International Journal of Engineering and Advanced Technology (IJEAT)
ISSN: 2249-8958 (Online), Volume-6 Issue-5, June 2017

Computation of Linear Elastic and Elastic Plastic Fracture Mechanics Parameters Using FEA

Renjin J Bright, Lokesh Kumar P. J

Abstract— Linear Elastic Fracture Mechanics (LEFM) parameter Stress intensity factor and Elastic Plastic Fracture Mechanics (EPFM) parameter J-Integral are the most imperative fracture parameters used to determine the structural integrity of components with flaws. This work deals with the evaluation of Stress intensity factor and J-integral for Center Cracked Tensile (CCT) specimens using the versatile Finite Element Analysis (FEA) package ANSYS. Alternate methods used for evaluating stress intensity factor such as stress extrapolation method and displacement extrapolation method have also been demonstrated. Different aspect ratios have been selected for the evaluation. The FEA results have been compared with that obtained by empirical means. An attempt has also been made to compute the critical crack length of a CCT specimen with a center through crack. Evaluation of critical crack lengths have been made under different loading cases, purely with the aid of FEA package ANSYS. J-integral evaluation depends on material non-linearity. The material curve for non-linear analysis have been generated using inverse Ramberg-Osgood relationship. ASTM A 36 steel used for common structural and plate applications is designated as the material for analysis. The ability of FEA to determine the fracture parameters has been successfully studied by this work.

Keywords— Stress intensity factor, J-Integral, Center Cracked Tensile Specimen, Stress Extrapolation, Displacement Extrapolation, Finite Element Analysis, Critical crack length.

I. INTRODUCTION

Fracture mechanics is the field of mechanics concerned with the study of the propagation of cracks in materials. It exploits the methods of analytical solid mechanics to calculate the driving force on a crack and those of experimental solid mechanics to characterize the material's resistance to fracture. In modern materials science, fracture mechanics is an important tool in improving the mechanical performance of materials and components. It applies the physics of stress and strain, in particular the theories of elasticity and plasticity, to the microscopic crystallographic defects, found in real life materials in order to predict the macroscopic mechanical failure of bodies. Fracture mechanics is based on the implicit assumption that there is an inbuilt crack in every structural component. Fracture mechanics are basically divided into two types namely, ductile and brittle fracture.

Manuscript published on 30 June 2017.
***** Correspondence Author (s)

Renjin J Bright, Assistant Professor, Mechanical Engineering Department, Jeppiaar SRR Engineering College, Chennai-603103, Tamil Nadu, India. PH-+91 9488752071, E-mail: renjin1990@gmail.com.

Lokesh Kumar P. J, Assistant Professor, Mechanical Engineering Department, RMK Engineering College, Chennai-601206, Tamil Nadu, India. E-mail: lokeshpj@gmail.com.

The three modes of fracture are, Mode I-the opening mode, mode II- the sliding mode and mode III - the tearing mode [1]. The basic energy balance approach of Griffith and the modified stress intensity approach of LEFM have been detailed in [2]. [3] presented a systematic technical review about various fracture toughness parameters and the associated testing methods. A novel approach of determining stress intensity factor widely known as displacement extrapolation approach has been presented in [4]. [5] presented a method named as stress extrapolation method which aids the determination of stress intensity factors. [6] conducted a detailed study on the evaluation of stress intensity factor of CCT specimen and presented a modified mathematical relation coined as the three parameter fracture criterion for correlating the fracture parameters of CCT specimen with pressure vessels. The stress intensity factor and extended stress intensity factor concept known as notch stress intensity factor have been studied for all the three modes of fracture in [7]. Mode I stress intensity factor of a square bar with a quarter circle edge crack has been explained in [8]. A sudden increase in the stress intensity factor value has been observed. [9] utilized the ability of FEA software ABAQUS in the evaluation of stress intensity factor and J-integral. [10] presented a method to evaluate the J-integral of a circumferential through wall cracked pipes for leak before break analysis of pressurized piping. [11] assessed the ability of fracture parameters stress intensity factor and J-integral for the determination of LBB. Experimental investigation of J-integral has been successfully presented in [12]. Crack opening stress equation for CCT and edge cracked specimen has been developed in [13]. The ability of J-integral to predict fatigue crack growth of modes I, II, III and mixed mode fracture has been predicted in [14].

This work deals with the ability of FEA in predicting the LEFM and EPFM parameters. This computational analysis has been done with the aid of a CCT specimen made of ASTM A 36 steel. Empirical solutions have been utilized to validate the FEA results.

