Int. J. Chem. Sci.: 14(4), 2016, 1886-1892
ISSN 0972-768X
www.sadgurupublications.com

EVALUATION OF STRESS INTENSITY FACTORS OF DIFFERENT MATERIALS: A FINITE ELEMENT APPROACH

V. CHITTIBABU, K. SANTARAO*, P. GOVINDA RAO and M. V. S. BABU

Department of Mechanical Engineering, GMRIT, RAJAM (A.P.) INDIA

ABSTRACT

Fracture toughness is an indication of amount of stress required to propagate a preexisting flaw. It is very important material property since the occurrence of flaws is not completely avoidable in the processing, fabrication, or service of a component. Flaws may appear as cracks, voids, and weld defects, etc. Engineers can never be totally sure that a material is flaw free, it is common practice to assume that a flow of some chosen size will be present in some number of components and use the Linear Elastic Fracture Mechanics (LEFM) approach to design critical components. This approach uses the flaw size and features, component geometry, loading conditions and the material property called fracture toughness to evaluate the ability of a component containing a flaw to resist fracture. This paper presents K_{1C} fracture toughness works well for very high strength materials exhibiting brittle fracture. Linear Elastic Fracture Mechanics has been applied to different materials. The Linear elastic fracture parameters i.e., Stress Intensity Factor has been determined using ANSYS for different materials like maraging steels, 2024-T3 Al alloy, Ti-6Al-4V, Al 7075-T651.

Key words: Linear elastic fracture mechanics, Stress intensity factor, ANSYS, Maraging steels.

INTRODUCTION

Fracture is a problem that society has faced for as long as there have been manmade structures. Fortunately advances in the field of fracture mechanics have helped to offset some of the potential dangers posed by increasing technological complexity. Researches in this field can help reducing the cost due to failure. The development of fracture mechanics started with Griffith's crack theory. After that many people did a lot of work in developing theories regarding fracture and its cause.

Origin of such type of work is several centuries earlier. Experiments were performed by Leonardo Da Vinci that provided some clues as to the root cause of fracture. He

*Author for correspondence; E-mail: ksantarao@gmail.com

measured the strength of iron wires and found that the strength is inversely related with wire length[1]. In this paper[2], a finite element representation of stable crack growth is discussed. Also an extended finite element method is used for the description of growth phenomena at a crack tip in plane stress. Simple macroscopic models of the fracture processes are discussed and the crack tip opening angle is chosen for determination of a fracture criterion in the finite element model. Under the somewhat arbitrary assumption of a constant crack opening angle as a fracture criterion, results are obtained that make clear the possibility of simulating stable crack growth with this method. A finite element analysis of stable crack growth in an aluminium alloy is carried out. Extensive stable cracking was observed in large test pieces of 25 mm thick weldable AlMgZn alloy, which is used in the construction of a portable bridge. Standard fracture specimens produced valid KIC values, with short cracks exhibiting unstable fracture. Finite element analysis of the large specimens determined a valid J-R curve that can increase the effective KC by several times the KIC value. The R-curve can be used to explain the stability of long cracks in full scale tests on a bridge prototype, compared with the instability of short cracks in small, standard test piece[3]. A predictive method for remaining component lifetime evaluation consists in integrating the crack growth law of the material considered in a finite element step-by-step process. The aim of the present work is to test several existing numerical techniques reported in the literature. Both the crack opening displacement extrapolation method and the J-integral approach are applied in 2D and 3D ABAQUS finite element models. The results obtained by these various means on CT specimens and cracked round bars are in good agreement with those found in the literature[4]. An experimental and computational study of HY-100 steel three-point bend specimens was performed. Two specimens were considered for experiments, differing with respect to thickness and the presence of side grooves. Plane stress and plane strain finite element analyses of the specimens were conducted to assess the relative role of constraint on load vs crack opening displacement response and crack growth initiation. A critical value of the strain energy density associated with local material fracture was used to predict the onset of crack growth. The experimental responses were bounded by the predicted plane stress and plane strain load Vs crack mouth opening responses[5]. Crack initiation and stable crack growth under monotonic loading in steels has been studied using an elastic-plastic FE analysis (2D). The fracture criterion used for crack initiation and stable crack growth was the critical strain energy density. In addition the shift core method for the analysis of crack extension was used. In this method, crack advance is simulated by moving the coordinates of the core region which surrounds the crack tip. Simultaneously, the core itself geometrically undergoes a simple rigid-body motion during the crack extension. The analytically calculated and experimentally measured load for crack initiation and the subsequent stable crack growth agreed well[6]. An experimental investigation was carried out to study the crack initiation and growth in a single-edge notched NiTi shape memory alloy sheet under tension.

