Aleksandar Grbović[1], Aleksandar Sedmak[1], Ljubica Lazić-Vulićević[2], Radzeya Zaidi[1], Nikola Vitas[3], Snežana Kirin[4]*

EXTENDED FINITE ELEMENT METHOD SIMULATION OF FATIGUE CRACK GROWTH IN A CHARPY SPECIMEN

SIMULACIJA RASTA ZAMORNE PRSLINE ŠARPI EPRUVETE PRIMENOM PROŠIRENE METODE KONAČNIH ELEMENATA

Originalni naučni rad / Original scientific paper
UDK /UDC:

Rad primljen / Paper received: 22.08.2023

Adresa autora / Author's address:
[1)] University of Belgrade, Faculty of Mechanical Engineering, Serbia [*]email: agrbovic@mas.bg.ac.rs
[2)] High Technical School of Professional Studies, Zrenjanin, Serbia
[3)] Ministry of Economy, Republic of Serbia
[4)] University of Belgrade, Innovation Centre of the Faculty of Mechanical Engineering, Belgrade, Serbia

Keywords
- extended finite element method
- fatigue crack growth
- Charpy specimen

Abstract

Research presented here demonstrates the application of the extended finite element method (XFEM) in modelling of fatigue crack growth. This method provides certain advantages to classic approaches since it does not need to generate a new finite element mesh around the crack tip, saving considerable time and making XFEM simulations very efficient. In this specific case, the focus is on 3D fatigue crack growth simulations in Charpy specimens, performed using ABAQUS© software and its XFEM extension, Morfeo.

Ključne reči
- proširena metoda konačnih elemenata
- rast zamorne prsline
- Šarpi epruveta

Izvod

U radu je predstavljena primena proširene metode konačnih elemenata na modeliranje rasta zamorne prsline. Ova metoda poseduje određene prednosti u odnosu na klasičan pristup, budući da ne zahteva generisanje nove mreže konačnih elemenata u okolini vrha prsline, što u velikoj meri štedi na vremenu i čini simulacije PMKE veoma efikasnim. U ovom slučaju je fokus na 3D simulaciji rasta prsline u Šarpi epruvetama, primenom softverskog paketa ABAQUS© i njegove ekstenzije za zamor, Morfeo.

INTRODUCTION

Modelling of cracks by using classical FEM, requires mesh to contain discontinuity of geometry, /1/. Problem is even more complicated if crack growth is simulated, requiring successive generation of a mesh. To overcome this problem, the Extended finite element method (XFEM) was introduced back in nineties, /2/, based on a mesh without discontinuity (crack), which is modelled by using additional approximation, i.e. enrichment functions:
- discontinuous Heaviside function - $H(x)$, defining the displacement field on crack surfaces;
- set of linear elastic asymptotic displacement functions, describing displacements at the crack tip, the so-called Near Tip functions - $N(x)$.

The XFEM, also known as the Generalised Finite Element Method (GFEM) or Partition of Unity Method (PUM), has proved to be a competent mathematical tool since it allows the presence of discontinuities in an element. In comparison to the classical finite element method, the XFEM provides significant benefits in the numerical modelling of crack propagation. In the traditional formulation of the FEM, the existence of a crack is modelled by requiring the crack to follow element edges. In contrast, the crack geometry in the XFEM needs not to be aligned with element edges which provides flexibility and versatility in modelling. In addition, nodes surrounding the crack tip are enriched with DOFs associated with functions that reproduce asymptotic LEFM fields. This enables the modelling of the crack discontinuity within the crack tip and substantially increases the accuracy in the computation of stress intensity factors (SIFs). There are numerous examples of XFEM applications to illustrate its versatility to solve practical problems, /3-9/.

In this paper the XFEM is used to simulate 3D fatigue crack growth experimentally observed on a standard Charpy specimen during testing with a RUMUL Cracktronic device. Results obtained by ABAQUS /10/ and MORFEO /11/ are used to compare with experimental results on a Charpy specimen.

ABAQUS AND MORFEO/ABAQUS

Regarding commercial software, 3D XFEM was first available in ABAQUS, after MORFEO/ABAQUS was introduced as a post-processing option to calculate stress intensity factors and number of cycles needed for crack growth from initial- to any other size. Two options are provided for crack growth: a forced crack growth in the plane (force in-plane propagation) which is controlled by the user, and a free crack growth, automatically done by the software. In the latter case, the first step in 3D crack growth analysis using XFEM is to calculate stresses and stress intensity factors, as well as the deflection angle in relation to the initial crack growth direction, in order to ‘open’ the crack. In any case, crack growth is simulated in a number of steps.

