Investigation of fracture of inhomogeneous cast iron specimens

E. Stupak
Vilnius Gediminas Technical University, Saulėtekio al. 11, 10223 Vilnius-40, Lithuania,
E-mail: Eugenius.Stupak@fm.vgtu.lt

1. Introduction

A major part of fracture mechanics problems [1-4] is related to stress strain fields in the vicinity of discontinuities, cracks, defects, etc. The main methods and techniques are analytical solutions, experimental and numerical.

The analytical models available in the literature allow predicting the stress-strain distribution either within or outside the inhomogeneity and for the further research of the problem. But it is better to use some numerical techniques, because an analytical solutions were obtained generally for simple geometries, e.g. ellipsoidal, cylindrical and spherical voids (cracks) in an infinite domain.

Among the numerical methods the most usually used is the finite element method (FEM). However, it was investigated and shown earlier that it is also necessary to assess the quality of the computed results, which depends on the FE mesh [5] and the assumptions about the stress-strain state [6].

The aim of this paper is the application of FE analysis technique for solving elastic-plastic problem of compact tension (CT) specimen of austempered cast iron. In the previous works [7-9] the analytical and experimental techniques have been described and results of solved elastic problems were discussed.

Murakami [10, 11] investigated interaction effects between small defects and cavities and proposed the simple prediction equation for fatigue limit, which is based on the square root of the equivalent projected area onto the principal stress plane of the defect. Costa et al. [12] proposed an improved technique for fatigue limit of nodular cast irons taking into account the size and positions of graphite nodules.

In the FEM solutions it is possible to account the inhomogeneities, but the solution scale is large since both the matrix and every inhomogeneity should be discretized. In future one may try to compare the obtained results with the results obtained using other techniques, actually boundary element method (BEM), discrete element method (DEM), etc.

In recent years in Lithuania many investigations were devoted for the analysis of different mechanical structures acted of high cycle or low cycle fatigue loading. Stonkus et al. [13] investigated using experiments the welded joints in the high cyclic loading, where the influence of different materials properties in the weld and main material junction (interaction) layer on cracking threshold were analysed.

Jakušovas and Daunys [14] investigated crack opening in the low cycle fatigue case using analytical, experimental and FEM techniques, where the influence of specimen size on crack tip opening displacement contour curve has been taken into consideration.

This paper is organized as follows: problem formulation, description of proposed technique for the calculation of J-integral in the vicinity of straight and piecewise linear straight crack front, analysis of obtained numerical results, conclusions and discussion.

2. Problem formulation

The analysis of a CT specimen produced of austempered cast iron, see Fig. 1 (thickness $B = 25$ mm), with an initiated defect and void placed in the different crack front positions is considered below. The size and position of the void has been changed during the investigations.

The external loading $F$ is assumed to be 3.0 kN.

![Fig. 1 Geometry and loading of a compact tension specimen](image)

After analyzing the fracture of the CT specimen of cast iron [8], the size and position of the defect in the specimen cross-section, an assumption was made that total dimensions of the defect on fracture surface do not exceed 10x6 mm and the distance to the notch tip is 10 mm. The crack-like defect had been initiated at the model having a rectangle shape as presented in Fig. 1. Few typical crack length values ($a = 18.8, 19.8, 22.8, 25.8$ mm and $a > 25.8$ mm, when the additional void is placed) were chosen alongside a few reference points ($A, B, C$ and $D$) where stress intensity factor (SIF) was supposed to be indicative.

The behaviour of the specimen’s matrix material is assumed to be elastic-plastic. The linear elastic part of the matrix material diagram is characterized by the elasticity modulus $E_1 = 175$ GPa, ratio $\sigma_{01}/E_1 = 0.00171$ and Poisson’s ratio $\nu_1 = 0.275$. Behaviour of the material above the yielding limit $\sigma > \sigma_{01} = 300$ MPa is taken as elastic-perfectly plastic.

The behaviour of the specimen’s inclusions or voids material (graphite) is assumed to be elastic, characterized by the elasticity modulus $E_2 = 27$ GPa and Poisson’s ratio $\nu_2 = 0.275$.

