Elisaveta Doncheva[1]*, Bojan Medjo*[2]*, Aleksandar Sedmak*[3]

FINITE ELEMENT ANALYSIS OF FRACTURE RESISTANCE PARAMETERS FOR STATIONARY SEMI-ELLIPTICAL SURFACE CRACKS IN HIGH STRENGTH STEEL

ANALIZA PARAMETARA OTPORNOSTI NA LOM METODOM KONAČNIH ELEMENATA, KOD STACIONARNIH POLUELIPTIČNIH PRSLINA U ČELIKU POVIŠENE ČVRSTOĆE

Originalni naučni rad / Original scientific paper
UDK /UDC: 669.15:539.42
Rad primljen / Paper received: 10.7.2015

Adresa autora / Author's address:
[1] Ss. Cyril and Methodius University, Faculty of Mechanical Engineering, Skopje, Macedonia
[2] University of Belgrade, Faculty of Technology and Metallurgy, Belgrade, Serbia
[3] University of Belgrade, Faculty of Mechanical Engineering, Belgrade, Serbia, asedmak@mas.bg.ac.rs

Keywords
- J integral
- finite element method
- plastic behaviour

Abstract

This study deals with numerical simulations based on elastic-plastic finite element method for nonlinear stress analyses on pre-cracked tensile specimens with high strength properties. The influence of crack geometry on fracture resistance properties is considered with numerical analysis on models with two different cracks. Results on fracture resistance parameters are obtained for both simulated tensile panels with different crack lengths and width, referred as short and long crack. The obtained numerical results for crack resistance behaviour of tensile specimens are compared with data results from existing experimental investigations on specimens with same shape and material properties. Also, comparison is made with results obtained from similar standard experimental procedure. It is shown, that a proper combination of numerical and experimental procedures can lead research to significantly accurate results when investigating the fracture response of materials.

Ključne reči
- J integral
- metoda konačnih elemenata
- plastično ponašanje

Izvod

U ovom radu se razmatraju numeričke simulacije na bazi elastoplastičnih konačnih elemenata kod nelinearnih naponskih analiza zateznih epruveta sa prslinom, od materijala povišene čvrstoće. Uticaj geometrije prsline na otpornost prema lomu je razmatran u numeričkoj analizi na modelima sa dve različite prsline. Rezultati parametara loma su dobijeni kako za simulaciju zateznih ploča različitih dužina prsline i širina, koje čine tzv. kratke i duge prsline. Poređenjem dobijenih numeričkih rezultata za otpornost na lom zateznih epruveta sa rezultatima odgovarajućih eksperimentalnih ispitivanja epruveta istih oblika i osobina materijala, kao i poređenje sa rezultatima dobijenim sličnim standardnim eksperimentalnim postupcima, pokazuje se da pravilna kombinacija numeričkih i eksperimentalnih procedura može usmeriti istraživanje u dobijanju mnogo preciznijih rezultata u istraživanju svojstava otpornosti na lom materijala.

INTRODUCTION

Initial cracks may occur in various types of structures, such as pressure vessels or pipelines due to welding procedure, fabrication, transportation and installation. There are various reasons and factors that may cause the crack to accelerate its propagation and reach its critical size, which will significantly reduce the load capacity of the structure and eventually will lead to catastrophic failure. Fracture resistance parameters as the J-integral and crack mouth opening displacement (CMOD) are fundamental parameters that are widely used to describe the behaviour of elastic-plastic materials when a crack is present. Investigation of tensile specimens with a rectangular cross section and semi-elliptical surface crack in the middle, can give an adequate and realistic image of the remaining strength capacity of a structure, if results obtained include measurements for J-integral and CMOD parameters. These parameters can be obtained through standard fracture mechanics tests. However, results for different specimens often reveal considerable differences due to the constraint effects that are very important even in homogeneous structures. These differences basically arise from the fact that material's fracture resistance is strongly dependent from crack geometry and type of loading. In welded joints these effects are much more pronounced than in homogeneous structures. Abovementioned constraints are the reason why transferring the fracture parameters from specimens to components is especially questionable. As a parameter characterizing crack tip field, the J-integral has played an important part in elastic-plastic fracture mechanics and assessment of homogenous structure, /1-4/. Many researchers /5-9/ have used the finite element method (FEM) which is an important tool to design a practical mechanical component, such as welded joints. Cracks subjected to loads respond in the same manner as a notch in a material, namely as stress raisers. Due to the sharp configuration at the tip, the crack creates severe concentrations of stress at the tip, /10/. Related to this behaviour is also the creation of a plastic zone in the vicinity of the tip, due to plasticity in the material from high stress. The stress field at the tip of the crack greatly depends on the mechanical behaviour of materials.

