The Study on Crack Growth using Extended Finite Element Method

Bindu.N.Swamy1 Dr.M.N.Hegde2

1P.G. Student 2Professor
1,2Department of Civil Engineering

Abstract— Cracks are small imperfections within a material that are created when it splits apart. These tend to propagate and cause further cracking by creating new surfaces, eventually splitting and causing the material to fail. In the field of Computational fracture mechanics the numerical methods have been employed and regarded as suitable tools to predict the fracture and failure of engineering structures. In the present work Abaqus 6.11 is used for crack analysis which includes crack initiation and propagation resulting in failure and the results are compared with the theoretical values. The Stress Intensity Factor (SIF) which defines the state of stress at the crack tip is used as an important parameter for estimating the life of the cracked structure. The crack growth simulations are presented using two XFEM techniques namely, Virtual crack closure technique (VCC) and cohesive segments method. Variation of stress intensity factor with different parameters like crack length and crack angle are discussed.

Key words: Crack Growth, Finite Element Method

I. INTRODUCTION

Fracture mechanics has developed into a major branch of science and engineering mechanics. Brittle materials like high strength steel, glass, concrete, etc, experiences brittle fracture and the study of this is in the area of Linear Elastic Fracture Mechanics (LEFM). LEFM, an elastic analysis of the stress field for small strains gives great results for fragile-elastic materials like high-strength steel and glass. In LEFM, plasticity does not play a vital role during fracture. With the occurrence of plasticity or viscoelasticity we enter into the field of nonlinear fracture mechanics. For fracture and crack that break this fundamental assumption, elastic-plastic fracture mechanics (EPFM) methodologies are utilized to portray the fatigue process and fracture process. This is the case for ductile materials like stainless steel, some aluminum alloys and polymers. The plasticity manifests itself in two ways: a) At the level of the plastic zone in front of the tip of the crack as it is the source of an history effect by virtue of the development of residual stresses and b) At the level of the mechanism of crack propagation by superposition of the mechanism of ductile fracture, which is from the instability resulting when large deformations occur in the vicinity of crystalline defects.

The theory of cracking phenomena or fracture mechanics depicts the conduct of solids/structures with macroscopic geometric discontinuities at the size of the structure. These discontinuities are lines in two-dimensional media and surfaces in three-dimensional media.

Linear Elastic Fracture Mechanics (LEFM) is the fundamental theory for crack/fracture, initially developed by Griffith (1921) and finished in its crucial frame by Irwin (1957) and Rice (1968). LEFM is a highly simplified, versatile theory that deals with sharp cracks in elastic bodies. As early as 1920, Griffith showed that the failure of a brittle elastic medium could be characterized by a variable, later called the energy release rate, whose critical value, independent of the geometry of the structure, was a characteristic of the material. This approach, called the global approach, showed that in all cases the essential phenomena occurs closeness to the crack front and that it is possible to study the macroscopically cracked medium with the help of intrinsic variables. This is due to the high stress concentration present at the tip of a crack which, for linear elasticity, can be represented by singularity of the stress field.

As long as the load is low enough, whether monotonically increasing or periodic, these effects can be neglected and LEFM continues to provide a good approximation to the physical reality. In contrast, for large and highly variable loads, the stable progression of ductile fracture cracks and the history effects due to overloads can be modeled only by taking plasticity into account.

II. EXTENDED FINITE ELEMENT METHOD (XFEM)

XFEM is a numerical technique that extends the classical finite element method (FEM) approach by extending the solution space for solutions to differential equations with discontinuous functions. Traditional FEM, introducing a discontinuity in the mesh, changing the material topology or drastically changing the shape of the material requires a new mesh. The process is laborious and difficult—XFEM arose as a solution to this impediment by applying enrichment functions at the position of the material interface or topology discontinuity instead of re-meshing the entire structure. Once a crack is present, its initiation advances under monotonic, static loading conditions.

In XFEM additional enrichment functions are used to provide an approximation solution in addition to the polynomial shape functions. Single mesh suffices for modelling. Re-meshing is not required when there is change in shape of structure during loading or at point where there is a discontinuity (crack) in the structure.

The XFEM approximation $u(x)$ is expressed as,

$$u(x) = \sum N_i(x) u_i + \sum N_i^e(x) b_i(x) a_k$$

Where $N_i(x)$ is the standard shape function associated to node $i$; $k$ is the set of all nodes in that domain; $k^*$is the set of enriched nodes and $u_i$ is the unknown (displacements) of the standard FEM at node $i$; $b_i(x)$ is the local enrichment function of node $i$.