### 3. Technique for $J$-integral calculation

A novelty technique of path independent $J$ integral calculation in piece-wise linear crack front is proposed. Two different $J$ integrals can be calculated as surface integrals along contours around: 1) crack tip (line 1-2, $J_1$ integral path) or 2) defect tip (line 3-4, $J_2$ integral path) as shown in Fig. 2. $J$ integral derives from an energy potential. It was considered the $J$ integral along a contour linked to the crack or defect tip which extends by an infinitesimal distance $da$ and which takes the contour with it. It was considered the expression of the $J$ integral [3]

$$J = \int_{\Gamma} W dy - \int_{\Gamma} T_i \frac{\partial u_i}{\partial x} ds$$

where $\Gamma$ is any path surrounding the crack tip, $W$ is strain energy density (that is, strain energy per unit volume), $T_i$ is traction vector, $u_i$ is displacement vector, $ds$ is distance along the path $\Gamma$.

Partition of $J$ into elastic and plastic component can be done

$$J = J_{el} + J_{pl}$$

where $J_{el}$ is elastic, $J_{pl}$ is plastic part of $J$ integral.

### 4. Numerical results

ANSYS [15, 16] simulation for a defective model was performed to evaluate the defect and void influence on stress intensity factor and stress-strain state. The FE mesh had been generated using triangular and quadrilateral elements in plane. After they were modified to 3D brick elements, using layers technique. FE mesh used for stress intensity factor calculation, having 25 elements layers through thickness of specimens, is presented in Fig. 3. Only a half of the specimen was investigated using the advantages of symmetric. Less important model volume which is far from the crack tip is meshed with triangle prismatic shape finite elements (56420 nodes, 77170 FE). This helps to thoroughly analyze the stress intensity factor distribution along the crack front and in the vicinity of the defect. FE mesh in the small depth voids volume is finer and had different material properties than others elements.

During numerical investigations it is better to use the full expression of Eq. (1) [16]

$$J = \int_{\Gamma} W dy - \int_{\Gamma} T_x \frac{\partial u_x}{\partial x} + T_y \frac{\partial u_y}{\partial x} ds$$

where $T_x$, $T_y$ are traction vectors along $x$ and $y$ axes respectively ($T_x = \sigma_x n_x + \sigma_y n_y$, $T_y = \sigma_x n_x + \sigma_y n_y$), $\sigma$ is component stress, $n$ is unit outer normal vector to path $\Gamma$.

The numerical accuracy is better the more contour is distant from the crack tip. One must verify that the derived value of $J$ is independent of the considered path. Crack extension $da$ has been selected as $1\%$ of initial crack length $a$. In order to minimize computational costs and taking into account symmetry only half of specimen was investigated, so the result of $J$ integral, obtained by Eq. (2) and/or Eq. (3) must be multiplied by 2.

The distributions of principal stresses in the vicinity of crack and defect tip are illustrated in Fig. 4. Simulation of possible fatigue crack propagation has been done using node releasing technique, which is used instead of remeshing technique. The aim was to determine maximum stress intensity factor (SIF) in the CT specimen. As soon as each elements layer is rather thin, so SIF can be estimated according plane stress assumption using the next formula

$$K = \sqrt{\frac{J}{E_1}}$$

where $E_1$ is modulus of elasticity of matrix material.

SIF values, obtained using Eq. (4), versus accumulated crack length $a$ are presented in Fig. 5.
such considerations the different results of $K$ for the different crack fronts with the same averaged crack length $a$ were obtained.

Fig. 5 is separated into 2 parts: 1) when crack length $a < 25.8$ mm only homogeneous matrix material has been taken in the investigation; 2) when crack length $a > 25.8$ mm some additional graphite void is placed. These models were entitled from void 1 till void 7 and obtained results are presented in the Table.

Void 1 means the defected model (without graphite voids) and was selected for the purpose of comparison. Void 2 is the case with graphite voids close to each other and interacted, so their effective area $A_{eff}$, proposed by Murakami [11], was selected as $\pm 10\%$ of defect area. Void 3 till void 7 have the same $A_{eff}$, equal to $\pm 23\%$ of defect area, but with the different piece-wise linear crack front through the specimen thickness.

Influence of the voids on SIF it is clear from the 5th column, $K_{max}$ increases by 15% from 7.41 MPa$^{1/2}$ till 8.74 MPa$^{1/2}$. The ratio between plastic component $J_{pl}$ of $J$-integral with result of $J$-integral increase from 17.7% to 35.5%, which explain the $J$-dominance against $K$-dominance in the vicinity of crack (void) tip in the elastic-plastic fracture mechanics.