II. MATERIAL USED

The material selected for the analysis is ASTM A36 steel, used for common structural and plate applications [15]. ASTM A36 plate is a low carbon steel that exhibits good strength coupled with formability. It is easy to machine and fabricate and can be securely welded. A36 structural steel plate can be galvanized to provide increased corrosion resistance. The major properties of the material are listed in table 1. 'E, μ, σ_{ys}, σ_{us} and K_{IC}' represents young's modulus, poisson's ratio, yield strength, ultimate strength and fracture toughness respectively.

Table-1 Material Properties of selected material [15]

E (GPa)	μ	σ_{ys} (MPa)	σ_u (MPa)	K_{IC} (MPa√m)
210	0.3	250	500	98.89

III. FINITE ELEMENT MODEL

Figure 1 depicts a CCT specimen subjected to a tensile load, 'σ' of 100 MPa. The CCT specimen has been modeled primarily as a plate by considering the dimensions 400 mm X 400 mm X 20 mm (2W X 2W X t) and a crack length, 2a of 50 mm. The meshed model of the CCT specimen is shown in figure 2.

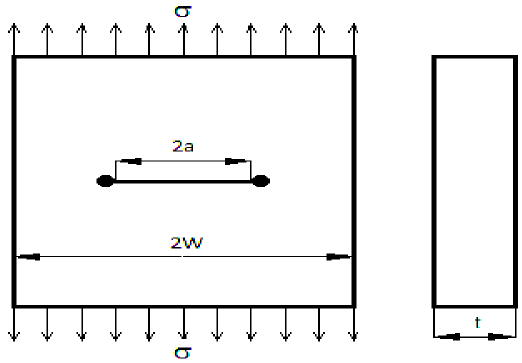

Figure 1. Center Cracked Tensile Specimen

To reduce the complexity of 3-D modeling, plane stress condition has been assumed. The symmetry of the CCT specimen further helps in utilizing quarter symmetric modeling technique available in ANSYS. Meshing of the model has been done using the 8 noded element, plane 183. Concentrate key point method [16] has been utilized for the meshing and computation of fracture parameters. The concentrated key point is generated at the crack tip. Computed value of stress intensity factors is depicted in figure 3. FEA values of stress intensity factor of the CCT specimen was found to be 43.146 MPa√m.

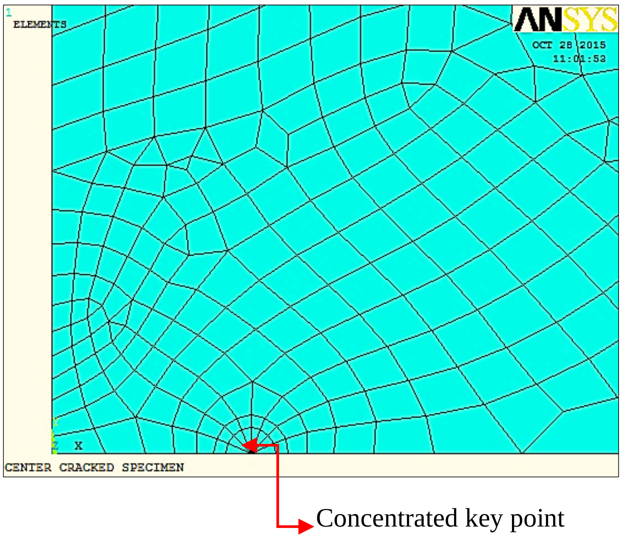

Figure 2. Meshed model of CCT specimen

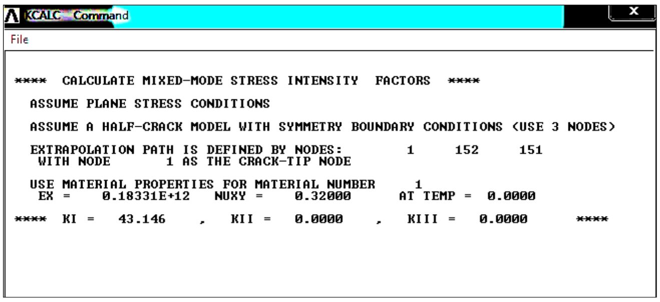

Figure 3. Stress Intensity Factor of CCT specimen Obtained from ANSYS

IV. COMPUTATION OF STRESS INTENSITY FACTOR

A. *Empirical Method*

The empirical relations for the evaluation of stress intensity factor for a CCT specimen [1] is explained in equations (1-3). 'K_I, σ, a, f(α), α' represents stress intensity factor, applied stress, crack length, correction factor and aspect ratio respectively.

$$K_I = \sigma\sqrt{\pi a}f(\alpha) \quad (1)$$

$$\text{where,} f(\alpha) = 1.12 - 0.128\alpha - 0.288\alpha^2 + 1.523\alpha^3 \quad (2)$$

$$\alpha = \frac{a}{W} = 0.25 \quad (3)$$

For an applied tensile stress of 100 MPa on the CCT specimen of dimensions specified in section III, stress intensity factor has been evaluated for the material considered. The empirical value of stress intensity for has been found to be 42.89 MPa√m. The effect of geometry could on the stress intensity factor could be evaluated by varying the crack lengths.

