Computational Modeling and Analysis of Stress Intensity Factor for Opening Mode Crack Propagation using Displacement Extrapolation Method

Syed Jabiulla[1], Kirthan L J[2*] , Ramakrishna Hegde[2], Girish Kumar R[2]

[1]Research Scholar, VTU, Belagavi
[2]Department of Mechanical Engineering, RV College of Engineering. Bengaluru, Karnataka

Abstract

Stress intensity factors play a crucial role in predicting the behavior of cracks in engineering structures. Accurate determination of stress intensity factors is vital for assessing the structural integrity and fracture behavior of materials. This paper presents an evaluation of Mode I stress intensity factor for an edge crack utilizing the displacement extrapolation method. The field extrapolation technique was employed to calculate the stress intensity factor, which involved measuring the displacement of nodes at the crack tip. To validate the accuracy and reliability of the proposed approach, a comprehensive comparison with experimental results is performed. The displacement extrapolation approach was found to be in good accord with the experimental data. The effect of element size and mesh arrangement near the crack is discussed.

Keywords*: Displacement extrapolation method, Stress Intensity Factor (SIF), FEM, Fracture Toughness.*

1.0 Introduction

The property of fracture toughness (K_{IC}) describes material's capacity to resist crack propagation. The reliable estimation of Stress Intensity Factors (SIF) is an essential aspect of fracture mechanics, enabling the assessment of crack growth behavior and structural integrity [1, 2]. Mode I stress intensity factors specifically characterize the crack opening mode, which is commonly encountered in numerous engineering applications. Accurate determination of these factors aids in predicting crack propagation rates, estimating remaining life, and designing fatigue-resistant components. Conventionally, analytical, and numerical methods have been employed to calculate SIF [3, 4]. Analytical approaches, such as the Westergaard equation and Newman-Raju equation provides closed-form solutions for specific geometries. However, these methods often require simplifying

**Mail address: Kirthan L J, Associate Professor, Department of Mechanical Engineering, RV College of Engineering, Bengaluru – 59*
Email: kirthanlj@rvce.edu.in, Ph: 99021 92679

assumptions, limiting their applicability to complex crack configurations. On the other hand, numerical techniques such as the Finite Element Method (FEM), offer greater flexibility but demand significant computational effort, particularly for large-scale problems.

The present crack configurations are three-dimensional ones. The use of plane stress/plane strain as a two-dimensional approximation to these cracked bodies is often unacceptable [2]. To analyze stress intensity components using multiple procedures and balance accuracy and computational efficiency, researchers have looked into a variety of approaches. Here are two types of estimating techniques: First, which employ field extrapolation near the crack tip and second, which uses energy release during crack propagation. Special post-processing techniques are required in the energy release method and in mixed mode situations, it is typically complex [5]. One of the field extrapolation methods is the displacement extrapolation method, which capitalizes on the displacement fields obtained from Finite Element Analysis (FEA). By extrapolating displacement data near the crack tip, this method enables the estimation of SIF without the need for complex analytical derivations or extensive computational resources. To adequately characterise the unique strain field near the crack tip, as recommended by Barsoum [6] an altered iso-parametric element is presented. The projected finite element model is made up of SINGULAR iso-parametric pentahedral solid elements at crack front [7].

This paper presents a comprehensive evaluation of the Mode I SIF for an edge crack using the displacement extrapolation method. The proposed technique leverages the displacement fields near the crack tip to calculate the SIF, providing an efficient and accurate alternative for engineering applications. The results from the displacement extrapolation methods are compared with experimental data to see how reliable they are. The displacement extrapolation method's dependability will be proven through comparison with experimental data, opening the door for a wider use in engineering practice. The influence of mesh arrangement and element size near crack on SIF obtained by the displacement method and numerical is presented. The findings of this study have the potential to enhance crack growth prediction models, improve structural design considerations, and ultimately enhance the safety and durability of engineering structures.

2.0 Development of Finite Element Model

In the displacement extrapolation method, the accuracy of SIF estimation relies on the proper selection of finite elements and the development of an appropriate finite element model. The choice of elements and the construction of the model are crucial to ensure reliable results.