An analysis of SIF distribution through the thickness of the specimen is one of the advantages in 3D FEM analysis. It allows to investigate the influence of any separate void (or even micro-void if the model allows it) on SIF in any part of the specimen in the vicinity of the crack, defect, void, etc. The obtained results of SIF versus specimen thickness $B$ are presented in Fig. 6, where only cases of void 1, 2, 5, 7 with increasing $K_{max}$ in the point $B$ (see Fig. 1) are selected for the illustration of results. Maximum value at the defect right point $A$ (distance $t = B$) $K_{max,A} = 5.97$ MPa$^{1/2}$ in the void 7 model. Maximum value at the specimen center point $C$ $K_{max,C} = 6.15$ MPa$^{1/2}$ in the void 7 model, while at the left specimen edge point $D$ $K_{max,D} = 5.77$ MPa$^{1/2}$ in the void 2 model.

The obtained results illustrate SIF changing, when crack length $a$ increased during many numerical tests. Crack length was determined in the all layers through the FE model thickness and then averaged. Taking into account

Table

<table>
<thead>
<tr>
<th>No.</th>
<th>Void effective area $A_{eff}$, mm$^2$</th>
<th>Averaged crack length $a$, mm</th>
<th>Ratio $a/W$</th>
<th>$K_{max}$, MPa$^{1/2}$</th>
<th>$J_{pl}/J$, %</th>
<th>Stress triaxiality $k_{SIT}$</th>
<th>Homogeneous $\varepsilon_{pl, hom}$</th>
<th>Inhomogeneous $\varepsilon_{pl, inh}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0</td>
<td>25.8</td>
<td>0.516</td>
<td>7.41</td>
<td>-</td>
<td>-</td>
<td>0.0085</td>
<td>0.00162</td>
</tr>
<tr>
<td>void 2</td>
<td>0</td>
<td>26.8</td>
<td>0.537</td>
<td>7.87</td>
<td>-</td>
<td>-</td>
<td>0.0084</td>
<td>0.00160</td>
</tr>
<tr>
<td>void 3</td>
<td>13.6</td>
<td>29.3</td>
<td>0.587</td>
<td>7.74</td>
<td>-</td>
<td>-</td>
<td>0.0087</td>
<td>0.00158</td>
</tr>
<tr>
<td>void 4</td>
<td>13.6</td>
<td>29.6</td>
<td>0.591</td>
<td>7.76</td>
<td>17.7</td>
<td>0.84</td>
<td>0.00169</td>
<td>0.00162</td>
</tr>
<tr>
<td>void 5</td>
<td>13.6</td>
<td>30.8</td>
<td>0.617</td>
<td>8.65</td>
<td>33.9</td>
<td>0.85</td>
<td>0.00219</td>
<td>0.00180</td>
</tr>
<tr>
<td>void 6</td>
<td>13.6</td>
<td>31.0</td>
<td>0.619</td>
<td>8.73</td>
<td>35.0</td>
<td>0.88</td>
<td>0.00219</td>
<td>0.00190</td>
</tr>
<tr>
<td>void 7</td>
<td>13.6</td>
<td>31.1</td>
<td>0.622</td>
<td>8.74</td>
<td>35.5</td>
<td>0.88</td>
<td>0.00219</td>
<td>0.00190</td>
</tr>
</tbody>
</table>
Some attention must be paid for the zone close to
the left specimen edge (distance $t = 0.1B$), where
the maximum $K_{\text{max,E}} = 8.26 \text{ MPa}\sqrt{\text{m}}$ is only 5.5% smaller than
$K_{\text{max}} = 8.74 \text{ MPa}\sqrt{\text{m}}$. These detailed investigations had
proved once more, that voids must be taken into account,
especially if their effective area $A_{\text{eff}}$ is $> 2.5\%$ of the
area of fractured surface.

If the influence of softer material is bigger than in
the solved problem one must try to substitute the reduced
modulus of elasticity into Eq. (4) instead of $E_1$. And this is
the main advantage of proposed technique in the compar-
ison to other research techniques [11, 12] or standard re-
quirements of ANSYS [16], where material should be ho-
ogeneous in the vicinity of the crack.

Graphite is rather soft in comparison with matrix
material, so the microcracks will initiate in the matrix con-
ected to graphite. Stress concentration between softer
graphite and surface of matrix material will cause plastic
strains, after that matrix material fractures. Different mate-
rial properties will cause so called material mismatching,
also the plastic strain mismatch between elastic-perfectly
plastic matrix material and elastic graphite material.

