International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 2 Issue 6, June - 2013

Investigation Of Fracture Parameters Of Compact Tension Specimen By FEA

[1]Ch.V.Sushma, [2]Dr. P. Ravinder Reddy, [3]P. Rama Lakshmi

[1]P.G.Student, [2]Prof&H.O.D, [3]Asst.prof, *Mechanical Engineering Dept., Chaitanya Bharathi Institute Of Technology, Hyderabad-75, AndraPradesh ,India.*

ABSTRACT

This paper is focused on the investigation of fracture parameters like J-integral of compact tension specimen by using ANSYS finite element analysis software which plays key role to estimate fracture toughness of materials. Here controlled displacement loading is applied on CT specimen and the results compared with ASTM standards.

Key wards: J-integral, Compact Tension Specimen, Controlled displacement loading.

1. INTRODUCTION

The standard compact specimen is a single-edge notched and fatigue-cracked plate loaded in tension. The specimen geometry which has been used successfully is Shown in Fig.1.

Compact tension specimen has been standardized by ASTM for use in experimental determination of fracture toughness of metallic materials[1,2]. Generally a clevis and pin arrangement is used to hold the specimen. The pre cracked specimen is loaded at a controlled rate, and the resulting load displacement data is recorded .Analysis of experimental data allows material fracture toughness to be determined in terms of stress intensity factor K or J-Integral. ANSYS12 includes features such as creating seam cracks, defining singularities, defining normal to the crack front and creating focused meshes. Using these features J-Integrals are estimated For CT specimen.

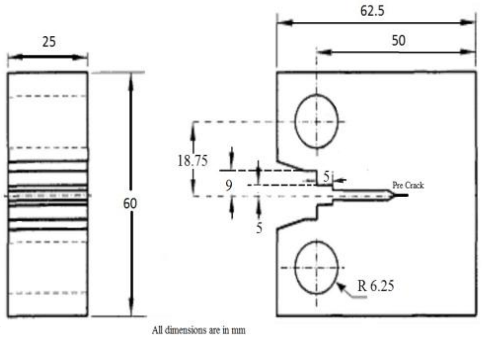


Fig1: Compact Tension specimen

2. J-INTEGRAL

To determine an energy quantity that describes the elastic-plastic behavior of materials, Rice[3] introduced a contour

Integral or line integral that encloses the crack front. The two-dimensional J-integral was originally defined (see Fig.2.)

$$J := \int_{\Gamma} \left(W\, dx_2 - \mathbf{t} \cdot \frac{\partial \mathbf{u}}{\partial x_1}\, ds \right)$$

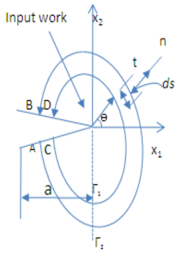


Fig2: J-Integral contours around a crack surface

J = Effective energy release rate (N/mm)

W = Elastic strain energy density or Plastic loading work (J/m^3)

ds = Differential element along the Contour

n = Outward unit normal to Γ

$t.(du/dx_1)$ ds = Input work

a= Crack length

t = Tension vector on the body bounded by Γ

Γ = Arbitrary counter clockwise contour

Around a contour Γ. It characterizes the stress-strain field around the crack front and therefore, it must be the energy release to the crack tip during crack growth. Due to this fact, the J-integral is used as failure criterion and it is a measure of the fracture toughness at the onset of slow crack growth for elastic and elastic-plastic metallic materials. The inherent characteristics of the J-integral exhibits a) remarkable path, contour size and shape independence, and b) an invariability in magnitude when the contour lies either inside or outside the plastic zone [4]. The calculations of J-Integral are made from load vs. load-line displacement curves. It begins by writing J in terms of its elastic and plastic components.

$$J=J_{el}+J_{pl}$$

$$\text{Or}\quad J=\frac{K^2\,(1-\nu^2)}{E}+\frac{\eta\,Apl}{B\,(W-a)}$$

Where $J_{el}=\frac{K^2\,(1-\nu^2)}{E}$ and

$$J_{pl}=\frac{\eta\,Apl}{B\,(W-a)}$$

$$\text{With}\qquad K=\frac{P}{B\,(W)\hat{}\,½}\,f(a/w)$$

$$f(a/w)= [2+(a/w)] * [(0.866+4.64\,(a/w)-13.32(a/w)^2+14.72\,(a/w)^3-5.6\,(a/w)^4)] * [1-(a/w)]^{-3/2}$$

And $\eta = 2+ 0.552 * B/W$

B= W-a

A_{pl} = plastic work

W= distance between point of application of load and end of specimen to right side in Fig1

a = distance between load line and the crack tip

B = thickness of the specimen

3. FINITE ELEMENT ANALYSIS

Finite element analysis includes three steps. (a) Preprocessing (b) analysis (c) post processing. Preprocessing includes modeling of CT specimen and applying boundary conditions like constraints, symmetry conditions, and loads. The CT specimen is considered as an isotropic two dimensional plane-strain model. The material selected is low alloy ferritic steel. The low alloy ferritic steel is an elastic –plastic material, not perfectly plastic but exhibits hardening. The properties of low alloy ferritic steel are Elastic modulus of specimen material is 213 GPa and poisson's ratio is 0.3.The yield stress is 715 MPa. Because of symmetry only half of the model is created.

