UDK: 625.785:539.42:514/539
COSATI: 11-13, 20-12, 01-03

Numerical Analysis of Damaged Stiffened Panels With Respects to Fracture Mechanics

Katarina Maksimović[1)]
Milutin Janković[2)]
Nenad Berić[3)]
Vera Nikolić-Stanojević[4)]

The main and primary structures in airframes are composed of stiffened panels in order to obtain high specific strength. This paper considers the finite element modelling to evaluate the fracture behaviour of cracked thin-walled structural components. The rapid development of computer technology has brought conditions for solving problems increasingly demanding in terms of numerical simulations. In the case of complex and large construction systems exposed to arbitrary loads, including complex boundary conditions, solving differential equations by analytic methods is very difficult or impossible. Then the solution requires using numerical methods, most often, using the finite element method (FEM). To determine stress intensity factors (SIF`s) of cracked thin-walled stiffened panels, singular finite elements are used. In this paper, the effects of the stiffeners on SIF`s are considered. The fatigue crack growth behaviour of 2219-T851 aluminum is characterized by constant-amplitude loading conditions. Fatigue crack growth analyses are performed on the cracked panel using Paris`s and Valker`s crack growth models. Analytical predictions are correlated with the test data.

Key words: **thin walled structure, aircraft structure, fracture mechanics, panel, crack, crack growth, numerical analysis, finite element method, singular finite elements, stress intensity factor.**

Introduction

AEROSPACE components must adhere to airworthiness regulations and satisfy damage tolerance requirements. Stiffened panels are metal sheets reinforced by stringers. They are widely used in the aerospace industry where light, highly resistant and damage tolerant structures are required. The fracture behaviour of cracked structures is dominated by the near-tip stress field. In fracture mechanics, most interest is focused on stress intensity factors, which describe the singular stress field ahead of a crack tip and govern fracture of a specimen when a critical stress intensity factor is reached.

Structural components that form a part of a structure are, in most cases, of complex geometric shapes. Examinations have shown that, in places where the cross section decreases, the stress increases. This phenomenon is called the stress concentration. The presence of cracks in a material generally reduces the static strength of the material because the stress and strain are highly magnified at the crack tip [1, 2]. Parameters deduced from linear fracture mechanics (LEFM) can be used to determine the stress and strain magnification at the crack tip. These parameters, the stress intensity factor (SIF), incorporate applied stress levels, geometry and crack size in a systematic manner and may be evaluated from the elastic stress analysis of cracked structures [3-7].

A new concept, called the damage tolerance approach [8-10], based on the principles of fracture mechanics, is to assume the initial damage in the critical zones of the elements, which can lead to structural failure during the planned life. In this paper a special attention is paid to the analysis of real structural elements with damage in a form of initial cracks. For that purpose, a cracked stiffened panel, representing the aircraft wing structure, is considered using singular finite elements. A structural component is damage tolerant if it can withstand reasonable loads without catastrophic failure or excessive deformation after the occurrence of serious fatigue damage. The numerical crack growth analysis via the finite element method (FEM) has been used extensively to predict the damage tolerance characteristics of stiffened panels. Several numerical analyses, each simulating a different crack length, have to be performed. The important results from the analyses are the stress intensity factors and the maximum stress levels at the sheet, stiffener and stiffener attachments. For the determination of the residual life of cracked structural components, various crack growth models have been proposed by numerous investigators in the past decades. These crack growth models use stress intensity factors and it is very important that we use correct formulae for SIF`s.

Determination of stress intensity factors using finite elements

Stress intensity factor determination plays a central role in linearly elastic fracture mechanics (LEFM) problems. Fracture propagation is controlled by the stress field near the crack tip. Because this stress field is asymptotic dominant or singular, it is characterized by the stress intensity factor.

1) City Administration of City Belgrade, Secretariat for Utilities and Housing Services Water Management, Kraljice Marije 1, 11000 Belgrade, SERBIA
2) VZ Moma Stanojlović, puk. Milenka Pavlovića 160, 11273 Batajnica, SERBIA
3) Special Brigade, 26000 Pančevo, SERBIA
4) State University of Novi Pazar, Vuka Karadžića bb, 36300 Novi Pazar, SERBIA

In this paper, special singular 2D finite elements are used for modeling the continuum in the plane condition of stress and strain, where plates belong to. Special singular 2D finite elements are provided in the top of the crack, singular strain. We get extremely accurate results with a relatively coarse finite element mesh around the tip of cracks.