The aim of this work is to analyse the influence of crack geometry characteristics on the mechanical behaviour of specimens produced of high strength low alloyed steel (HSLA). The J-integral method is used for numerical investigation of fracture behaviour on two tensile specimens of rectangular cross sections with different crack geometries in the central section. The tensile specimens are simulated in Abaqus 6.13-1 using 3D finite elements. Two models are considered, the base metal with small crack (BMSC) and the base metal with the large crack (BMLC).

FINITE ELEMENT MODELLING OF SPECIMENS

Computational methods such as finite element method are widely accepted as an important tool used to investigate crack behaviour. Contributing to this field, we analysed the fracture behaviour of HSLA steel by computing J-integral at the crack front. Finite element simulations are done using Abaqus 6.13 platform in elastic-plastic domain.

Taken into account the symmetry of loading and geometry, only a quarter of the model is simulated with reduction on the side-ends in order to reduce calculation time. Figure 1 shows the geometry of the model and the finite element mesh. The quadratic 20-node isoparametric brick elements with reduced integration (C3D20R) are used to construct the quarter models of the tensile panels. The resulting finite element models consist of different number of elements and nodes, which is due to the difference in crack geometry in each model. The number of elements and nodes for the models is given in Table 1 and the geometry of each crack is given in Fig. 2.

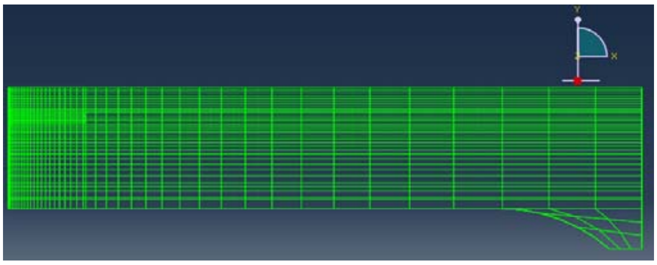

Figure 1. Finite element mesh of the model.

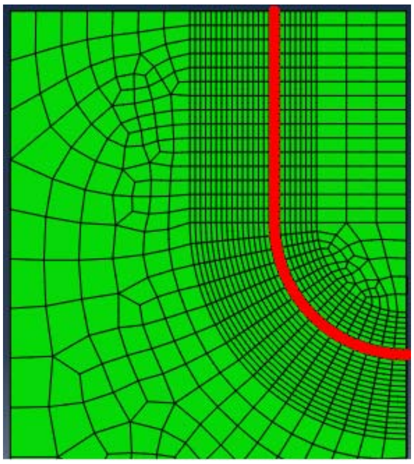

Figure 2. Semi-elliptical small and large surface crack.

The singularity at the vicinity of the crack tip is modelled by small elements with 0.2×0.2 size in order to increase the precision of calculation. Also, smooth mesh transition from smaller to larger mesh elements is applied, so a finer mesh is used in the region of the crack and it proceeds continuously into mesh elements that are larger. This transition reduces the number of mesh elements in the region that is located at a considerable distance from the crack. The crack tip is designed with the focused small elements of a different number of layers around cracks which also depends on crack geometry.

Details concerning the geometries of tensile panels with semi-elliptical surface cracks in the central region are given in Table 2. In the FE computation the base material used is Sumiten 80P. The mechanical properties of the material are introduced in the finite element code as a set of numerical data from true stress-strain curves drawn from previous tensile tests.

Table 1. Mesh details for 3D tensile panels.

Model description	Element type	Elements	Nodes
BMSC	C3D20R	16350	71791
BMLC	C3D20R	22555	97287

Table 2.Geometries of centre cracked tensile panels.

Specimen	t	$2W$	$2c$	a_o	a_o/t	a_o/c	c/W
BMSC	15	75	16	2.5	0.33	0.313	0.21
BMLC	15	75	26	5.0	0.17	0.384	0.35

The true yield stress is calculated 797 MPa, as shown in Fig. 3. The elastic properties of the material are assigned in a different module in the program through elastic modulus $E = 206843$ MPa and Poisson's ratio $\nu = 0.3$ /11/.

