Nenad Gubeljak

Associate Professor
University of Maribor
Faculty of Mechanical Engineering

Mustafa Koçak

European Fitness-for-Service Network –
FITNET coordinator
GKSS Research Centre
Institute for Materials Research, Geesthacht

Michell Huther

Bureau Veritas, Paris

Tomaz Valh

Walch d.o.o, Maribor

FITNET Fitness-for-service Fracture Module SOFTWARE

The paper gives concepts of FITNET software for Fracture module. In order to perform structure integrity assessment for component with real or assumed crack, it is necessary to vary material properties and crack geometry. Such approach can provide statistically reliable results for maximum loading capacity vs. crack size. Software can be useful and simple tool which helps to avoid calculation of polynom's parameters in equations for limit loads and stress intensity factors. The new concept of software is useful for education and introduction to FFS procedure in company and departments with base engineering knowledge.

Keywords: fitness-for-purpose, structural integrity, FITNET.

1. INTRODUCTION

Today, decent software for structure integrity is available [1,2]. Usually, the application of such software and interpretation of results require good knowledge about fracture mechanics, mechanics and materials. Within the framework of FITNET project, a Fracture education version is developed. The aim of the software is not to replace the mentioned softwares. The main goal of the software is to spread application to user with different knowledge level and skills. It is acheived with friendly operate enviroment, simple explainations and blocked unnecessary operations and steps. With this approach, the shortcut between input data and results is established. This helps the beginners and students to start to apply Fitness-for-Service concept for structure in design, manufacturing and service. However, usage of software helps to establish a new philosophy regarding to structure with flaws or defects. Only few case studies are included in the software.

Software is established with the module structure which incorporates each component as a module with known limit load solutions and stress intensity factor solution. In the software SINTAP concept [3] is applied for structure integrity analysis. Software checks validity conditions for solutions and provides possibility to change stress intensity factor with new solution or add a new component. The material module includes routine for input mechanical properties and fracture toughness. The output routine enables to print and safe results as report or plots and text file appropriate for further analysis.

2. FITNET FFS FRACTURE MODULE

The FITNET procedure represents with its sequential approach an ideal basis for computer manipulation. The procedure might be seen as a collection of recommendations meant for engineering purposes in industry mainly for engineers with practical experiences

but limited knowledge of fracture mechanics. This application considers only the standard options of FITNET procedure; these are **Default** option, **Basic** option, **Mis-match** option and **Advanced** option. When increasing the option, the demand for the entry parameters of the material increases. There's also another issue, on which this application depends in addition to the "standard" FITNET procedure. This is the one of available methods (e.g. ETM [4], ETM-MM [5], BS7910 [6], R6 [7]) for assessing the significance of crack-like defects in engineering practice that offers the results of the stress intensity factor and limit loading for different configurations. The application already contains a few examples.

The main requirement when planning the application was the universality and openness of the tool for manifold of elementary configurations in addition to the already included ones. One should also be able to export the results, data and graphs in standard formats into other standard applications.

Though the FITNET procedure is standardised, each solution holds for an individual configuration, on the other hand. Its result in the simplest form is only an information whether the construction is safe or there is a potential danger of a failure (i.e. fracture). This is basic information only. Consider that a crack can be safe for the prescribed load but unsafe in case of overloading. The application enables repeating the calculations with different properties of the crack or different loadings for that purpose. It remains up to the user to decide which is the critical parameter.

The application is built up from two program languages that coincide with the philosophy of universality of the tool with respect to the user and configurations. While MS Visual Basic serves mainly as a graphical interface, C++ is used for loading additional geometrical and loading configurations. The latter enables the new configurations to be added on-line without the need of annoying compilation. The configurations are usually based on problem solving; the FITNET procedure option depends on the availability of the parameters of the material. The boundary values (conditions) are selected by the user, while the validity conditions depend on the stress intensity factor and limit loading.

