FRACTURE TOUGHNESS NUMERICAL MODELLING OF THE MARTENSITIC CHROMIUM STEEL

DUNDULIS Gintautas, JANULIONIS Remigijus, MAKAREVIČIUS Vidas, GRYBĖNAS Albertas

Lithuanian Energy Institute, Kaunas, Lithuania, Gintautas.Dundulis@lei.lt

Abstract

To evaluate the resistance to failure of the constructions with cracks usually the fracture criterions are used. One of such criterion used to evaluate failure is J-integral. Calculated value of failure criteria in construction with crack is compared with its critical value for the material the construction is made of. The values of J-integral for the materials are usually experimentally determined. This article presents the methodology for the determination of J-integral using numerical methods. For this purpose the Finite Element (FE) code ABAQUS was used.

The modelling of the J-integral of steel P91 was performed. 2D and 3D FE models were used for this purpose. The experimental stress - strain curve was used as material description. For crack growth calculation the crack opening displacement as a function of crack length is presented. The prognosis results of J-integral were compared with experiment data.

As results of investigation the J-integral of steel P91 was determined. Prognosis results were compared with experiment. Numerically determined J-integral has acceptable coincidence with experiment results.

Keywords: Finite element method, fracture mechanic, J-integral

1. INTRODUCTION

Traditional approach in structural design and material selection is when design stress is compared to mechanical properties such as yield or tensile strength of candidate material. The material is assumed to be adequate if its strength is greater than the applied stress. Such approach together with imposed safety factor is widely used in construction designing. The approach is good for the constructions and materials with minor imperfections. However if the construction has a flaw fracture mechanic approach has to be used. It has three important variables such as applied stress, flaw size and fracture toughness. Fracture mechanics quantifies the critical combinations of these three variables [1].

There are brittle and ductile types of fracture and each type is analyzed by linear elastic fracture mechanics (LEFM) or elastic-plastic fracture mechanics (EPFM) theories [1].

LEFM refers to linear elastic material behavior when plastic strain is possible in small areas and close to crack tip. The stress intensity factor (SIF) is the parameter used in LEFM. SIF depends on construction shape and sizes and place of crack.

EPFM is not limited by plastic strain and it can be produced in whole section of analyzed construction. Crack tip opening displacement (CTOD) and J-integral are the parameters used to describe the conditions of crack tip in elastic-plastic materials and each can be used as fracture criterion.

To use SIF, CTOD or J-integral as a criterion it is necessary to determine their critical values. Usually it is done by experiment and the procedures are described in standards such as ASTM, ISO and etc. There are a lot of examples of experimental determination of critical J-integral for various materials. However it is not always possible to conduct an experiment. Sometimes it is required to test the specimen in certain environment which cannot be achieved in the laboratory, the material can be irradiated or even the volume of the sample material can be an issue because the machined specimens should be of certain size. Therefore alternative methods for determination of critical values of fracture parameter are necessary. Usually finite element method (FEM) is used as alternative method. A number of papers can be found on fracture parameters calculation using FEM [2-3]. However these papers do not present the methodology of determination of critical fracture parameter determination and only limited amount of papers can be found which discuss the current issue. Jun-Young J. et. al. [4] presents sophisticated crack growth modeling technique which is used for critical J-integral determination. However this technique requires damage analysis to be added in the model and additional three material constants have to be found. Therefore this method is complex for estimation of J-integral values.

In this paper the methodology in which J-integral resistance curve was modelled applying finite element method is presented. J-integral values were estimated according ASTM E1820 standard.

2. METHODOLOGY

The idea of numerical investigation of J-integral is numerically simulate the experiment according to actual procedures/instructions used for experiment conduction. For this purpose instructions described in ASTM E1820 standard [5] were used and computer code ABAQUS v6.11 [6] which uses finite element method (FEM) has been chosen for numerical simulation. ASTM E1820 instructions for critical stress intensity factor determination are presented in the following section.

