Research Journal of Engineering Sciences ____ ISSN 2278 – 9472
Vol. **2(7),** 38-42, July **(2013)** *Res. J. Engineering Sci.*

Computational Analysis of Stress Intensity Factor for a Quarter Circular Edge Crack under Mode-I loading

Naresh S, Bharath Naik L., Madhu S.K and Mohan A
Department of Mechanical Engineering, R V College of Engineering, Bangalore, INDIA

Available online at: www.isca.in
Received 27th June 2013, revised 14th July 2013, accepted 25th July 2013

Abstract

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

Keywords: Quarter circular crack, singularity, stress intensity factor.

Introduction

Failure of a material can be broadly classified as fracture, fatigue or creep. Fracture refers to the failure of a structure by means of propagation of a crack, breaking it into more than one different parts. Fracture crack propagation takes place through any of the three modes (Mode I, Mode II or Mode III) or through a combination of the three modes. Mode I represents the crack propagation under normal in plane loading where the crack is positioned perpendicular to the applied load. Mode II represents crack propagation due to shear type failure where the load applied is transverse to crack length. Mode III represents crack propagation due to tear type failure where the load applied is parallel to the crack length[1]. The fracture toughness of a specimen can be measured through the following parameters- Stress Intensity Factor (K), Energy Release Rate (G), Path independent integral (J) and Crack Tip Opening Displacement (CTOD)[2].

Surface cracks are normally seen in welded joints like as in case of ships and other marine structures. Quarter circular cracks may result due to improper welding which might cause the failure of the structure. Hence we are analysing such type of crack using Computational fracture mechanics.

LEFM (Linear Elastic Fractuce Mechanics) is considered for a brittle material where only elastic analysis is carried out to determine stress and displacement fields near the crack tip with characterizing parameters like SIF. By conforming our attention only to brittle materials, we would be able to obtain closed form solution to many problems and we would be able to learn how to play with the singularity (infinite stresses at the crack tip). There is another advantage of finding solutions to these problems. If plastic zone size is small in compared to crack length, the influence of plastic zone in the elastic analysis may be neglected[3].

Von-Mises stress distribution and plastic zone near the crack tip for plate under mode-I loading is shown in figure-1.

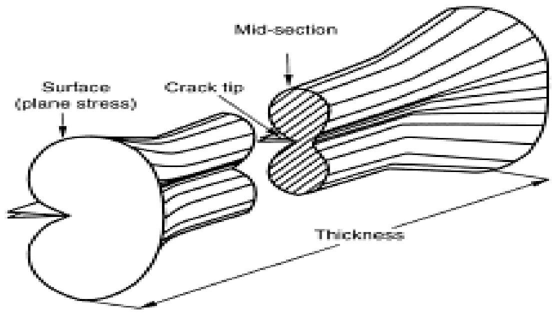

Figure-1
Variation of plastic zone near the crack tip[4]

Fracture Toughness for a quarter circular edge crack in a square bar as shown in figure-2 is considered in our research paper. It is subjected to mode-I loading as shown in figure-3. The dimensions and material properties of the specimen are given in table-1. The critical crack length when the stress intensity factor is equal to the fracture toughness is to be determined.