The calculation is performed in two phases. The first one is the preparation of the stress intensity factor and limit loading. The second phase is a standard FITNET procedure carried out in FAD or CDF diagram.

3. GEOMETRICAL AND LOADING CONFIGURATION

Selection of the geometrical and loading configuration is the basis for the calculation. At the left side of the dialog all registered configurations with their number and description are listed. Most of the dialogs are set to the default values, therefore the history of previous calculations is erased when a new configuration is selected.

4. ENTERING THE MATERIAL PARAMETERS

This dialog contains different fields to enable entering the material properties. There are several options which depend on the option of the FITNET procedure. In addition to other material parameters, the information about the shape of stress-deformation curve has to be given (whether material exhibits continuous yielding or Lüders plateau).

Parameters of the material with respect to the FITNET procedure option are given hereafter:

Description	Unit	Option			
		D	B	MM	A
Young's Modulus of Elasticity, E	GPa	✓	✓	✓	✓
Poisson's ratio, ν	–		✓	✓	✓	✓
Charpy impact energy, C_v	J	✓			
or					
Working temperature, T	ºC	✓			
Temperature at 28J of Charpy impact energy, T_{28J}	ºC	✓			
Probability of failure, P_f	%	✓			
Yield strength at 0.2 % plastic strain, $R_{p0.2}$	MPa	✓	✓	✓	✓
Tensile strength, R_m	MPa		✓	✓	✓
Crack fracture resistance in terms of K, J or δ (K_{mat}, J_{mat}, δ_{mat})	N/mm$^{1.5}$, N/mm, mm		✓	✓	✓
Engineering stress-deformation curve from single axis tensile test, R-e	File				✓

D – Default option; MM – Mis-match option; B – Basic option
A – Advanced option.

The parameters of the material are valid at the operating temperature. When entering the material data the dialog at the FITNET advanced option behaves in a slightly different way. All data fields except the crack fracture resistance as a material property are locked. This is due to the fact that all material parameters are entered into a special dialog together with engineering (R-e) stress-deformation curve.

The engineering stress-deformation curve is needed at FITNET Advanced option. The presence of boundary conditions depends on an individual geometrical and loading configuration. These conditions are mainly user selectable and define equations for stress intensity factor and limit load calculation.

There are maximum three groups of different boundary conditions:
- Stress/Strain condition (plane stress or plane strain);
- Geometrical point;
- Geometry or Shape.

Each of the boundary condition groups can have at the most three different options for selection. This dialog is not shown if a particular configuration doesn't have boundary conditions to select.

8.1 Entering geometry and loading parameters

The prepared FITNET application supports external loads in terms of forces, momentums or arbitrary stress distributions. The residual, internal or secondary stresses are optional in calculation. The dialog can be used in two different ways with respect to the method of describing the load.

8.2 Simple loading case

The first parameter or the first two parameters are considered as the key parameters of geometry or loading. Considering the geometry, these are the crack parameters. The crack size can be described only by length or by length (c) and depth (a). These are the key loading parameters. A simple load is usually described by one parameter (e.g. force or momentum); combined loads are often given by two parameters (e.g. tensile and bending stress). There's an optional selection right from the first two geometrical and the first two loading parameters, which represents a decision for primary and secondary parameter of the crack geometry and loading. The optional selection isn't always available. When the crack growth curve or loading curve is drawn, only the selected primary parameter is increasing while the secondary remains constant.

