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

Elisaveta Doncheva¹, Bojan Medjo², Aleksandar Sedmak³

Vol. 15, No 3 (2015), pp. 131–134

Original scientific paper

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.

Keywords

- J integral
- finite element method
- plastic behaviour

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 consider-
Finite element analysis of fracture resistance parameters for ... Analiza parametara otpornosti na lom metodom konačnih ... 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.](image1)

![Figure 2. Semi-elliptical small and large surface crack.](image2)

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>
<thead>
<tr>
<th>Model description</th>
<th>Element type</th>
<th>Elements</th>
<th>Nodes</th>
</tr>
</thead>
<tbody>
<tr>
<td>BMSC</td>
<td>C3D20R</td>
<td>16350</td>
<td>71791</td>
</tr>
<tr>
<td>BMLC</td>
<td>C3D20R</td>
<td>22555</td>
<td>97287</td>
</tr>
</tbody>
</table>

Table 2. Geometries of centre cracked tensile panels.

<table>
<thead>
<tr>
<th>Specimen</th>
<th>t</th>
<th>W</th>
<th>Z</th>
<th>a₀</th>
<th>a₀/t</th>
<th>a₀/c</th>
<th>c/W</th>
</tr>
</thead>
<tbody>
<tr>
<td>BMSC</td>
<td>15</td>
<td>75</td>
<td>16</td>
<td>2.5</td>
<td>0.33</td>
<td>0.313</td>
<td>0.21</td>
</tr>
<tr>
<td>BMLC</td>
<td>15</td>
<td>75</td>
<td>26</td>
<td>5.0</td>
<td>0.17</td>
<td>0.384</td>
<td>0.35</td>
</tr>
</tbody>
</table>

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 Sumiten 80P.](image3)

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.

![J-CMOD exp. vs. J-CMOD num.](image)

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

![J-Strain exp. vs. J-strain Num.](image)

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.

![J-Strain exp. vs. J-strain Num.](image)

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.
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