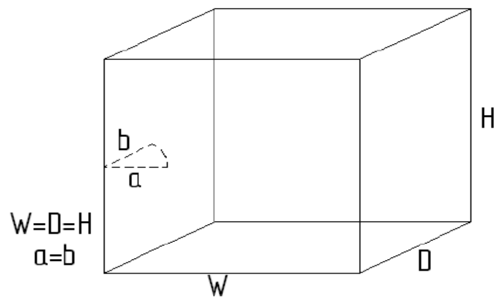

Figure-2
Square bar specimen with a quarter circular crack

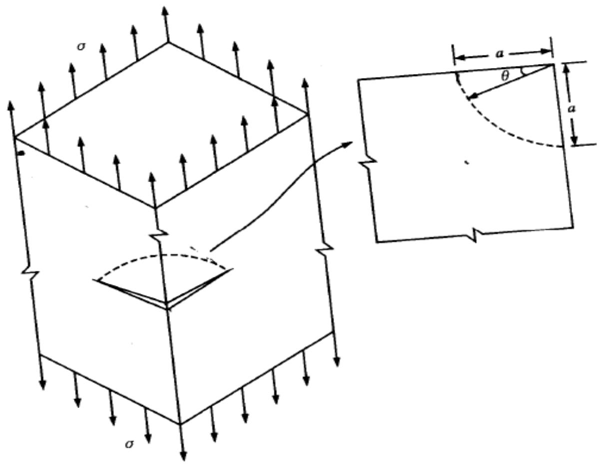

Figure-3
Specimen subjected to mode-I loading

Table-1
Material properties and geometric properties considered for the specimen

Material Properties	**Geometric Properties**
Cast Iron Young's Modulus(E)=2.11x10^5 Mpa Poisson's Ratio(ν=0.28)	W=100mm D=100mm H=100mm a=b=10mm to 70mm θ=0^0 to 90^0

Benchmark Analysis: Failure analysis are mainly based on experimental analysis recently Finite element analysis has become a novel and supplementary approach to failure analysis[5]. This is applied in many of the mechanical engineering problems and results obtained are found to be agreeable with the experimental results. For eg., Numerical simulations of tubular shell subjected to combined loading were conducted. The analytical solutions showed excellent agreement with the numerical results predicted by FEM[6]. Design and stress-deflection analysis of a multi leaf spring is carried out by finite element approach using CAE tools (i.e CATIA, ANSYS) where the results obtained using FEM were in agreement with experimental results[7].

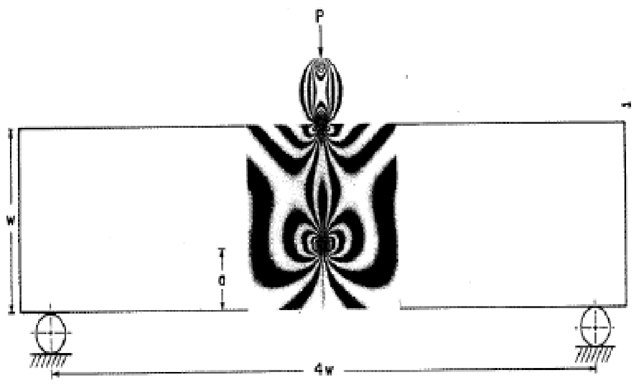

Figure-4
Three-point bend specimen with photo elastic pattern insert

The analysis is done using ANSYS as a platform. The analysis of Single Edge Notch Bar (SENB) specimen subjected to a three point bending load is conducted on ANSYS and the results of the analysis are verified with the results obtained from the photo elastic analysis of the three-point bent specimen[8]. The specimen model is created in ANSYS and is meshed.

A progressively refined Finite Element mesh of Singular Iso-parametric Pentahedral solid element (SPENTA15) with user specified number (NS) and size (a) from one crack face to another and a number of such segments (NSEG) along a surface crack front is created using a pre-processing options in ANSYS. The rest of the domain under consideration is discretized using a compatible mesh of regular elements namely Iso-parametric Pentahedral solid (PENTA15) and Iso-parametric Hexahedral solid (HEXA20). The meshed model is shown in figure-5.