8.3 Distributed stress case

In this case the arbitrary stress distribution (ASD) profile through the cross section of the component is the loading input information. There are two more command buttons available in addition to the common ones. The numerical values are simply entered into the corresponding data fields. Proper units have to be considered. The verification of the values is performed upon confirmation. With arbitrary stress distribution profile the elementary loads, e.g. force, momentum or more often membrane σ_m and bending σ_b stress and polynomial coefficients of the stress distribution profile (S_0, S_1, S_2, ...) are present. These are not entry but intermediate calculated data. However, these values can also be inserted manually. There's also a more reliable, applicable and more often used choice. This dialog enables the entering of the ASD profile through the uncracked cross-section of the component. The stress distribution profile is always referring to a structural component without any crack. The stress distribution

profile is given in a form of individual points – usually along the thickness of the component cross-section. The stress distribution profile is always parallel to one of the characteristic dimensions of the component, while the direction of the stress is orthogonal to the crack surface. The position of the stress point is to be given always as a ratio with respect to the characteristic dimension of the cross-section, e.g. the position x with respect to the thickness T (x/T). The points of the profile are not completely arbitrary: a specific point has to be placed within the limits of the chosen characteristic dimension, e.g. thickness. The stress distribution profile first point's position is usually 0, the last point's position is 1.

5. VERIFICATION OF GEOMETRICAL AND STRESS VALIDITY CONDITIONS

The validity conditions discussed here are related to the specific geometrical and loading configuration and are to be used only within this configuration. The conditions from stress intensity factor and limit load solution originate the corresponding equations. The solutions differ from one to another geometrical and loading configuration. The conditions also differ due to different solution. The details and explanation of individual conditions are available in the original stress intensity factor and limit load solutions compendium. Each condition is described by mathematical expression or by description. The result expresses the fulfilment of the condition. The individual validity conditions are statically verified for the operating point only, i.e. for the nominal geometry and load. They aren't verified through the calculation with the crack growth or with increasing load when loading curve or crack growth curve is drawn. Therefore, though the nominal conditions are fulfilled, they might not be fulfilled far from the operating point. For that reason, the calculation has to be repeated for the critical crack length and critical load as a new operating point. This is the main advantage that allows the computer software calculation tool.

6. ENTERING THE CALCULATION PARAMETERS

This dialog enables decision about the assessment with respect to the critical load or critical crack (Critical value assessment). The calculation is repeated with only one parameter being changed, while the other remains constant. Thus the loading curve or the crack growth curve is drawn. This method enables us to determine not only the operating or nominal point, but also the critical or failure point. There are two parameters that define the increase of selected value, the step for increasing the load or crack length and the maximum load or crack length. The lower limit of the increasing parameter is selected as a small positive number. The step for increasing the normalized load L_r has to be given, too. The lower limit of the increasing normalized load is always zero, while the plastic collapse limit defines the upper limit. With this dialog the final act of the parameter entry is presented. It usually erases the history of previous calculations, if they exist. The history of calculations can be preserved for future comparisons by means of the control, but this can only

be done within one geometrical and loading configuration. The above mentioned control is disabled whether there is no history or whether this is the first calculation. The history is also erased with a choice of a new geometrical and load configuration or if one reselects the same configuration. The plastic collapse limit of the normalised load is displayed in this dialog. It usually cannot obtain the value that's less than one. If under certain conditions it becomes less than one, the red colour indicates a probable fault. A selection of the calculation with respect to the critical load or crack is in case of an arbitrary stress distribution profile limited to the crack growth and therefore the increasing parameter cannot be freely chosen in the dialog.

7. GRAPHIC INTERPRETATION OF RESULTS

The basic purpose of the FITNET procedure and the graphical representation of the results is the assesment of the structural integrity of the component. The location of two points is determined with the use of various calculation methods. The first point is called the nominal or operating point, while the second is known as a critical or failure point. Both are presented in both graphs (FAD and CDF) and give identical results. The position of the nominal point with respect to the failure curve is of our interest. This indicates the failure safety of the structural component. The failure point expresses the critical state, having two parameters: critical crack length and the failure load.

After the last dialog is confirmed, the FITNET calculation is performed in two stages. The first stage is performed in Python environment. Firstly, the selected parameter is increased, then the calculation of stress intensity factor, limit load, reference stress and normalised load is performed. The data are transferred into the main application before the second stage follows. The second stage is the standard FITNET procedure with prescribed equations. Their selection depends on the option of the procedure and on the type of material yielding (Lüders plateau). The application pop-ups a short message box with a results summary. The message box also contains the coordinates of nominal and critical point with respect to both interpretation types and the critical length of the crack or failure load, obtained from the graphs.