It is observed that a crack initiated at the tip of a V-shape notch before the peak axial load was reached and it grew steadily across the width of the NiTi sheet until final fracture. In-plane crack-tip deformation fields at various stages of the crack growth were measured based on an image correlation technique and the crack-tip opening displacement (CTOD) and crack-tip opening angle (CTOA) were subsequently determined[7]. A normalized calibration curve using clip-gauge measurements of (COD) at the crack mouth has been developed for CT specimen. This procedure provides a simple, indirect determination of crack length during cyclic loading and is well adapted for automation of fatigue crack-growth-rate (FCGR) tests. The specific CT specimen configuration studied was the same as that employed in a recent inter laboratory FCGR testing program conducted by ASTM Sub-committee E24.04 on Subcritical Crack Growth. Alloys of Al, Ti and steel were used for tests[8]. The fracture behavior in Cr, Mo low alloy structural steels has been studied and two alloy steels, 2.25Cr-1Mo and 0.5 Mo, have been employed to investigate the methods for determination of J-integral and CTOD parameters for critical events using R-curve approach. The study has been conducted over a range of 300-400^{0}C temperatures[9]. Cohesive zone models have been employed to simulate fracture and delamination in solids. The formulation for incorporating cohesive zone models within the framework of a large deformation FE procedure is presented. A special Ritz-finite element technique is employed to control nodal instabilities that may arise when the cohesive elements experience material softening and lose their stress carrying capacity. Quasi-static crack growth along the interface in an adhesively bonded system is simulated employing the cohesive zone model[10]. Improved formulae for estimating crack tip opening displacement (CTOD) from test records of compact tension (CT) specimens are developed. Two-dimensional, plane strain, finite element analyses of 1T C(T) specimens are made for normalized crack depths, a/W, of 0.40-0.70 using Ramberg-Osgood material behavior with strain hardening coefficients n = 5, 10, and 20. Finite element predictions of J, CTOD, crack mouth opening displacement (CMOD) and load are used to obtain proportionality constants relating the area under the load-CMODpl curve to Jpl, and J to CTOD in terms of a/W and n. Improvements in CTOD estimates of 25% over existing estimation methods are obtained. Correction factors for displacements measured on the specimen front face instead of the load line are also examined[11]. A two-region empirical formula is proposed for the transformation of crack tip opening displacement (CTOD) gauge to the CTODstd and the J-integral. Coefficients of the approximation are fitted to data sets of three-dimensional elastic-plastic finite element solutions for plane side and side-grooved three-point bend and compact tension specimens. Solution parameters include also specimen size, crack front curvature and strain hardening of the material. A window for equality between (CTOD) gauge and (CTOD) stud was defined[12].

Based on the past literature, finite element simulation procedure has been used for the evaluation of stress intensity factors of different materials.

Finite element analysis

Materials considered for analysis are Maraging Steels, 2024-T3 Al alloy, Ti-6Al-4V, and Al 7075-T651. PLANE 183, SOLID 185, plane 182 and MASS 21 are used to carry out the analysis. Stress Intensity Factor is evaluated by considering different geometries viz., Center crack, Edge Crack, Double Edge crack and Compact Tension Specimen.