B. *Displacement Extrapolation method*

The displacement extrapolation method is based on the nodal displacements along the crack tip [4]. In this method, the displacements were determined for certain nodes along the crack path. Stress intensity factor is computed for the displaced nodes using the empirical relation explained in [1], as shown in equation (4). A plot is made with 'r' along x-axis and 'u_y' along y-axis as depicted in figure 4. The data is used to create a trend line and the y-intercept gives the value for stress intensity factor. The stress intensity factor value has been found to be 43.708 MPa√m.

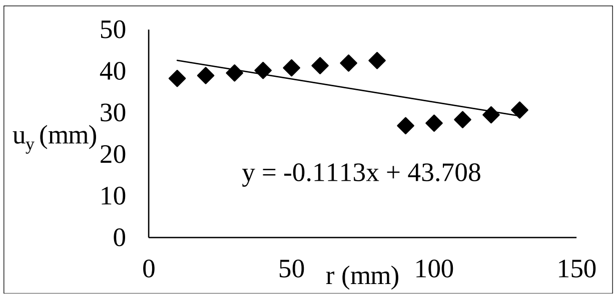

Figure 4. Stress Intensity factor from Displacement Extrapolation method

Published By:
Blue Eyes Intelligence Engineering and Sciences Publication (BEIESP)

International Journal of Engineering and Advanced Technology (IJEAT)
ISSN: 2249-8958 (Online), Volume-6 Issue-5, June 2017

$$K_I = \lim_{r \to 0} \frac{Eu_y}{4} \sqrt{\frac{2\pi}{r}} \quad (4)$$

where, u_y = y-component displacement and r = distance of node from the crack tip. The displacement distribution of the quarter symmetric model of CCT specimen is depicted in figure 5.

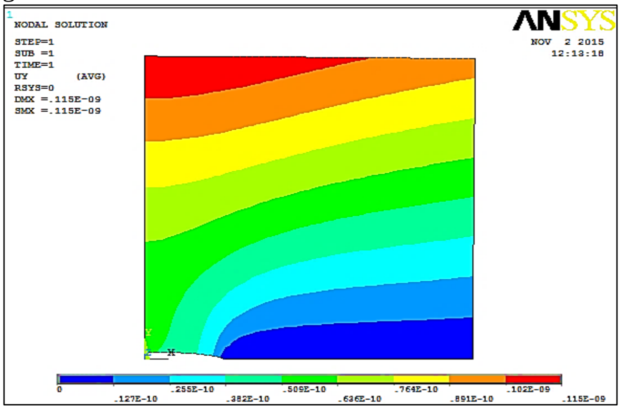

Figure 5. Displacement Distribution

C. Stress Extrapolation Method

The stress extrapolation method utilizes the values of stress distribution near the crack tip. These value are extrapolated to the crack tip. In the crack tip, the relevant trend curve is placed [5], as depicted in figure 6. The y-intercept gives the value of stress intensity factor. In stress extrapolation method the stresses were determined for certain nodes in the crack path. Stress intensity factor is computed using the empirical relation explained in [1], as shown in equation (5).