In order to represent point singularities, the quadrilateral must be degenerated into a triangle. This is done by coalescing grids 1, 4 and 8 as it can be done for standard 8-node isoparametric elements and is schematically presented in Fig.1. Barsoum [2] also showed that a triangular quarter-point element, shown in Fig.1, exhibits the $r^{-1/2}$ singularity both on the boundary of the element and the interior. In this work, the 6-node, quarter-point triangular element, which degenerated from the 8-node quadrilateral element, is used around the crack tip.

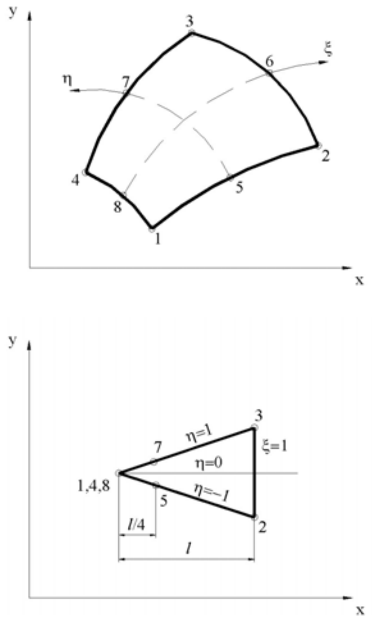


Figure 1. Quadrilateral isoparametric and collapsed quarter point 2D finite element

The finite element, which approximates the displacement field, must have nodes lying in the *xy* plane with displacements having components in the direction of the *x* and the *y* axis. The interpolation of geometry can be written in the following form, [1, 2].

$$x = \sum_{i=1}^{8} h_i(\xi,\eta)\cdot x_i \text{ ; } y = \sum_{i=1}^{8} h_i(\xi,\eta)\cdot y_i \text{ ; or}$$

$$x_i = \sum_{k=1}^{N} h_k X_i^k \tag{1}$$

where h_i is the interpolation matrix or the shape function and x_i and y_i are the coordinates, respectively, at the point *i* in the element. Thus, the displacement field of the points can be defined by a vector field, [1], in the following way:

$$u = \sum_{i=1}^{8} h_i(\xi,\eta)\cdot u_i \text{ ; } v = \sum_{i=1}^{8} h_i(\xi,\eta)\cdot v_i \text{ or}$$

$$u_i = \sum_{k=1}^{N} h_k U_i^k \tag{2}$$

Once a finite element solution has been obtained, the values of the stress intensity factor (SIF) can be extracted from it. Three approaches to the calculation of SIF can be used: the direct method, the indirect method and the J-integral method. In this study the indirect method has been selected. In this method, the values of stress intensity factors are calculated using the nodal displacements in the element around the crack tip. The displacements can be expressed in terms of the nodal displacements A, B and C, Fig.2.

$$\begin{aligned} u(r) &= u_A + (-3\cdot u_A + 4\cdot u_B - u_C)\cdot\sqrt{\frac{r}{l}} + \\ &\quad + (2\cdot u_A - 4\cdot u_B + 2\cdot u_C)\cdot\frac{r}{l} \\ v(r) &= v_A + (-3\cdot v_A + 4\cdot v_B - v_C)\cdot\sqrt{\frac{r}{l}} + \\ &\quad + (2\cdot v_A - 4\cdot v_B + 2\cdot v_C)\cdot\frac{r}{l} \end{aligned} \tag{3}$$

When r becomes small, the stress intensity factors can be obtained by comparing the $\sqrt{r}$ in the corresponding equations. Based on the results of the displacements obtained using the FEM, one can determine the stress intensity factors

$$\begin{aligned} K_I &= \frac{2\cdot\sqrt{2\cdot\pi}\cdot G\cdot(4\cdot v_B - v_C - 3\cdot v_A)}{(k+1)\cdot\sqrt{l}} \\ K_{II} &= \frac{2\cdot\sqrt{2\cdot\pi}\cdot G\cdot(4\cdot u_B - u_C - 3\cdot u_A)}{(k+1)\cdot\sqrt{l}} \end{aligned} \tag{4}$$