2.1. Instructions for critical J-integral determination

According ASTM E1820, to determine the value of J-integral it is necessary to develop so called J-R curve which consist of J-integral values at a series of measured specimen crack extensions. The actual result after the experiment is data points which later are used for J-R curve construction. However not all points are good to be used for J-R curve but the points which are in the area limited by 0.15 mm and 1.5 mm exclusion lines and by J_{limit} line. To draw an exclusion line in the first place it is necessary to determine a construction line which is calculated by the following equation:

$$J = 2\sigma_Y \Delta a \quad (1)$$

where σ_Y - effective yield strength (the average of yield strength and ultimate strength) of the material, MPa; Δa - crack extension, mm.

The exclusion lines are just the parallel lines to the construction line with offset of 0.15 mm and 1.5 mm. J_{limit} is calculated by equation:

$$J_{limit} = b_0 \sigma_Y / 7.5 \quad (2)$$

where b_0 - uncracked ligament of the specimen, mm.

Using selected data points power law regression line can be constructed using a method of least squares. Also the offset line has to be determined. The offset line, the same as exclusion lines, is the line parallel to the construction line with offset of 0.2 mm. This line is used to determine the conditional J_Q value which is determined at the intersection of regression line with offset line. After J_Q is determined and if effective yield strength, specimen size and J_Q meet ASTM E1820 conditions it is possible to state that $J_Q = J_{IC}$.

2.2. Numerical models

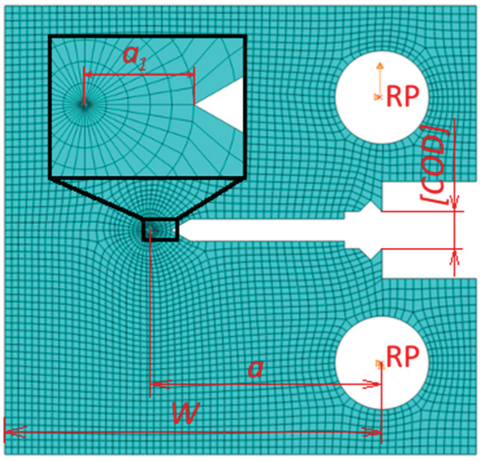
2D and 3D finite element models of Compact Tension (CT) specimen was prepared for J-integral modelling. The mesh of the models is shown in **Figure 1**. The dimensions of FE models are the same as dimension of CT specimen used in the experimental testing, which was a standard CT specimen described in ASTM E1820 with W = 50 mm and with side grooves. The only difference between 2D and 3D models is that 3D model has side grooves as the specimen used in experiment what in 2D model was not possible to model. 2D FE model can be meshed with two type elements: CPE8R and CPS8R [6]. Both are 2D plane biquadratic 8 nodes elements; however one is evaluating plane strain and other plane stress state. Plane strain state means that strains are evaluated in 2 directions and stresses are evaluated in three directions while plane stress state is in opposite. The calculation of both plane strain and plane stress states was carried out. The arithmetic average of load, crack opening displacement and J-integral of both calculations are presented as 2D model results in the article. 3D FE model was meshed with C3D20R elements [6]. These elements are quadratic brick shape and have 20 nodes.

The length of fatigue crack in FE models were the same as in CT specimen used in experiment and it was equal to a_1 = 2.78 mm (see **Figure 1b**).

For 2D model boundary conditions were added to the Reference Points (RP) placed at the centers of the holes used for pins in experiment. The displacements of one RP were restricted in two directions, i.e. along the X and Y axes, and the other RP displacements were restricted only in X axis direction. The displacement along Y axis added to last mention RP was used as load.

As the specimen is symmetric about the XZ plane only half of the model was created for 3D model. The similar boundary conditions were added as in 2D model. The displacement of RP point was restricted in two directions, i.e. along X and Z axis. The displacement as load was added to RP along Y axis. The difference is that symmetry boundary conditions were added to the red colored surface showed in **Figure 1b)**.

As calculation results the following parameters were received: reaction force at the RP where displacement was added, crack opening displacement [COD] between points, were extensometer was attached in experiment, and J-integral at crack tip. In case of 3D model the calculated J-integral values were averaged along the crack front.