8. APPLICATION OF SOFTWARE

To illustrate the use and the possibilities offered by the software, we shall present a case built up from an industrial case. The 3 options presented in FITNET documents are applied with the same service conditions.

8.1 Case study

A project for a Canadian Arctic harbour requires the design of a steel box beam structure which dimensions are: width = 2.0 m, height = 1.5 m, thickness 25 mm.

The designer selects an HS steel with the following characteristics:
- Yield Strength 410 MPa;
- Ultimate Strength 630 MPa.

The stress-strain curve is known and given by the following measurement data (σ in MPa):

ε	0	0.002	0.01	0.02	0.04	0.07	0.08	0.09	0.1	0.12
σ	0	410	460	510	560	590	600	610	620	630

The applied loads are an axial force and 2 local loads (Fig. 1) leading at the weld toe to the following stresses:

global plus local tension: 310 MPa
local plate bending: 20 MPa
(tension at the outer tube surface)

The operational environment conditions in winter correspond to a minimum air temperature of – 50 °C.
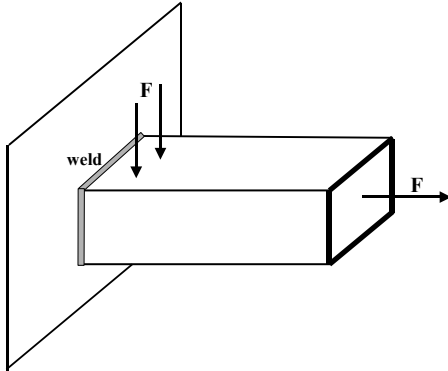

Figure 1. Steel box beam shape and loadings

The return experience of similar structures leads to assume that surface cracks can develop without being detected at the weld toe and that their shape factor is $a/c = 0.2$.

The beam integrity will be continuously checked by internal pressure monitoring.

The raised question is what brittleness requirements have to be introduced in the steel specification to prevent the risk of brittle fracture before leakage or ductile failure in service.

The 3 options of risk of brittle fracture presented in FITNET will be applied using the developed above presented demo software. The presented example corresponds to the "Surface Cracks in Plates" (ASD) case.

8.2 Default Option

The Default option assessment requires only to know the steel yield strength and the service temperature and allows only to calculate C or T_{28J} temperature for a given crack size.

For the application we decided to determine the necessary T_{28J} value.

The T_{28J} is found equal to – 88 °C without brittle fracture and with a critical crack length of 19.1 mm.

8.3 Basic Option assessment

The Basic option assessment requires to know the steel yield and ultimate strength, the service temperature and allows to determine K_{mat} or J or the CTOD.

For the application we decided to determine the necessary CTOD value.