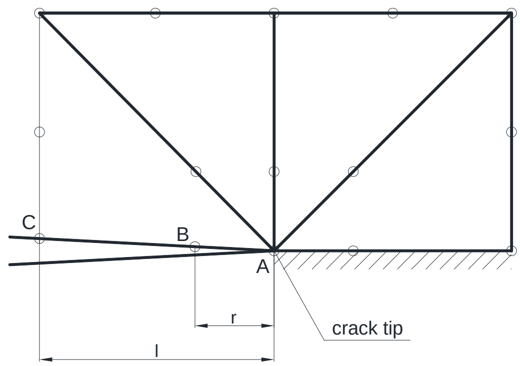


Figure 2. Element nodes along the crack surface

The authors have tested the accuracy of this singular finite element analysis and other available solutions. We obtained a remarkable accuracy of the finite element with a relatively coarse finite element mesh in the zone of damage.

Fracture criterion

In the region surrounding the tip of the crack (for mode I only), the singular stresses are characterized by the stress intensity factor K_I. It is postulated that crack growth will occur when the equality

$$K_I = K_{Ic} \tag{5}$$

holds. As for K_{Ic}, which behaves as a threshold value for K_I, it is called the critical stress intensity factor. It is a material parameter, also known as mode I fracture toughness. It may be determined experimentally.

Numerical results

To illustrate the computation procedure for the determination of the parameters of fracture mechanics, based on FEM, here is considered a cracked stiffened panel subjected to tensile load, Fig.3. This panel is a representative of aircraft wing tip structures. A simplified cracked wing skin panel between two stringers is considered here. The effects of these stringers on SIFs are considered using singular finite elements.

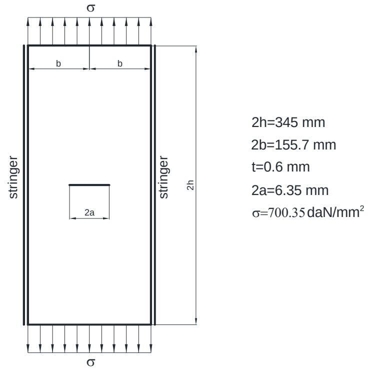


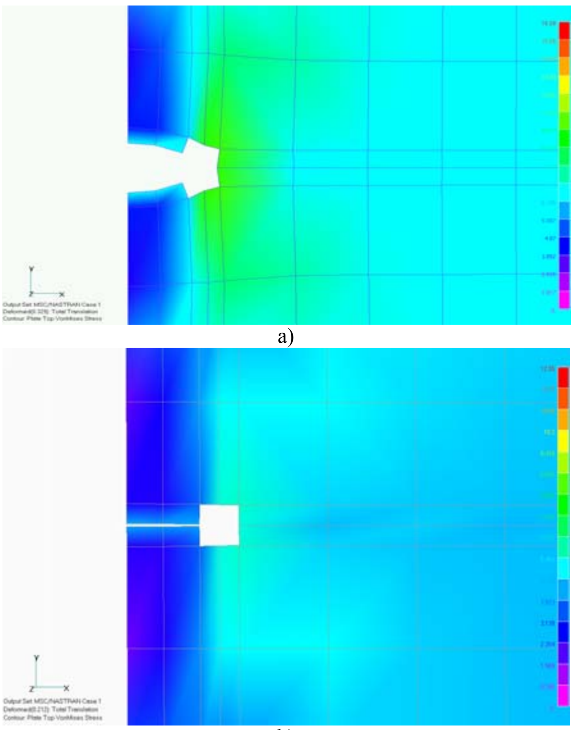
Figure 3. A Cracked stiffened panel

The graphical illustrations of stress distributions of un-stiffened and stiffened panels using singular finite elements are given in Fig.4. As a stiffener in this analysis, two L-type profile are used as shown in Fig.3. The analysis was carried out using special 6-node singular finite elements, as described in paragraph 2 of this paper.

Table 1 gives the comparisons of the presented finite element and the analytic results of SIF`s for the stiffened panel of the wing skin defined in Fig.3.

