CALIBRATION OF A TEST METHOD TO MEASURE MODE I, MODE II, AND MODE III FRACTURE TOUGHNESS OF ENGINEERING MATERIALS

L.J. Kirthan, H.V. Lakshminarayana and R.S. Shivashankar
Research and Development Centre
Department of Mechanical Engineering
Dayananda Sagar College of Engineering
Bangalore-560 000, India
Emails : contactkirthan@gmail.com; hvl_mech2007@rediffmail.com and soryes@gmail.com

Abstract

A refined finite element model and a new post processing sub program to determine mixed mode stress intensity factors and their variation along a surface crack front in components and structures is presented. The proposed finite element model employs a fine mesh of singular isoparametric pentahedral solid element with user specified number and size from one crack face to another and a number of such segments along a surface crack front. A compatible mesh of regular elements namely hexahedral solid element and pentahedral solid element is used to discretize the rest of the domain under consideration. Consistent with the use of the singular element, formulae to compute the Mode I, Mode II and Mode III stress intensity factors using displacements only of flagged nodes on flagged singular elements are implemented in a special purpose post processing sub program named SIF 1-2-3. In the present work the finite element models developed using ANSYS, a commercial FEA program, and the stress intensity factors determined using SIF 1-2-3 are validated using benchmarks. This finite element model is then used to calibrate a proposed test method to measure Mode I, Mode II and Mode III fracture toughness of engineering materials.

Introduction

Fracture is a failure mode due to unstable propagation of a crack due to applied stress. Fracture mechanics provides a methodology for prediction, prevention, and control of fracture in materials, components and structures subjected to static, dynamic, and sustained loads. Fracture mechanics analysis is the basis of damage tolerance design methodology. The objectives of fracture mechanics analysis are determination of (1) Stress intensity factor (K), (2) Energy release rate (G), (3) Path independent integral (J), (4) Crack tip opening displacement (CTOD), and prediction of (1) Mixed mode fracture, (2) Residual strength and (3) Crack growth life.

A comprehensive review of structural failures revealed that the origin of failures due to cracks, in order of frequency of occurrence to be (1) Surface cracks,(2) Through thickness cracks ,(3) Corner cracks, and (4) Cracks emanating from fastener holes. Such cracks are truly three dimensional crack configurations. Two dimensional approximations to these cracked bodies as plane stress/plane strain are usually unsatisfactory and inaccurate. The focus of this work is on the determination of mixed mode stress intensity factors and their variations along a surface crack front in complex components and structures. Accurate stress intensity factor solutions for these configurations therefore can be obtained only by solving 3D boundary value problems.

An exhaustive survey of methods for three dimensional analysis of cracked solids and structures is presented by Raju and Newman in [1]. An up to date survey and assessment of published literature identifies the finite element method in general and a commercial FEA software in particular for fracture mechanics analysis of solids and structures with surface cracks.

Paper Code : V64 N2/756-2012. Manuscript received on 05 Sep 2011. Reviewed and accepted as a Full Length Contributed Paper on 16 Apr 2012

Finite element modeling using Singular and regular iso-parametric elements is recommended to be used in practice because it permits the use of commercial FEA programs with minimum enhancement. However the mixed mode stress intensity factors have to be calculated posteriori. Validation of finite element models featuring both singular and regular element meshes and the chosen SIF evaluation procedures demand benchmarks with known target solutions that resemble the current cracked configurations being studied.

In the design against fracture of structures and components, complex three dimensional configurations with surface cracks are encountered. Recent advances in the FEM, commercial FEA programs and availability of large, fast computers have led to more refined finite element modeling and fracture mechanics analysis of complex cracked bodies.

Several investigators have proposed Special crack tip elements which yield SIF (K) as part of the solution. These are enriched elements, stress hybrid elements and displacement hybrid elements. Unfortunately these elements are not implemented in general purpose FEM systems and therefore cannot be used in practice on a day to day basis.

Using the singular and regular iso-parametric elements implemented in commercial FEA program to FE modeling of cracked components and structures, three methods are generally used to extract SIF (K). They are Crack Opening Displacement (COD) method [2],Virtual crack closure techniques and its variants [3, 4, 5] and Virtual Crack Extension (VCE) method [6]. Comparisons between SIF solutions for several surface crack problems shows that all the three methods yield nearly identical solution when plane strain conditions are assumed along crack front.

Despite the complexity of many cracked components and structures, comparison between available methods of analysis has shown that accurate SIF (K) can be obtained. The choice of particular method is governed by the analyst's expertise, the available FEA program and hardware resources to obtain the solution.

As the analysis of surface crack problem is completed, compendia of SIF like the ones available for 2D configurations can be developed. Such a compendia should be the focus of research in the future. The compendia of SIF solution to surface crack problems can help engineers design structures which are safe, economical and damage tolerant.

The overall aim of this paper is to present the development and validation of a refined finite element model and a new post processing subprogram to calculate mixed mode stress intensity factors (K_I , K_{II}, K_{III}) for arbitrarily located and oriented surface/corner cracks in components and structures. Its application to the calibration of a proposed test method to measure Mode I, Mode II and Mode III fracture toughness of engineering materials turns out to be a case study.

The proposed finite element model involves a very fine mesh of singular isoparametric pentahedral solid element (SPENTA15) with user specified number (NS) and length (Δa) from one crack face to another and number of segments (NSEG) along the surface crack front. A compatible mesh of regular elements PENTA15 and HEXA20 is used to discretize the rest of the domain.

Consistent with the use of SPENTA15 elements, formulae to compute mixed mode stress intensity factors (K_I, K_{II}, K_{III}) using the displacements of flagged nodes in flagged singular elements have been derived[2] and these are implemented in a special purpose post processing subprogram called **SIF1-2-3**.

In the present work, the finite element model is created using commercial FEA software ANSYS and validated using benchmarks. Extensive stress intensity factor solutions covering a range of crack lengths are graphically presented and discussed for the compact tension shear test specimen.

Finite Element Model Development

Finite element modeling is defined here as the analyst's choice of material models, finite elements (type/shape/order), meshes, constraint equations, pre and post processing options, governing matrix equations and their solution methods available in a chosen commercial FEA program for the intended analysis.