2-D 8 node Plane 82 element is selected for the model. The 8-node elements have compatible displacement shapes and are well suited to model curved boundaries. The 8-node element is defined by eight nodes having two degrees of freedom at each node: translations in the nodal x and y directions. The element may be used as a plane element or as an axisymmetric element. The element has plasticity, creep, swelling, stress stiffening, large deflection, and large strain capabilities.

The loading pins are modeled as rigid bodies. To create model first key points are created. These key points are connected by lines. Then area is created with these lines. Then solid circles of radius 6.25 mm are created and subtracted from area. This represents CT specimen. Then at the crack tip by using KSCON command in ANSYS stress field singularity is defined. Then the specimen is meshed. Again solid circle of radius 6.25 is created to represent as pin. Then surface to surface contact between pin and specimen is obtained by selecting contact pair option, pick rigid target as solid circle's outer periphery and contact element as CT specimen's hole inside periphery. Now rigid to flexible contact is established.

Here target elements are TARGE169 and contact elements are CONTA172. Then symmetric boundary conditions are applied at the bottom of the specimen leaving the crack. Here analysis is done on 5 mm Pre cracked CT specimen. Two analysis steps are used. In the first step contact is established between the pins and the specimen by applying a small displacement in vertical direction. In the second step controlled displacement loading of the pins is applied. The reaction force for each displacement load at pins can be determined.

Table1: key points

Key point no	X (mm)	Y (mm)
1	0	0
2	5	0
3	30	0
4	30	30
5	-32.5	30
6	-32.5	12.35
7	-20	9
8	-20	5
9	-15	5
10	-15	1.25
11	-2.165	1.25

3.1. Creating lines

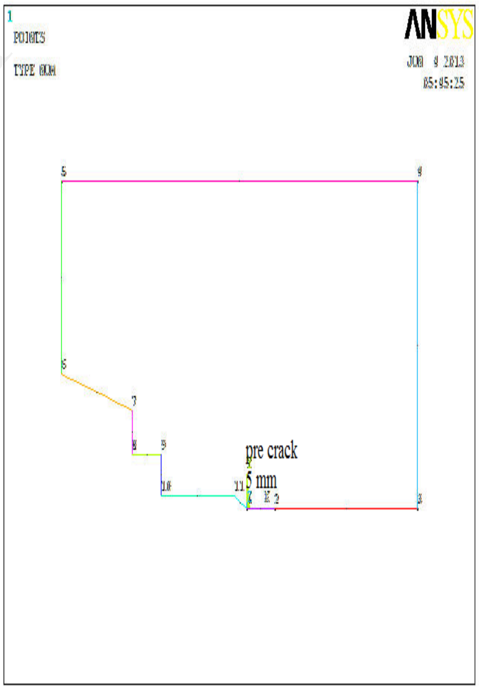


Fig3: Creating lines in ANSYS

3.2. Creating area

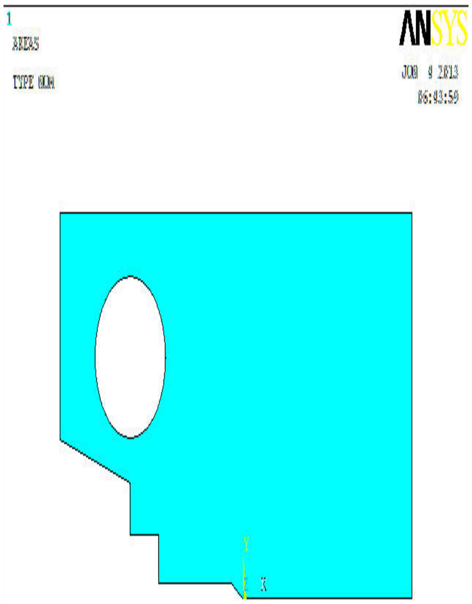


Fig4: Creating CT specimen area

3.3. Meshing area

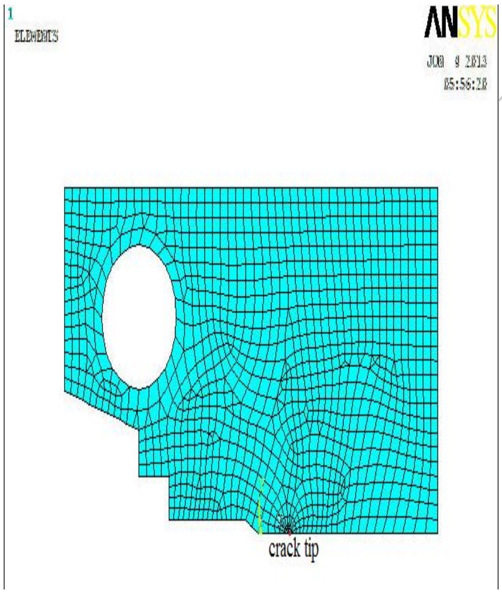


Fig5: Meshing specimen area

3430 nodes are created and 1126 elements are created.