$$\lim_{r \to 0} K_I = \sigma_y \sqrt{2\pi r} \quad (5)$$

where, r= distance from crack-tip, σ_y = y-component stress

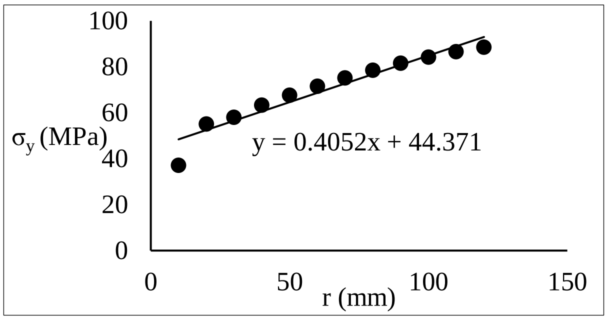

Figure 6. Stress Intensity Factor from Stress Extrapolation Method

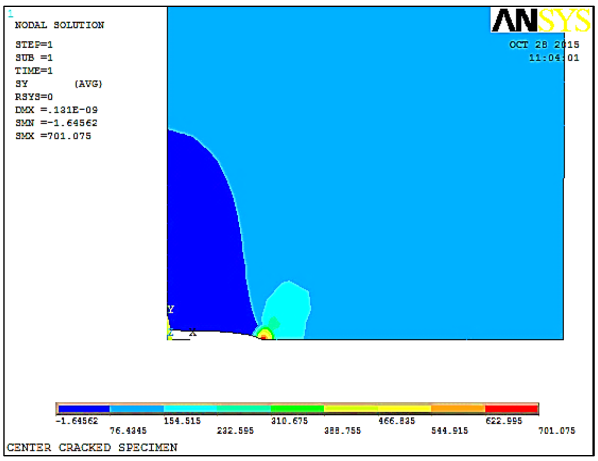

Figure 7. Stress Distribution

The value of stress intensity factor using stress extrapolation method has been found to be 44.371 MPa√m. The stress distribution along the crack path for a quarter symmetric CCT plate is shown in figure 7. Figure 8 depicts the expanded view of CCT specimen modeled in ANSYS. Table 2 depicts the comparison between the stress intensity values obtained by different computational methods.

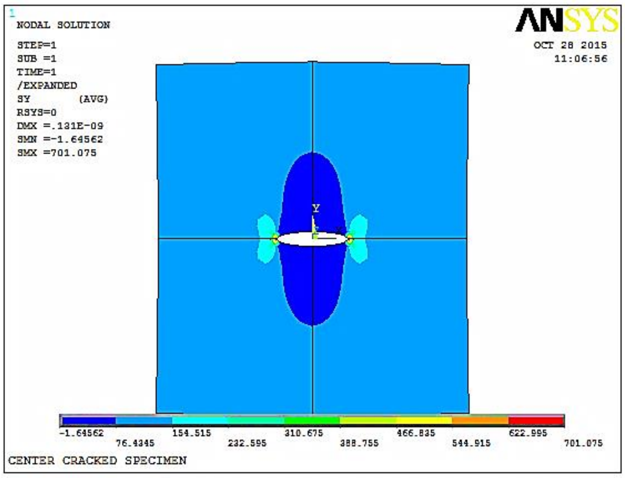

Figure 8. Expanded View of CCT specimen

Table 2 Comparison of Stress Intensity Factor Values

K_I (MPa√m)			
Concentrate Keypoint (FEA)	Stress Extrapolation	Displacement Extrapolation	Empirical
43.145	44.371	43.708	42.89

From the comparison it could be observed that FEA provides the closest solution with the empirical value at an error of 1.87%. Thus it could be concluded that FEA package ANSYS could be successfully employed for the evaluation of fracture parameters.

D. Stress Intensity Factor Values for varying 'a/b' Ratio

The effect of geometry and applied load on stress intensity factor could be analyzed by varying the crack length and applied loads. The variation in crack length is mentioned in terms of aspect ratio (a/b ratio).

The comparison of stress intensity factor values for varying aspect ratio and applied load is explained in table 3 and graphically depicted in figure 9. Computation of stress intensity factor has been purely determined by ANSYS.

Table 3Comparison of Stress Intensity Factor for varying aspect ratio and applied load

K_I (MPa√m) ASTM A36 Steel			
a/b	100 MPa	200 MPa	400 MPa
0.125	25.763	51.527	103.05
0.25	43.146	77.37	154.74
0.375	51.865	103.73	207.46
0.5	66.838	133.68	267.93

Published By:
Blue Eyes Intelligence Engineering
and Sciences Publication (BEIESP)

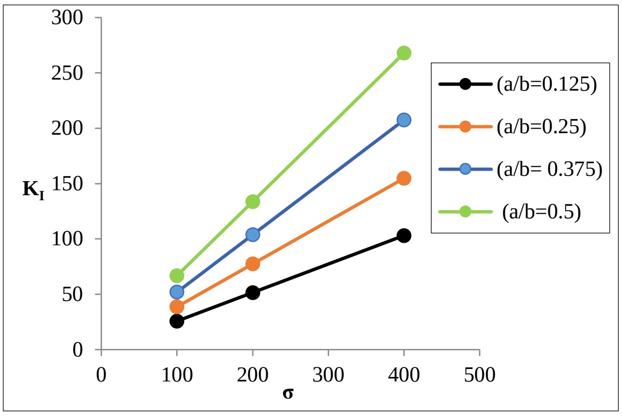

Figure 9. Comparison of stress intensity values with varying aspect ratio and applied load

From the comparison it could be inferred that stress intensity values upsurges with increase in crack length. This paves the path for critical crack length evaluation for various loads.