Table 1. Comparisons of the finite element and the analytic SIF results for the cracked panel

$j=1.5$	$2b$ [mm]	a_0 [mm]	K_I^{ANAL} $[\mathrm{N/mm^{3/2}}]$	K_I^{FEM} $[\mathrm{N/mm^{3/2}}]$	Panel of wing skin
$\sigma=71.7$ N/mm^2	155.7	3.175	229.9 110.5	206.2 113.6	Panel without stringers Panel with stringers



a)

b)

Figure 4. Von Mises stresses distributions around the crack of the un-stiffened panel (a) and the stiffened panel (b)

The effects of the stiffeners on the stress intensity factors are evident, Table 1. Good agreement between the finite element with the analytic solutions is obtained.

Fatigue crack growth behavior and life prediction

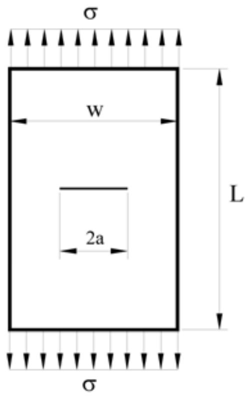
Various crack growth models have been proposed by numerous investigators in the past decades. A few of them in the literature have demonstrated the ability to provide consistently good predictions for certain types of variable-amplitude loading. To study residual lives of cracked structural components here, two fatigue crack growth equations are used, including the Paris (6) and Walker [11] (7) equations:

$$\frac{da}{dN}=C(\Delta K)^m \tag{6}$$

$$\frac{da}{dN}=C[(1-R)^{m-1}(\Delta K)]^n \tag{7}$$

where: 2a- the crack length, N- the number of cycles, K-the stress intensity factor, C and m are Paris`s dynamic properties of material, $R$- the cyclic stress ratio and $m$ is Walker`s stress ratio layer collapsing factor.

This test compares computation with experiments using previous relations (6) and (7) with experiments [12]. The geometry of the specimen and the dynamic properties of the material are given in Fig.5.



Geometry of the Specimen:

$w=152.4\,\mathrm{mm}$
$L=457.2\,\mathrm{mm}$
$2a=7.62\,\mathrm{mm}$
$t=5.35\,\mathrm{mm}$ t

Cyclic Properties of 2219-T851:

$C=8.367\cdot10^{-10}$
$n=3.64$
$K_C=71.5\ \mathrm{MPa}\sqrt{\mathrm{m}}$

Figure 5. Model of a wing skin panel with an initial crack

Table 2 shows the numerical and experimental results of the crack growth analysis with residual lives. The results of Valker`s model possess better agreement with the experiment than Paris`s model. The effect of the stress ratio, R, which is incorporated into Valker`s model, is evident.

Table 2. Comparisons of the computation crack growth model of the panel with the experiments [12]

Specimen No.	$\sigma_{\max}$ [MPa]	$\sigma_{\min}$ [MPa]	N_f [$Exp.$]	N_f [$Pariss$]	N_f [Val ker]
M-1	55.16		355810	382405	339000
M-3	55.16	-55.16	129240		96000
M-5	275.8	0	846	906	803
M-6	275.8	82.74	1695	3320	1537

The previous results shows that Valker`s model in which the effect of the stress ratio, $R$, is incorporated gives better results than the conventional Paris`s model. The obtained residual fatigue lives of the cracked structural element, Table 2, are conservative. This is good in practical design and analysis with respect to fracture mechanics and residual life estimations.

Conclusions

Numerical methods are necessary for the SIF evaluation of 3-D planar cracks because analytical solutions are limited to simple geometries with special boundary conditions. This paper considers the finite element formulation to evaluate fracture behavior of cracked thin-walled structural components. Singular finite elements are used for modeling cracked thin-walled structural elements. It is shown here that finite elements can solve very complex thin-walled problems in fracture mechanics. Good agreement between present computation SIF's using singular finite elements and analytic solutions is obtained. Based on the obtained results for singular displacements at the crack tip, the stress intensity factor can be determined with sufficient accuracy. This paper considers two computation models for the fatigue crack growth analysis and residual life estimations. Valker`s computation model in which the stress ratio is incorporated gives good agreement with experiments.

Acknowledgement

Parts of this research were supported by the Ministry of Science, Technology and Development of the Republic of Serbia through the Mathematical Institute SANU, Belgrade, and the Mathematical institute, Belgrade: Grants No. ON174001 The dynamics of hybrid systems of complex structures. Mechanics of Materials and TR 35011 (2011.-2014.).

