SIE 2022
8th International Conference on Industrial Engineering
29th-30th September 2022

APPLICATION OF FINITE ELEMENT METHOD FOR ANALYSIS THE PARAMETERS OF FRACTURE MECHANICS

Milovan Paunić[1], Vladimir Černicin[2],
[1]Innovation Center, Faculty of Mechanical Engineering, University of Belgrade, Serbia
[2]Faculty of Mechanical Engineering, University of Belgrade, Serbia
mpaunic@mas.bg.ac.rs

Abstract. *When making mechanical components, geometric imperfections are common, which in certain cases can cause a weak spot or some form of crack. Depending on the intensity and position of the action of forces on the mechanical component, its bearing capacity or load that the structure can carry with a given degree of safety directly results. The paper will compare the critical values of the stress intensity factor, as a basic element of fracture mechanics, obtained by empirical formulas and the values obtained by applying the finite element method. Using Abaqus, the values were verified numerically and the results of the analysis can be considered validated. The results showed deviations in the acceptable range and the application of the finite element method to the analysis of cracks in the material, confirmed that Abaqus can also be used as an excellent tool in the analysis of fracture mechanics.*

Key words: *finite element method, stress intensity factor, crack, fracture mechanics*

1. INTRODUCTION

The finite element method is a modern method of numerical analysis and falls within the methods of discrete analysis. The application of FEM first began in the field of engineering systems calculations. The main idea is that the area which is representing a continuum with infinitely many degrees of freedom can be replaced by a discreet model of interconnected elements with the final number of degrees of freedom. FEM is based on the physical discretization of the system, unlike other numerical methods where mathematical discretization is performed[5]. Therefore, instead of elements of differential small dimensions, we have areas of final dimensions or final elements. The state of individual elements is then described in ordinary algebraic equations, which is much simpler than in the case of differential or integral equations when applying mathematical discretization.

2. STRESS CONCENTRATION

In this paper, there will be analyzed application of the finite element method to examine the deviations of fracture mechanics parameters. All engineering components and structures contain certain geometric irregularities or discontinuities [1,2]. The size and shape of these irregularities directly affect the integrity of the mechanical component in working conditions[4].

The integrity of the component which contains the defect is calculated by analyzing stress concentrations caused by the existence of discontinuity. Depending on the tip of the defect, the results may vary widely – if it is a circular opening, it has a much less effect on the reduction of structural integrity compared to the sharp tip defect (crack).

When the crack starts to grow, there are different techniques by which crack propagation can be stopped [3, 6]. Fracture mechanics represents a group of theories that describe the behavior of structures that have geometric discontinuity or study fracture phenomena. Fracture mechanics can be divided into linear elastic fracture mechanics and elastoplastic fracture mechanics. Linear elastic fracture mechanics obtain excellent results for materials such as steel, glass, and concrete, while for materials such as low-carbon steel, stainless steel, aluminum alloys, and polymers, significant plasticity will occur before breaking. Anyway, when the loads are low enough, linear elastic fracture mechanics provide a good assessment of physical reality.

The stress state at the tip of the crack is described as the ratio of maximum (elastic) stress at the crack tip and nominal (peripheral) load that acts on the plate. One practical formula used as a good approximating of elliptical irregularities in the material is given by the expression

$$K = \frac{\sigma_{max}}{\sigma_{nom}} = 1 + 2\sqrt{\frac{a}{\rho}}$$

Where the value *a* represent the size of a semi-major axis while *ρ* represent tip radius. In the case that it was a circular opening instead of an ellipse, the values *ρ* and *a* would be equal to$\sigma_{max} = 3\sigma_{nom}$

It is important to note that the local maximum stress at the crack tip, in which the failure occurs, is not strictly constant but directly depends on the size of the ellipse semi-axis which gradually increases as the force grows through the material. For ductile materials that express a characteristic of high plasticity, the cracks are less dangerous because their growth is limited by local plastic deformations at the crack tip that prevents further spread.