E. Evaluation of Critical Crack Length

Fracture toughness is the critical property of material which is used to determine the critical crack length (a_c) [2]. Critical crack length characterizes the value of load at which the crack growth will get arrested and would be held firm by the plastic enclaves as shown in figure 1. The stress intensity factor of CCT specimen with previously specified parameters have been found out for various crack lengths at different loading cases of 100 MPa, 200 MPa and 400 MPa. Stress intensity factors have been computed using FEA package ANSYS and is depicted in table 4. The fracture toughness value, 'K_{IC}' is 98.89 MPa√m [16].

Table 4 Comparison of Stress Intensity Factor values under the load of 100MPa, 200MPa, 400MPa, of varying crack lengths (K_{IC}=98.89 MPa√m, W=200 mm)

σ =100 MPa		σ =200 MPa		σ = 400 MPa	
a (mm)	K_I (MPa√m)	a (mm)	K_I (MPa√m)	a (mm)	K_I (MPa√m)
10	17.9	10	35.67	**10**	**71.34**
20	25.5	20	50.957	**20**	**101.91**
30	31.8	30	63.47	30	126.94
40	37.5	40	74.95	40	149.97
50	43.14	50	86.925	50	172.59
60	48.8	**60**	**97.76**	60	195.5
70	53.3	**70**	**109.6**	70	219.35
80	61.1	80	122.01	80	244.02
90	67.7	90	135.34	90	270.2
100	74.8	100	149.62	100	299.24
110	82.52	110	165.06	110	330
120	**90.8**	120	181.9	120	363.9
130	**100.4**	130	200.87	130	401.74
140	111.3	140	222.6	140	445.26
150	124.43	150	248.85	150	497.71

From table 4 it could be observed that for a load of 100 MPa the material attains fracture toughness value at a crack length value in between 120 mm and 130 mm. Similarly fracture toughness is attained at a crack length value in between 60 mm and 70 mm and between 10 mm and 20 mm for the tensile loads of 200 MPa and 400 MPa respectively. In order to find the critical crack length further iterations have been done on the crack lengths for obtaining the fracture toughness value, as explained in table 5.

Table 5 Iterative Value of SIF for selected crack length under the load of 100 MPa, 200 MPa, 400 MPa, with cracks of varying lengths (K_{IC}=98.89MPa√m, W=200mm)

σ =100 MPa		σ =200 MPa		σ =400 MPa	
a (mm)	K (MPa√m)	a (mm)	K (MPa√m)	a (mm)	K (MPa√m)
120	90.8	60	97.76	10	71.34
121	91.98	**61**	**98.93**	11	74.86
122	92.79	62	100.1	12	78.247
123	93.89	63	101.27	13	81.523
124	94.82	64	102.46	14	84.68
125	95.76	65	103.65	15	87.73
126	96.71	66	104.84	16	90.71
127	97.67	67	106.04	17	93.608
128	**98.65**	68	107.24	**18**	**96.437**
129	99.64	69	108.45	19	99.197
130	100.40	70	109.6	20	101.91

From table 5 critical crack lengths of each loading cases could be obtained as depicted in table 6.

Table 6 Critical Crack Length Values

Load (σ) in MPa	Critical Crack Length (a_c) in mm
100	128
200	61
400	18

Figure 10 explains the variation of stress intensity factor values with respect to varying crack length and applied tensile loads. The critical crack length under varying loading cases are also depicted in the same figure. The ability of predicting critical crack lengths using FEA package ANSYS has been effectively demonstrated. The reduction in critical crack length value with increasing load has been noticed from the analysis. This analysis could be further proceeded towards the evaluation of safe load computation of components in service.

Published By:
Blue Eyes Intelligence Engineering and Sciences Publication (BEIESP)