3.4. Creation of contact pair

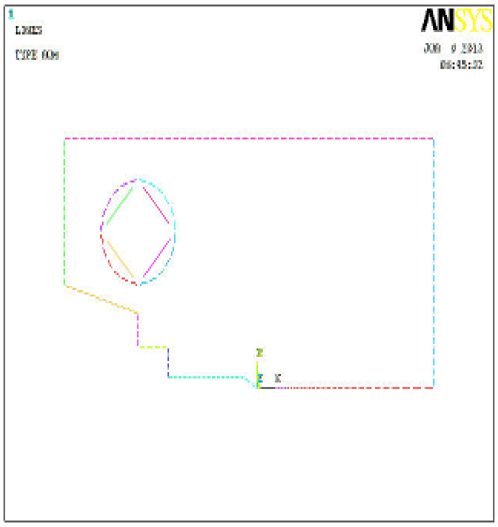


Fig6: Contact pair

3.5. Solution

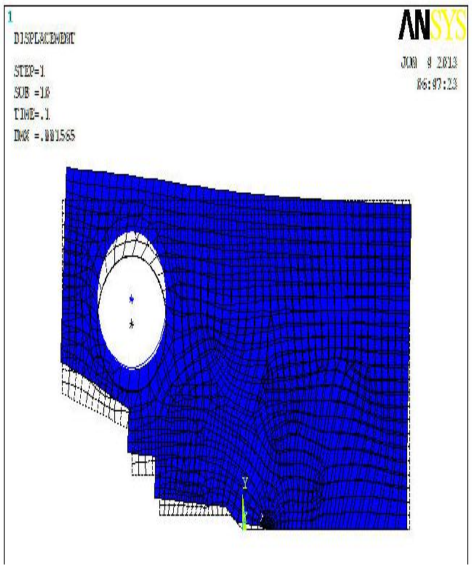


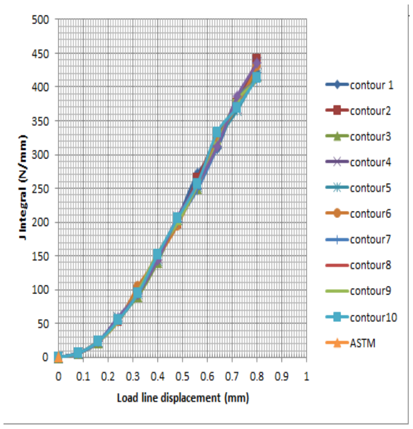
Fig7: solution for 5 mm crack and 0.639 mm vertical load.

4. RESULTS AND DICUSSIONS

By using J-Integral post processing technique in ANSYS the J-Integral values for 0.5 mm crack at various displacement loadings for 10 contours is noted. The average value of J-Integral is compared with ASTM standard values. The comparison is shown in Table2.They are in very close conformance with results computed using ASTM standards. In Graph1 the J-Integral values for all the analysis methods are plotted. Usually the J-Integral for the first contour is ignored because of numerical inaccuracies in the stresses and strains at the crack tip. The effect of inaccuracies is less pronounced in small strain problems than in finite strain problems.

Table2: comparison of j-integral values in (N/mm)

Load line displacement in mm	ASTM (E1737-96)	ANSYS results
0	0	0
0.0797	6.19	6.231
0.159	23.87	23.2741
0.239	56.53	56.0204
0.319	98.88	96.1
0.399	148.19	148.05
0.479	203.19	202.57
0.559	259.01	256.9
0.639	312.7	318.9
0.719	371.86	376.13
0.8	425.78	422.96



Graph1: J-Integrals results obtained for 10 contours using small-strain analysis

5. CONCLUSIONS

In this paper the J-Integral values for 5mm pre cracked compact tension specimen is measured by controlled displacement loading technique. The J-Integral values obtained by ANSYS are compared with ASTM values are found to be very close to each other and are satisfactory.

6. REFERENCES

1. ASTM Standards,—Test Method for ***J***-integral characterization of Fracture Toughness, ASTM E1737-96, American Society for Testing and Materials, Philadelphia, 1996.

2. ASTM Standards, —Standard Test Method for Plane-Strain Fracture Toughness of Metallic Materials, ASTM E399- 90,‖ American Society for Testing and Materials, Philadelphia, 1997.

3. J.R. Rice, J. Appl. Mech. 35 (1968) 379-386.

4. E.M. Morozov and G.P. Nikishkov, “Finite Element Method in Fracture Mechanics,” Nauka, Moscow, (1980), referencecitedin.

5. W. F. Brown, Jr., J. E. Srawley, "Plane Strain Crack Toughness Testing of High Strength Metallic Materials", ASTM STP-410, 1966.