References

[1] ZIENKIEWICZ,O.C., TAILOR,R.L.: *Finite Element Method*, Butterworth Heinemann, Oxford, 2000.

[2] BARSOUM,R.S.: *On the use of isoparametric finite elements in linear fracture mechanics*, Int. J. Numerical Methods in Engineering, 1976, Vol.10, pp.25-37.

[3] BOLJANOVIĆ,S., MAKSIMOVIĆ,S.: *Fatigue crack growth modeling of attachment lugs*, International Journal of Fatigue, 2014, Vol.58, pp.66-74.

[4] MAKSIMOVIĆ,K., NIKOLIĆ-STANOJEVIĆ,V., MAKSIMOVIĆ,S.: *Efficient computation method in fatique life estimation of damaged structural components*, Facta Universitatis, Series Mechanics, Automatic Control and Robotics, 2004, Vol.4, No.16, pp.101-114.

[5] MAKSIMOVIĆ,K., NIKOLIĆ-STANOJEVIĆ,V., MAKSIMOVIĆ,S.: *Modeling of the surface cracks and fatique life estimation*, ECF 16 2006, Minisymposium integrity of dynamical systems, Alexandroupolis, GREECE, July 3-7, 2006, pp.113-116.

[6] MAKSIMOVIĆ,S., POSAVLJAK,S., MAKSIMOVIĆ,K., NIKOLIĆ-STANOJEVIĆ,V., DJURKOVIĆ,V.: *Total Fatigue Life Estimation of Notched Structural Components Using Low-Cycle Fatigue Properties*, J. Strain, 2011, 47 (suppl.2), pp.341-349.

[7] NIKOLIĆ-STANOJEVIĆ,V., DOLIĆANIN,C., RADOJKOVIĆ,M.: *Application of Finite Element Method of thin steel Plate with holes*, Technical Gazette*,* 2011, Vol.18, No.1, pp.57-62.

[8] STAMENKOVIĆ,D., MAKSIMOVIĆ,S., MAKSIMOVIĆ,K., NIKOLIĆ-STANOJEVIĆ,V., STUPAR,S, VASOVIĆ,I.: *Fatigue Life Estimation of Notched Structural Components*, Strojniški vestnik - Journal of Mechanical Engineering 2010, Vol.56, No.12, pp.846-852, UDC 629.7:620.178.3, http://www.sv-jme.eu/current-volume/sv-jme-56-12-2010/

[9] MAKSIMOVIĆ,S., VASOVIĆ,I., MAKSIMOVIĆ,M., DJURIĆ,M.: *Some Aspects of Damage Tolerance Analysis of LASTA Training Aircraft Structures*, Scientific Technical Review, ISSN 1820-0206, 2013, Vol.63, No.2, pp.70-74.

[10] Damage tolerance assessment handbook, vol. II, Airframe damage tolerance evaluation, DOT/FAA/CT-93/69.II. Springfield, IL: Federal Aviation Administration, US Department of Transportation, 1993.

[11] WALKER, K.: *In Effects of Environment and Complex Load History on Fatigue Life*, ASTM STP 462, American Society for Testing and Materials, 1970.

[12] CHANG,J.B., STOLPESTED,J.H, SINOZUKA,M., VAICATIS,R.: *Improved methods for predicting spectrum loading effects-Phase I Report*, Vol. I – Results and Discussions, AFDL-TR-79-3036, Ohio, SAD, 1979.

Received: 22.01.2014.

Numerička analiza ojačanog panela sa inicijalnim oštećenjem, a sa aspekta mehanike loma