III. METHODOLOGY

A. Crack propagation using XFEM based approach

In Crack propagation simulation there will be no need for re-meshing near the tip of the crack. Though for majorly curved cracks, necessity of re-meshing will be near the tip of crack to obtain accurate results.
B. Crack growth simulations using Abaqus

Crack growth simulation occurs in three steps: (i) Crack initiation (ii) Crack propagation and (iii) Failure. All these three steps are simulated using Abaqus. Since Abaqus is one of the best tools to analyze the crack growth simulation with varied analysis techniques. The graphical user interface is a simple interface which creates, submits, monitors, and evaluates. It includes from defining part and its dimensions, geometry, material properties, loading pattern, boundary conditions, meshing and thus finally building the module. The module then is subjected for analysis and creates an output data base. The results are read and viewed using visualization module.

The onset of cracking can be studied in quasi-static problems by using contour integrals in two or three-dimensional problems but the crack propagation cannot be studied because focused meshes are generally required. The crack propagation allows a quasi-static analysis of the crack growth along pre-defined paths. Cracks de-bond along user-defined surfaces. Several crack propagation criteria are available, and multiple cracks can be included in the analysis.

Extended finite element method (XFEM) models a crack as a special feature by including degrees of freedom in elements with special enriched functions. The discontinuities in the geometry need not match the mesh in. The crack simulation can be done without the necessity of remeshing. And also there will be no need of refining of mesh near the tip of the crack in performing contour integral evaluation.

IV. Problem Definition

The problem chosen for analysis is Single Edge Notch (SEN) specimen, made of 6061-T6 aluminum material. The plate is used for analysis of mode-I crack type in Abaqus. Fig 5.1 represents the geometry of the plate with loading condition. Breadth of the specimen, b is 50mm, height of the specimen, h is 75mm, thickness is taken as unity, a is the crack length varied from 5mm to 20mm and loading, uniaxial loading of 1MPa is applied at the top and bottom edge of plate. Young’s modulus is 70GPa, Poisson’s ratio is 0.33, Maximum Principal Stress is 146.3MPa, Critical Stress Energy Release rate is 24.2KN/m.

![Fig. 1: A schematic presentation of the plate and its loading method](image1)

The problem chosen for static analysis is slant-cracked specimen, made of 6061-T651 aluminum material. The plate is used for analysis of mode-I and mode-II crack type in Abaqus. Width, w is 20 mm, height, h is 200 mm and thickness (t) is 6 mm, a is 2.5 mm cantered in plate, \( \theta \) is the Crack angle varied for different angles. Loading, \( \sigma_0 \) is Stress of 250 MPa is applied at the top edge of the plate. Plate is fixed in y-direction at the bottom of the plate. Young’s modulus is 70GPa, Poisson’s ratio is 0.33, Maximum Principal Stress is 146.3MPa, Critical Stress Energy Release rate is 24.2KN/m.

![Fig. 2: A schematic presentation of the plate and its loading method](image2)

V. Results and Discussions

The mode-I SIF values were calculated for standard 3D XFEM with the help of Abaqus capabilities are presented in table 1. The obtained values are compared with the theoretical values. For three dimensional cases thickness is taken as unity. It is not possible to calculate SIF for two dimensional XFEM in Abaqus so three dimensional XFEM was chosen for analysis.

![Fig. 3: Comparison of stress intensity factor (SIF) values for different Crack length](image3)

<table>
<thead>
<tr>
<th>Ratio of crack length to width of plate</th>
<th>Theoretical value</th>
<th>XFEM 3D (Three dimensional extended finite element plate)</th>
</tr>
</thead>
<tbody>
<tr>
<td>a/w</td>
<td>KI (mode –I stress intensity factor)</td>
<td>KI (mode –I stress intensity factor)</td>
</tr>
<tr>
<td>0.1</td>
<td>4.6937</td>
<td>4.5950</td>
</tr>
<tr>
<td>0.2</td>
<td>7.6958</td>
<td>7.2690</td>
</tr>
<tr>
<td>0.3</td>
<td>11.4320</td>
<td>11.2600</td>
</tr>
<tr>
<td>0.4</td>
<td>16.7525</td>
<td>16.5810</td>
</tr>
</tbody>
</table>

Table 1: Comparison of stress intensity factor (SIF) values

Fig 3 represents the variation of stress intensity factor (SIF) along crack length. As the crack length increases the stress intensity factor increases for mode I type of cracked plate element. All the values of stress intensity factor (SIF) are in MPa/\( \sqrt{\text{mm}} \).
After the analysis of plate for different crack length for mode-I type of crack. The stress profile for each crack length have been extracted using abaqus 6.11. Fig 4-7 represent the Von-misses stress for 5,10,15 and 20 mm crack length.