From the previous equation, it is observed that if we reduce the value of the radius of the crack tip to a very small value (atom size 10^{-9}m), the value of factor K tends to be infinite. In this case, the maximum stress expression:

$$\sigma_{max} = \lim_{\rho\to 0} \sigma_{nom}(1 + 2\sqrt{\frac{a}{\rho}}) \approx 2\sigma_{nom}\lim_{\rho\to 0}\sqrt{\frac{a}{\rho}}$$

because the first member of the bracket is negligibly small compared to the second. If both sides of equalition are expanded by $\frac{1}{2}\sqrt{\pi\rho}$

$$\frac{1}{2}\sqrt{\pi\rho}\,\sigma_{max} = \frac{1}{2}\sqrt{\pi\rho}\left(2\sigma_{nom}\sqrt{\frac{a}{\rho}}\right) = \sigma_{nom}\sqrt{\pi a}$$

3. CRACK ANALYSIS

In this chapter, the previously mentioned case of the crack in the material will be analyzed and how it is possible to obtain a match of theory and simulation by using Abaqus. The analysis will be carried out for 3 cases of the crack position:

a) central crack in an infinite plate
b) central crack in a plate of limited width
c) edge crack in an infinite plate

In all three cases, the peripheral stress $\sigma = 100$ MPa was given while the semi-major axis of crack was a=5mm. In the second case, the width of the plate is *W=40mm*. Under each type of crack, there are given theoretical formulas. For the first case, if the semi-axis of the crack is 5mm, the following results are obtained:

$$K_I = \sigma\sqrt{\pi a}$$

$$K_I = 100MPa\sqrt{3.14 * 0.005} = 12.53314 = 12.53\ \text{MPa}\sqrt{m}$$

Unlike previous cases, steps in Abaqus will not be detailed. The idea is to show that FEM can be used as a tool for fracture mechanics to analyze cracks in the material. The system is modeled as a 2D deformed shell large enough not to affect the results of the analysis. The remote stress is 100 MPa, and the half-axis of the crack is 5mm. The crack is modeled in the special tab in the interaction module.

The results of the simulation for the first case were presented in figure 1. The model is shown only partially, to zoom in on a piece of interest. In boundary conditions, the lower part of the plate is fixed to the vertical axis, while pressure with the sign minus is selected to achieve stretching, so -100MPa is set to operate on the upper surface, which can be seen in figure 1. The left vertical edge of the plate is set to represent body symmetry – on the model this is the edge crack, but with this condition this crack is central.

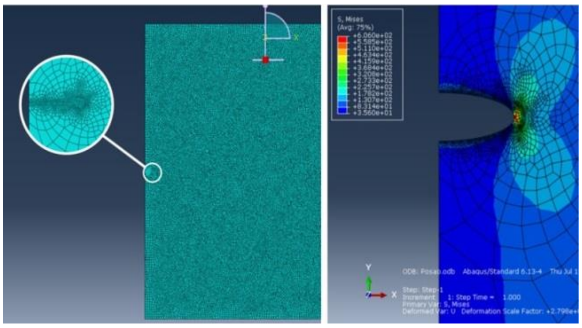

***Figure 1**. Results for the first case of central crack in an infinite plate*

The results are 394 MPa $\sqrt{mm}$ which can be seen in figure 1. As units are used in the program in accordance with millimeters, the resulting value should be multiplied by $\sqrt{0.001m}$ and then a value of 12.45 MPa $\sqrt{m}$ is returned. Compared to the firstly calculated 12.53 that's an acceptable deviation. For the second case of a *W=40mm*, the input parameters are the same as in the first case. The formula for obtaining stress intensity factors is given to the following formula:

$$K_I = \sqrt{sec\frac{\pi a}{W}}\sigma\sqrt{\pi a} = 1.0823 * 12.53314 = 13.56462 = 13.56\ \text{MPa}\sqrt{m}$$

The results of the simulation were presented in figure 2. The model is also shown only partially with prominent details of interest. Identical to the previous case, the left edge is defined in boundary conditions as body symmetry so the program looks at the model as the case of the central crack.