The proposed finite element model involves a very fine mesh of singular isoparametric pentahedral solid elements (SPENTA15) with user specified number NS and length Δa from one crack face to another and number of segments (NSEG) along the surface crack front.

A compatible mesh of regular elements (NREG) namely isoparametric pentahedral solid element (PENTA15) and isoparametric hexahedral solid element (HEXA20) are used to discretize the rest of the domain under consideration. A brief insight into the formulation of these elements is given.

Figure 1 shows quadratic order hexahedral solid element of the so called Serendipity family. The element has twenty 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.

Any variable Φ is approximated over the parent element domain using the incomplete quadratic order polynomial. Explicit shape functions N_i (ξ, η, ζ) ($i = 1 \ldots 20$) have been derived for this polynomial basis and are readily available. Using iso-parametric formulation, the parent element can be distorted to have straight or curved edges and flat or curved faces as illustrated in Fig.1. Use of a three-point Gauss quadrature formula in each of the ξ, η, ζ coordinate directions is recommended to compute the element matrices and vectors. The required gauss point locations and weighting factors are readily available. 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 (15Nodes) is shown in Fig.2. This is designed by further distorting the HEXA20 element as illustrated in Fig.2. 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.

One more distortion of the PENTA15 element creates SINGULAR element called SPENTA15 for computational fracture mechanics [7] and is shown in Fig.3. Specifically, the edge with nodes 3-19-7 is located along a curved crack front. The mid-side nodes 10, 12, 14, 16 are moved to quarter point locations closer to the crack front. The number of the SPENTA15 elements (NS) from one crack face to the other and the number of segments (NSEG) along a crack front can be progressively increased and the size of the singular elements Δa can be decreased to achieve convergence in computed stress intensity factors.

The rest of the domain under consideration is discretized using a compatible mesh of regular elements (NREG). Numerical experiments are necessary to arrive at satisfactory values for NS, Δa, NSEG and NREG for each problem to ensure convergence of computed stress intensity factors.

In this work the finite element model is created using ANSYS program. However, in ANSYS the required singular element is not listed in the element library. Therefore the preprocessing commands and user experience is essential for the concurrent creation of SPENTA 15 element mesh along any curved crack front with specified NS, Δa and NSEG.

Post Processing Sub Program : SIF 1-2-3

Mixed Mode Stress Intensity Factors denoted by K_I, K_{II}, K_{III} at any point along a surface crack front have to be calculated posteriori. This involves derivation of formulae to calculate K_i (i = I, II, III) using nodal displacements at flagged nodes of flagged elements only. In the sequel setting up of a unique crack tip coordinate system (x, y, z) with the x-axis normal to the crack front and in the plane of the crack, with the y-axis normal to the crack front and normal to the crack plane, and with the z-axis tangential to the crack front in the crack plane at user selected nodes along the surface crack front is a prerequisite. The computation of direction cosines of the x, y, z axes needs to be automated. The known nodal displacements U_i, V_i, W_i with respect to the global Cartesian coordinates (X, Y, Z) are transformed to the crack tip coordinates. Using the relative displacements it is then possible to identify the opening (Mode I), in plane sliding (Mode II) and anti-plane shear (Mode III) modes and calculate K_I, K_{II}, K_{III} using SIF evaluation formulae given in [2]. A new Post Processing Sub Program named SIF 1-2-3 has been developed and used in this study with ANSYS as the solver.

In the development of SIF 1-2-3 a local cartesian coordinate system (x,y,z) is defined at each crack tip node and the computation of the direction cosines of x,y,z is automated. These direction cosines are used to create a transformation matrix [λ] defined as

$$[\lambda] = \begin{bmatrix} Cos\,(x\,,X)\,,\,Cos\,(x\,,Y)\,,\,Cos\,(x\,,Z) \\ Cos\,(y\,,X)\,,\,Cos\,(y\,,Y)\,,\,Cos\,(y\,,Z) \\ Cos\,(z\,,X)\,,\,Cos\,(z\,,Y)\,,\,Cos\,(z\,,Z) \end{bmatrix} \tag{1}$$

where X,Y,Z denote global cartesian coordinates used in the solver. The nodal degrees of freedom in the crack tip coordinate system (x,y,z) are then computed as

$$\begin{bmatrix} u_i \\ v_i \\ w_i \end{bmatrix} = [\lambda] \begin{bmatrix} U_i \\ V_i \\ W_i \end{bmatrix} \quad (2)$$

Figure 4 shows the flagged SPENTA 15 elements one on each crack face. The stress intensity factors are calculated using the following formulae consistent with the use of SPENTA15 elements [2];

$$K_I = \frac{E}{4(1-\nu^2)} \sqrt{\left(\frac{\pi}{2L_1}\right)} \begin{bmatrix} 2v_B - v_C + 2v_E - v_F + v_D - 2v_{B'} + v_{C'} - 2v_{E'} + v_{F'} - v_{D'} + \\ \frac{1}{2}\eta(-4v_B + v_C + 4v_E - v_F + 4v_{B'} - v_{C'} - 4v_{E'} + v_{F'}) + \\ \frac{1}{2}\eta^2 (v_F + v_C - 2v_D - v_{F'} - v_{C'} + 2v_{D'}) \end{bmatrix} \quad (3)$$

$$K_{II} = \frac{E}{4(1-\nu^2)} \sqrt{\left(\frac{\pi}{2L_1}\right)} \begin{bmatrix} 2u_B - u_C + 2u_E - u_F + u_D - 2u_{B'} + u_{C'} - 2u_{E'} + u_{F'} - u_{D'} + \\ \frac{1}{2}\eta(-4u_B + u_C + 4u_E - u_F + 4u_{B'} - u_{C'} - 4u_{E'} + u_{F'}) + \\ \frac{1}{2}\eta^2 (u_F + u_C - 2u_D - u_{F'} - u_{C'} + 2u_{D'}) \end{bmatrix} \quad (4)$$

$$K_{III} = \frac{E}{4(1+\nu)} \sqrt{\left(\frac{\pi}{2L_1}\right)} \begin{bmatrix} 2w_B - w_C + 2w_E - w_F + w_D - 2w_{B'} + w_{C'} - 2w_{E'} + w_{F'} - w_{D'} + \\ \frac{1}{2}\eta(-4w_B + w_C + 4w_E - w_F + 4w_{B'} - w_{C'} - 4w_{E'} + w_{F'}) + \\ \frac{1}{2}\eta^2 (w_F + w_C - 2w_D - w_{F'} - w_{C'} + 2w_{D'}) \end{bmatrix} \quad (5)$$