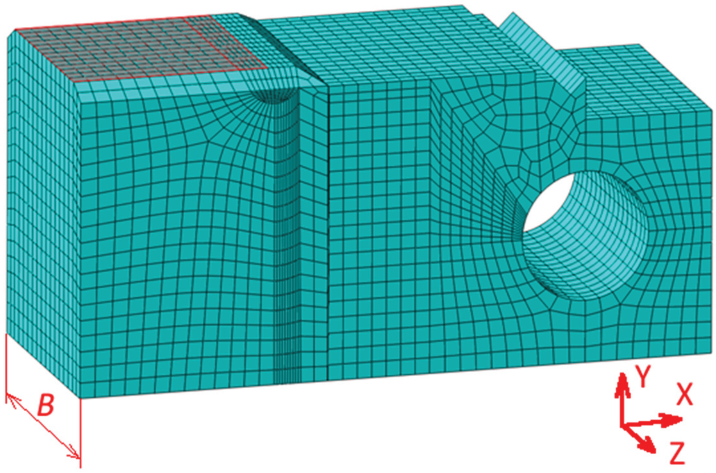


Figure 1 Finite element mesh of CT specimen (W = 50 mm): **a)** 2D model; **b)** 3D model

It is very important to mesh the crack tip correctly for modelling of fracture parameters. It is recommended [6] to perform the analysis using rectangular shape elements created in circular pattern around crack tip. In 2D model to create such mesh the nodes of one edge of elements closest to the crack tip have to be collapsed to a single node. Collapsing the nodes of one edge to the sing node we are getting the elements which still have 8 nodes but have triangle shape (see **Figure 1a**). The same procedure is done for 3D model and here you end up with prism shape elements around crack front (see **Figure 1b**).

Experimentally and numerically investigated material was steel P91. Steel P91 is a ferritic-martensitic class steel. It has higher thermal conductivity, lower thermal expansion, high resistance to swelling and high thermal resistance comparing to austenitic stainless steels [7].

To describe the elastic part of the mechanical properties of material the following parameters have been entered: Young's modulus E = 215 GPa; Poisson's ratio v = 0.3; yield stress $\sigma_{0.2}$ = 456 MPa; ultimate strength σ_U = 620 MPa.

3. CALCULATION RESULTS AND PROCESSING

Comparison of CT specimen simulation result with experiment is presented in **Figure 2**. The figure presents *P as* load versus crack opening displacement curves for experiment and for 2D and 3D FE models. The load values for 2D model are slightly higher comparing to 3D model case. It is because 2D model do not evaluate side grooves in CT specimen.

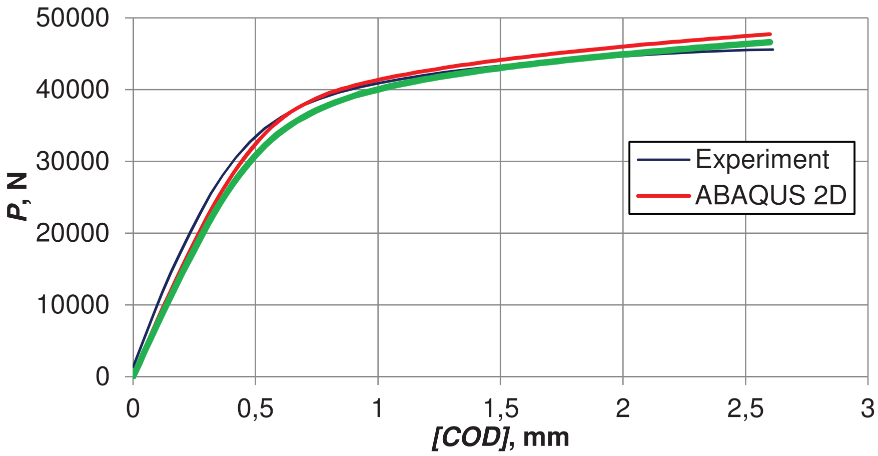


Figure 2 Numerically determined Load vs. COD comparison with experiment

According to methodology presented in section 2.1., it is necessary to construct J-R curve for J-integral determination. For this reason the crack extension should be measured during the tension of CT specimen. However, for the FE models presented in the section 2.2., the actual crack extension is not obtained. Therefore neither direct crack length measurement nor the elastic compliance measurement method suggested in ASTM code can be used. In this article the crack growth was determined from of the crack opening displacement and load ratio dependency on crack extension. In 1977 A.M. Sullivan and T.W. Crooker have presented the crack opening displacement technique for crack length measurement [8]. The authors suggest to use the polynomial function *a / W = f (E, B, [COD], P)* they determined where *E* is Young's modulus, Pa; *B* - specimen thickness, m; *[COD]* - crack opening displacement, m; *P* - load, N. However the function that authors recommend is only good to be used for materials which produce linear load vs COD trace.

