*International Journal on Emerging Technologies* (Special Issue on ICRIET-2016) **7**(2): 52-56(2016)

**ISSN No. (Print) : 0975-8364**
**ISSN No. (Online) : 2249-3255**

# Finite Element modeling for computational Elastic-Plastic Fracture Mechanics: Crack Tip Opening Displacement (CTOD)

***Pradeep Kumar***
*Assistant Professor, Department of Mechanical Engineering,*
*BKIT, Bhalki Karnataka, India.*

*(Corresponding author: Pradeep Kumar)*
*(Received 16 September, 2016 Accepted 19 October, 2016)*
*(Published by Research Trend, Website: www.researchtrend.net)*

**ABSTRACT: EPFM applies to materials that exhibit time independent, non-linear behavior (i.e. plastic deformation). There are two Elastic-Plastic fracture parameters i.e. Crack Tip Opening Displacement (CTOD) and Non-Linear Energy Release Rate J. This investigation deals with Elastic-Plastic Fracture Mechanics Parameters CTOD. 2-D plane strain Elastic-Plastic Finite Element stress analysis of Center Cracked Tension Specimen (CCT) is carried out using finite element commercial code ABAQUS. The material behavior considered is Elastic-Perfectly Plastic. Focused mesh of plastic singularity elements is considered around the crack tip with independent nodes at the tip so as to model blunting of the crack tip as load is applied so as to compute CTOD by 90 degree intercept procedure from the deformed mesh. Mesh Design adopted is "spider-web" configuration. The proposed methodology for CTOD computation is validated using NAFEMS benchmark namely a centre cracked rectangular panel subjected to in-plane tension. Master curve is obtained by plotting normalized CTOD v/s Normalized load. Incidentally the relation between J and CTOD is also explored. Parametric studies include different material models as bilinear isotropic hardening material with different values of tangent modulus and Ramberg Osgood Material model for different hardening exponents and different crack lengths.**

***Keywords**:* LEFM, EPFM, CTOD, ABAQUS, NAFEMS

## I. INTRODUCTION

Fracture Mechanics is defined as the study of unstable propagation of crack under effect of applied loads. It is divided into two classes as Linear Elastic Fracture Mechanics (LEFM) and Elastic Plastic Fracture Mechanics (EPFM) Linear elastic fracture mechanics (LEFM) is valid as long as nonlinear material deformation is confined to a small region surrounding the crack tip. In many materials, it is virtually impossible to characterize the fracture behavior with LEFM, and an alternative fracture mechanics model is required. Elastic-plastic fracture mechanics applies to materials that exhibit time-independent, nonlinear behavior (i.e., plastic deformation). Two elastic-plastic parameters defines the fracture in elastic-plastic regime i.e. the crack-tip-opening displacement (CTOD) and the *J* (Non-linear Energy Release Rate).Both parameters describe crack-tip conditions in elastic-plastic materials, and each can be used as a fracture criterion. Critical values of CTOD or *J* give nearly size-independent measures of fracture toughness, even for relatively large amounts of crack-tip plasticity. There are limits to the applicability of *J* and CTOD but these limits are much less restrictive than the validity requirements of LEFM. CTOD can be defined as finite displacement ($\delta$) at the crack tip. When an initially sharp crack blunts with plastic deformation.

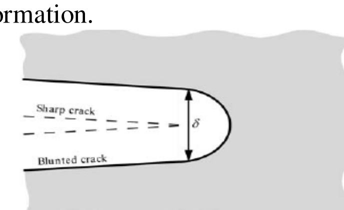



**Fig. 1.** Definition of CTOD as suggested by SHIH.

S.K. Kudari, K.G. Kodancha [1] estimated CTOD using 90 degree intercept procedure and by plastic hinge model for Compact Tension and SENB specimen. The CTOD values obtained by both the methods are found to be linearly proportional to J-integral. The proportionality constant $d_n$ between CTOD and J is found to be strongly depend on the method of estimation of CTOD, specimen geometry and a/w ratio of the specimen.