During numerical nonlinear analysis plastic values
in all the points with different results are averaged in
order to satisfy the strain compatibility requirement be-
tween elements. Such inequality must be satisfied
\[ e_{pl,inh}^* < e_{pl,\text{hom}} \]  
(5)
where $e_{pl,\text{inh}}^*$ is averaged plastic strain for inhomogeneous
material, $e_{pl,\text{hom}}$ is plastic strain for homogeneous material.

This inequality can be transformed into rela-
tionship taking into account so called stress triaxiality ratio $k_{\text{ST}}$
\[ e_{pl,inh}^* = k_{\text{ST}} \cdot e_{pl,\text{hom}} \]  
(6)
\[ k_{\text{ST}} = \frac{\sigma_0}{\sigma_{eqv}} \]  
(7)
where $\sigma_0$ is mean stress or hydrostatic pressure, $\sigma_{eqv}$ is
equivalent (von Mises) stress.

Failure of ductile materials is often related to coa-
lescence of microscopic voids. The stress triaxiality is one
of the primary factors that influence the coalescence [17,
18].

Contours of equivalent plastic strains for some
selected cases (void 5 and void 7) are presented in Fig. 7,
while results are summarised in the 7-9 columns of Table.

\[ \text{Fig. 7 Illustration of equivalent plastic strains for:} \]
\[ a – \text{void 5, b – void 7} \]

After performed analysis it can be concluded, that
increased stress triaxiality increases the equivalent plastic
strains of inhomogeneous material, which maximum value
0.00190 rates to 72% of total strain equal to 0.00263.

5. Discussion and conclusions

The results of the simulation of inhomogeneous
cast iron CT specimen may be summarised as follows:
1. The proposed technique of $J$-integral calcula-
tion allows determination of SIF values through the thick-
ness of the CT specimen with defect and voids and piece-
wise linear fatigue crack front.
2. The investigation of graphite voids influence
on SIF showed increasing by 15% from 7.41 MPa m$^{1/2}$ (in
the specimen without voids) till 8.74 MPa m$^{1/2}$ (including
voids – void 7 case).
3. Plastic strains in the inhomogeneous specimen
were determined using stress triaxiality ratio.

References
1. Panasiuk, V.V., Andreikiv, A.E., Parton, V.C. Funda-
mentals of Fracture Mechanics. Fracture Mechanics
793p.

E. Stupak

NEVIENALYČIŲ KETAUS BANDINIŲ IRIMO TYRIMAS

Rezimė

Naudojant pasiūlytą J-integralo nustatymo nuovarginio dalinio tiesiško pliaško franto metodiką, atlikti nevienalyčių ketaus bandinių su defektais ir grafito interpalsų įtęmių intensyvumo koeficiento skaičiavimai.

Ištirta, kaip keičiasi įtęmių intensyvumo koeficientas priklausomai nuo pliaško gylies, franto konfiguracijos, interparų dydžio ir padėties. Nustatyta 15% įtęmių intensyvumo koeficiento prieaugus.

Nevienalyčių ketaus bandinių įtęmių analizei naudota BE programa ANSYS.

E. Stupak

INVESTIGATION OF FRACTURE OF INHOMOGENEOUS CAST IRON SPECIMENS

Summary

The proposed J-integral calculation technique in the piece-wise linear fatigue crack front has been used investigating SIF in the inhomogeneous cast iron specimens with defect and graphite voids.

The influence of crack length and front, size and position of voids on SIF have been investigated and the increase of SIF by 15% has been obtained.

Stress-strain state analysis of inhomogeneous cast iron specimens was carried out using FE code ANSYS.

Э. Ступак

АНАЛИЗ РАЗРУШЕНИЯ НЕОДНОРОДНЫХ ЧУГУННЫХ ОБРАЗЦОВ

Резюме

В работе при помощи предложенной методики для получения J-интеграла в окрестности усталостной трещины с кусочно-прямолинейным фронтом рассчитывался коэффициент интенсивности напряжений (КИН) в неоднородных чугунных образцах.

В итоге расследований по влиянию роста трещины, размера и положения частиц графита на КИН, получено 15% увеличение КИН.

Программа КЭ ANSYS была использована для расчета напряженно-деформированного состояния неоднородных чугунных образцов.