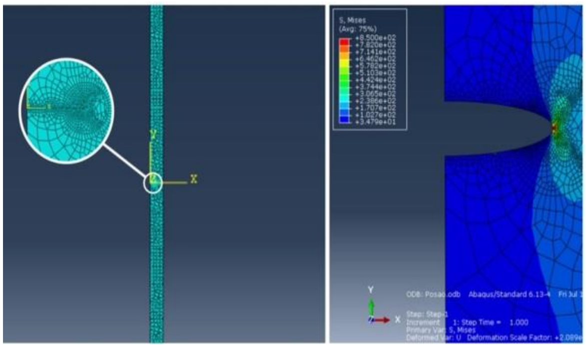

***Figure 2**. Results for the second case of central crack in a plate of limited width*

After the simulation, a results of 417 MPa $\sqrt{mm}$ was obtained. As units are used in the program in accordance with millimeters, the resulting value should be multiplied by $\sqrt{0.001m}$ and then returned a value of 13.18 MPa $\sqrt{m}$. Compared to the previously calculated 13.56 that's a deviation of about 3% where the possible cause can be mesh density.

For the third case or case of an infinite plate with an edge crack, the input parameters are identical and the calculation formula is given in the following form:

$$K_I = 1.12\sigma\sqrt{\pi a}$$

$$K_I = 1.12 * 100MPa\sqrt{3.14 * 0.005}$$

$$= 14.0371168 \text{ MPa}\sqrt{m}$$

Compared to the previous two cases, the only limit is a fixed lower part of the plate. This time, it shouldn't be used the boundary requirement that the left vertical edge represents body symmetry, which directly defines the edge crack. The results of the analysis were presented in figure 3.

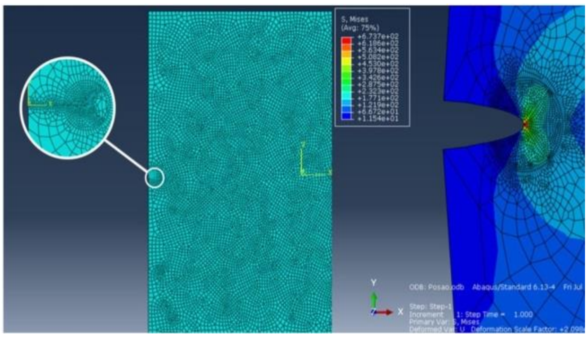

***Figure 3**. Results for the second case of edge crack in an infinite plate*

It is interesting to note that the edge crack has opened up more (figure 3) than the central ones (the reason is the edge conditions of body symmetry). In the first two cases of the central crack, the form of the ellipse was preserved and the edge is vertical in that area, while in the case of the edge crack (figure 3), the form of the ellipse is changed and the left edge is no longer vertical.

After the simulation, the result is 452 MPa $\sqrt{mm}$. As units are used in the program in accordance with millimeters, the resulting value should be multiplied by $\sqrt{0.001m}$ and then a value of 14.29MPa $\sqrt{m}$ is obtained. Compared to the previously calculated 14.03 this is a deviation of about 2% which can be considered an acceptable deviation.

As you can conclude from several previous examples, Abaqus is an exceptional program for calculating structures and it is suitable for analysis parameters of fracture mechanics. The simplest examples can be calculated using basic formulas, but for complex models, programming is the only optimal way to resolve them..

4. ADVANTAGES AND DISADVANTAGES

In general, the advantage is that you can obtain very accurate results and the disadvantage is that the engineer must be experienced enough to know if the results are appropriate. Linear analysis is considered valid in the small deformation zone (0.2%) and this zone can be approximated by the linear relationship between stress and deformation. In this case, small deformations do not affect the behavior of the structure and there is no stability failure resulting from the fact that the stiffness matrix does not change. In this case, the calculation is significantly simplified because the rigidity matrix needs to be calculated only once.

At the moment when the stress causes the yield of material, linear analysis cannot be considered because the material is located in the zone where the nonlinear analysis must be applied. Yang's module directly affects the stiffness matrix, therefore it is necessary to write and adjust the matrix from moment to moment. Accepting information obtained by linear analysis after the yield stress leads to the incorrect results.

