Computational Analysis of Stress Intensity Factor for Chevron Notched Specimen under Mode II fracture

Ganesh G.A., Gaurav Dave & Abanikash Rayaji

Department of Mechanical Engineering, R V College of Engineering, Bangalore, India

E-mail: ganesh2691ga@gmail.com, gaurav_dave91@yahoo.co.in, abanikash1122@gmail.com

Abstract - The paper aims at evaluating the stress intensity factor for a mode II type crack propagation for a standard chevron notched bar test specimen which is taken as a case study. Finite element analysis is used to determine the stress intensity factor and the simulation is performed on specimens with various crack-length to specimen-width (a/W) ratios. The software used for the analysis is APDL 12.1[ANSYS]. The reliability of the software has been proven using SENB specimen as the benchmark problem. The stress intensity factors are presented and observed along the crack front for different a/W ratios. The stress intensity factor is found to have the maximum value at the edges along the crack front and minimum at the midplane for different a/W ratios.

Keywords - Chevron notch, Singularity, Stress intensity factor.

I. INTRODUCTION

Failure of a material can be broadly classified as either fracture, fatigue or creep. Fracture refers to the failure of a structure by means of propagation of a crack, breaking it into more than one different parts. Fracture of any component can be classified as opening, sliding or tearing on the basis of the displacement of the crack. The fracture toughness of a specimen can be measured through the following parameters- Stress Intensity Factor (K), Energy Release Rate (G), Path independent integral (J) and Crack Tip Opening Displacement (CTOD).

The chevron-notched specimen (Fig 1 and 2) is a fracture toughness specimen being considered for use in standard tests by the American Society for Testing and Materials (ASTM) Committee E24. The unique features of a chevron-notched specimen, over conventional fracture-toughness specimens, are: 1) The extremely high stress concentration at the tip of the chevron-notch. 2) The development of a minimum stress-intensity factor as the crack grows. Currently, these specimens can only be used for low toughness brittle materials, like gray iron or ceramics. Further advances in elastic-plastic fracture mechanics are needed to use these specimens for ductile materials [1].
The actual specimen (Fig 3) employed for the simulation is a modified version of the ASTM specimen. The change is made to accommodate the demands of loading involved in Mode II fracture. It is more practical to apply a shear load on a surface without a stepped shape, and hence the step is removed from the specimen. From Fig 3, the highlighted portion represents the cracked surface.

II. BENCHMARK ANALYSIS

The analysis is done using ANSYS as a platform. The analysis of Single Edge Notch Bar (SENB) specimen subjected to a three point bending load is conducted on ANSYS and the results of the analysis are verified with the results obtained from the photo elastic analysis of the three-point bent specimen[2]. The specimen model is created in ANSYS and is meshed.

A progressively refined Finite Element mesh of Singular Iso-parametric Pentahedral solid element (SPENTA15) with user specified number (NS) and size (Aa) from one crack face to another and a number of such segments (NSEG) along a surface crack front is created using a pre-processing options in ANSYS. The rest of the domain under consideration is discretized using a compatible mesh of regular elements namely Iso-parametric Pentahedral solid (PENTA15) and Iso-parametric Hexahedral solid (HEXA20) [3].

The meshed model is shown in Fig 4.
The results of the photo elastic experiment and also of the analysis conducted on the ANSYS software are a close match. This establishes the credibility of the software, and thus it can be employed for testing a chevron notched bar specimen under Mode II fracture.

III. FINITE ELEMENT MODELING

The methodology followed involved defining the FE model, followed by formulation of the global matrix and solving using the iterative method, and results in extraction of the stress intensity factor and Von Mises stress. The dimensions of the specimen are shown in Fig 8, which are in accordance with available literature[4].

The complete FE model after meshing is shown in Fig 10.

For a Mode II type fracture, only sliding of one surface over the other is to be considered. The loading effects in other directions should be neglected. For achieving this, the constraints are applied on the Finite Element model before application of the load as shown in Fig 11.

Fig. 8 : Dimensions of Chevron notched bar specimen

After the creation of the singular points the whole model is meshed. First a plane along a singular element is meshed using 8 noded element. A 2D mesh is created. Then the whole body is meshed using the sweep option to create a 3D mesh using 20 noded elements. The FE model thus generated consisted of 20768 elements and 86432 nodes. The generated mesh is shown in Fig 9.

Fig. 10 : Complete FE model for a/W=0.5

Fig. 9 : Mesh using 8 noded element and extrusion to entire volume using 20 noded element.

Fig. 11 : Load and Boundary Conditions: (1) Bottom surface (Base) all DOF constraint; (2) Side bottom surface all DOF constraint; (3) Top surface constrained in y-direction; (4) Load Applied

Fig. 12: Von Mises Plastic zone created at the crack front
Table 1: Values of SIF for mode 2 fracture for different a/W ratios

<table>
<thead>
<tr>
<th>a/W</th>
<th>0.45</th>
<th>0.5</th>
<th>0.55</th>
<th>0.6</th>
<th>0.65</th>
<th>0.7</th>
</tr>
</thead>
<tbody>
<tr>
<td>K_II</td>
<td>Plane-1</td>
<td>7830.9</td>
<td>7419.2</td>
<td>7308.8</td>
<td>7385.6</td>
<td>7438.3</td>
</tr>
<tr>
<td></td>
<td>Plane-2</td>
<td>8352.6</td>
<td>7475.6</td>
<td>6910.3</td>
<td>6554.8</td>
<td>6394.6</td>
</tr>
<tr>
<td></td>
<td>Plane-3</td>
<td>8672.9</td>
<td>7168.0</td>
<td>6618.8</td>
<td>6313.5</td>
<td>6167.6</td>
</tr>
<tr>
<td></td>
<td>Plane-4</td>
<td>8262.7</td>
<td>7198.5</td>
<td>6643.9</td>
<td>6336.8</td>
<td>6195.3</td>
</tr>
<tr>
<td></td>
<td>Plane-5</td>
<td>8597.3</td>
<td>7635.9</td>
<td>7021.6</td>
<td>6662.8</td>
<td>6510.2</td>
</tr>
<tr>
<td></td>
<td>Plane-6</td>
<td>7942.4</td>
<td>7532.2</td>
<td>7558.8</td>
<td>7684.5</td>
<td>7824.1</td>
</tr>
</tbody>
</table>

IV. RESULTS

SIF was evaluated for specimens of different a/W ratios ranging from 0.4 to 0.7 in steps of 0.05. SIF was obtained for each and every model separately and are tabulated as in Table 1.

The variation of the stress intensity factor (K_II) versus Plane number is plotted for different a/W ratios, a sample of which is shown in Fig 13.

![Fig. 13 : Variation of SIF along the crack front for a/W=0.7](image)

The SIF for the middle plane for different a/W ratio specimens is plotted against the a/W ratio and is shown in Fig 14.

![Fig 14 : Variation of SIF against different ratios of a/W](image)

V. CONCLUSION

Referring to the results obtained (Figs. 13,14) the following points can be inferred.

- The stress intensity factor (SIF) is found to have the maximum value at the edges along the crack front, which are characterized by Planes 1 and 6 in the graphs.
- The stress intensity factor (SIF) is minimum at the midplane for all the a/W ratios. This refers to Planes 3 and 4 in the graphs.
- The Von Mises plastic zone along the crack front is created as shown in Fig 12.

Referring to the variation of SIF against a/W (Fig. 14), it was observed that the SIF decreases as the a/W ratio increases. It infers that with the increase in blunting of the crack, the stress intensity factor decreases.

VI. REFERENCES