CHARPY SPECIMEN MODELLING

The simulation of crack growth using XFEM is used to model the standard Charpy specimen made of API J55 steel, Fig. 1, as described in /12, 13/. The Charpy specimen model is defined in ABAQUS as well as the characteristics of the material (Young's modulus is 2.1×10^5 MPa, Poisson's coefficient is 0.33), bending stress value 7 MPa, and corresponding boundary conditions, /12, 13/.

The finite element mesh consisted of 67254 hexahedral elements and 72699 nodes, Fig. 1 (top). The notch, 2 mm deep, is treated as the initial crack length. Figure 1 (bottom) shows the initial crack (notch) on a standard Charpy specimen and the mesh around it, /12/.

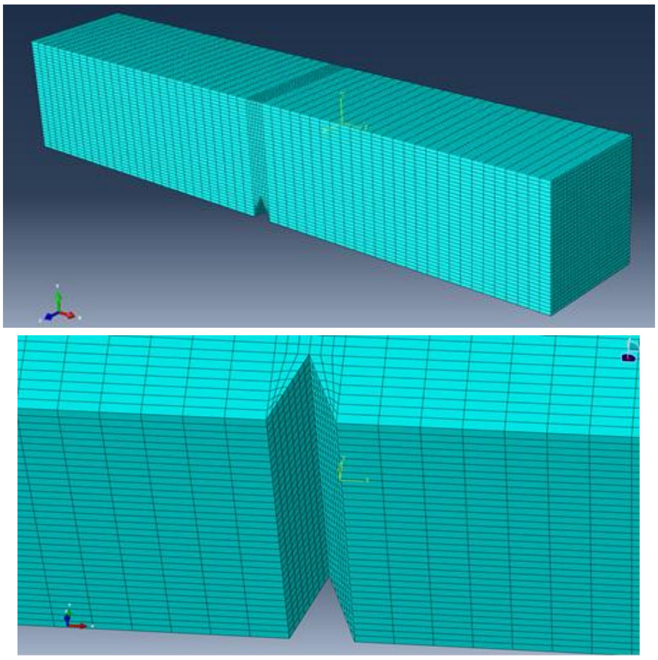
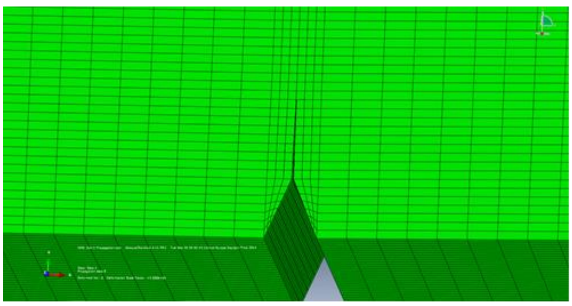


Figure 1. Finite element mesh of Charpy specimen model (top); initial crack (notch) in a standard Charpy specimen (bottom).

Here, the option of free crack growth with a step of 0.3 mm is chosen and its growth is monitored in 28 steps. The crack grew through an unchanged mesh of elements in the vertical plane (y direction in the figure), almost exactly between two rows of hexahedral elements (Fig. 2), slight deviation from that direction to the right (in the first few steps, by 0.006 mm, and from the tenth step all the way to the end, to some 0.03 mm), and then continued to grow through hexahedral elements. Stress states around the crack tip after the eighth, fourteenth, and twenty-seventh propagation steps are shown in Fig. 3.

a)



b)

c)

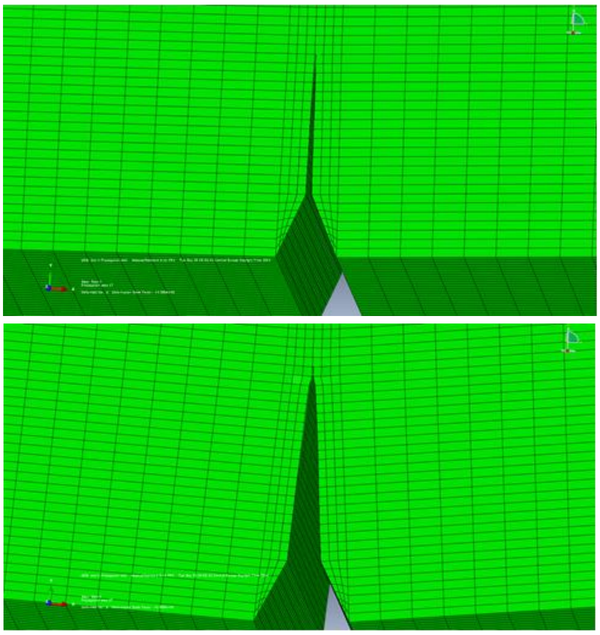


Figure 2. Crack growth after steps: a) 8th; b)17th; c) 27th.