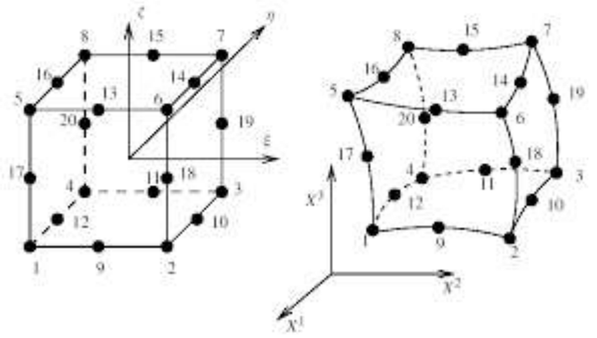

Fig.1. Quadratic order hexahedral solid element

In 3D models, tetrahedral or hexahedral elements are commonly used. Tetrahedral elements, with their simplicity and ease of meshing, are frequently employed for general 3D crack problems. However, hexahedral elements offer superior accuracy and efficiency for crack modeling, especially when the crack plane is well-aligned with the mesh. For the displacement extrapolation method, the commonly used elements are typically those that can accurately capture the displacement fields near the crack tip. In many cases, singular elements, also known as crack-tip elements or enrichment elements are employed. Fig. 1 shows quadratic order hexahedral solid element of the Serendipity family.

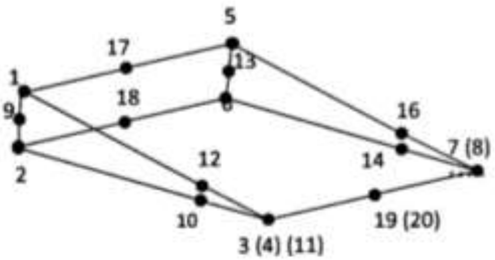

Fig. 2. Singular pentahedral element

The element has 20 nodes. Eight nodes are located at the vertices and the others are at mid-side points of the parent element which is a bi-unit cube. The HEXA20 element is widely used in practice and is implemented in every commercial FEM system. This element is employed as regular element. Pentahedral solid element of the serendipity family of quadratic order (15 nodes) is shown in Fig. 2. This is designed by further distorting the HEXA20 element. Specifically, it involves collapsing a face and constraining the nodes that are collocated to have identical degrees of freedom. This element is called PENTA15 and is also used as regular element. The projected finite element model involves a very fine mesh of SINGULAR iso-parametric pentahedral solid elements (SPENTA15) along the surface crack front [8, 9]. A compatible mesh of regular elements namely iso-parametric pentahedral solid element (PENTA15) and iso-parametric hexahedral solid element (HEXA20) are used to discretise the rest of the domain under consideration [10]. These elements are specifically designed to model the stress singularity that occurs at the crack tip. The most widely used singular element is the quarter-point element or quarter-point singular element, which is a higher-order element capable of accurately capturing stress gradients and displacements near the crack tip.

In this work the finite element model was created using ANSYS program. However, in ANSYS the required singular element is not listed in the element library. Therefore, the pre-processing commands and user experience is essential for the concurrent creation of SPENTA 15 element mesh along any curved crack front. The element's shape functions are derived based on the serendipity concept, which allows for efficient and smooth interpolation of displacements within the element. It is important to note that the choice of elements and the refinement of the mesh near the crack tip are critical for accurate displacement extrapolation. Fine mesh refinement in the vicinity of the crack tip ensures that the displacement field is adequately captured, leading to reliable SIF estimation.

3.0 Displacement Extrapolation Method

The displacement extrapolation method leverages displacement data of nodes obtained from Finite Element Analysis (FEA) to estimate SIF. It is based on the assumption that the displacement field near the crack tip follows a power-law singularity. To obtain decent depiction at the crack front edge, Quarter-point iso-parametric elements are employed. Fig. 3 depicts element arrangement and crack's progression on the x-axis.

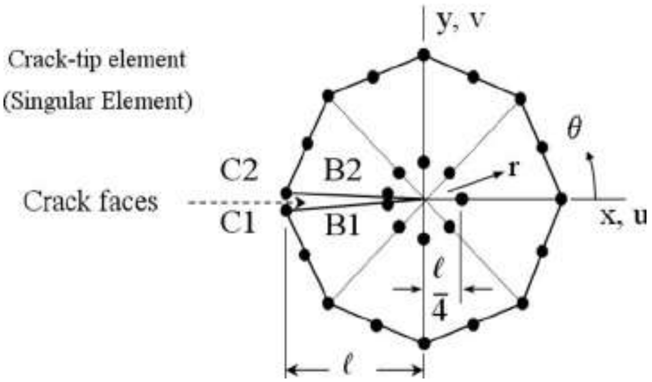

Fig. 3. Progression of crack in x-direction

The asymptotic equation for bi-dimensional crack under in-plane loading displacement normal to the crack plane$v$, is given by equation (1) [5].