D M Kulkarni, Ravi Prakash and A.N. Kumar [2] developed a technique for determining fracture criteria in sheet metals, results are generated on critical CTOD and fracture toughness. Finite Element analysis was performed to support the results on various fracture parameters. T.L. Anderson [3] discusses regarding the theoretical relationship between CTOD and J-integral. He also discusses computational aspects of fracture mechanics. S.K. Kudari, B. Maiti, K.K. Ray [4] investigated the extent of plastic zone size ahead of the crack tip in SENT, CT specimen experimentally by micro-hardness technique.S.A. Krishnan, B. Shashank Dutt, A Moitra, G. Sasikala, S.K. Albert and A.K. Bhaduri [5] used CTOD as the fracture criteria to simulate crack growth behaviour of laboratory specimens as well as for structural components for their integrity assessment.

This investigation deals with computation of CTOD using 90 degree intercept procedure and J using domain integral procedure in ABAQUS for Centre Cracked Tension Panel (CCT) in Plane Strain for elastic-perfectly plastic, bilinear isotropic material model and Ramberg Osgood Material Model.

## II. FINITE ELEMENT ANALYSIS

A wide range of finite element models (FEM's) were created for a matrix of different crack sizes and hardening values and material models. The FEM's for the 2-D geometries were created using the part module of ABAQUS and crack-tip elements were added using the mesh module of ABAQUS. The modeling recommendations by Anderson [4] are followed for designing the finite element mesh. Eight-Node diso-parametric quadrilateral quadratic elements with reduced integration were used for all geometries with 2 DOF'S at each Node i.e. $u_i$ and $v_i$. ABAQUS recommends using reduced integration to prevent element locking when fully plastic (incompressible) deformation is modeled. At the crack tip region, a series of concentric rings of second-order quadrilateral elements around the crack tip was used. The quadrilateral elements of the innermost ring were collapsed to triangles (STRIA6), so that three nodes of each element were coincident at the crack tip. Initially, coincident nodes at the crack tip were left unconstrained for elastic-plastic analysis to model blunting of the crack tip. The side nodes were left at the midpoints of each side for elastic-plastic analysis. This configuration gives a $1/r$ singularity at the crack tip.

### *A. Benchmark: Center-Cracked Rectangular Panel*

CCT FEM's were generated using the modeling procedure described above for a/w ratio of 0.2 as suggested in NAFEMS Manual and for Elastic-perfectly plastic Material Model. Eight-Nodediso-parametric quadratic elements with reduced integration were used i.e (type CPE8R in ABAQUS) Plane strain state is assumed. As illustrated in Figure 2, full model of the CCT panel was considered in this study. Appropriate partitions were made in the model so that finer and focused meshes can be created near the crack tip and coarse mesh elsewhere. Boundary conditions and uniformly distributed loading were imposed on the model using predefined node sets in ABAQUS as shown in fig 2.

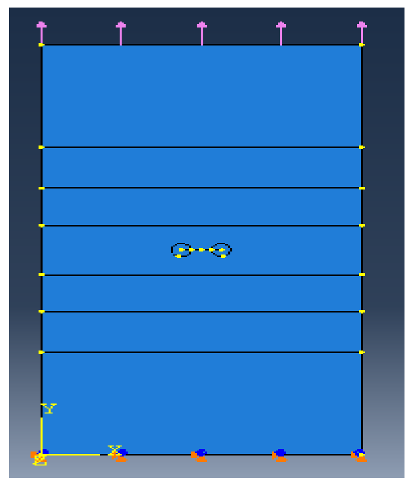


**Fig. 2.** Geometric model of CCT Specimen along with Boundary Conditions and Loading.

The CCT FEM has four distinct areas as illustrated in Fig. 3. and 4 the first area (A) consists of a series of concentric rings of elements. The element size near the crack tip was kept at 0.001 mm and number of elements around the crack tip was set at 36. The fine mesh has concentric element rings of elements and element widths are biased towards the crack tip to ensure that the smallest elements will be in the area of highest stress. The second area (B) is the region where uniform pressure is applied 8 elements high. The other two areas, C and D, are transition meshes.