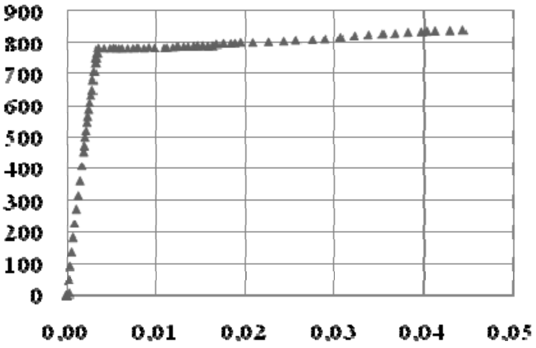

Figure 3. True stress-strain curve of Summiten 80P.

The boundary conditions in both 3D models are symmetrically applied on boundary surfaces in X-Z plane (Y = 0) and the Y-Z plane (X = 0) in the finite element model. The uniform 5 mm displacement, representing the force in X direction, is applied to the remote end.

In order to ensure the accuracy of performed analyses, all models are previously calibrated numerically on tensile specimens without crack, through assigning changes in data input and mesh techniques, as shown in /12/.

RESULTS AND DISCUSSION

J-integral values obtained for models BMSC and BMLC are plotted vs. strain and their values are compared with experimentally obtained data. Also, comparison between experimental and numerical results for J-CMOD are plotted for both tensile panels with large and small cracks. Results for models BMSC and BMLC are shown in Figs. 4-5 and in Figs. 6-7, respectively.

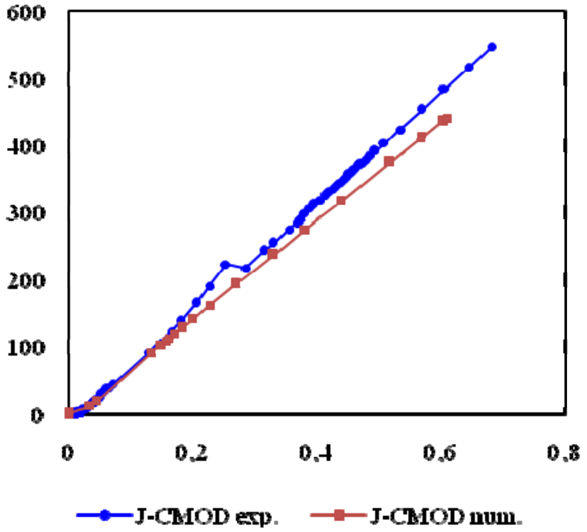

Figure 4. J-CMOD for tensile panel with small crack.

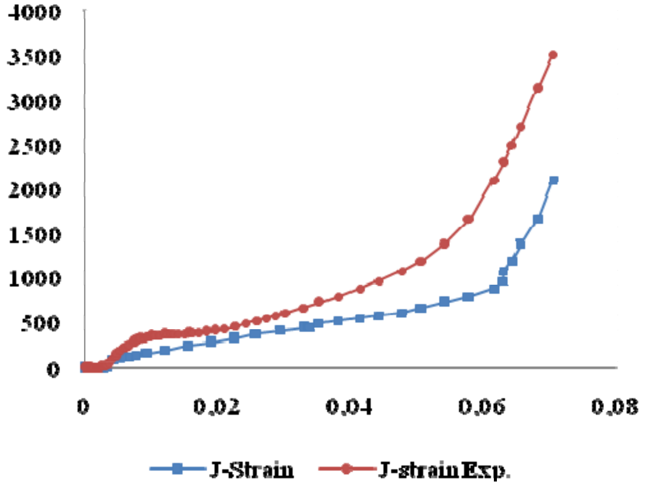

Figure 5. J vs. strain for tensile panel with small crack.

It should be noted that the calculated parameters result from elastic-plastic analyses without considering ductile tearing. Although the prevailing tendency of increasing toughness with increasing strain is evident for both cases, the assessment of obtained data is influenced by degree of scatter. In general, the results are promising and show good agreement with experimental results, but for more accurate results further corrections in properties and mesh density should be done.

It is clear that J-integral values increase with CMOD values when the load increases. However, the rate of increase in J-integral values is different for tensile panels with small and large cracks. Relationship between J and CMOD is almost linear for tensile panel with a small crack. For a tensile panel with large crack this relationship appears to be curvilinear. For all of the investigated specimens, it appears that part of the curves are almost overlapping at low load levels. With further increases in load, the curves tend to separate. This is especially the case for specimens with large cracks. In addition, the rate of increase of J-integral values are approximately the same for both models.