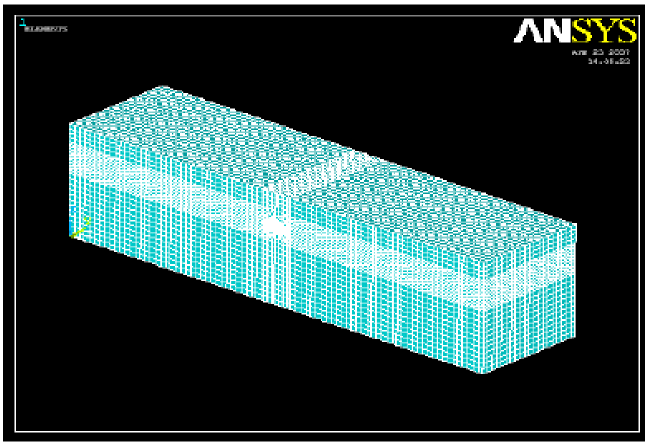

Figure-5
FE Model of SENB

The result of the analysis is plotted in figure-6. f(a/W) in the graph refers to the normalized value of SIF.

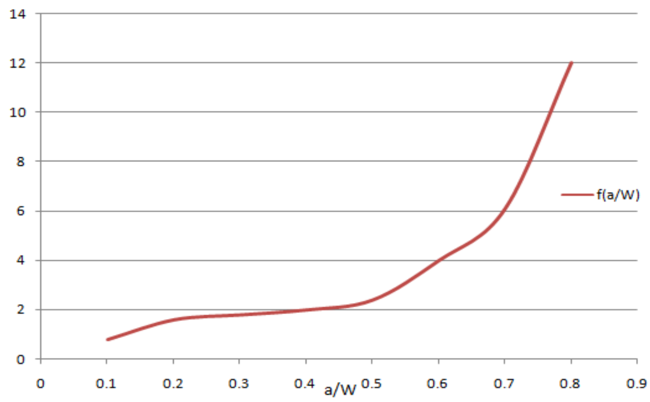

Figure-6
Variation of f(a/W) with a/W

Geometry of the three-point bend specimen with photo elastic pattern insert for one crack length is shown in the figure-4.
The result of the photo elastic experiment is plotted in figure-7.

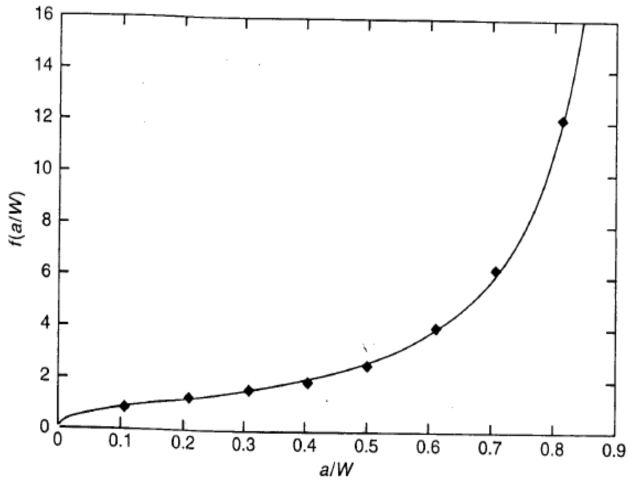

Figure-7
Results from the photo elastic analysis of the three-point bend geometry for the full range of crack lengths

The results of the photo elastic experiment and also of the analysis conducted on the ANSYS software are a close match. This establishes the credibility of the software, and thus it can be employed for testing a square bar specimen with quarter circular crack under Mode I fracture.

Finite Element Modeling: The methodology followed involved defining the FE model, followed by formulation of the global matrix and solving using the iterative method, and results in extraction of the stress intensity factor and Von Mises stress[9]. The dimensions of the specimen are given in table-1.

After the creation of the singular points the whole model is meshed. First a plane along a singular element is meshed using 8 noded element as shown in figure-8. A 2D mesh is created. Then the whole body is meshed using the sweep option to create a 3D mesh using 20 noded elements. The generated mesh is shown in figure-9.