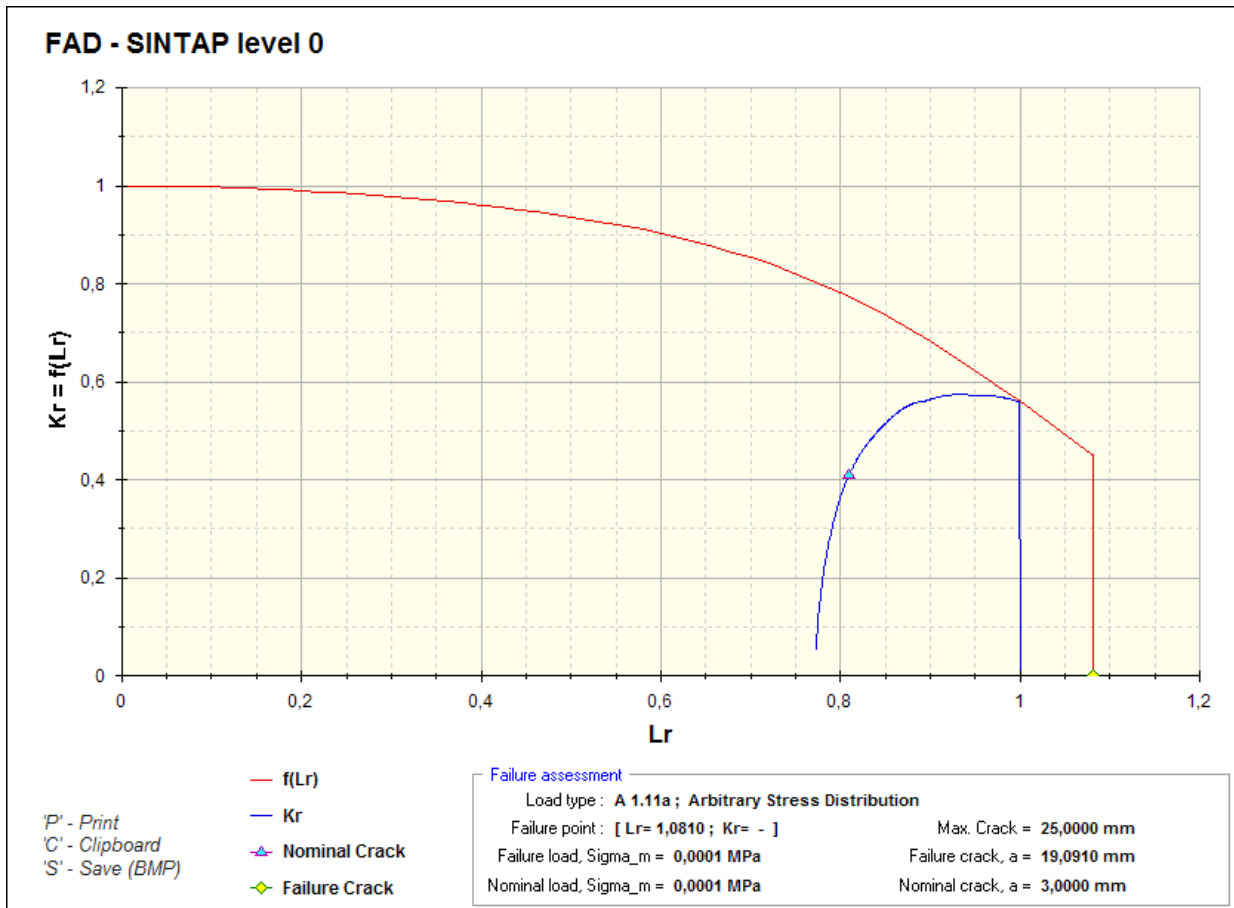To start a crack size is fixed, equal to $a = 5$ mm and a CTOD value is given equal to 0.1.

Figure 2. FAD curve for Default option limit state

With the same stress distribution data than previously, a first calculation provides a critical crack length of 11.8 mm.

This value does not correspond to a fail safe condition, the crack length is smaller than the plate thickness, so brittle fracture will occur before leakage.

Using the "Repeat calculation" function, the CTOD value will be changed until obtaining by iteration a failure point on the limit state curve. The FAD is given in the Figure 3.

The CTOD is found equal to 0.75 without brittle fracture and with a critical crack length of 16.5 mm.

8.4 Advanced Option assessment

The Advanced option, as the Basic option assessment, allows to determine K_{mat} or J or the CTOD, requires the service temperature but also to know the steel stress-strain curve.

For the application we decided to determine the necessary CTOD value.

The stress-strain data are entered, but due to their small number, the "Approximate engineering R-e curve" function is applied.

To start a crack size is fixed, equal to a = 5 mm and a CTOD value is given equal to 0.1.

With the same stress distribution data than previously, a first calculation provides a critical crack length of 14.1 mm.