$$v = K_I \frac{1+v}{4E}\sqrt{\frac{2r}{\pi}}\left\{(2\kappa+1)\sin\frac{\theta}{2} - \sin\frac{3\theta}{3}\right\} + A_1\frac{(1+v)r}{E}(\kappa-3)\sin\theta + A_2\frac{(1+v)r^{\frac{3}{2}}}{E}\left\{\frac{(2\kappa-1)}{3}\sin\frac{3\theta}{2} - \sin\frac{\theta}{2}\right\} + \cdots \quad (1)$$

where E is the young's modulus, v the Poisson's ratio, κ for plane stress $(3-v)$ / $(1+v)$ and for plain strain $(3-4v)$. A_i are parameters that depend on load and the specimen's geometry, while r and Θ are the polar coordinates, as shown in Fig. 3. At fracture tip the normal displacement v(r=0) is zero, as prescribed to the symmetry of mode I.

To improve the accuracy of the extrapolation, equation (1) only incorporates terms in $r^{1/2}$, $r^{3/2}$, $r^{5/2}$, etc. when the displacement v is measured along the fracture faces ($\theta = \pm\pi$). At the upper face of crack for node A and node B of singular element by specialising equation (1) we have,

$$V_A = K_I\sqrt{\frac{2}{\pi}}\frac{(1+v)(\kappa+1)}{4E}\sqrt{l} - \frac{A_2\ (1+v)(\kappa+1)}{12E}l^{\frac{3}{2}} + O(l)^{\frac{5}{2}} \quad (2)$$

$$V_B = K_I\sqrt{\frac{2}{\pi}}\frac{(1+v)(\kappa+1)}{4E}\sqrt{l} - \frac{2A_2\ (1+v)(\kappa+1)}{3E}l^{\frac{3}{2}} + O(l)^{\frac{5}{2}} \quad (3)$$

where l is the element side TB's length. Equations (2) and (3) can be solved or K_I and $A_1$by ignoring higher order terms. The SIF value is thus:

$$K_I = \frac{E}{3(1+v)(\kappa+1)}\sqrt{\frac{2\pi}{l}}(8v_A - v_B) = \frac{E'}{12}\sqrt{\frac{2\pi}{l}}(8v_A - v_B) \quad (4)$$

Where E′ is the effective young's modulus, which is defined as E/(1 – v^2) for plane strain and E for planar stress. If the higher terms ($l^{\frac{3}{2}}$) ignored in equation (2), the quarter node displacement can be used to estimate K_I more easily.

$$K_I = \frac{2E}{(1+v)(\kappa+1)}\sqrt{\frac{2\pi}{l}}v_A = \frac{E'}{2}\sqrt{\frac{2\pi}{l}}(v_A) \quad (5)$$

Finally, an alternative estimation of K_I may be produced by matching the term in r $^{1/2}$ of the displacement expansion along the top crack face with the equivalent term of the element interpolation function for $v(r)$.

The displacement field θ = π at the crack edge for a single six-node or eight-node iso-parametric element is a function of the nodal displacements v_A and v_B, and is given by:

$$V(r) = (4v_A - v_B)\sqrt{\frac{r}{l}} - (4v - 2v_B)\frac{r}{l} \quad (6)$$

By recognising terms with $\sqrt{r}$ in equations (1) and (7) and by changing θ = π in equation (1) we get:

$$K_I = \sqrt{\frac{2}{\pi}}\frac{(1+v)(\kappa+1)}{2E}\sqrt{r} = (4v_A - v_B)\sqrt{\frac{r}{l}} \quad (7)$$

Now the SIF is,

$$K_I = \frac{E}{(1+v)(\kappa+1)}\sqrt{\frac{2\pi}{l}}(4v_A - v_B) = \frac{E'}{4}\sqrt{\frac{2\pi}{l}}(4v_A - v_B) \quad (8)$$

The quarter point element's nodal displacements on the top side of the crack are used in Equations (4), (5), and (8) to calculate K_I . Due to symmetry, a similar outcome would be achieved for the bottom face element. The performance of these three K_I estimations is evaluated in the following section.

4.0 Numerical Simulation of Fracture Toughness

4.1 Methodology of the Numerical Simulation of Fracture Toughness

The numerical research of fracture toughness was done by numerically simulating experimental tests according to the ASTM E 399 standard. Here is a brief overview of the methodology [11]

Geometry and Mesh Preparation: Geometry of the specimen prepared as per ASTM E399.The most commonly used specimen geometries for fracture toughness testing are the standard Compact Tension (CT) and Single-Edge Notched Tension (SENT) specimens. Create a finite element mesh that accurately represents the geometry of the specimen. The mesh should be refined near the crack tip to capture the stress gradients accurately.