**Table 1: Mesh details.**

| | |
|---|---|
| No of Elements | 3170 |
| No of Nodes | 16174 |
| No of Variables | 29178 |

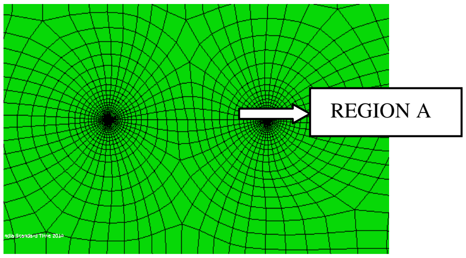



**Fig. 3.** Spider Mesh Configuration Near the Crack Tip.

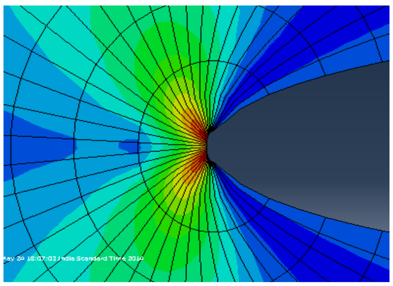


**Fig. 4.** Blunting of the sharp crack tip and plastic zones formed near the crack tip.

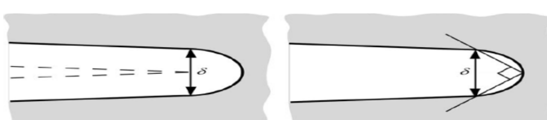
*B. Material Model used in ABAQUS*

Material models considered for the study is.

1. Elastic-Perfectly Plastic.
2. Isotropic Bi-linear Hardening
3. Ramberg-Osgood Constitutive model

*C. Crack Tip Opening Displacement (CTOD)*

There are a number of alternative definitions of CTOD. The two most common definitions, which are illustrated in Fig. 5, are the displacement at the original crack tip and the 90 degree intercept. The latter definition was suggested by Rice and is commonly used to infer CTOD in finite element measurements. Note that these two definitions are equivalent if the crack blunts into semicircle.




(a) (b)

**Fig. 5.** Definitions of CTOD: (a) displacement at the original crack tip and (b) Displacement at the intersection of a 90 degree vertex with the crack flanks.

*D. Relationship between J and CTOD*

The relationship between CTOD and *J* in the limit of small-scale yielding is given by:

$$\delta=\frac{Jd_n}{\sigma_y} \qquad (2.1)$$

Where $d_n$ is a dimensionless constant that depends on the stress state and material properties and can be read off from the following fig. 6.

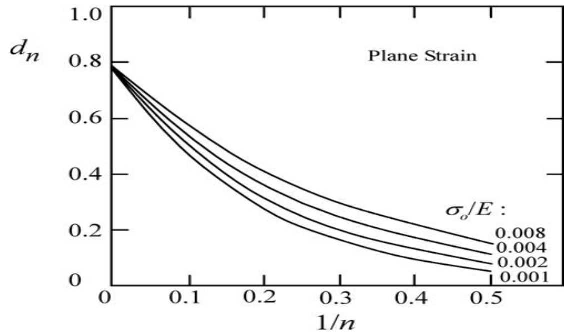



**Fig. 6.** Predicted *J*-CTOD relationships for plane stress and plane strain, assuming $a$ = 1. For $a \neq 1$, the above values should be multiplied by $\alpha^{1/n}$. Taken from Shih, C.F. ''Relationship between the *J*-Integral and the Crack Opening Displacement for Stationary and Extending Cracks.'' *Journal of the Mechanics and Physics of Solids,* Vol. 29, 1981, pp. 305–326.

## III. RESULTS AND DISCUSSIONS

Various load steps have been applied on the specimen with a/w = 0.2.To study the distribution of Von-Mises stresses in the specimen domain. At each Load step the value CTOD is computed using 90 degree intercept procedure. Then plots of Normalized J v/s Normalized load has been done for maximum load level of 0.922 of yield strength to study the behavior of master curve. The curve depicts a rising trend which resembles the curve in the NAFEMS Manual which is the Bench Mark. No Bench Mark exists for CTOD in NAFEMS Manual. Hence Bench Mark for CTOD is set up using the J and CTOD Relation.