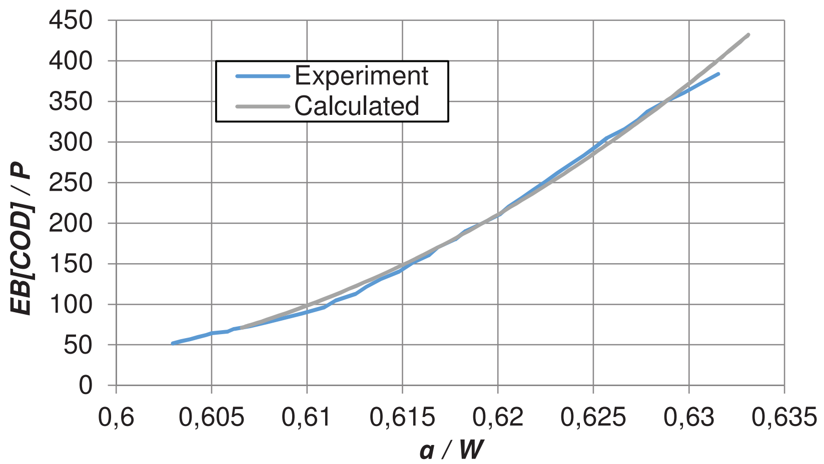


Figure 3 *EB[COD] / P* versus *a / W*

For crack length extension of steel P91 the quadratic polynomial expression (3) of experimentally determined curve *EB[COD] / P* versus *a / W*, which is shown in **Figure 3**, has been found. The figure presents experimentally measured curve and calculated curve which was created from numerical simulation result and by the following expression:

$$\frac{EB[COD]}{P} = 244104\left(\frac{a}{W}\right)^2 - 289003\left(\frac{a}{W}\right) + 85559 \qquad (3)$$

It should be noted that an expression (3) have been found while a / W ranged from 0.603 to 0.631. To increase the accuracy of the expression more experiments with different a / W ranges have to be carried out.

Figure 4 represents calculated and experimentally determined J-R curves of steel P91. Experimental data are shown as square and round points. J_Q values are determined at intersection points of curves and offset line. The ASTM E1820 indicates that for qualification of J_Q as elastic plastic fracture toughness J_{Ic}, $J_{Ic} = J_Q \leq min(b_0\sigma_y / 10; B\sigma_y / 10)$. Applying this criterion determined J_Q values could be qualified as J_{Ic}.