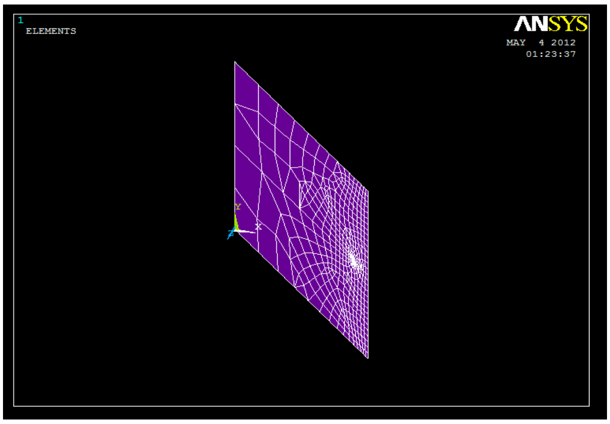

Figure-8
Diagonal plane meshed with 8 noded element

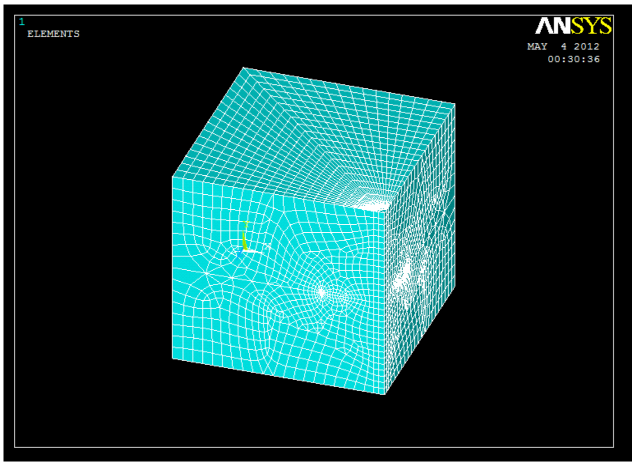

Figure-9
Square bar specimen mesh using 20 noded element and having singular elment around crack front

For a Mode I type fracture, only opening of the crack surface is to be considered. The loading effects in other directions should be neglected. For achieving this, the constraints are applied on the Finite Element model before application of the load as shown in figure-10.

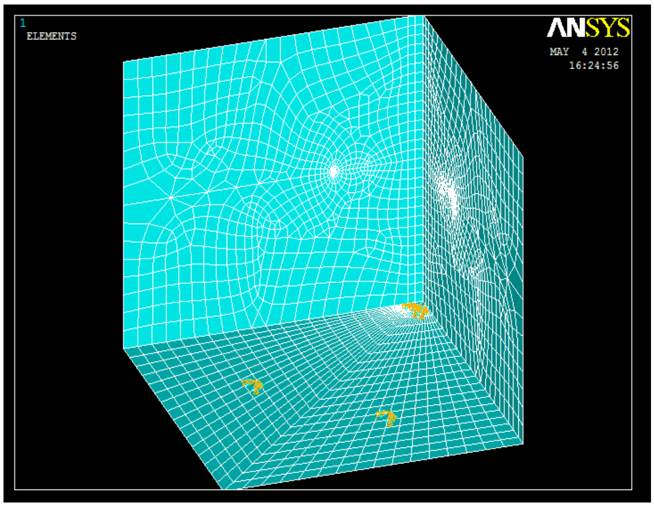

Figure-10
Contraints applied on the bottom face

After applying the contraints on one of the face parallel to the crack surface, pressure is applied on the opposite face as shown in figure-11. The value of the pressure applied is 100 Mpa.

The Von-Mises stress distribution at the crack tip is shown in figure-12. It can be seen that the shape of the plastic zone around the crack tip is bean shaped which confirms with the theoretical study. Hence the obtained result is reliable.