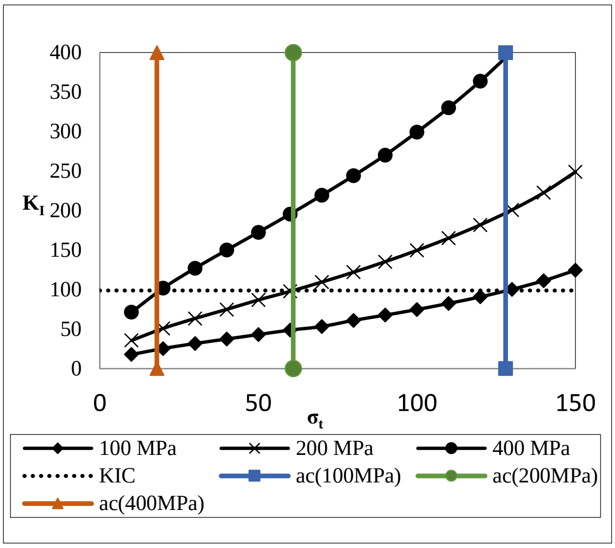

Figure 10. Variation of SIF with Crack Length and Critical Crack Length

V. Computation Of J-Integral

J-Integral is a parameter to characterize crack not only for linear elastic materials but also for non-linear elastic-plastic materials. Elastic-plastic behavior makes the analysis of fracture mechanics complex because of the existence of two zones, elastic-plastic zone near the crack tip and elastic zone surrounding it [1]. J-Integral represents a way to calculate strain energy release rate, or work per unit fracture surface area in a material. J-Integral is equal to strain energy release rate when subjected to monotonic loading. This is true under quasi-static conditions, both for linear elastic materials and for materials that experience small scale yielding at the crack tip [2]. The path may be chosen arbitrarily within the material, but it must be smooth and continuous. Figure 11 depicts the path along the crack tip with outward normal and traction, ‘Γ’ [2].

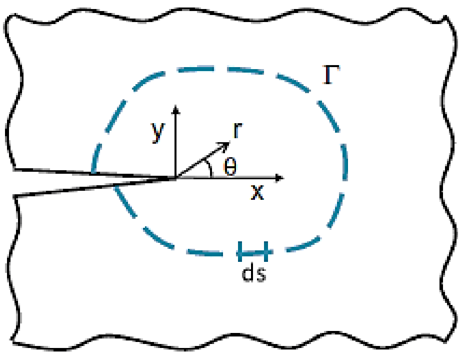

Figure 11. Path around the crack tip with outside normal and traction [2].

The two-dimensional J-Integral [1] has been originally defined as shown in equation (6) and its nomenclature is explained from equations (7-9).

$$J = \int (W dy) - \int \left(t_x \frac{\partial u_x}{\partial u_y} + t_y \frac{\partial u_y}{\partial u_x} \right) ds \qquad (6)$$

where, $W = \int \sigma_{xy} d\varepsilon_{xy}$ (7)

W = Strain Energy Density

u = Displacement vector at a point along the path.

t_x = traction vector along x axis

$t_x = \sigma_x n_x + \sigma_{xy} n_y$ (8)

t_y = traction vector along y axis

$t_y = \sigma_y n_y + \sigma_{xy} n_x$ (9)

σ = component stress

n = unit outward normal vector to path г

s = distance along the path г

By knowing the value of component stresses, unit outward normal vector along the crack-tip and distance along the crack-tip traction vector can be calculated and finally the J-Integral can be evaluated.

Two important features of J-Integral are [1]:

i. J-Integral is path independent; that is, Γ can be chosen arbitrarily within the body of the component.

ii. For Linear–Elastic bodies, J-Integral represents energy release rate ‘G’.

The total J-Integral could be evaluated by adding elastic J-integral and plastic J-integral

$$J = J_{elastic} + J_{plastic} \qquad (10)$$

A. Elastic J-Integral

For Linear–Elastic bodies, J-Integral represents strain energy release rate ‘G’ [1]. Equation (11) explains the empirical relation for evaluating G.

$$J_{elastic} = \frac{\beta^2 K_I^{\,2}}{E} = \frac{\beta^2 \pi \sigma^2 a}{E} = G \qquad (11)$$

where, ‘β’ accounts for the geometry of the crack and the component (for general cases β=1),

K_I= Stress Intensity Factor

σ= Stress

a=Crack Length

E= Young’s Modulus.

The elastic J-integral value has been computed as 0.00918 J/mm^2 by empirical means while that determined by FEA is 0.010067 J/mm^2. The nodal path around crack tip for the determination of J-integral is depicted in figure 12. The J-integral evaluation by FEA is an extension of concentrate key point method as explained in [16].

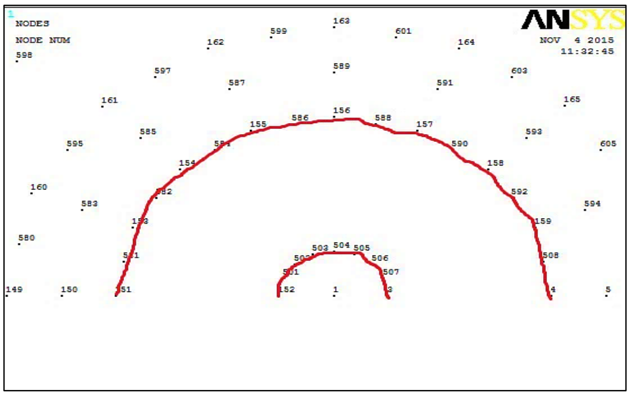

Figure 12. Nodal contour around the crack tip for J-Integral evaluation

B. Plastic J-Integral

The non-linear behavior of material is conveniently formulated by Ramberg-Osgood which is found to be reasonably well in agreement with the material behavior [1]. For one dimensional case the relation is shown in equation (12),

Published By:
Blue Eyes Intelligence Engineering
and Sciences Publication (BEIESP)

$$\varepsilon = \frac{\sigma}{E} + \frac{\sigma^n}{F} \tag{12}$$

where,
E= Young's Modulus,
ε = Strain
n and *F* are material constants which are determined from the stress-strain relationship.