Material Properties: Material properties of the specimen, including elastic modulus, Poisson's ratio are defined. It is important to ensure that the material properties used in the simulation are representative of the actual material being tested.

Loading Conditions: Appropriate loading conditions as per ASTM E399 standard are applied. The most common loading condition for fracture toughness testing is a linearly increasing load applied to the specimen, resulting in crack propagation.

Crack Growth Simulation: The crack was initiated by introducing a small initial crack or notch in the specimen geometry and loading conditions were defined to simulate crack growth. The appropriate fracture mechanics-based criteria, such as SIF (K) or the J-integral were used to determine the crack growth behaviour.

Analysis and Results: displacements, stress, and crack lengths, at specific intervals or load increments were monitored and recorded. The recorded data was analysed to calculate the critical SIF (K_{IC}) or fracture toughness value, typically using established fracture mechanics equations or methods. Comparative study of calculated fracture toughness with the specified value was done to evaluate the specimen's fracture resistance and to determine if it meets the ASTM E399 standard criteria.

Validation: The numerical simulation results were validated by comparing them with experimental data obtained from physical testing of the same specimen geometry and material and ensured that the simulated fracture toughness values are within an acceptable range of the experimental results, indicating the accuracy and reliability of the numerical model.

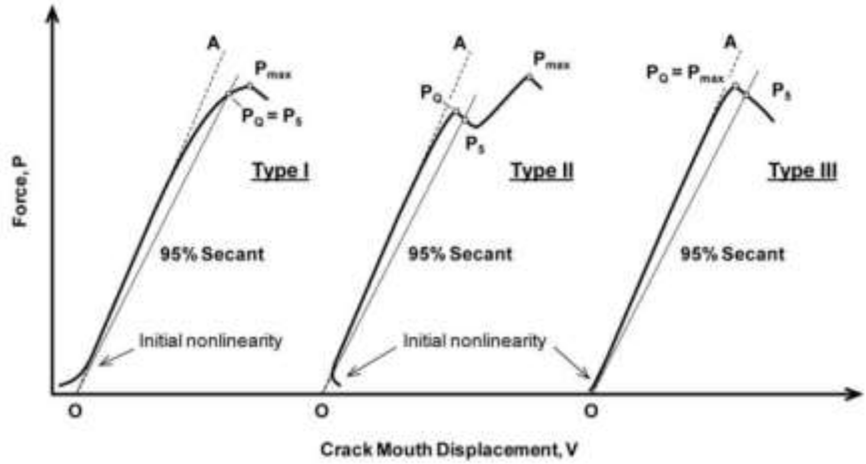

Fig. 4. Principal Types of Force-Displacement (CMOD) Records

For K_Q determination, a load vs. crack mouth opening displacement (CMOD) graph must first be generated. By monitoring the movement of the notch edge during numerical modelling, the CMOD can be determined. The loads P_Q and P_{max} are determined using a built curve. P_{max} is the load value at the curve's peak load point. The load at which the SIF reaches the K_Q value is defined as P_Q. The OA line is a tangent line to the linear part of the curve (Fig. 4), where O point is at the origin of the curve. Line OP_5 is a secant line with a 5% slope to the OA line and found by $(P/v)_5 = 0.95(P/v)_O$. In the case of a Type I curve, P_Q is found at the intersection point of the curve and the secant line. For Type II and Type III curves, the P_Q load is found at the first vertex. Knowing the P_Q load, K_Q can be analytically calculated by using the following equation for a standard C(T) specimen.

$$K_Q = \left(P_Q/BW^{1/2}\right) f(a/W) \tag{9}$$

where:

$$f(a/W) = \frac{\left(2+\frac{a}{W}\right)}{\left(1-\frac{a}{W}\right)^{3/2}} \left[0.886 + 4.64\frac{a}{W} - 13.32\left(\frac{a}{W}\right)^2 + 14.72\left(\frac{a}{W}\right)^3 - 5.6\left(\frac{a}{W}\right)^4\right] \tag{10}$$

Where P_Q is the load at which K_Q is found, B is the thickness of the specimen, W is the width of the specimen, a is the length of the crack. K_{max}

is calculated using the same Equations (9) and (10) as for the K_Q calculation, only replacing load P_Q with load P_{max}.

4.2 Numerical Model

The methodologies for determining K_I that were discussed in the preceding section are now applied to compact tension specimens (CTS). ANSYS was used to simulate mode 1 fracture propagation and assess the SIF along the crack front. The CT specimen's dimensions were measured according to ASTM standards, as indicated in Fig. 5. The impact of the number of elements and their sizes was investigated.