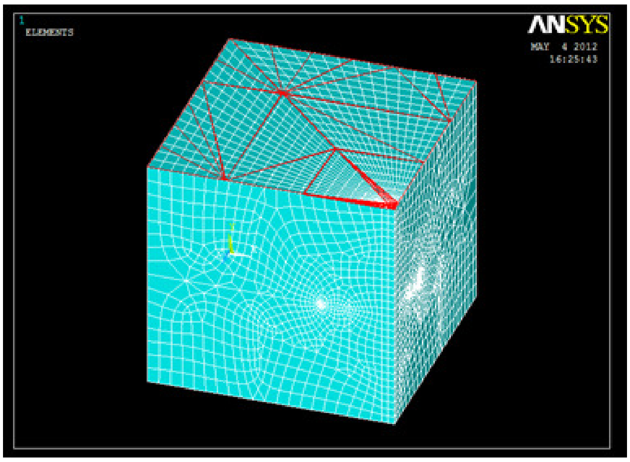

Figure-11
Pressure applied on top face

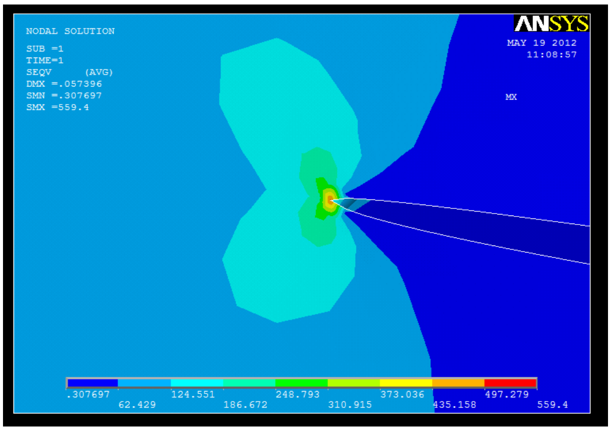

Figure-12
Von-Mises stress distribution near the crack tip

Table-2
SIF values for different a/W ratio along the crack front

a/W ratios	0.1	0.3	0.5	0.7
θ (in degrees)	Stress Intensity Factor (K_I)			
0	371.71	745.78	1267.3	3381.25
15	338.54	649.47	1072.84	2034.6
30	334.62	634.91	973.83	1623.82
45	331.81	583.46	874.81	1494.8
60	335.18	636.85	978.84	1632.72
75	338.54	648.5	1079.1	2047.94
90	371.71	745.58	1268.35	3381.55

Results: SIF was evaluated for specimens of different a/W ratios ranging from 0.1 to 0.7 in steps of 0.2. SIF was obtained for each and every model separately and are tabulated as in table-2.

The variation of the stress intensity factor (K-I) versus plane angle is plotted for different a/W ratios shown in figure-13.

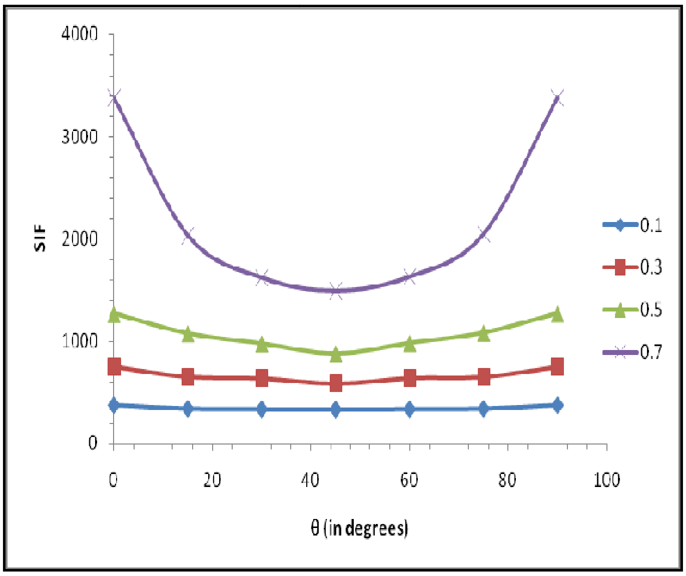

Figure-13
Variation of SIF along the crack front for different a/W ratios

Variation of SIF for the edge plane i.e., for θ=0° or 90° for different a/W ratios is shown in figure-14.