Here, 'n' is known as hardening exponent and its value is unity for linear-elastic materials and is greater than one for elastic - plastic materials. Material constant 'F' is larger than 'E' and therefore second term of Ramberg-Osgood becomes negligible for small values of 'σ'.

Also,

$$\varepsilon = \varepsilon_p + \varepsilon_e \tag{13}$$

Where,

$$\varepsilon_e = \frac{\sigma}{E} \tag{14}$$

$$\varepsilon_p = \frac{\sigma^n}{F} \tag{15}$$

For engineering applications Ramberg-Osgood Equation can be written in a slightly different way as explained in equation (16).

$$\frac{\varepsilon}{\varepsilon_0} = \frac{\sigma}{\sigma_0} + \alpha\left(\frac{\sigma}{\sigma_0}\right)^n \tag{16}$$

This relation has four material constants, 'n', 'α', 'σ_0', 'ε_0'. Here 'σ_0' can be chosen from the stress-strain curve and corresponding 'ε_0' is evaluated by the relation explained in equation (17),

$$\sigma_0 = \varepsilon_0 E \tag{17}$$

In many cases 'σ_0' is assumed to be equal to 'σ_{ys}' of the material. Realizing the existing relation between 'σ_0'and 'ε_0' a new formulation can be made as explained in equation (18) and corresponding J-integral value could be expressed as shown in equation (19).

$$F = \alpha\left(\frac{\sigma}{\sigma_0}\right)^n \tag{18}$$

Corresponding J-Integral is expressed as,

$$J_p = \alpha\sigma_0\varepsilon_0 b g_1 h_1 \left(\frac{P}{P_0}\right)^{n+1} \tag{19}$$

where,
P=Applied Load per unit thickness of the plate.
P_0= Limit or collapsed load of the plate based on the yield stress. Its variation under plane stress and plane strain conditions are expressed in equations (20) and (21) respectively.
'g_1' and 'h_1' = geometric factors which depends on 'a/W' and n, where 'a' and 'W' are crack length and width of the plate respectively.

$$P_0 = 2b\,\sigma_0 \text{ (For plane stress conditions)} \tag{20}$$

$$P_0 = \frac{4}{\sqrt{3}} b\sigma_0 \text{(For plane strain conditions)} \tag{21}$$

Empirical evaluation of plastic J-integral has been done with the aid of equation (19). For the computational demonstration of J-integral, the CCT specimen made of ASTM A 36 steel explained in figure 1 has been utilized. The plastic J-integral value has been found to be 0.00291 J/mm^2. The total J-integral value has been evaluated as 0.001209 J/mm^2 using equation (10).

Finite element Analysis of plastic J-integral required a nonlinear route. The non-linear analysis has been done by means of generating the material curve of ASTM A 36 steel shown in figure 13. The material curve was generated using the inverse Ramberg-Osgood relation [17], as expressed in equation (22). The value of plastic J-integral obtained by FEA is 0.002894 J/mm^2. The total J-integral value has been evaluated as 0.0012961 J/mm^2. The comparison between J-integral obtained by FEA and by empirical route is demonstrated in table 7.

Inverse Ramberg-Osgood Relationship,

$$\sigma = \frac{E\varepsilon}{\left[1 + \left(\frac{\varepsilon}{\varepsilon_0}\right)^n\right]^{1/n}} \tag{22}$$

where,

$$n = \frac{-\log_e 2}{\log_e\left(\frac{\sigma_y}{E\varepsilon_0}\right)} \tag{23}$$

$$\varepsilon_0 = \frac{\sigma_{uts}}{E} \tag{24}$$

The nomenclature of the above relationship is same as that of Ramberg-Osgood equation.

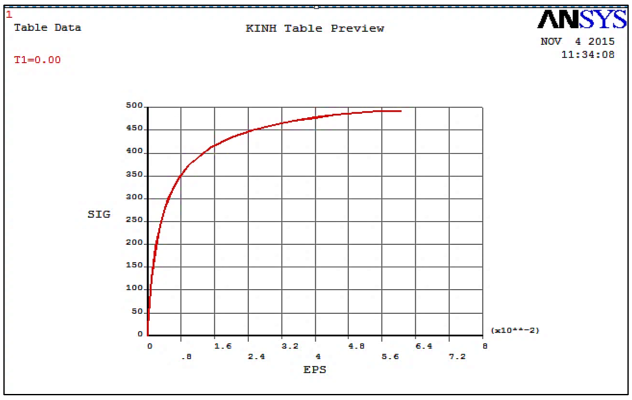

Figure 13. Material Curve for ASTM A36 Steel generated using ANSYS

Table 7 Critical Crack Length Values

J-Integral (FEA)	J-Integral (Empirical)	% Error
0.001296 J/mm^2	0.001209 J/mm^2	-6.75

The ability of FEA to determine EPFM parameter J-integral has been successfully computed using the FEA package ANSYS.