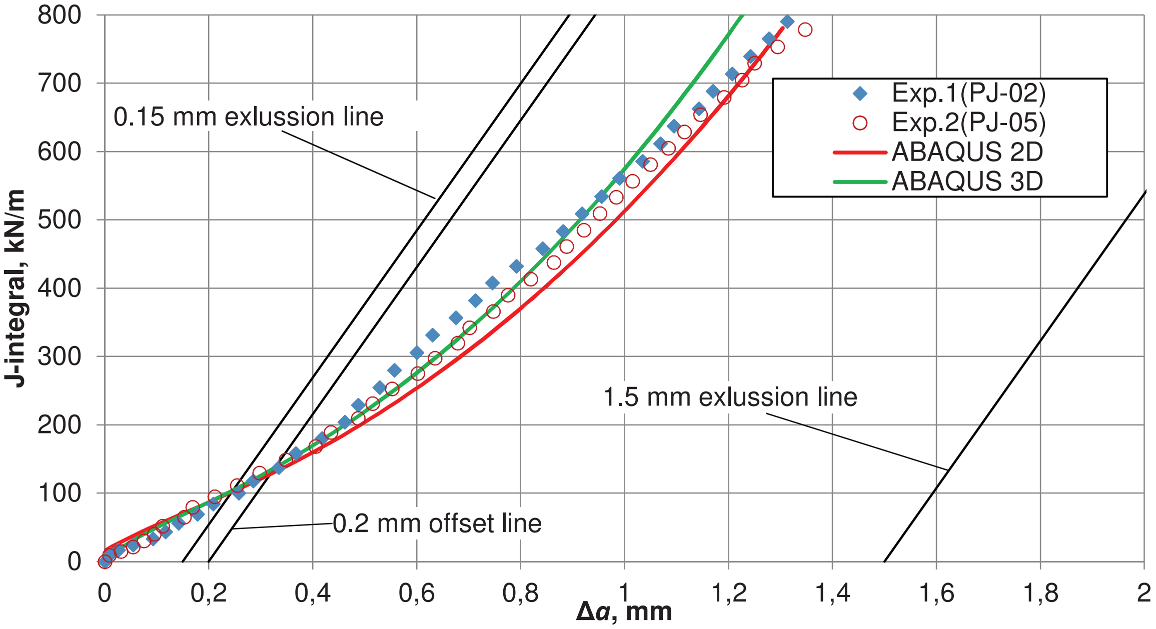


Figure 4 J-R curve of steel P91 at room temperature

The comparison of experimentally determined and numerically modeled J-integral results at different offset lines are presented in **Table 1**.

Table 1 J-integral values at different offset line distances of steel P91

	Experiment average, kN / m	2D model		3D model	
		J-integral, kN / m	Deviation from experiment, %	J-integral, kN / m	Deviation from experiment, %
J_Q	141.6	127.7	9.8	137.2	3.1
$J_{0.3}$	256.6	195.2	23.9	223.5	12.9
$J_{0.4}$	398.2	292.8	26.5	367.7	7.6

4. CONCLUSION

The numerical investigations of the Compact Tension (CT) specimen were carried out for the estimation of J-R curve for the ferritic-martensitic class steel (P91). The finite element method was used for the numerical investigation using the state-of-the-art ABAQUS/Standard code. The values of J-integral were determined using finite element analysis simulating the experiment according to instructions described in ASTM E1820 standard. The numerical investigation results have been compared with the experimental test.

The quadratic polynomial expression for crack extension calculation for steel P91 was suggested on experimentally determined curve *EB[COD] / P* versus *a / W*.

2D and 3D finite element models of CT specimen were prepared for J-R curve modelling. Comparison of the simulation results using both models shows that prognosis results of the 2D model are more conservative than 3D model. However calculated J_Q values do not deviate from experimentally determined more than 10 % in both 2D and 3D model cases.

ACKNOWLEDGEMENTS

This research was funded by a grant (No. MIP-023/2014) from the Research Council of Lithuania.

REFERENCES

[1] ANDERSEN, T.L. *Fracture Mechanics. Fundamentals and Applications*. 1st ed. Boca Raton, Ann Arbor, Boston: CRC Press Inc., 1991. 793 p.

[2] ZARE, A., KOSARI, E. S. M., ASADI, I., BIGHAM, A., BIGHAM, Y. Finite Element Method Analysis of Stress Intensity Factor in Different Edge Crack Positions, and Predicting their Correlation using Neural Network method. *Research Journal of Recent Sciences*, 2014, vol. 3, no. 2, pp. 69-73.

[3] LIKEB, A., GUBELJAK, N., MATVIENKO, Y. Stress Intensity Factor and Limit Load Solutions for New Pipe-Ring Specimen with Axial Cracks. *Procedia Material Science*, 2014, vol. 3, pp. 1941-1946.

[4] JUN-YOUNG, J., YUN-JAE, K., SA-YONG, L., JIN-WEON, K. Extracting ductile fracture toughness from small punch test data using numerical modeling. *International Journal of Pressure Vessels and Piping*, 2016, vol. 76, pp. 1-16.

[5] ASTM E1820-15a. *Standard Test Method for Measurement of Fracture Toughness*. ASTM International, West Conshohocken, PA, 2014. www.astm.org

[6] ABAQUS/Standard User's Manual, Version 6.11.

[7] IAEA. *Structural materials for liquid metal cooled fast reactor fuel assemblies - operational behaviour*. IAEA nuclear energy series No. NF-T-4.3, Vienna, 2012. 87 p.

[8] SULIVAN, A. M., CROOKER, T. W. A crack-opening-displacement technique for crack length measurement in fatigue crack growth rate testing - development and evaluation. *Engineering Fracture Mechanics*, 1977, vol. 9, pp. 159-166.