Where E is the young's modulus of elasticity

υ is the Poisson's ratio

L_1 is the length of the singular element

η is the variable along the crack front and $\eta = 1$ at the crack tip

B, C, E, F, D, and B', C', E', F', D', are the flagged nodes identified in Fig.4. u_B, v_B, w_B, $u_{B'}$, $v_{B'}$, $w_{B'}$ etc denote the relative nodal displacements at the flagged nodes with respect to the crack tip nodes A or G.

In the post processing sub program SIF 1-2-3 the user has the option to input the matrix [λ] or to automatically compute the same with additional inputs. The nodal degrees of freedom at nodes identified in Fig.4 as A, B, C, D, F, E, G and A, B', C', D', F', E', G are extracted from the solver and transformed to the crack tip coordinate system and used to calculate the stress intensity factors. The computed stress intensity factors are normalised using $K_o = \sigma_o \sqrt{\pi a}$ where σ_o is a reference stress input by the user and 'a' denotes crack length.

Finite Element Model Validation

Benchmark is a standard test problem [11] with a known target solution in the form of formulae, tables and graphs obtained using analytical methods, experimental techniques and computational procedures.

Figure 5 shows a square bar with a quarter circular corner crack. For very small crack length a (in comparison with the width W) the Mode I stress intensity factor K_I and its variation along the crack front is well known [12]. Fig.6 shows the dimensions of the bar used for computation and its finite element model is presented in Fig.7. A refined mesh of SPENTA 15 elements with NS = 30, Δa = a/100, NSEG = 30 is used along the crack front. A compatible mesh of HEXA20 and PENTA15 elements is used to discretize the rest of the domain. Nodes along the bottom surface restrained against the loading direction. Uniform pressure load is specified along the top surface. The finite element model shown in Fig.7 is created using pre-processing capabilities in the ANSYS program.

The post processing sub program SIF 1-2-3 is used to calculate the stress intensity factors. The Mode I stress intensity factor K_I and its variation along the circular crack front is presented in Fig.8 for different values of (a/w). It's evident that as (a/w) tends to zero the present solution approaches the target solution. For larger values of a/w the free surface effect on the stress intensity factor is noticeable. The maximum value of K_I (which occurs at the free surfaces is plotted against a/w in Fig.9).

The graphical post processing capabilities in ANSYS is demonstrated by isolating the mesh of SPENTA15 elements along the curved crack front and capturing the von Mises equivalent stress contours at three different sections namely ϕ = 0, 45 and 90 degrees as illustrated in Fig.10. Given the yield strength of the material it is then possible to study the variation of the plastic zone shape and size along the crack front.

Verification of these numerical results using experimental techniques such as 3D photo elasticity is identified as a research opportunity.

Figure 11 reproduced from reference [8] shows the geometry of SEN specimen in three-point bending with photo elastic pattern insert for one crack length. Fig.12 also taken from reference [8] presents the results from 2D photo elastic analysis of the above geometry along with those of the boundary collocation analysis for the full range of crack lengths. This is the target solution for this benchmark.

A refined mesh of SPENTA15 elements with NS=60, $\Delta a = (a/100)$ and NSEG = 40 is used along the crack front between crack faces. A compatible mesh of HEXA20 and PENTA15 elements is used to discretize the rest of the domain. The complete FE model created using ANSYS pre-processing capabilities is shown in Fig.13.

The post processing sub program SIF 1-2-3 is used to calculate the stress intensity factors. The Mode I stress intensity factor is expressed as $K_I = \frac{PS}{BW^{\frac{3}{2}}} f\left(\frac{a}{w}\right)$, where P is the applied load, S is the span, B is the specimen thickness and W is the specimen width. The normalised Mode I stress intensity factor K_I and its variation along the crack front is presented in Fig.14 for a/w = 0.5 and 0.7. Near the surfaces the free surface effect on K_I is significant and in fact increases with the crack length. It has been verified that the present 3D solution is comparable with the target solution at the centre of the specimen for all crack lengths.

Case Study

A unique test method to measure Mode I, Mode II and Mode III fracture toughness of engineering materials is proposed [9,10]. Using a compact tension shear (CTS) specimen and a pair of loading fixtures bolted to the specimen it is possible to measure K_{IC}, K_{IIC} and K_{IIIC}. It is also possible to simulate mixed mode fracture as illustrated in Fig.15.

Finite element modeling to simulate Mode I fracture is displayed in Fig.16. A refined mesh of SPENTA15 element is used along the crack front with NS = 60, $\Delta a = (a/100)$ and NSEG = 60. A compatible mesh of regular elements namely HEXA20 and PENTA15 is used to discretize the rest of the CTS specimen domain and fixtures. Compatibility is ensured at the interfaces between CTS specimen and fixtures. Different material properties can be assigned to the specimen and the fixtures. The single point constraints and nodal forces used in the analysis are also highlighted in Fig.16.

The special purpose post processing sub program SIF 1-2-3 is used to compute the stress intensity factors at user specified nodes along the crack front. The results are expressed as $K_I = \frac{P}{B\sqrt{W}} f\left(\frac{a}{w}\right)$ where P is the applied load, B is the specimen thickness and W is the specimen width.

Figure 17 presents the normalised Mode I stress intensity factor K_I and its variation along the crack front for (a/w) = 0.5. Fig.18 presents the contour plots of von Mises equivalent stress in the immediate vicinity of the crack tip. Knowing the yield strength of the material it is therefore possible to study the size and shape of the plastic zone all along the crack front.