This value does not correspond to a fail safe condition, the crack length is smaller than the plate thickness, so brittle fracture will occur before leakage.

Using the "Repeat calculation" function, the CTOD value will be changed until obtaining by iteration a failure point on the limit state curve. The FAD is given in the Figure 4.

The CTOD is found equal to 0.25 without brittle fracture and with a critical crack length of 16.5 mm.

8.5 Results summary

The application of the 3 FITNET´s option provides the following requirements for the steel brittleness specification:
Default Option
minimum Kv = 28 J at – 88 °C
Basic Option
minimum CTOD = 0.75 mm at – 50 °C
Advanced Option
minimum CTOD = 0.25 mm at – 50 °C

9. CONCLUSIONS

The application of the software of the Fracture module of the FITNET FSS Procedure to real case gives simple but important information about materials properties requirements. The software is possible to use in design stage (choice of right material) and in-service, (evaluation of risks from detected cracks). Software is established with module structure which incorporates each component as module with known limit load solutions and stress intensity factor solution. In the software the FITNET concept [3] is applied for structure integrity analysis. Software checks valid conditions for
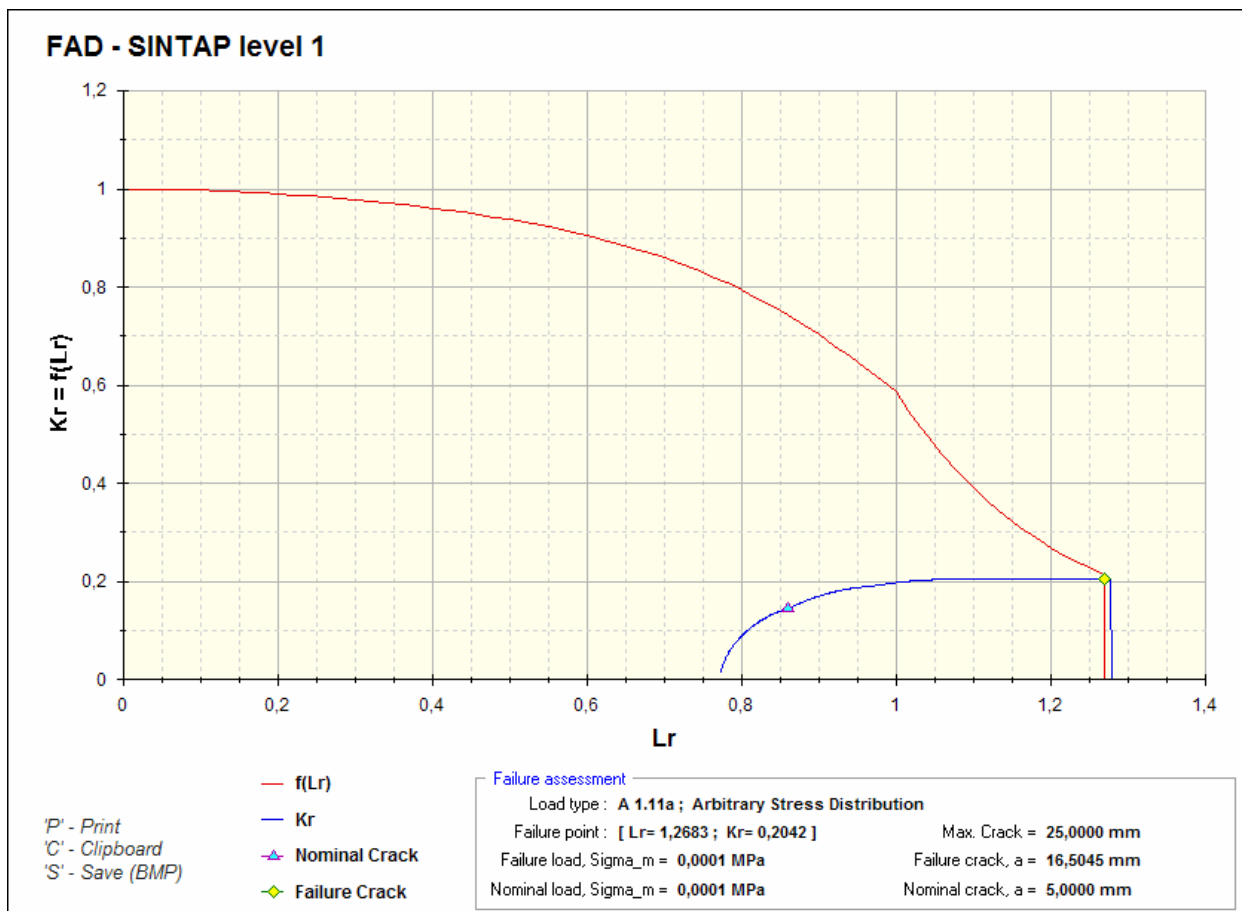