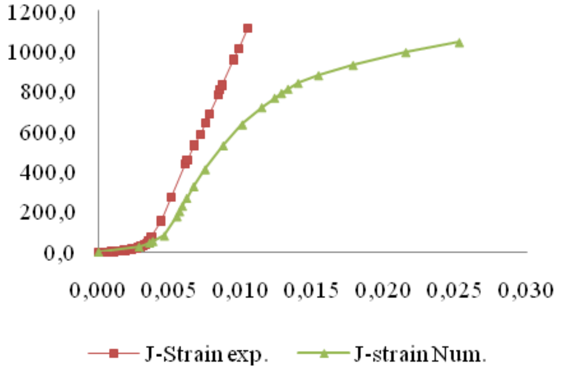

Figure 6. J vs. strain for tensile panel with large crack.

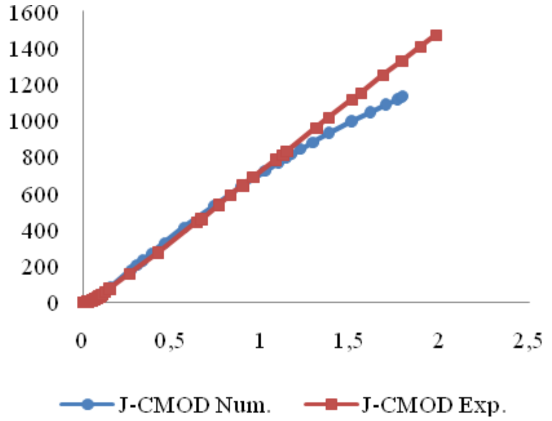

Figure 7. J-CMOD comparison for tensile panel with large crack.

As indicated from the above the difference in crack size may have a weak influence on the relationship J-integral and CMOD values at low load levels. Strong effects exist at high load levels, where local deformation plays an important role. Thus, it is expected that the relationship between J-integral and CTOD can vary significantly depending on material properties and load conditions. Both models have different mesh qualities which is eminent from the results scatter when comparing both numerical models. Figures 8-9 show stress distribution at the crack tip for BMLC and BMSC models, respectively.

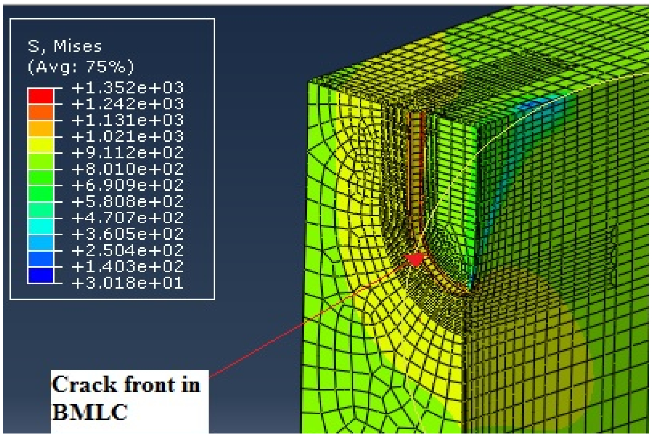

Figure 8. Results obtained from BMLC, with details on mesh and stress distribution on the tip of the crack.

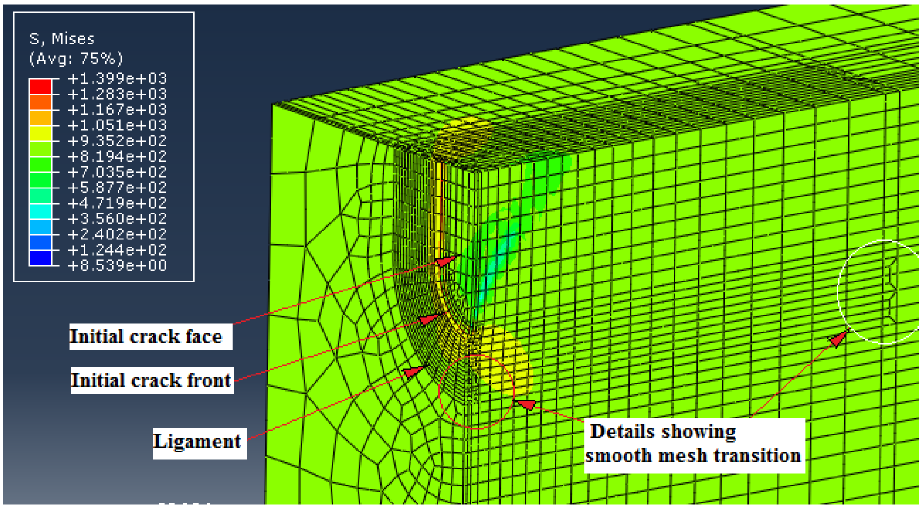

Figure 9. Results obtained from BMSC, with details on mesh and stress distribution on crack tip.