Normalised stress intensity factor K_I as a function of crack length is presented in Fig.19. This data is essential to reduce the measured critical load and crack length at fracture to Mode I fracture toughness K_{IC}. An empirical relation between the Mode I stress intensity factor K_I and the crack length (a/w) is presented here as $K_I = \frac{P}{B\sqrt{W}} f\left(\frac{a}{w}\right)$ and

$$f(a/w) = 52.167\,(a/w)^3 - 29.58\,(a/w)^2 + 11.54\,(a/w) - 0.1942 \qquad (6)$$

Finite element modeling to simulate Mode II fracture is displayed in Fig.20 and this is identical to the one used to study Mode I fracture. However the single point constraints and nodal forces used in this analysis are different as highlighted in Fig.20.

Figure 21 presents the normalised Mode II stress intensity factor and its variation along the crack front for (a/w) = 0.5.

Normalised Mode II stress intensity factor K_{II} as a function of (a/w) is presented in Fig.22. This data is essential to arrive at Mode II fracture toughness denoted by K_{IIC}. An empirical relation between the Mode II stress intensity factor K_{II} and the crack length (a/w) is presented here as $K_{II} = \frac{P}{B\sqrt{W}} f\left(\frac{a}{w}\right)$ and

$$f(a/w) = 50.587\,(a/w)^3 - 41.359\,(a/w)^2 + 14.337\,(a/w) - 0.8897 \qquad (7)$$

Finite element modeling to simulate Mode III fracture is displayed in Fig.23 and this is identical to the one used to study Mode I fracture. However the single point constraints and nodal forces used in this analysis are drastically different as highlighted in Fig.23. Figure24 presents the normalised Mode III stress intensity factor and its variation along the crack front for (a/w) = 0.5.

Normalised Mode III stress intensity factor K_{III} as a function of (a/w) is presented in Fig.25. This data is essential to arrive at Mode III fracture toughness denoted by K_{IIIC}. An empirical relation between the Mode III stress intensity factor K_{III} and the crack length (a/w) is presented here as $K_{III} = \frac{P}{B\sqrt{W}} f\left(\frac{a}{w}\right)$ and

$$f(a/w) = 1769.8\,(a/w)^3 - 1283.8\,(a/w)^2 + 330.63\,(a/w) - 7.6687 \quad (8)$$

A critical discussion of the results presented above is in order. The stress intensity factors presented here are based on converged finite element solutions and therefore believed to be accurate. The variation across the thickness of stress intensity factors is quite different and characteristic of the three modes of fracture. The variation along the thickness of the stress intensity factors shows the inadequacy of 2D analysis. The superposition of deformed over undeformed meshes when animated clearly identified the opening (Mode I), in plane sliding (Mode II) and anti-plane shear (Mode III) are correctly simulated in the analysis. This was also supported by the numerical results for K_I, K_{II} and K_{III} for all crack lengths considered.

The contour plots of the von Mises equivalent stress on the surface for Mode I, Mode II and Mode III do resemble the well-known shapes presented in text books. However, an in depth post processing to capture their variation along the specimen thickness using the results of elastic-plastic analysis will provide quantification of plastic zone shape and size.

The calibration of the proposed test method using CTS specimen and two loading fixtures bolted to the specimen is essential to reduce the measured fracture load and crack length into fracture toughness values denoted by K_{IC}, K_{IIC}, K_{IIIC}. The test method also enables mixed mode fracture tests to be performed under all possible combinations of Mode I, Mode II and Mode III. The data from such tests can be used to verify predictability of mixed mode fracture criteria.

Conclusion

Isoparametric solid elements of the serendipity family, hexahedra and pentahedra in shape and quadratic in order implemented in ANSYS, a general purpose FEA program, are effectively used for finite element modeling of surface cracked components and structures. The pre-processing commands in ANSYS enable the creation of a refined mesh of singular isoparametric pentahedral solid element (SPENTA15) with user specified number (NS) and size (Δa) from one crack face to another and a number of such segments (NSEG) along a surface crack front. A compatible mesh of HEXA20 and PENTA15 can be used to model the rest of the domain under consideration. However determination of mixed mode stress intensity factors and their variation along a crack front demanded the development and use of a new post processing sub program SIF 1-2-3.

The finite element models developed using ANSYS and the stress intensity factors determined using SIF 1-2-3 together are validated using two benchmarks.

The calibration of the proposed test method using CTS specimen and two loading fixtures bolted to the specimen presented here is essential to reduce the measured fracture load and crack length into fracture toughness values denoted by K_{IC}, K_{IIC}, K_{IIIC}.

Acknowledgement

Authors are thankful to esteemed reviewers whose comments and suggestions have significantly improved the quality of presentation.

References

1. Raju, I. S and Newman, J. C., "Methods for Analysis of Cracks in Three Dimensional Solids", Journal of Aeronautical Society of India, Vol.36, No.3, August, 1984.

2. Ingraffea, A. R and Manu, C., "Stress Intensity Factor Computation in Three Dimensions with Quarter Point Elements", Int. J. Numerical Methods in Engineering, Vol.15, 1980, pp.1427-1445.

3. Krueger, R., "Virtual Crack Closure Technique: History, Approach and Applications", Applied Mechanics Reviews, Vol.57, 2004, pp.1092-143.

4. Badari Narayana, K., Dattaguru, B., Ramamurthy, T.S and Vijaya Kumar, K., "A General Procedure for Modified Crack Closure Integral in 3D problems with Cracks", Eng. Fracture Mech., 48, 1994, pp.167-176.

5. Dattaguru, B., Lok Singh, K and Palani, G.S., "Genelarisation of MVCCI Approach for LEFM Problems Using Numerical Integration", Journal of Aerospace Sciences and Technologies, Vol.61, No.1, 2000, pp.100-110.

6. Parks, D. M., "A Stiffness Derivative Finite Element Technique for Determination of Crack Tip Stress Intensity Factors", Int. J. Fracture, 109,1974, pp.487-502.

7. Koers, R. W. J., "Use of Modified Standard 20-node Isoparametric Brick Elements for Representing Strain/Stress Fields at a Crack Tip for Elastic and Perfectly Plastic Materials", Int. J. Fracture, Vol.40, 1989, pp. 70-110.