a)

b)

c)

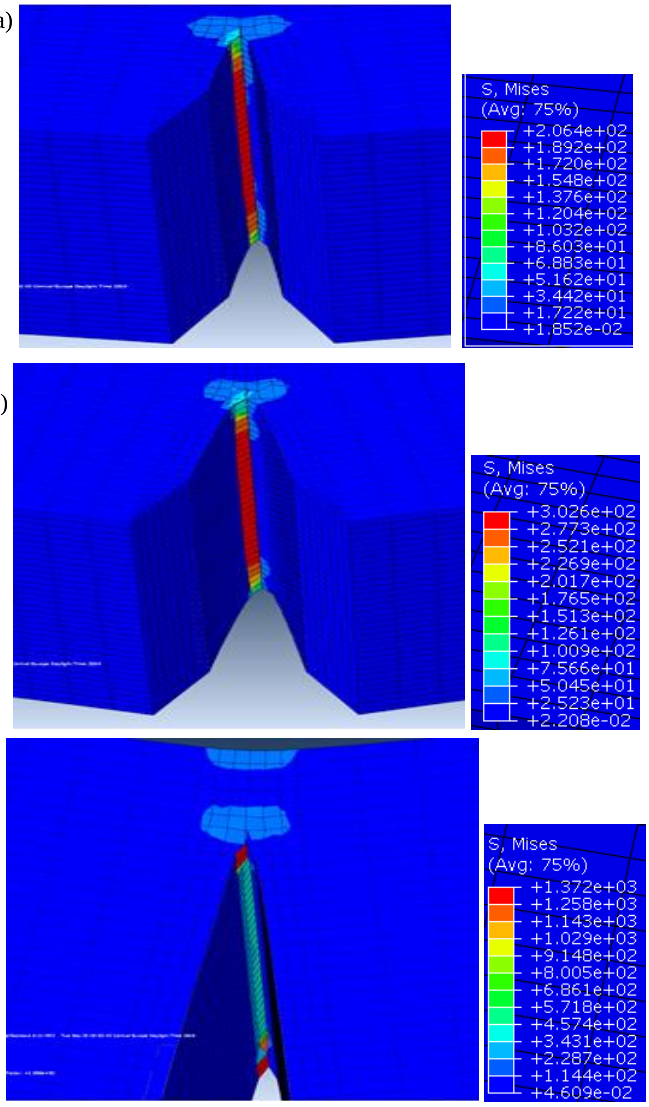


Figure 3. Von Mises stress state after: a) 8; b) 14; c) 27 crack growth steps.

The number of crack front points is the same in each crack propagation step (from 66 to 68, as shown in /12/), which means that the crack spreads through an equal number of network elements. As can be seen from Fig. 4, the crack growth per step was smaller than that initially given, 0.3 mm. This given step of 0.3 mm is the maximal value the crack growth per step should reach, but Morfeo/ABAQUS ‘chose’ the growth that might be more appropriate (within the given limits).

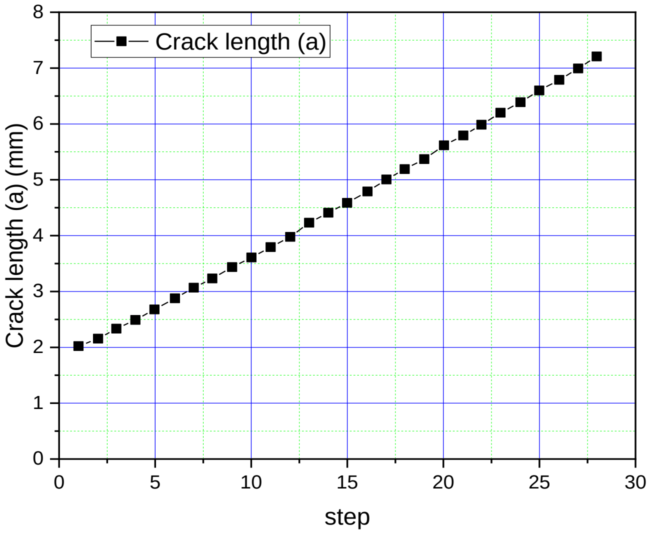


Figure 4. Crack growth per step.