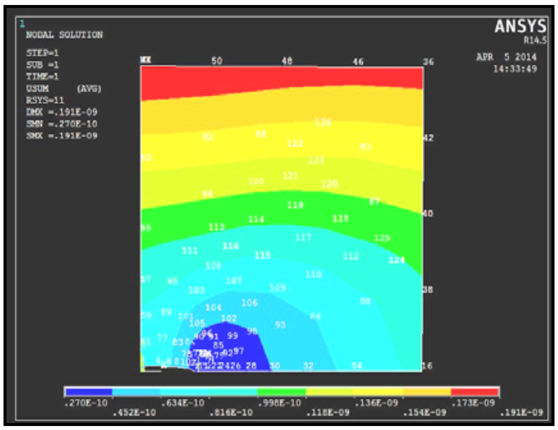

Fig. 1: Nodal solution of centre crack

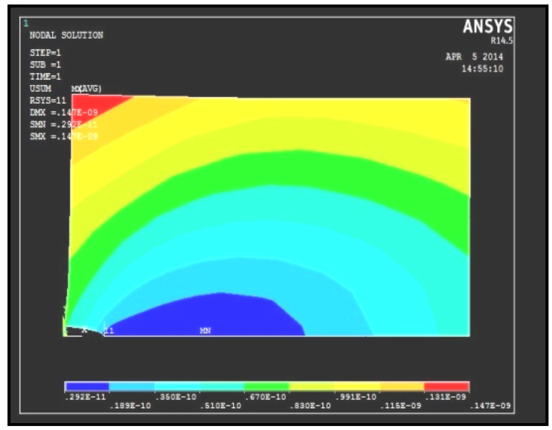

Fig. 2: Nodal solution of edge crack

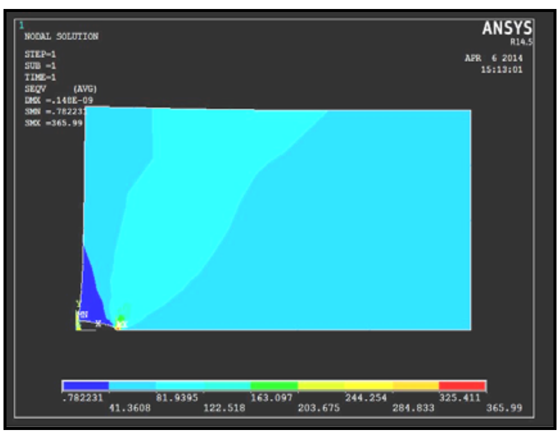

Fig. 3: Nodal solution of double edge crack

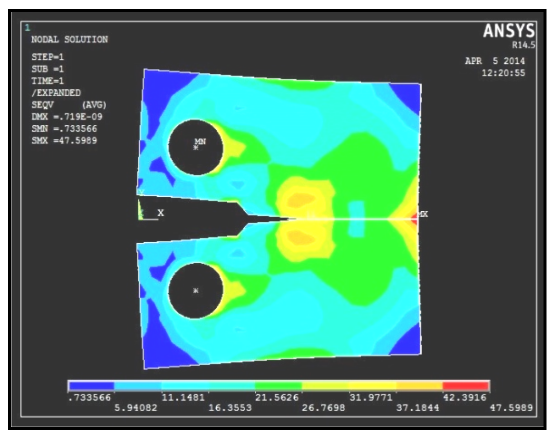

Fig. 4: Nodal solution of compact tension specimen

Fig. 1 to 4 depicts nodal solutions of center crack, edge crack, double edge crack and compact tension specimen.

RESULTS AND DISCUSSION

Table 1 shows comaparison between experimental data taken from the literature and the data obtained from finite element simulation of center crack, edge crack, double edge crack and compact tension specimen using Ansys. It has been cleared from tabulated results that there is a good agreement between the experimental results and finite element simulated results for all specimens.