8. Sanford, R.J., Principles of Fracture Mechanics, Pearson Education, NJ, 2003.

9. Richard, H.A and Benitz, K., "A Loading Device for the Creation of Mixed Mode in Fracture Mechanics", Int. J. Fracture, Vol.22, 1983, pp.R55-R58.

10. Buchholz, F. G., "Finite Element Analysis of a 3D Mixed Mode Fracture Problem by Virtual Crack Closure Integral Methods, Fracture Mechanics", Proceedings of the Indo-German Workshop, Indian Institute of Science, Bangalore, March, 1994, pp. 7-12.

11. Judge, R. C. B and Marsden, B. J., "Three-Dimensional Test Cases in Linear Elastic Fracture Mechanics", NAFEMS, UK, 1993.

12. Simha, K. R. Y., "Fracture Mechanics for Modern Engineering Design", Universities Press, Hyderabad, 2001.

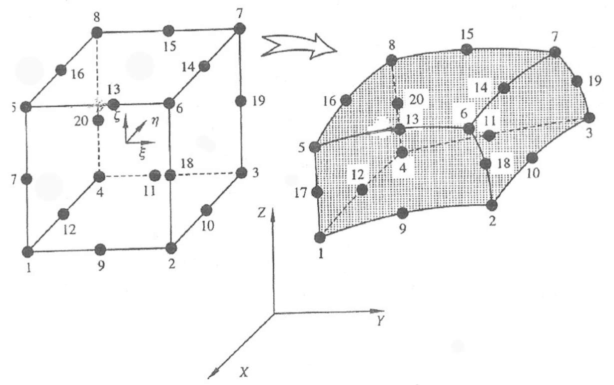


Fig.1 HEXA 20 Element

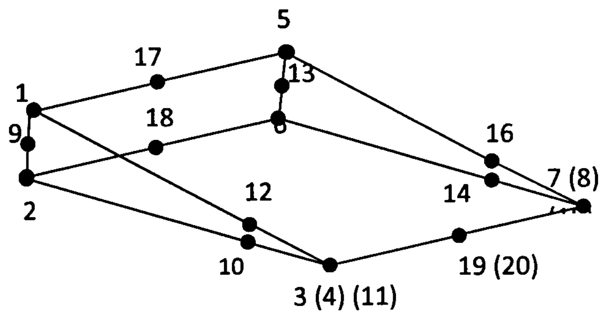


Fig.3 SPENTA 15 Element

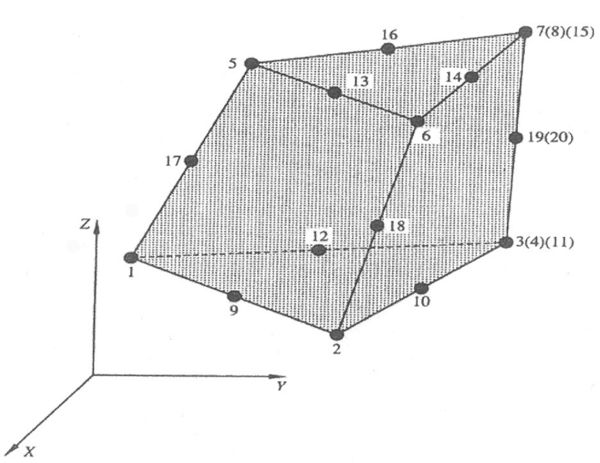


Fig.2 PENTA 15 Element

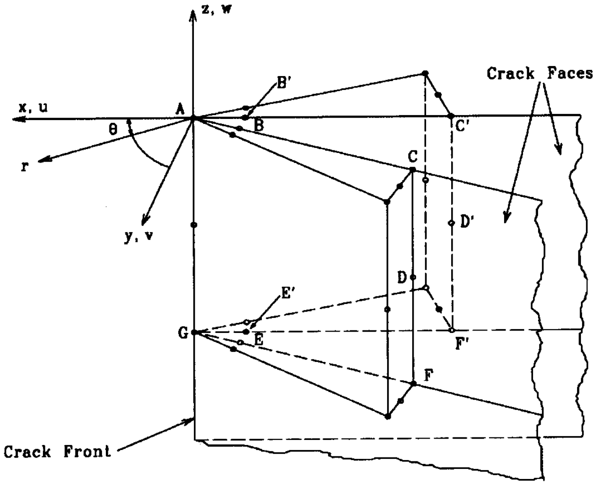
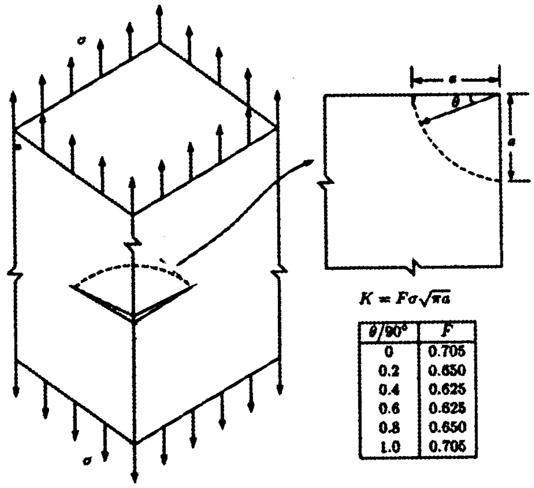