FAD - SINTAP level 1

Kr = f(Lr)

Lr

'P' - Print
'C' - Clipboard
'S' - Save (BMP)

— f(Lr)
— Kr
△ Nominal Crack
◇ Failure Crack

Failure assessment
Load type : A 1.11a ; Arbitrary Stress Distribution
Failure point : [Lr= 1,2683 ; Kr= 0,2042]
Failure load, Sigma_m = 0,0001 MPa
Nominal load, Sigma_m = 0,0001 MPa
Max. Crack = 25,0000 mm
Failure crack, a = 16,5045 mm
Nominal crack, a = 5,0000 mm

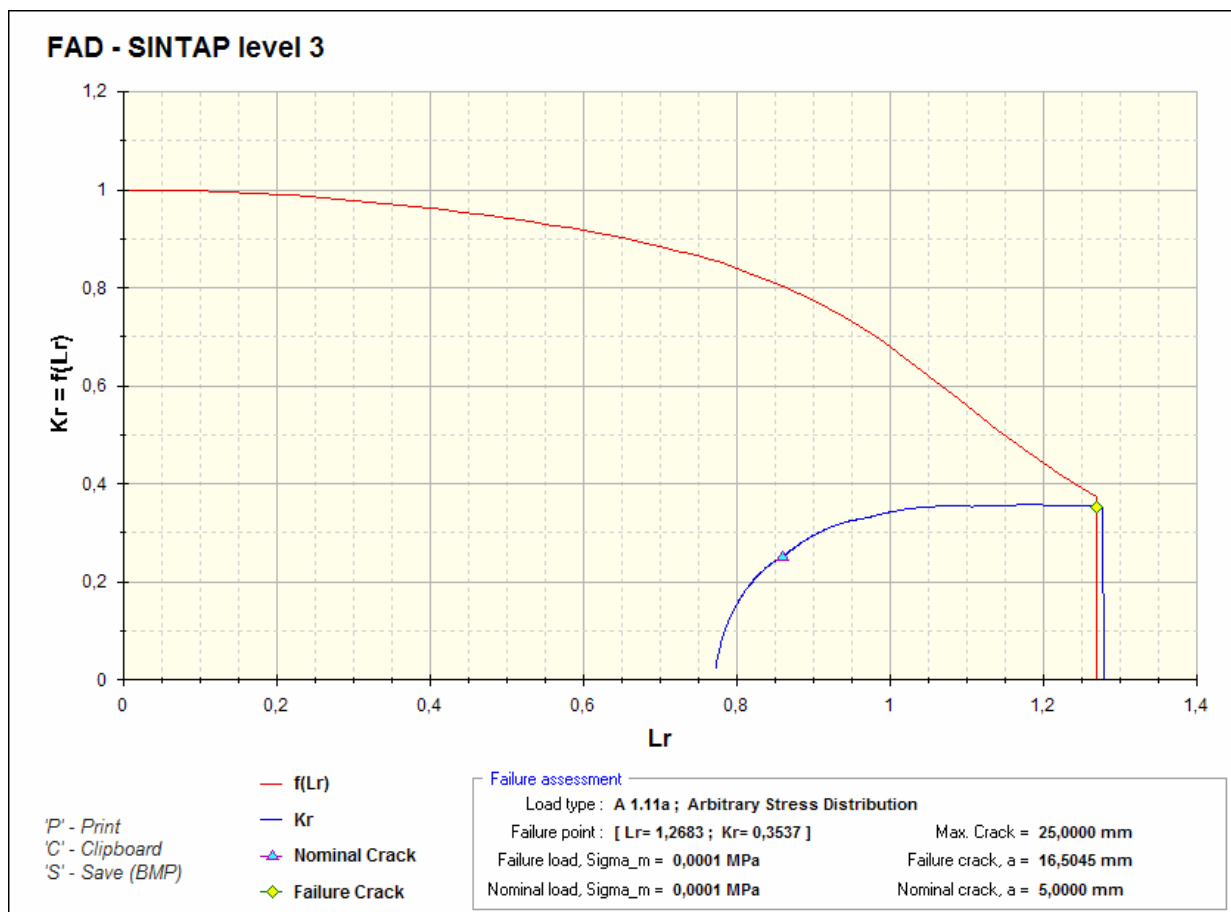**Figure 3. FAD curve for Basic option limit state**