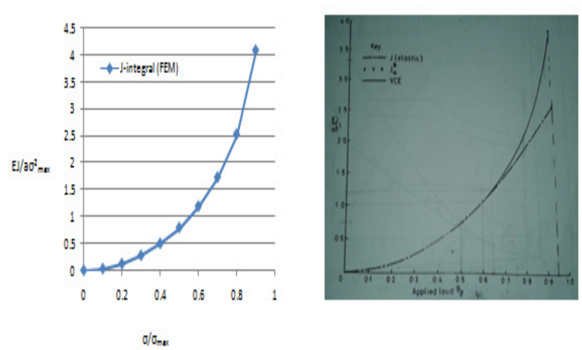



**Fig. 7.** Shows Plot of normalized J v/s normalized load for a/w =0.2 using FEM and Plot of normalized J v/s normalized load (NAFEM Manual) for a/w=0.2.

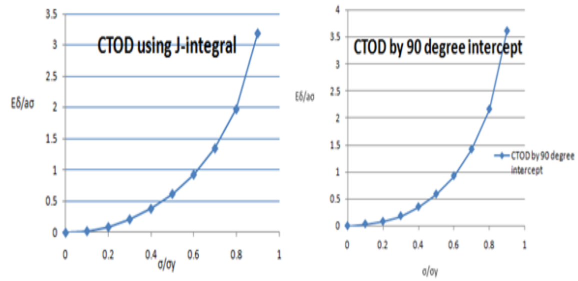



**Fig. 8.** Graph of normalized CTOD v/s normalized load for a/w= 0.2.

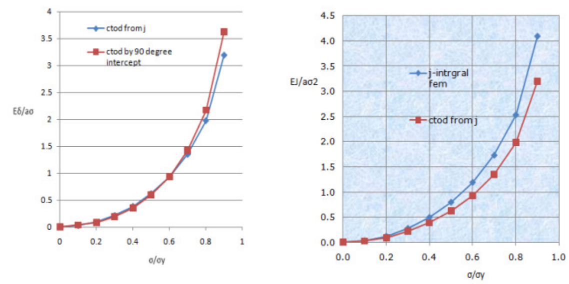



**Fig. 9.** Comparison of different plots.

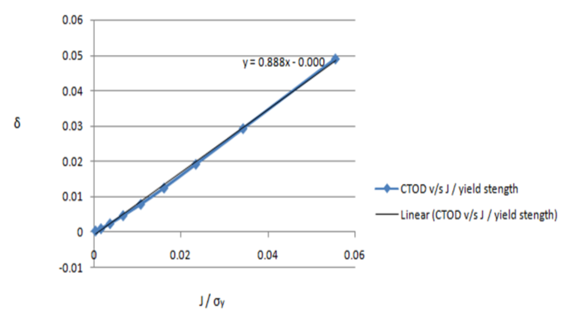



**Fig. 10.** Typical plot showing variation of CTOD v/s $J/\sigma_y$

The value of dn (plastic constraint factor) as determined by Shih is 0.78 and by using FEM is 0.88.

**Parametric Studies**: Material Model: Bilinear Isotropic Hardening Modulus:

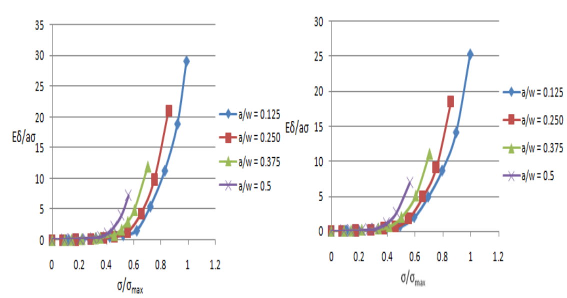



**Fig. 11.** Plot of Normalized δ v/s Normalized load $\sigma/\sigma_{max}$for $E_t$ = 15 Gpa and $E_t$ = 20 Gpa.

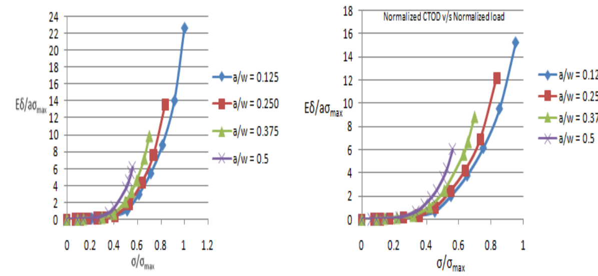



**Fig. 12.** Plot of Normalized δ v/s Normalized load $\sigma/\sigma_{max}$for $E_t$ = 25 Gpa and $E_t$ = 30 Gpa.

The plots indicate that normalized δ depends quadratically on the load level in the elastic region and linear in the elastic-plastic region. It can also be inferred that at the same load level as the value of a/w increases Normalized δ tends to increase. When the CCT specimen was loaded to full plastic collapse i.e. the load at which the entire width of the plate becomes plastic. It was found that as a/w ratio increases plastic failure load tends to decrease.