Fig.4 Flagged Singular Elements and Flagged Nodes



θ/90°	F
0	0.705
0.2	0.650
0.4	0.625
0.6	0.625
0.8	0.650
1.0	0.705

Fig.5 Square Bar with Quarter Circular Crack

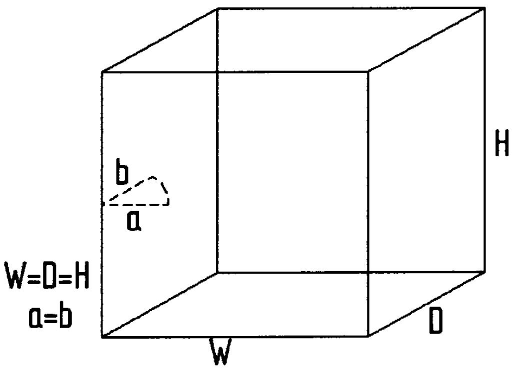


Fig.6 Square Bar with a Corner Quarter Circular Crack

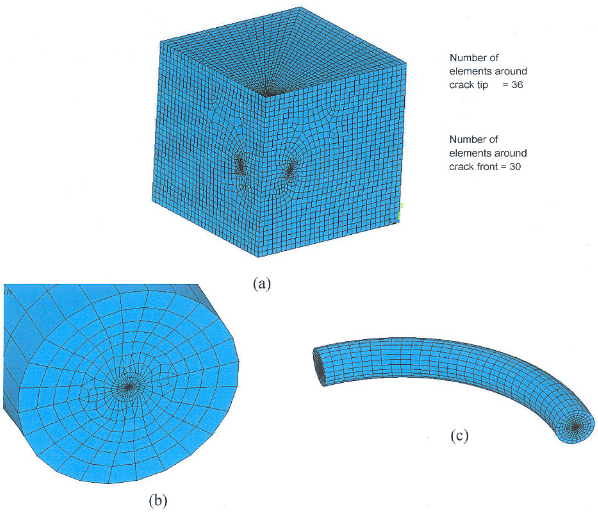


Fig.7 Square Bar with a Corner Quarter Circular Crack
(a) FE Model (b) Singular Elements Around Cracktip
(c) FE Mesh Around Crack Front

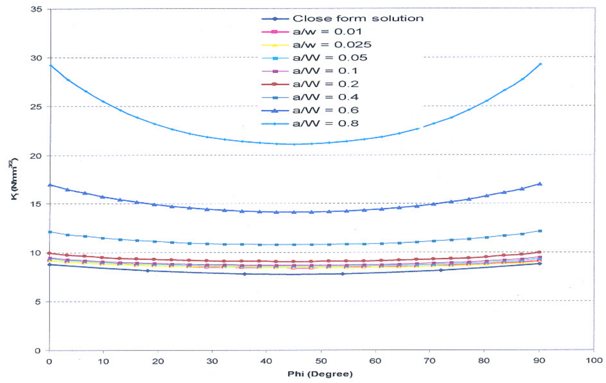


Fig.8 Variation of K_I with Different a/W Ratio

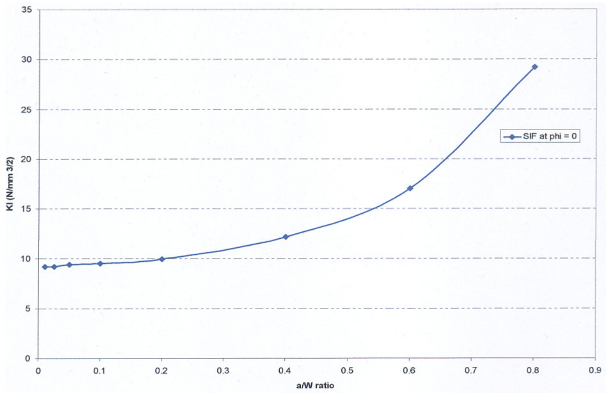


Fig.9 Variation of K_I with Different a/w Ratio @ ϕ = 0°

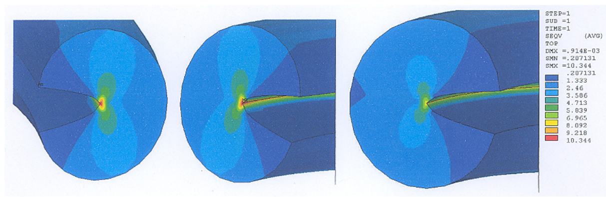


Fig.10 Von Mises Stress Plots for ϕ = 0, 90 and 45°

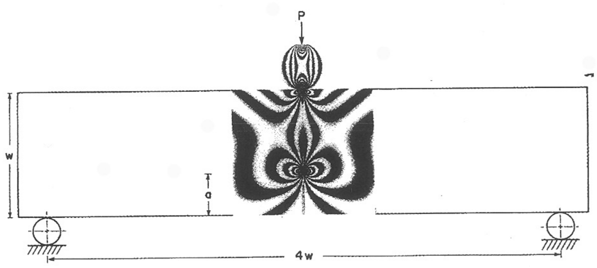


Fig.11 Geometry of the Three-point Bend Specimen with Photoelastic-pattern Insert for One Crack Length

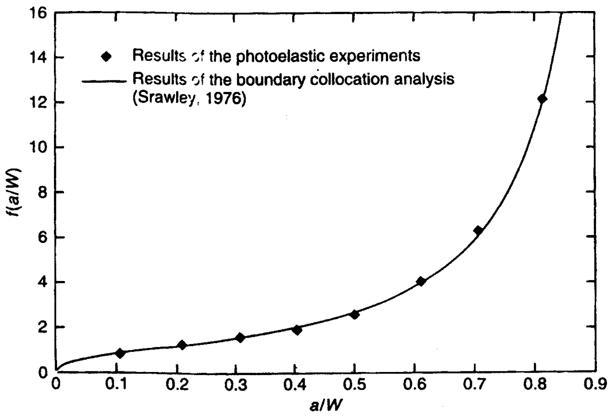


Fig.12 Comparison of the Results from the Photoelastic Analysis of the Three-point Bend Geometry with Those of the Boundary-collocation Analysis for the Full Range of Crack Lengths