Figure 5 shows the obtained dependence of the equivalent stress intensity factor K_{eq} and crack length a. One should notice that the use of K_{eq} is the most suitable in this case because it has sublimated the SIF values for all three forms. As presented and explained in more detail in /12/, it is obvious that the values for K_I are dominant, so it also would not be a mistake to consider only its values. The required number of cycles for crack propagation steps is shown in Fig. 6. Finally, comparison between experimental and numerical results is shown in Fig. 7, indicting excellent agreement for small and large number of cycles. Although some discrepancies appear at mid-range, the overall agreement is still excellent, since practically the same remaining life is obtained in the numerical simulation as well as in the experiment.

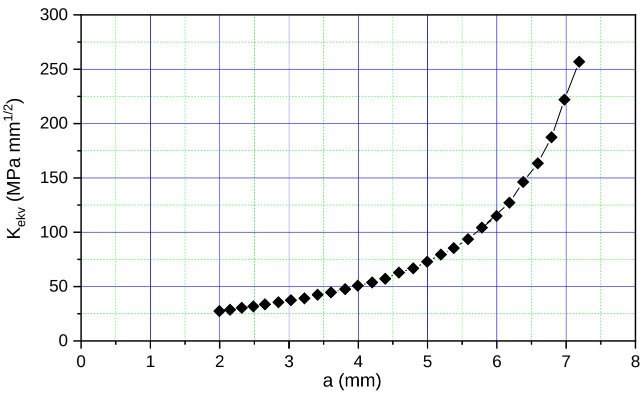


Figure 5. Mean value of K_{eq} vs. crack length a.

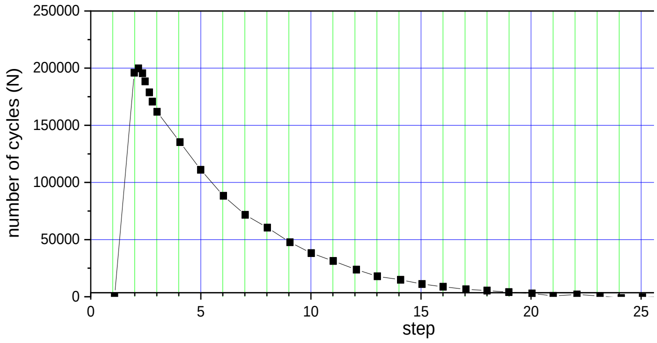


Figure 6. Required number of cycles for crack propagation per step (3D simulation)

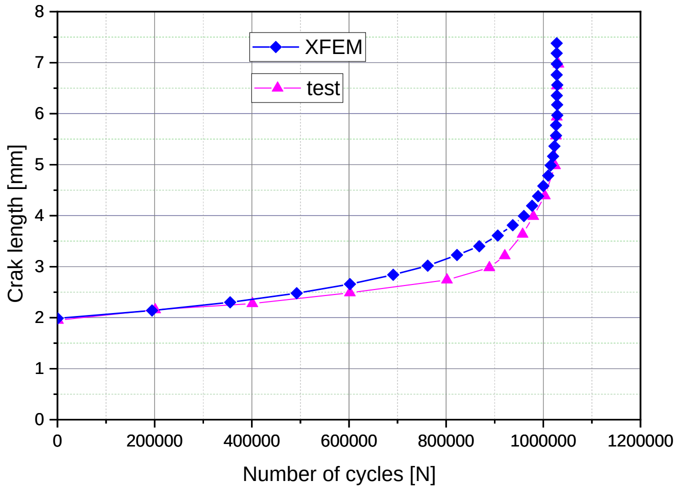


Figure 7. Comparison - experiment and XFEM results.

