence

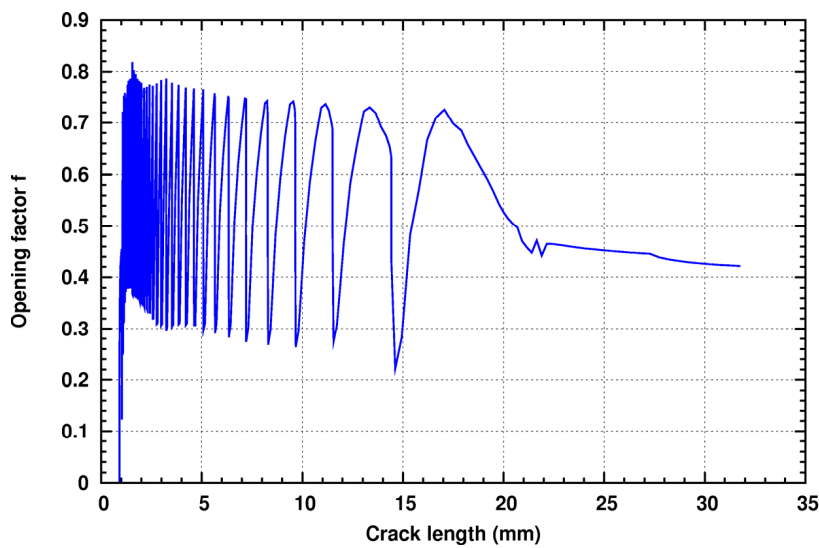


Fig. 16 Opening factor as function of the crack length determined with the Strip-Yield model for a spike load sequence

Another important issue is the difference in constraint factors of the variable constraint model applied in the Strip-Yield code that differ from the α applied in the Newman opening function (constant constraint model). The variable constraint model depends on a constraint factor α_{new} that represents the ratio of the cyclic yield limit in tension over its value in compression, which is a different value than the α applied in the Newman opening function! The value of α_{new} is of influence on the value of the opening factor. It is therefore important to use the opening factors computed by the Strip-Yield model.

5 Conclusions

A number of new developments have been made to ESACRACK software packages in the frame of an ESA study addressing various aspects of structural integrity and fracture control of pressurised structures.

These new developments, as listed in this paper, can be found in the field of

- New modules and capabilities for analysis of pressurised structures, like the rotational acceleration and interface displacements, residual stresses from welding, new SIF solutions, and a simple more conservative low-cycle fatigue behaviour;
- Development of advanced user interfaces to increase the program user-friendliness;
- Modernisation and improvements to the data storage and handling capabilities, like improved integration of NASGRO[®].

The values of the coefficients of the NASGRO2 crack growth equation strongly depends on the opening factor, which itself depends on the crack growth model used! Therefore, in a curve fit the same opening model should be applied to calculate the opening factor as used in the crack growth analysis. Otherwise large deviations are possible due to mismatch of the applied model and coefficients. This is true for any crack growth equation which depends on the opening factor.

Acknowledgement

The authors would like to express their appreciation to the contributions by Dr. Arij de Koning (formerly NLR), ESA/ESTEC and DOREA.

Registered users can download new releases of ESACRACK, available patches and documentation via the website <http://www.esacrack.com>. The ESACRACK distribution of NASGRO[®]5 may only be used on ESA funded project. Commercial licences for other applications may be obtained from the NASGRO Consortium.

References

1. Sinnema, G., Henriksen, T.K.; *ESACRACK User's Manual version 4, ESALOAD User's Manual version 4.1.6*, TOS-MCS/2000/14/ln Issue 1, Annex 1, ESTEC, May 2000.
2. Sinnema, G., Henriksen, T.K.; *ESACRACK User's Manual version 4, NASGRO3 User's Manual (ESACRACK version)*, TOS-MCS/2000/14/ln Issue 1, Annex 2, (original document JSC22267B), ESTEC, May 2000.
3. Sinnema, G., Henriksen, T.K.; *ESACRACK User's Manual version 4, ESAFATIG User's Manual version 4.1*, TOS-MCS/2000/14/ln Issue 1, Annex 3, ESTEC, May 2000.
4. Schijve, J.; *Fatigue of Structures and Materials*, Kluwer Academic Publishers, Dordrecht, 2001.
5. Aarhus, K., Henriksen, T.K.; *ESACRACK User's Manual (version 3) - Residual Stress Option Implementation In NASFLA*, ESA PSS-03-209 Issue 2, Annex 3, January 1995.
6. Tada, H., et al., *The Stress Analysis of Crack Handbook*, Third Edition, ASME Press, New York, 2000.
7. Murakami, Y.; *Stress intensity factors handbook Volume 1*, Reprinted edition, Pergamon Press, Oxford, 1988.
8. Brunetti, F; *European Space Agency FRAMES-2 software user manual*, SIPS/DOC/011, 2006.