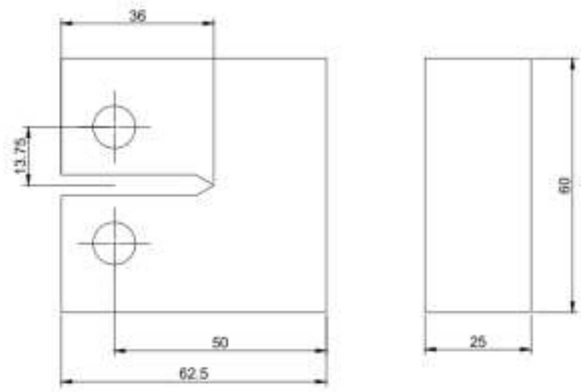

Fig. 5. Compact testing specimen according to the ASTM E 399 standard [11]

Iso-parametric SPENTA15 elements were used to discretise near the crack tip for investigation of singularity stress near crack tip, as illustrated in Fig. 6, and the remaining region was discretised using second order 20 noded hexahedron elements.

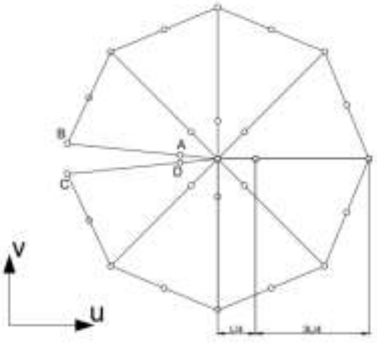

Fig. 6. SINGULAR element around the crack tip

The instructions mentioned above were then applied to the finite element model to simulate fracture toughness numerically. Since the specimen is symmetric, just half of it was generated to save time and effort in modelling and computation work. For all simulation conditions, a crack length of 1.8 mm was chosen. The analysis was done for two different loadings of P_Q 42.4 KN and P_{max} 82.7 KN. In a symmetric condition, displacement constraints were implemented. At the loading co-ordinate, a 3DMASS element was formed. The CT model's opening mode was ensured by the boundary and loading parameters, as illustrated in Fig. 7. The elements' compatibility conditions were ensured. The material of choice for the analysis was P91steel, E= 214 GPa and poisons ratio is 0.3.

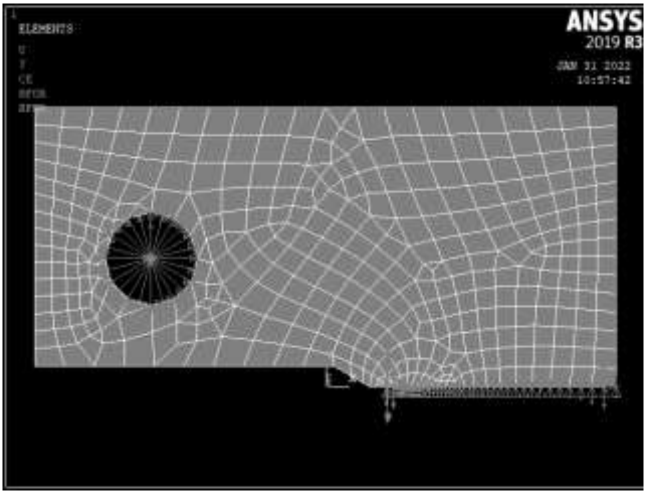

Fig. 7. Mesh, Loading and boundary conditions.

The crack was given an appropriate local coordinate system. The SIF was calculated in plane stress condition and local coordinate system. For the analysis of SIF, a Preconditioned Conjugate Gradient (PCG) solver was used. The element matrix formulation is the first step in this solver. PCG solvers collect the entire global stiffness matrix and iterate to convergence to calculate the DOF solution, instead of factoring the global matrix.

The impact of the number of elements and element size on the SIF was investigated using many mesh patterns around the fracture tip. As shown in Fig.8 SPENTA15 elements were used for configurations E-1 to E-3. Each of this number of elements around the crack tip varies. In configuration E-1, four elements (If the symmetry is taken into account, the total number is eight.) were meshed around the crack tip, eight (Sixteen) in E-2 and twelve (twenty-four) in E-3. The rest of the specimen was mesh with 15 node pentahedron and 20 node hexahedron elements.