Glavna i noseća struktura vazduhoplova su projektovane u osnovi od ojačanih panela kako bi se postigla visoka specifična čvrstoća. Ovaj rad razmatra primenu metode konačnih elemenata (MKE) u analizi tankozidnih strukturalnih elemenata sa inicijalnim prskotinama sa aspekta mehanike loma. Intezivan razvoj računarske tehnologije tokom poslednjih dekada stvorili su uslove i omogućili rešavanje problema sa aspekta mehanike loma primenom metoda numeričkih simulacija. U slučajevima kompleksnih i velikih strukturalnih sistema izloženih dejstvu složenih opterećenja i graničnih uslova rešavanje diferencijalnih jednačina primenom analitičkih metoda je veoma teško i praktično nemoguće. Stoga rešavanje ovih problema zahteva primenu numeričkih metoda, najčešće MKE. Za određivanje faktora inteziteta napona (FIN) kod tankozidnih konstrukcija tipa ojačanih panela korišćeni su specijalni singularni konačni elementi. Analiza širenja prskotine za razmatrane elemente od duraluminija 2219-T851 je izvršena za ciklična opterećenja konstantne amplitude. Numeričke analize širenja prskotine u ovom slučaju za panel sa inicijalnom prskotinom, pod dejstvom cikličnih opterećenja konstantne amplitude, je realizovana primenom Parisovog i Valkerovog modela. Dobijeni rezultati numeričkih simulacija su u saglasnosti sa eksperimentima.

Ključne reči: **tankozidna konstrukcija, struktura letelice, mehanika loma, panel, prskotina, rast prskotine, numerička analiza, metod konačnih elemenata, singularni konačni elementi, faktor intenziteta napona.**

Численный анализ армированных панелей с первоначальным повреждением с точки зрения механики разрушения

Основная и несущая конструкция летательного аппарата спроектированы на базе усиленных панелей для достижения высокой удельной прочности. В данной статье обсуждается применение метода конечных элементов (МКЭ) в анализе тонкостенных элементов конструкции с начальными трещинами в плане механики разрушения. Интенсивное развитие компьютерных технологий за последнее десятилетие создало условия и позволило решать проблемы с точки зрения механики разрушения с помощью метода численного моделирования. В случаях сложных и масштабных структурных систем, подвергающихся воздействию сложных нагружений и граничных условий, решения дифференциальных уравнений с помощью аналитических методов очень сложно и практически невозможно. Таким образом, решение этих проблем требует применения численных методов, обычно МКЭ. Для определения коэффициента интенсивности напряжений (КИН) в тонкостенных конструкциях типа армированных панелей, использованы специальные единичные конечные элементы. Анализ роста трещины на рассмотреных элементах из дюралюминия 2219-T851 выполнен для циклических нагружений постоянной амплитуды. Численные анализы роста трещины в этом случае для панели с начальным ростом трещины при циклическом нагружении с постоянной амплитудой, реализуются с помощью моделей Париса и Валкера. Результаты численного моделирования согласуются с экспериментами.

Ключевые слова: **тонкостенные конструкции, конструкция самолёта, механика разрушения, панельное строительство, трещина, рост трещины, численный анализ, анализ методом конечных элементов, особые конечные элементы, коэффициент интенсивности напряжений.**

Analyse numérique du panneau renforcé à l'endommagement initial sous l'aspect de la mécanique de fracturé

La principale structure primaire des aéronefs se compose des panneaux renforcés pour avoir une très grande force spécifique. Ce papier considère l'application de la méthode des éléments finis (MKE) dans l'analyse des éléments structuraux aux parois minces fissurés initialement sous l'aspect de la mécanique de fracture. Le développement intensif de la technologie informatique pendant les dernières décades a créé les conditions et a permis de résoudre le problème sous l'aspect de la mécanique de fracture à l'aide des méthodes des simulations numériques. Chez les grands systèmes structuraux et complexes qui sont exposés aux effets composés et aux conditions limites, la résolution des équations différentielles par les méthodes analytiques est très difficile et pratiquement impossible. Pour cette raison la résolution de ce problème exige l'emploi des méthodes numériques, le plus souvent la méthodes des éléments finis. Pour déterminer le facteur d'intensité de tension (FIN) chez les constructions des panneaux renforcés à parois minces on a utilisé les éléments singuliers finis. L'analyse de la croissance de fissure pour les éléments considérés en aluminium dur 2219-T851 a été réalisée pour les charges cycliques à l'amplitude constante. Les analyses numériques de la croissance de fissure chez les panneaux à la fissure initiale qui sont sous l'effet de la charge cyclique à l'amplitude constante ont été faites à l'aide du modèle Paris-Valker. Les résultats des simulations numériques sont en bon accord avec les résultats des essais expérimentaux.

Mots clés: **construction à parois minces, structure d'aéronef, mécanique de fracture, panneau, fissure, croissance de fissure, analyse numérique, méthode de éléments finis, élément singuliers finis, facteur d'intensité de tension.**