CONCLUSION

The comparison between experimental and numerical results for tensile panels with different crack length and width reveals understandable differences that arise from many factors among which is the crack itself, since it is the place where stress concentrations occur. The J-CMOD curves for experimental and numerical results are different due to differences in a/W ratio that in a way describes crack geometry. In the elastic-plastic regime for the large crack specimen, plastic flow is confined to the ligament of the specimen and high stress concentration is maintained at the tip. However, for the small crack specimen the plastic flow spreads to the free surface behind the crack tip and the yielding to free surface causes a loss of crack tip constraint. In this case the small crack specimen must undergo more crack tip blunting and plastic deformation than large crack specimens in order to develop the same critical stress and strain at the crack tip required to cause the onset of crack growth, /13/. Thus, crack size, which is related to plastic constraint of the crack tip, must have an important influence on the relationship between J-integral and CMOD. Although some recent results show that the crack length appears to have very little effect on the plastic constraint factor /14/, in this investigation it has been shown that crack lengths have a strong influence on the relationship between the J-integral and CMOD for both numerical and experimental results.

The considered numerical analysis has an important role in further development of numerical models that will provide solid estimation of mechanical properties in ductile weld metals which are considered to be homogenous, as well as in heterogeneous materials such as the narrow heat affected zone (HAZ) in complex asymmetric joints.

ACKNOWLEDGEMENT

Authors wish to thank the Ministry of Education, Science and Technological Development of Serbia for supporting the paper as part of the project: 174004 ON.

REFERENCES

1. Lee, H., Cho, J., *Overload analysis and fatigue life prediction of spot-welded specimens using an effective J-integral*, Mechanics of Materials 2005, 37(1):19-32.
2. Hyungyil, L., Yun-Jae, K., *Interfacial crack-tip constraints and J-integral for bi-materials with plastic hardening mismatch*, Int. J of Fracture, 2007; 143:231-243.
3. Milouda, M.H., Bahria, O., Alia, B., Mohamed, B., Benattouc, B., *Effect of the mechanical properties and mode loading on the mechanical behaviour of weldment: a numerical analysis*, Materials Research. 2013, 16(4): 853-859.
4. Jianqing, F., Yaowu, S., *Mechanical heterogeneity and validity of J-dominance in welded joints*, Int. J of Fracture, 2000; 106:311-320.
5. Rice, J.R., *A path independent integral and the approximate analysis of strain concentration by notches and cracks*, J of Applied Mechanics. 1968; 35(2):379-386.
6. Casavola, C., Pappalettere, C., Tattoli, F., *Experimental and numerical study of static and fatigue properties of titanium alloy welded joints*, Mechanics of Materials. 2009, 41:231-243.
7. Jiang, W.C., Wang, B.Y., Gong, J.M., Tu, S.T., *Finite element analysis of the effect of welding heat input and layer number on residual stress in repair welds for a stainless steel clad plate*, Materials & Design, 2011; 32:2851-2857.
8. Panda, S.K., Kumar, D.R., *Experimental and numerical studies on the forming behavior of tailor welded steel sheets in biaxial stretch forming*, Materials & Design. 2010; 31:1365-1383.
9. British Standards Institution - BSI. Guide on methods for assessing the acceptability of flaws in structures. BSI; 1999. Guide B57910.
10. R6 - Assessment of the integrity of structures containing defects. 3rd rev. British Energy Generation Ltd; 1999.
11. Petrovski, B., Određivanje preostale čvrstoće nosivosti suda pod pritiskom sa povišenom prslinom, Univerzitet u Beogradu, Tehnološko-metalurški fakultet, Beograd 1985.
12. Doncheva, E., D.Sc. thesis: Modeling and computer simulation of initiation processes and propagation of cracks in welded structures (in Macedonian), University of Skopje, 2015.
13. Shia, Y., Suna, S., Murakawab, H., Uedab, Y., *Finite element analysis on relationships between J-integral and CTOD for stationary cracks in welded tensile specimens*, Int. J of Pres. Ves. and Piping, 1998, 75:197-202.
14. Sorem, W.A., Dodds, R.H., Jr, Rolfe, S.T., *A comparison of the J-integral and CTOD parameters for short crack specimen testing*. Welding Research Council Bul., 1990, No.351: 24-34.