Ramberg Osgood Material Model:

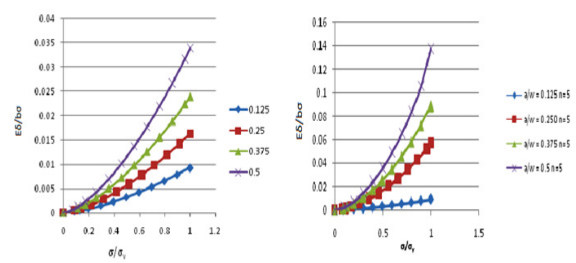



**Fig.13.** Variation of Normalized δ V/s Normalized Load for a/w=0.125, 0.250, 0.375, 0.5 for n=3 & n=5.

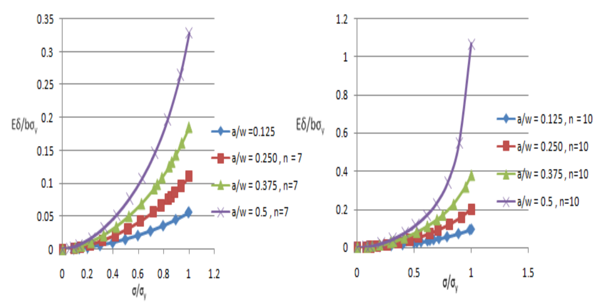



**Fig. 14.** Variation of Normalized δ V/s Normalized Load for a/w=0.125, 0.250, 0.375, 0.5 n= 7 and n=10.

The plots indicate that normalized δ depends quadratically on the load level in the elastic region and linear in the elastic-plastic region. It can also be inferred that at the same load level as the value of a/w increases Normalized δ tends to increase.

When the CCT specimen was loaded to full plastic collapse i.e. the load at which the entire width of the plate becomes plastic. It was found that as a/w ratio increases plastic failure load tends to decrease

## IV. CONCLUSIONS

1. The present study has demonstrated convincingly finite element modeling using ABAQUS for computational Elastic-Plastic Fracture Mechanics with focus on the CTOD. Incidentally the relationship between J and CTOD can be used to compute CTOD also.
2. Extensive results of parametric study to document the effects of crack length, tangent modulus and strain hardening exponent are also included.
3. The end objective of this research is EPFM analysis of pressurized tube with an axial trapezoidal crack. This problem is 3-dimensional and demands computing CTOD and its variation along the crack front. Significant results of this analysis will be reported elsewhere.

## REFERENCES


[1]. S.K. Kudari , K.G. Kodancha , " On the relationship between J-integral and CTOD for CT and SENB specimens, S*.K. Kudari et al., FratturaedIntegrita Strutturale,* ***6****(2008)* 3-10;

[2]. D.M. Kulkarni , Ravi Prakash and A.N Kumar , " Experimental and finite element analysis of fracture criterion in general yielding fracture mechanics" , *sadhana vol.* ***27*** , part 6, Dec 2002, pp. 631-642;

[3]. T.L Anderson, *Fracture Mechanics Fundamentals and Applications* CRC. Press, USA (1995), 2-Edition.

[4]. S.K. Kudari , B. Maiti, K.K.Ray, " Experimental investigation on possible dependence of plastic zone size on specimen geometry*" S.K. Kudari et alii, FratturaedIntegritàStrutturale,* **7** (2009) 57-64; DOI: 10.3221/IGF-ESIS.07.04;

[5]. S.A. Krishnan , B. ShashankDutt , A Moitra, G. Sasikala, S.K.Albert and A.K. Bhaduri, "Evaluation of CTOD/CTOA Scheme for Characterization of SCG through AISI type 316L(N) material", *Material Development and Technology Group, Indira Gandhi Centre for Atomic Research, Kalpakkam, Tamil Nadu, India.*