The origin of this stress concentration may be due to inadequate design or due to irregularities in the material. Stress concentration is a common occurrence in more complex geometry, so this issue is of great importance. If it is assumed that the irregularity of the material (crack) caused the stress concentration, the obtained results cannot be considered valid. Cracks are a common occurrence in materials and there is great importance of their analysis in fracture mechanics. Cracks are "weak points" which activate if yield stress is achieved for crack propagation.

5. VALIDATION

In this chapter, a comparative analysis of numerical results and results obtained by Abaqus will be performed. If the results deviate in an acceptable percentage, it will be confirmed that Abaqus generates accurate results in a much simpler and faster way if the engineer is able to adequately set the input parameters. This paper has shown that the deviations are in an acceptable range. If the design needs to be modified, all previous cases can be easy modifyed in the program while changing the numeric part is a time-demanding job, and this reflects the importance of using the software that applies the finite element method. Since all result deviations relative to numerical analysis are in an acceptable range of up to 3%, a review of the results has been performed and they are validated.

6. CONCLUSION

The maximum error in this work was in an acceptable range of about 3%, which is great given the rounding of values for a simpler calculation. Also, a check on the finite element mesh convergence was done with the aim of obtaining approximately the same results. The application of Abaqus verified numerical analyses and the results can be considered valid. As previously mentioned in the work, a simulation without validation is just a colorful image and the solver must not be seen as a black box. The best way is to have precalculations to have some predictions before simulation.

In order for an engineer to really stands by the simulation results, validation is the only possible path that involves proving minimal deviations. In the case of complex geometry, numeric analysis cannot be expected - a program solver is the only and optimal process. As you can see from the analysis of previous examples of plates, Abaqus quickly generated results with acceptable deviations, and the application of FME to analyze the crack in the material confirmed that Abaqus can also be used as an excellent tool in the analysis of the fracture mechanics.

ACKNOWLEDGMENTS

This work was supported by the Ministry of Education, Science and Technological Development, Republic of Serbia and European Union's Horizon 2020 research and innovation program.

REFERENCES

[1] Byskov, E. The calculation of stress intensity factors using the finite element method with cracked elements. *Int J Fract* **6,** 159–167 (1970). https://doi.org/10.1007/BF00189823

[2] Naoki Miura, Yukio Takahashi, Hiroshi Shibamoto, Kazuhiko Inoue, Comparison of stress intensity factor solutions for cylinders with axial and circumferential cracks, Nuclear Engineering and Design, Volume 238, Issue 2, 2008, Pages 423-434, ISSN 0029-5493, https://doi.org/10.1016/j.nucengdes.2006.09.015.

[3] S.M.J. Razavi, M.R. Ayatollahi, C. Sommitsch, C. Moser, Retardation of fatigue crack growth in high strength steel S690 using a modified stop-hole technique, Engineering Fracture Mechanics, Volume 169, 2017, Pages 226-237, ISSN 0013-7944, https://doi.org/10.1016/j.engfracmech.2016.11.013.

[4] W. Yi, Q-h. Rao, S. Luo, Q-q. Shen, Z. Li, A New Integral Equation Method for Calculating Interacting Stress Intensity Factor of Multiple Crack-hole Problem, Theoretical and Applied Fracture Mechanics (2020), doi: https://doi.org/10.1016/j.tafmec.2020.102535

[5] Wenyang Liu, Jung-Wuk Hong, A coupling approach of discretized peridynamics with finite element method, Computer Methods in Applied Mechanics and Engineering, Volumes 245–246, 2012, Pages 163-175, ISSN 0045-7825, https://doi.org/10.1016/j.cma.2012.07.006.

[6] Yue, Z.W., Yang, L.Y., Wang, Y.B., 2013. Experimental study of crack propagation in polymethyl methacrylate material with double holes under the directional controlled blasting. Fatigue and Fracture of Engineering Materials and Structures, 36, 827-833.