VI. CONCLUSION

In this work, methods of computational fracture mechanics in the determination of LEFM and EPFM parameters have been successfully studied. Effective utilization of FEA in fracture prediction has been analyzed and its reliability was checked with empirical models. LEFM parameter, stress intensity factor has been evaluated for varying aspect ratio values. Critical crack lengths have been evaluated for a CCT specimen under different loading cases by means of FEA, which could be utilized for in service testing of components.

An attempt has been made to demonstrate J-Integral by means of FEA and the results were compared with that of the analytical solution. Future scope of this work deals with the detailed study about the potential of FEA packages to predict Crack arrest mechanisms.

REFERENCES

1. Prashanth Kumar, "Elements of Fracture Mechanics", Wheeler publishers, New Delhi.", 1999, pages-2-151.
2. David Roylance, "Introduction to Fracture Mechanics", 2001.
3. Xian-Kui Zhu, James A. Joyce "Review of fracture toughness (G, K, J, CTOD, CTOA) testing and standardization", Engineering Fracture Mechanics 85 (2012) 1–46
4. Gustavo V. Guinea, Jaime Planas, Manuel Elices, "K_I evaluation by the displacement extrapolation technique", Engineering Fracture Mechanics 66 (2000) 243-255.
5. Boštjan Zafošnik, Gorazd Fajdiga, "Determining Stress Intensity Factor K_I With Extrapolation Method", Tehnički vjesnik 23, 6(2016), 1673-1678
6. T. Christopher, K. Sankaranarayanasamy, and B.N. Rao, (2004) "Failure behaviour of maraging steel tensile specimen and pressurized cylindrical vessels", Journal of Blackwell publishing ltd. Fatigue Fract. Engg.Mat Struct 27,177-186.
7. D. Radaj and M. Vormwald, "Advanced Methods of Fatigue Assessment", Springer-Verlag Berlin Heidelberg 2013, chapter-2, pages 101-265.
8. Naresh S, Bharath Naik L., Madhu S.K and Mohan A, "Computational Analysis of Stress Intensity Factor for a Quarter Circular Edge Crack under Mode-I loading", Research Journal of Engineering Sciences, Vol. 2(7), 38-42, July (2013).
9. G. Venkatachalam & R. Harichandran & S. Rajakumar &C. Dharmaraja & C. Pandivelan, "Determination of J-integral and stress intensity factor using the commercial FE software ABAQUS in austenitic stainless steel (AISI 304) plates", Int J Adv Manuf Technol, 2008, DOI 10.1007/s00170-008-1872-z.
10. Yuan J Kim, Nam-Su HUH and Young-Jin KIM, "Engineering J-estimation methods for Leak-Before-Break Analysis of Nuclear piping." JSME International Journal, Sereies A, Vol.48, No.1, 2005, pages. 41-50.
11. Ľubomír Gajdoš and Martin Šperl, "Application of a Fracture-Mechanics Approach to Gas Pipelines", World Academy of Science, Engineering and Technology 73 2011.
12. Daniel Vavrik, Ivan Jandejsek, "Experimental evaluation of contour J integral and energy dissipated in the fracture process zone", Engineering Fracture Mechanics 129 (2014) 14–25.
13. Tong Di-Hua, Wu Xue-Ren, "Analysis of crack opening stresses for center- and edge-crack tension specimens", Chinese Journal of Aeronautics, (2014),27(2): 291–298.
14. Tomáš Vojteka, Jaroslav Pokludaa, Jana Horníkováa, Pavel Šanderaa, Karel Slámečkaa, "Description of fatigue crack growth under modes II, III and II+III in terms of J-integral", Procedia Materials Science 3 (2014) 835 – 840.
15. ASM Handbook, Volume 1, Properties and Selection: Irons, Steels, and High Performance Alloys.
16. ANSYS Structural Analysis Guide, ANSYS Release 9.0
17. V. Murty , G. V. Rao, "An Inverse Form of Ramberg- Osgood Formula", NTIS, 1969.

Published By:
Blue Eyes Intelligence Engineering
and Sciences Publication (BEIESP)