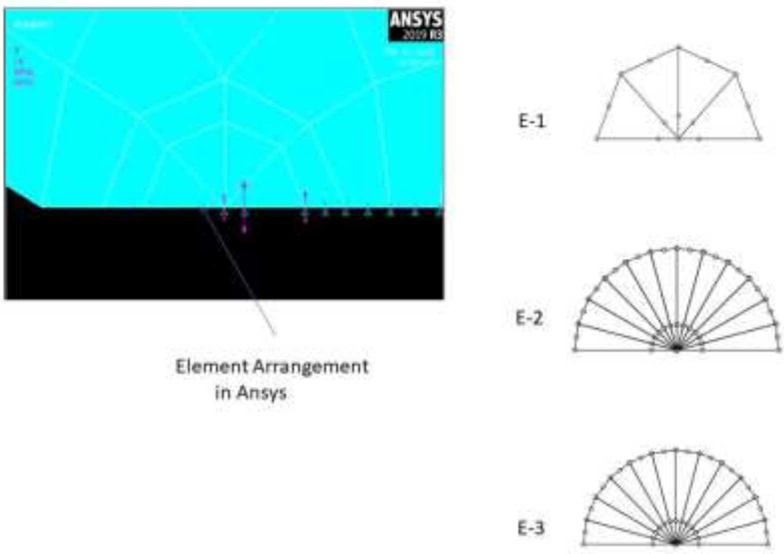

Fig. 8. Crack-tip modeling by configuration of number of elements

As shown in Fig. 9, varying size of elements were used to configure C-1 to C-3. C-1 with element size of 0.4, C-2 with element size of 0.6 and C-3 with element size of 0.8.

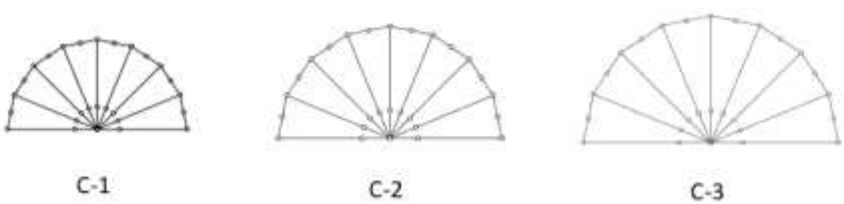

Fig. 9. Crack-tip modeling by configuration of size of elements

5.0 Results and Discussion

For varied loads, the SIF was calculated using displacement extrapolation method and FEM at the crack front. The displacement elements u, v, and w were extracted using ANSYS along the x, y, and z axes, which were then included in the equation. The experimental results were compared to the outcomes of both methods. [12].

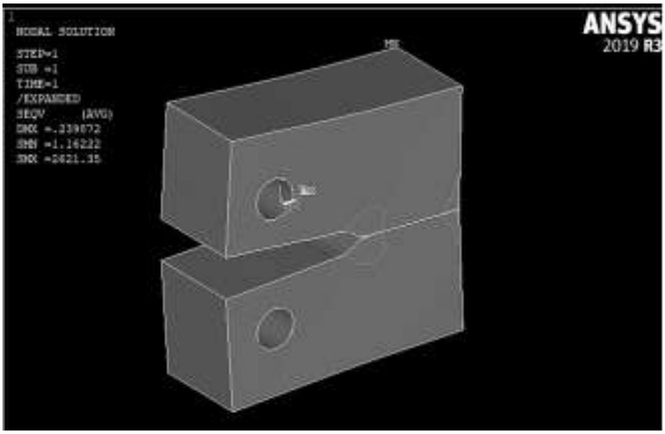

Fig. 10. Stressed model

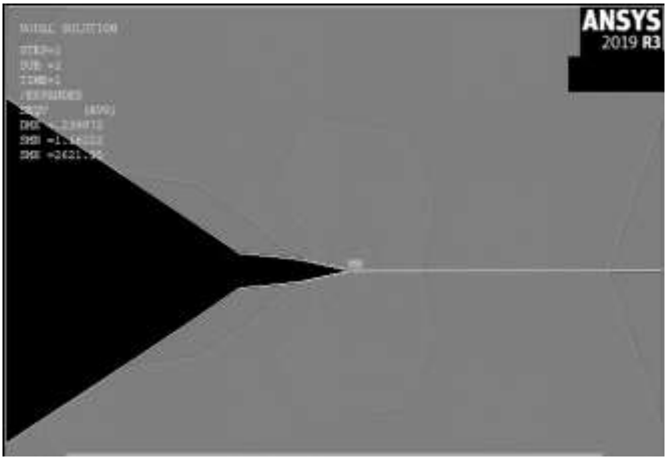

Fig. 11. Critical stress region and Plastic zone near the crack tip

The stress state and plastic zone size at the fracture tip are depicted in Fig. 10 and Fig. 11. It's worth noting that K_I accuracy is excellent for all setups under 5%. In this problem, the most accurate answers were obtained using the simplest estimating method, which is based just on the quarter point's displacement.