Table 1: Comparison of results for all specimens

S. No.	Material	Centre crack		Edge crack		Double edge crack		Compact tension specimen	
		A	B	A	B	A	B	A	B
1	Maraging steel	90	95.268	90	93.604	90	93.465	90	84.503
2	2024-T3 Al alloy	26	27.525	26	27.042	26	27.004	26	20.375
3	Ti-6Al-4V	57	60.346	57	59.287	57	59.213	57	45.031

Where A: Theoretical solution and B: ANSYS solution

Material Al7075-T651 is taken, whose experimental results are not available. Similar procedure is adopted to determine the stress intensity factor. In this procedure, load in steps is applied until crack propagation occurs. The critical load corresponding to the crack propagation for this material has been used for determining stress intensity factor K_{1C}. The results for different types of cracks for this material are shown Table 2.

Table 2: Finite element results for the material Al7075-T651

S. No.	Specimen	Critical load (MPa)	Results from Ansys
1	Centre crack	118	31.422
2	Edge crack	100	30.890
3	Double edge crack	104	30.870
4	Compact tension specimen	0.54	23.322

CONCLUSION

The finite element simulation procedure has been applied successfully on materials like Maraging steel, 2020-T3 Al alloy, Ti-6Al-4V and Al7075-T651 by using specimens of centre crack, edge crack and double edge crack and also by using compact tension specimen. The finite element results of stress intensity factors are found to be in good agreement with experimental results reported in the literature. The same procedure can be applied to determine the stress intensity factors of materials, whose experimental results are not available.

REFERENCES

1. T. L. Anderson, Fracture Mechanics, Fundamentals and Applications, Second Edition, Deptt. Mech. Engg., Texas A & M University (1994).
2. H. Andersons, Finite Element Representation of Stable Crack Growth, J. Mech. Phys. Solids, **21**, 337-356 (1973).
3. S. Cheung and A. R. Luxmoore, A Finite Element Analysis of Stable Crack Growth in an Aluminium Alloy, Engg. Fracture Mech., **7(9)**, 1153-1169 (2003).
4. C. Gardin, S. Courtin, G. Be´zine and Ben Hadj Hamouda, Advantages of the J-Integral Approach for Calculating Stress Intensity Factors when using the Commercial Finite Element Software ABAQUS, Engg. Fracture Mech., **72**, 2174-2185 (2005).
5. V. G. Degiorgi and P. Matic, Investigation of Crack Growth Initiation in a Three Point Bend Fracture Specimen, Mechanics of Materials Branch (Code 6380), Naval Research Laboratory, Washington, DC 20375-5000, U.S.A.
6. G. P. Zhu, G. M. C. Lee and A. A. Mufti, An Experimental and Computational Investigation of Crack Initiation and Stable Crack Growth of Ductile Materials, Mining and Metallurgy Department, Technical University of Nova Scotia, Halifax Canada (1993).
7. T. Wei, H. Tao and N. Zhang, Crack Initiation and Growth in a Notched NiTi Shape Memory Alloy Sheet, Mater. Res. Soc. Symposium Proceedings, **785**, 213-218 (2004).
8. A. M. Sullivan, Crack-Length Determination for the Compact Tension Specimen Using a Crack-Opening-Displacement Calibration, Naval Research Lab, Washington D.C. (1975).
9. P. Sundaram, Fracture Studies in Cr, Mo Low Alloy Structural Steels, Ph.D. Thesis, IIT Delhi (1989).
10. S. R. Chowdhary and R. Narsimhan, A Cohesive Finite Element Formulation for Modelling Fracture and Delamination in Solids, IISc, Bangalore, **25**, Part 6, 561-587 (2000).

11. T. L. Panontin, A. Makino and J. F. Williams, Crack Tip Opening Displacement Estimation Formulae for C(T) specimens, Materials and Failure Analysis Group, NASA Ames Research Center, MS 213-4, Mo.ett Field, CA 94035, USA (2000).

12. G. P. Nikishkov, J. Heerens and K. H. Schwalbe, Transformation of $(CTOD)_{g5}$ to $CTOD_{std}$ and J-Integral for 3PB- and CT-specimens, Institute of Materials Research, GKSS Research Center, Max Planck Straûe, D-21502, Geesthacht, Germany (1999).

Revised : 01.07.2016 *Accepted : 04.07.2016*