Figure 4. FAD curve for Advance option limit state

solutions and provides possibility to change stress intensity factor with a new solution or adds new components. Material module includes routine for input mechanical properties and fracture toughness. Output routine enables to print and save results as report or plots and text file appropriate for further analysis.

ACKNOWLEDGMENTS

REFERENCES

[1] *Fatigue Crack Growth Program "NASGO" version 3.0 – reference manual JSC-22267B*, NASA Lyndon B. Johnson Space Center, Houston, 2000.

[2] *CRACKWISE® 4 – Automation of Fracture and Fatigue Assessment Procedures (BS7910) for Engineering Critical Assessment*, TWI Ltd., Granta Park, Great Abington, Cambridge, 2005.

[3] *SINTAP: Structural Integrity Assessment Procedure – Final reversion*, EU-Project BE-1462, Brite Euram Programme, 1999.

[4] FITNET – European Fitness-for-Service Network, G1RT-CT-2001-05071, http://www.eurofitnet.org.

[5] Schwalbe, K.-H., Zerbst, U., Kim, Y.-J., Brocks, W., Cornec, A., Heerens, H. and Amstutz, H.: *EFAM ETM-97 The ETM method for assessing the significance of crack-like defects in engineering structures comprising the versins ETM 97/1 and ETM97/2*, GKSS Research Centre Geesthacht, GKSS 98/E/6, Geesthacht, 1998.

[6] Schwalbe, K.H., Kim, Y.-J., Hao, S., Cornec, A. and Koçak, M.: *EFAM ETM-MM 96: The ETM Method for Assessing the significance of crack-like defects in joints with mechanical heterogeneity (strength mismatch)*, GKSS Research Centre Geesthacht, GKSS 97/E/9, Geesthacht, 1997.

[7] R6 Assessment of the Integrity of Structures Containing Defects, Revision 4, British Energy, 2000.

[8] BS7910:2005 Guide to methods for assessing the acceptability of flaws in metallic structures, 2005.

FITNET – СОФТВЕРСКИ МОДУЛ ЗА ПРОЦЕНУ ПОГОДНОСТИ-ЗА-УПОТРЕБУ

Nenad Gubeljak, Mustafa Koçak, Michell Huther, Tomaz Valh

Овај рад даје концепт *FITNET* софтвера за модуле лома. Уобичајено када се изводи оцена структурног интегритета компоненти са реалном или симулираном прслином неопходно је мењати особине материјала и геометрију прслине. Такав приступ може обезбедити статистички поуздане (веродостојне) резултате за максимум оптерећења капацитета наспрам величине прслине.