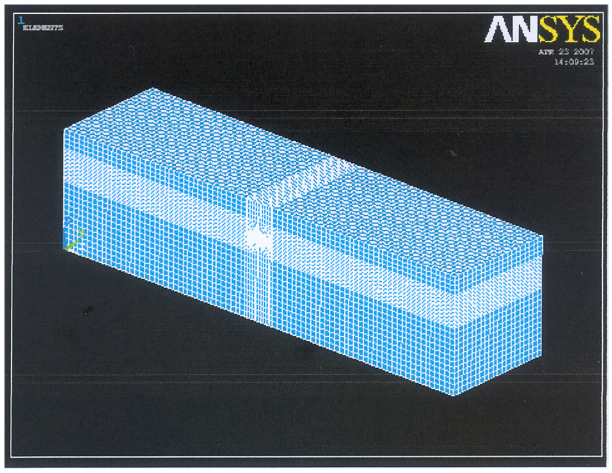


Fig.13 Single Edge Notch Bar and its FE Model

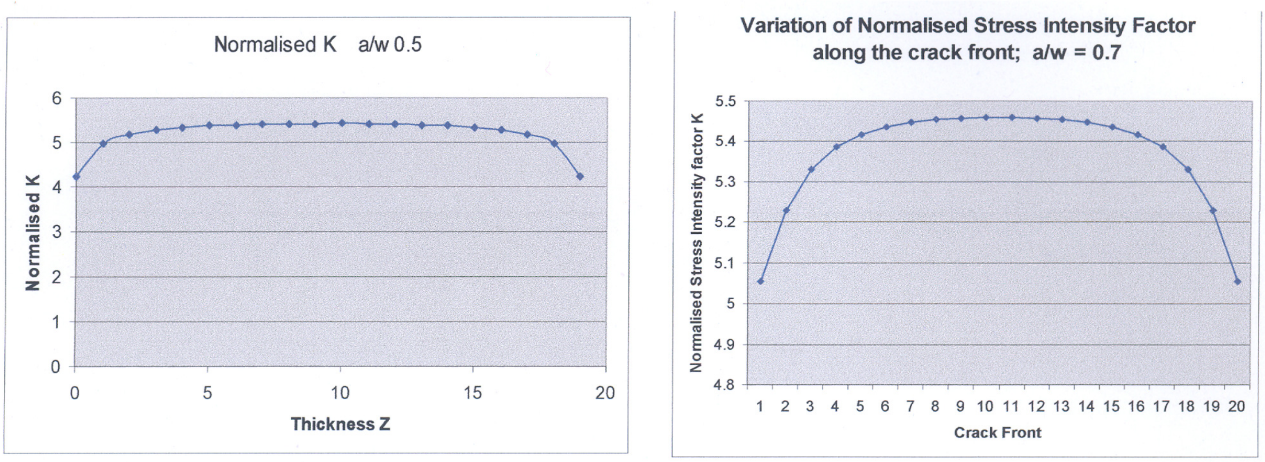


Fig.14 Normalised Stress Intensity Factor : Variation Along Thickness

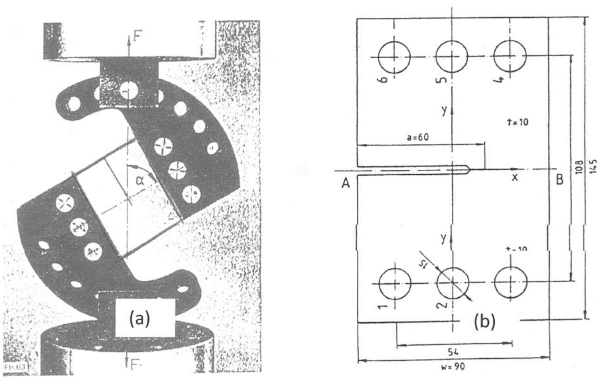


Fig.15 (a) Loading Device and (b) CTS Specimen Mounted in a Standard Testing Machine (in-plane tension/shear loading, α = 60°, a/w = 2/3, w = 90, d = 68, t = 10 mm, F = 40 kN)

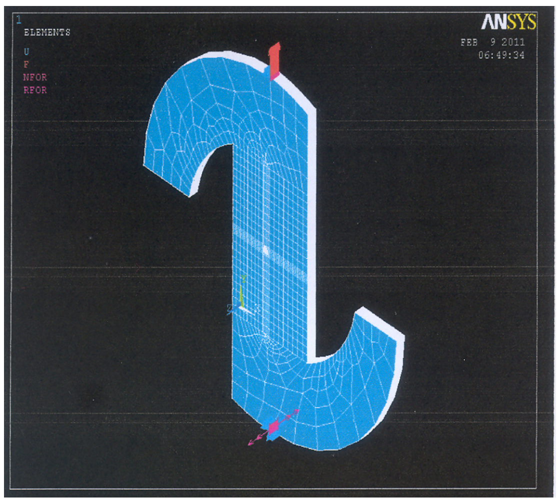


Fig.16 Finite Element Modeling of CTS Specimen and Loading Fixture to Study Mode I Fracture

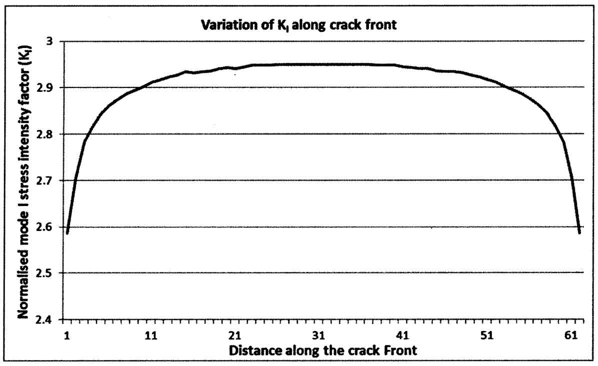


Fig.17 Normalized Mode I Stress Intensity Factor K_I and its Variation Along Crack Front

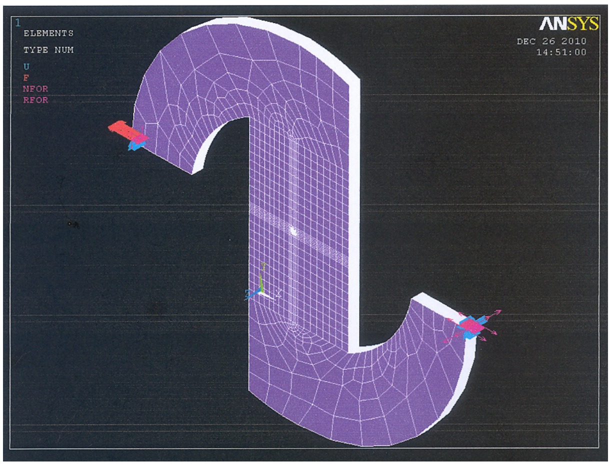


Fig.20 Finite Element Model of CTS Specimen and Loading Fixture to Study Mode II Fracture