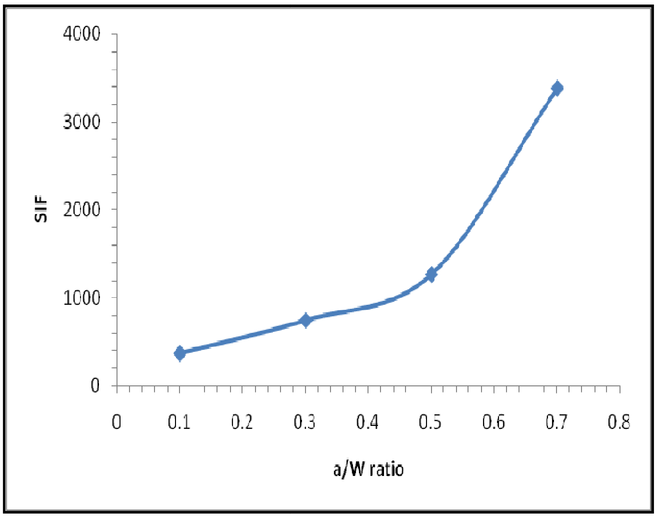

Figure-14
Variation of SIF against different ratios of a/W

Conclusion

In our analysis of the brittle material (Cast Iron), as the crack length increases the value of the SIF also increases. There was a sudden shoot up in the value of SIF when the crack length was between 60 to 70mm. Hence the critical crack length lies somewhere between 60 and 70 mm. Once this length is reached the catastrophic failure occurs. After obtaining the stress information from FEA based methods fatigue life can be calculated[10].

The stress intensity factor (SIF) is found to have the maximum value at the edges along the crack front, which are inferred from the gragh given in figure-13.

Von Mises stress distribution at the crack tip is given in figure-12. The shape of the plastic zone is found to be in accordance with figure-1. Since Cast Iron is a brittle material, the plastic zone is small compared to crack length. Hence, LEFM is applied where effect of plastic zone is neglected.

References

1. Chaitanya G., Srinivas K.and Kumar J. Suresh, Effect of Fiber Orientation on Mode I Crack Opening Stress Intensity of an Orthotropic Laminate, *Research Journal of Engineering Sciences*, **2(5)**, 30-34 (**2013**)
2. Prashanth Kumar, Elements of Fracture Mechanics, *Tata McGraw Hill Publications*, **(2009)**
3. Alan Liu, Mechanics and Mechanisms of Fracture-An Introduction, *ASM International Publications*, **(2005)**
4. Sanford R.J., Fundamentals of Fracture Mechanics, *Prentice-Hall Publications*, **(2003)**
5. Purkar T. Sanjay and Pathak Sunil, Aspect of Finite Element Analysis Methods for Prediction of Fatigue Crack Growth Rate, *Research Journal of Recent Sciences*, **1(2)**, 85-91 **(2012)**
6. Shariati Mahmoud, Fereidoon Abdolhosein and Akbarpour Amin, Investigation on Buckling Behavior of Tubular Shells with Circular Cutout, Subjected to Combined Loading, *Research Journal of Recent Sciences*, **1(7)**, 68-76 **(2012)**
7. Kumar Krishan and Aggarwal M.L., A Finite Element Approach for Analysis of a Multi Leaf Spring using CAE Tools, *Research Journal of Recent Sciences*, **1(2)**, 92-96 **(2012)**
8. Kirthan L.J., Refined Finite Element Models to determine Mixed Mode Stress Intensity Factors for Surface Crack Problems, *Proceedings on International Conference on Advances in Mechanical and Building Sciences in the 3rd millennium*, **(2009)**
9. Kirthan L.J., Calibration of test methods for Mode I,II and III fracture toughness, *Journal of Aerospace Science and Technology*, **64**, 33-42 **(2012)**
10. Purkar T. Sanjay and Pathak Sunil, Analysis of Crack Initiation in Fretting Fatigue Specimen, *Research Journal of Engineering Sciences*, **1(1)**, 26-34 **(2012)**