5.1 Influence of element size

Table 1 and Table 2 show the influence of the size of the elements on K_I for 42.4 KN and 82.7 KN, respectively. The element size to crack length ratio ranged from 0.2 to 0.4. The three approaches examined in this research appear to converge to the experimental values adequately [12]. Up to a particular element size, a local mesh refinement in the fracture tip zone can enhance K_I estimation. A finer mesh yields worse outcomes beyond this stage. This is the most important takeaway from the study.

Table 1. SIF for different element size at crack tip for P_Q 42.4KN

Configuration	**FEM K_I, MPa √m**	**Displacement extrapolation method, K_I, MPa √m**	**Experimental, K_I, MPa √m [12]**
C-1	72.504	72.77	75.1
C-2	72.63	72.97	
C-3	72.64	73.04	

Table 2. SIF for different element size at crack tip for P_{max} 82.7KN

Configuration	**FEM K_I, MPa √m**	**Displacement extrapolation method, K_I, MPa √m**	**Experimental, K_I, MPa √m [12]**
C-1	141.41	141.94	146.4
C-2	141.67	142.33	
C-3	141.67	142.46	

5.2 Influence of number of elements (Angular discretization)

The angular discretization around the fracture tip has the most impact on K_I estimation. Table 3 and Table 4 show the results for 42.4 KN and 82.7 KN, respectively. A bad meshing with least number of singular elements, two/three elements (four/six if symmetry is taken into account) can result in an erroneous K_I value. 4 (C-1), 8 (C-2) and 12 (C-3) element configurations were used for analysis. SPENTA 15 elements were used to create all crack tip meshes. The results are shown in Tables 3 and Table 4, which compare K_I values to experimental results for all setups. Figure 12 depicts the element arrangement in ANSYS in terms of configuration. It's worth noting that a mesh refinement with a bad angular discretization yield results that appear to converge to an incorrect estimate of K_I.

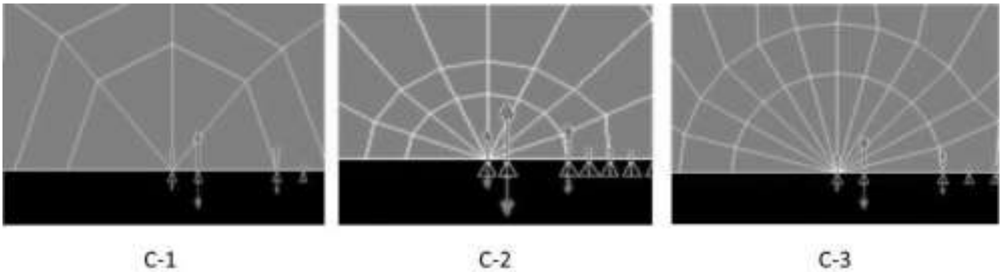

Fig. 12. Angular discretization at crack tip

Table 3. Stress intensity factor for different number of elements at crack tip for P_Q 42.4KN

Configuration	FEM K_I, MPa √m	Displacement extrapolation method, K_I, MPa √m	Experimental, K_I, MPa √m [12]
E-1	71.85	73.02	75.1
E-2	72.63	72.04	
E-3	72.64	72.94	

Table 4. Stress intensity factor for different number of elements at crack tip for P_{max} 82.7KN

Configuration	FEM K_I, MPa √m	Displacement extrapolation method, K_I, MPa √m	Experimental, K_I, MPa √m [12]
E-1	140.15	142.42	146.4
E-2	141.67	142.46	
E-3	141.673	142.26	