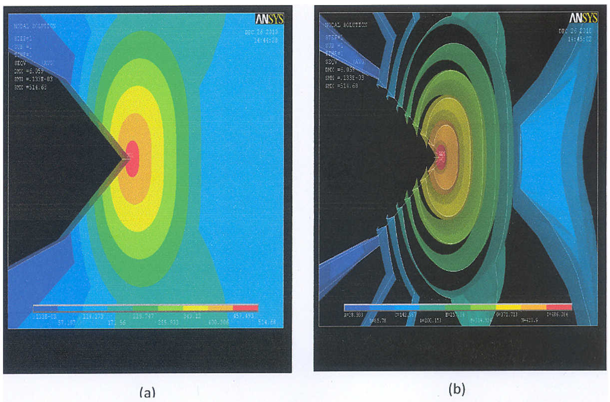


Fig.18 Von Mises Stress Contours Around the Crack Tip : Mode I (a) Stress Contours (b) Stress Line Contours

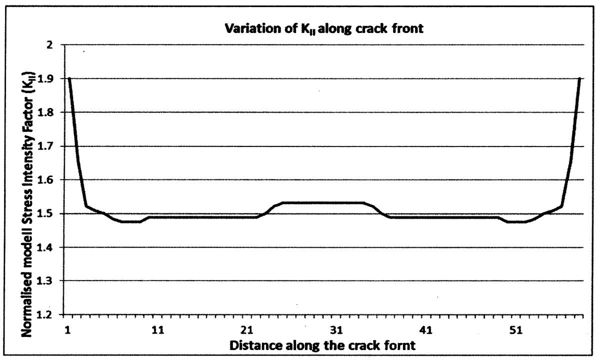


Fig.21 Normalized Mode II Stress Intensity Factor KII and its Variation Along Crack Front

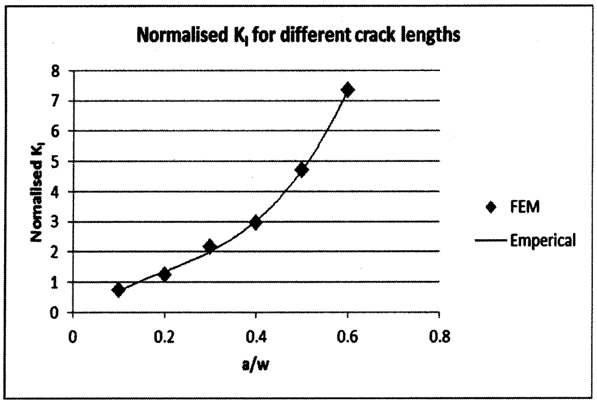


Fig.19 Normalized Mode I Stress Intensity Factor K_I for Different a/w Ratios

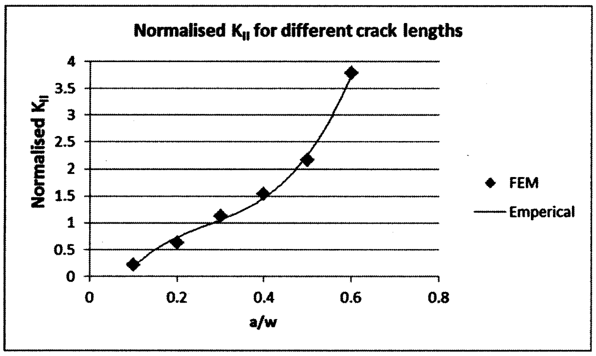


Fig.22 Normalized Mode II Stress Intensity Factor K_{II} for Different a/w Ratios

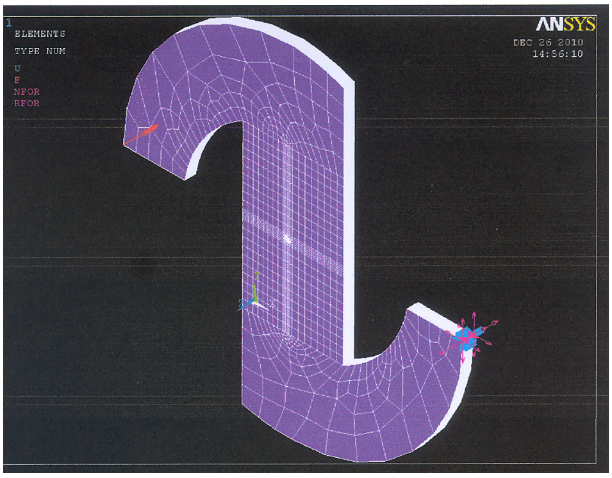


Fig.23 Finite Element Modeling of CTS Specimen and Loading Fixture to Study Mode III Fracture

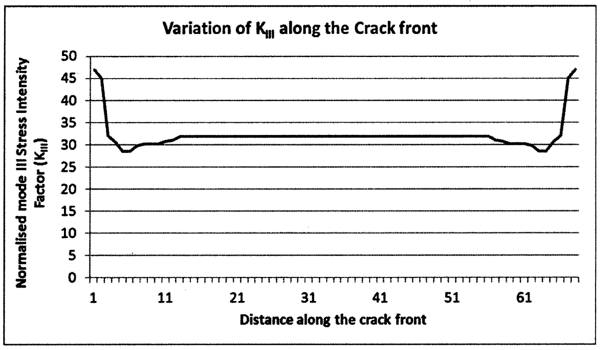


Fig.24 Normalized Mode III Stress Intensity Factor K_{III} and its Variation Along Crack Front

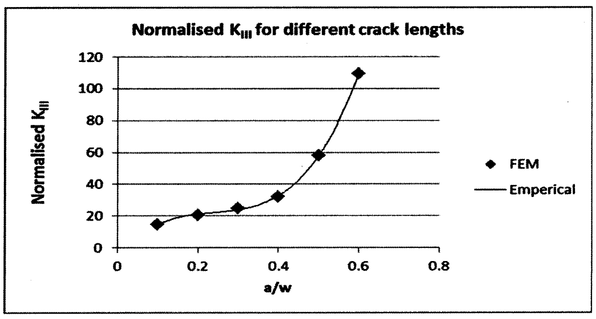


Fig.25 Normalized Mode III Stress Intensity Factor K_{III} for Different a/w Ratios