DISCUSSION AND CONCLUSIONS

By comparing the results obtained by 3D simulation and the results obtained by experiment on a Charpy test tube, a very good match is observed, especially from the moment when the crack increases over a value of 4 mm. For shorter cracks, a small deviation of results exists which can be attributed to the fact that the simulation was done for an ideally homogeneous material, and experimental values are obtained for real. Experimental and 3D results match for crack length values above 4 mm up to fracture. The simulation is excellent, which is of great importance because fracture occurred at the same time in the experiment and in the numerical simulation. On the other hand, the deviation of crack growth rate in the part of the diagram cannot be considered as a disadvantage. It just contributes to prediction of life to failure by XFEM, i.e., the reliability of this method, because it can be seen that the crack grows slower in real conditions than according to 3D simulation. This means that the results of lifetime-to-fracture prediction of specimen obtained by XFEM from this material could certainly be taken as relevant and sufficiently reliable because they represent conditions that are stricter than the realistic. As already mentioned, XFEM is a relatively new method, and in order to gain affirmation and more applications in practice, the results obtained by this method must be supported by practical results. The aim of this comparison of experimental- and data obtained by 3D simulation is precisely the verification of this new method with some certainty. Otherwise, as for the use of XFEM for simple geometries as is the case with this test tube, in order to obtain relevant fatigue parameters, it does not make any excessive sense, because these values can be obtained in a significantly shorter time by using an analytical method.

REFERENCES

1. Sedmak, A. (2018), *Computational fracture mechanics: An overview from early efforts to recent achievements*, Fatigue Fract. Eng. Mater. Struct. 41(12): 2438-2474. doi: 10.1111/ffe.12912
2. Moës, N., Dolbow, J., Belytschko, T. (1999), *A finite element method for crack growth without remeshing*, Int. J Numer. Meth. Eng. 46(1): 131-150. doi: 10.1002/(SICI)1097-0207(19990910)46:1<131::AID-NME726>3.0.CO;2-J
3. Kirin, S., Sedmak, A., Zaidi, R., et al. (2020), *Comparison of experimental, numerical and analytical risk assessment of oil drilling rig welded pipe based on fracture mechanics parameters*, Eng. Fail. Anal. 114: 104600. doi: 10.1016/j.engfailanal.2020.104600
4. Sedmak, A., Čolić, K., Grbović, A., et al. (2019), *Numerical analysis of fatigue crack growth of hip implant*, Eng. Fract. Mech. 216: 106492. doi: 10.1016/j.engfracmech.2019.106492
5. Sedmak, A., Grbović, A., Petrovski, B., et al. (2022), *The effects of welded clips on fatigue crack growth in AA6156 T6 panels*, Int. J Fatigue, 165: 107162. doi: 10.1016/j.ijfatigue.2022.107162
6. Alshoaibi, A.M. (2022), *Fatigue crack growth analysis under constant amplitude loading using finite element method*, Materials (Basel), 15: 2937. doi: 10.3390/ma15082937
7. Kumar, S., Singh, I.V., Mishra, B.K. (2015), *A homogenized XFEM approach to simulate fatigue crack growth problems*, Comp. Struct. 150: 1-22. doi: 10.1016/j.compstruc.2014.12.008
8. Solob, A., Grbović, A, Božić Ž., Sedmak, S.A. (2020), *XFEM based analysis of fatigue crack growth in damaged wing-fuselage attachment lug*, Eng. Fail. Anal. 112: 104516. doi: 10.1016/j.engfailanal.2020.104516
9. Balać, M., Grbović, A., Petrović, A., Popović, V. (2018), *FEM analysis of pressure vessel with an investigation of crack growth on cylindrical surface*, Eksploatacja i Niezawodność - Maint. Reliab. 20(3): 378-386. doi: 10.17531/ein.2018.3.5
10. Abaqus 6.12, Abaqus/CAE User's Manual. Dassault Systèmes, 2012, Simulia Corp., Providence, RI, USA.
11. Cenaero. Morfeo/crack for Abaqus, https://dial.uclouvain.be/pr/boreal/fr/object/boreal%3A134972/datastream/PDF_01/view
12. Zaidi, R., *Application of fracture mechanics parameters to residual life assessment of welded pipes exploitation under fatigue loading*, Ph.D. Thesis, University of Belgrade, Faculty of Mechanical Engineering, Belgrade, 2021.
13. Lazić-Vulićević, Lj., Rajić, A., Grbović, A., et al. (2016), *Fatigue life prediction of casing welded pipes by using the extended finite element method*, Frattura ed Integrità Strutturale, 10(36): 46-54. doi: 10.3221/IGF-ESIS.36.05