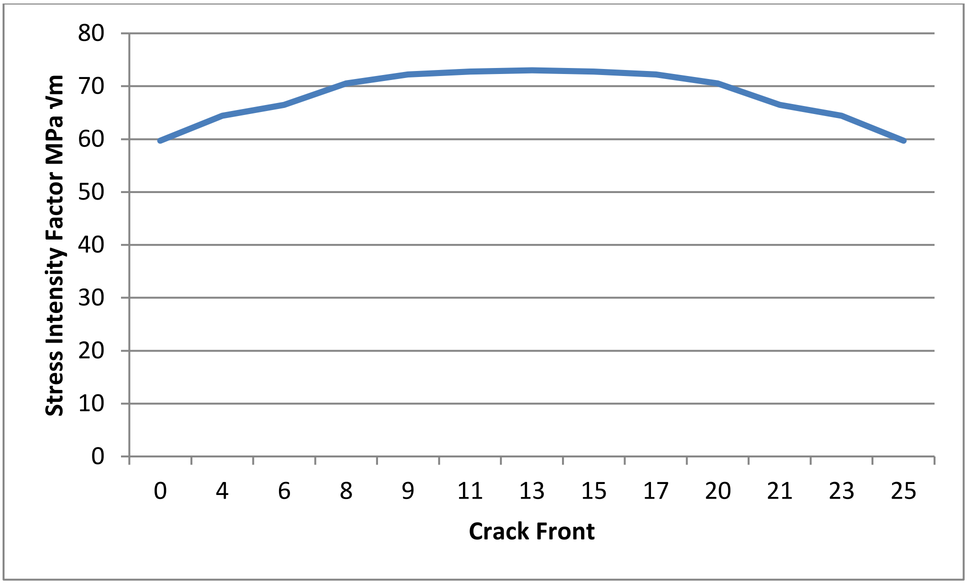

Fig. 13. Variation of SIF along the crack edge.

The stress intensity factor solution is shown in Fig. 13 along the crack edge. The SIF difference at the crack tip edge will follow the similar pattern for all configurations. The SIFs described here are regarded to be valid because they are based on comparisons with experimental data. The difference in SIF with respect to thickness of specimen is significantly variable. The insufficiency of 2D analysis is demonstrated by the fluctuation in the thickness of the SIFs as seen in the graph.

6.0 Conclusion

The serendipity family's Iso-parametric solid elements, pentahedral and hexahedral in form and quadratic in order are efficiently employed for edge-cracked component and structures are simulated using finite element in ANSYS, a general-purpose FEA application. The pre-processing command in ANSYS allows create specified size and number of singular iso-parametric pentahedral solid element (SPENTA15) the remaining of the volume under consideration can be discretise using a standard element of HEXA20 and PENTA15.

Experimental results were compared to the SIFs calculated using the displacement extrapolation method and finite element models developed with ANSYS. The results were discovered to be in perfect accord with the experimental results. When angular discretization (number of elements) around the fracture tip is done well, even for coarse meshes, the displacement extrapolation method can yield remarkably accurate predictions. If the angular discretization is excessively rough, such as 90^o

or 60^o elements, the results will be incorrect. A mesh refinement that just considers the element length size l, on the other hand, has no discernible effect on the accuracy of K_I predictions.

References

01. Anderson, T L Fracture Mechanics: Fundamentals and Applications, 4th ed., Taylor and Francis, Florida.2017

02. Kirthan L J , Hegde, Ramakrishna & Girisha, V A & Kumar, R Evaluation of mode 1 stress intensity factor for edge crack using displacement extrapolation method. *International Journal of Materials and Structural Integrity*. 10, 2016

03. Ali, Zare & Meysam, Kosari & Asadi, Iman & Aydin, Bigham & Yashar, Bigham, 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*. 3, 69-73, 2014

04. S K Chan, I S Tuba, W K Wilson, On the finite element method in linear fracture mechanics. *Engineering Fracture Mechanics*. 2(1), 1-17, 1970

05. Guinea, Gustavo & Guinea, V & Planas, Jaime & Elices, ManuelKI, Evaluation by the displacement extrapolation technique, *Engineering Fracture Mechanics*. 66. 243-255, 2000

06. Barsoum RS, Application of quadratic iso parametric finite elements in linear fracture mechanics, *International Journal of Fracture*. 10. 603-605, 1974

07. Ariza, M.P, Sfiez, A and Dominguez, J, A singular element for three-dimensional fracture mechanics analysis, *Engineering Analysis with Boundary Elements*, 20,275–285, 1997

08. R. D. Henshell and K. G. Shaw. Crack tip finite elements are unnecessary. *Int journal of numerical methods in engineering*. 9(3), 495-507, 1975

09. Kanninen, Melvin & Popelar, Carl & McEvily, A. Advanced Fracture Mechanics. *Journal of Engineering Materials and Technology*. 108. 199, 1986

10. Ingraffea, A R and Manu, C, Stress intensity factor computation in three dimensions with quarter point elements, *Int. J. Numerical Methods in Engineering*, 15, 1427–1445, 1980

11. ASTM E399. Standard Test Method for Linear-Elastic Plane-Strain Fracture Toughness KIc of Metallic Materials; ASTM International: West Conshohocken, PA, US, 2012.

12. Janulionis, Remigijus & Dundulis, Gintautas & Grybėnas, Albertas Numerical Research of Fracture Toughness of Aged Ferritic-Martensitic Steel, Metals. 10. 1686, 2020