Fig 4(a): Von-misses stress profile of 5 mm crack length of 3D XFEM and (b) close view 5 mm crack length

Fig 5: (a) Von-misses stress profile of 10 mm crack length of 3D XFEM and (b) close view of 10 mm crack length

Fig 6: (a) Von-misses stress profile of 15 mm crack length of 3D XFEM and (b) close view of 15 mm crack length

Fig 7 (a): Von-misses stress profile of 10 mm crack length of 3D XFEM and (b): close view of 10 mm crack length

Since the plate is having inclined crack the concept of second mode of stress intensity factor comes into effect along with the first mod of failure. If load applied is not along the direction of the crack the mode-I and mode-II stress intensity factor (SIF) values are calculated. The SIF values are calculated for standard two dimensional plate using FEM with the help of Abaqus 6.11 software. The obtained values are compared with the theoretical values. It is found that obtained values are almost near to the theoretical values thereby exists a good agreement confirming the robustness and accuracy of the Abaqus capabilities. Table 2 represents the stress intensity factor for different crack angles having same crack length. All the values of stress intensity factor (SIF) are in MPa√m.

<table>
<thead>
<tr>
<th>θ (in degree)</th>
<th>a (in mm)</th>
<th>Theoretical values</th>
<th>Abaqus values</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>KI (mode-I stress intensity factor)</td>
<td>KII (mode-II stress intensity factor)</td>
<td>KI (mode-I stress intensity factor)</td>
</tr>
<tr>
<td>25</td>
<td>2.5</td>
<td>18.1968</td>
<td>8.4853</td>
</tr>
<tr>
<td>55</td>
<td>2.5</td>
<td>7.2883</td>
<td>10.4808</td>
</tr>
<tr>
<td>60</td>
<td>2.5</td>
<td>5.5384</td>
<td>9.5927</td>
</tr>
<tr>
<td>80</td>
<td>2.5</td>
<td>0.6680</td>
<td>3.7884</td>
</tr>
</tbody>
</table>

Table 2: The mode-I and mode-II stress intensity factor (SIF) values compared with the theoretical values

The graph of stress intensity factor (SIF) versus different crack angle (θ) were plotted with the values obtained by numerical simulation in abaqus and theoretical value. As from the graph it is observed that the mode-I stress intensity factor (SIF) represented by ‘KI’ values obtained by Abaqus and theoretical formula decreases as the crack angle increases. The mode-II stress intensity factor (SIF) represented by ‘KII’ values obtained from both Abaqus and theoretical formula increases to certain angle and then decreases.

From the crack analysis of two dimensional plate using finite element capabilities of Abaqus 6.11 software. The Von-mises stress and displacement profile of different crack angle are obtained using Abaqus analysis output. Fig
9-10 represent the stress and displacement profile for crack angles of 25°, 55°, 60° and 80°.

**Fig. 8:** Comparison of KI and KII values for different crack angle

**Fig 9 (a)** Von-misses stress profile at the crack tip of Crack angle 25° and (b) of crack angle 55°

**Fig 10:** (a) Von-misses stress profile at the crack tip of Crack angle 60° and (b) of crack angle 80°

**VI. CONCLUSIONS**

The computations of stress intensity factor (SIF) are done for two specimens. The first specimen had crack length with zero crack angle for mode-I type of stress intensity factor and the second specimen had an inclined crack for different crack angles for both mode-I and mode-II stress intensity factor. From the above work following conclusion have been deduced.

- The SIF values obtained for mode-I crack type using Abaqus analysis for 3D XFEM are approximately near to the theoretical values.
- SIF values are calculated for different crack length. Graph of SIF vs. Crack length are plotted. It is observed that as the crack length increases the SIF value at the crack tip increases.
- The fringe pattern for von-misses stress profile around the discontinuity represents butterfly pattern at the crack tip.
- The SIF values are computed for two dimensional plate with plane stress element in Abaqus. Since the crack is inclined mode-I and mode-II SIF values are computed and found to approximately near to the theoretical values.
- The SIF values are computed for different crack angles keeping crack length constant. The SIF values vs. different crack angles are plotted. KI value decreases as the crack angle of crack increase but KII value increases to certain angle and then decreases.

**REFERENCES**


