Scientific Journal of Impact Factor (SJIF): 5.71

e-ISSN (O): 2348-4470
p-ISSN (P): 2348-6406

International Journal of Advance Engineering and Research Development

Applications of Nanotechnology In Civil Engineering-2019.

Volume 6, Special Issue 01, Feb.-2019.

STUDY OF FRACTURE BASED PARAMETERS ON CT SPECIMEN WITH THICKNESS VARIATION

Dr. Ganesh Doiphode[1], Pooja Shah[2]

[1] *Associate Professor, Applied Mechanics Department, Faculty of Technology and Engineering, Maharaja Sayajirao University, Vadodara, Gujarat, India.*
[2]*M.E student (Fracture Mechanics),Applied Mechanics Department, Faculty of Technology and Engineering, Maharaja Sayajirao University, Vadodara, Gujarat, India.*

ABSTRACT— *Fracture mechanics is the field of mechanics which is associated with the study of the propagation of cracks in structural materials. It is the prime driving force on a crack-based studies in various types of materials. It is also part of experimental solid mechanics which is intended to characterize the material's resistance during fracture. It is the analysis of flaws and initial defects which liable to propagate as cracks during different types of loading that causes sudden or catastrophic failure of the flawed structure. Thus, fracture mechanics properties are foremost to know to develop actual prototype which is sustainable, durable & long lasting. Fracture mechanics-based properties always differ for various materials as well it also may exhibit changes with distinct geometrical properties. Nowadays many finite element methods based softwares are also available to examine and study fracture properties. This research exhibits the analysis of different fracture properties such as stress intensity factor, J integral for materials which are of importance aspect in construction industry, steel industry, automotive field, aerospace and many more respectively. Here in this research fracture properties (i.e. stress intensity factor, J integral) for many materials such as aluminium, steel, concrete, glass and GFRP (Glass Fibre Reinforced Polymer) with varying geometrical properties are evaluated with the help of CT specimen as per ASTM E-399 by finite element method-based software "Abaqus" as well as obtained results are validated by derived analytical results.*

Keywords- *Fracture Mechanics, Compact Tension (CT) specimen, Abaqus , Finite element method , Stress intensity factor, J-integral*

I. INTRODUCTION

Applied Mechanics is the branch, which deals with study of external response of structure whenever a structure is subjected to various external forces. Fracture Mechanics is the young branch of applied mechanics, which deals with the material response and behaviour of a defined crack. Continuum Damage Mechanics (C.D.M.) attempts to deal with the circumstances, when multiple cracking or void age is prominently the factor of degradation and failure mechanism. These theories help to characterize structural element with their life span and workability. A structural element which is subjected to multiple varying dynamic-non-harmonic loading, in that case it is necessary to predict behaviour of the materials that may appear to be homogeneous, in fact, seen to be highly inhomogeneous when viewed through an electron microscope. Design engineer, geometric irregularities alone are not sufficient for defining the strength of material. Geometrical and other deformation-based information can be obtained from the specimens with simple geometries which is subjected to tensile or compressive loads under controlled loading rates. The resulting data plotted in terms of uniaxial stress can be linear or nonlinear depending on the combined effect of specimen size and geometry, loading rate and material type.

A structure will not fail due to any single reason; its failure may occur due to cumulative effect of various parameters. If we know the property of every structural element which is used, one can predict its individual performance for that effects and hence make the structural life as long as possible. For minimizing failure from the structures, we must know fracture properties of the different materials.

II. CRITICAL LITERATURE REVIEWS

Nikhil Gupta and Praveen Pachauri, "An Experimental and Computational Investigation of Crack Growth Initiation in Compact Tension (CT)Specimen" (International Journal of Scientific and Research Publications, pp-365-380 Volume 2, Issue 8, August 2012)
This paper highlights experimental and computational study of EN-31 steel Compact Tension specimen. Stress intensity factor, Crack tip opening displacement (CTOD) & J integral were main parameter found by the experimental work and

validated by the FE code ("Abaqus"). A comparison with an Experimental Vs Computational investigation on compact Tension specimen suggested similarities

Niharika Sharma, "Influence of specimen thickness on fracture toughness of mild steel" (International Journal of Advance Research in Engineering, Science & Technology, pp-12-13, Volume 3, Issue 9,2016)
In this paper fracture toughness parameters of mild steel were investigated experimentally by using cracked specimens with varying thickness varying from 12, 15, 20, 25 mm. respectively. The coupled effects of specimen thickness and delamination upon fracture toughness and the mechanism of delamination were revealed. It concludes that as thickness increases stress KIC and JIC decreases.

M.O.LAI and W.G.FERGUSON, "Effect of specimen thickness on fracture toughness" (Engineering Fracture Mechanics, pp 649-659 vol.23,no.4,1986)

This paper shows variation of fracture toughness with specimen thickness for steel EN-25. Compact Tension (CT) specimens of thickness ranging from 4 mm to 25 mm were used to investigate fracture properties using models proposed by bluhm proc. and Hahn et al. Hahn's model is more cumbersome to use because measurement of the surface depression width is involved, both models have the advantage that data from any two specimens can be used in the analysis. They have observed that sufficient accuracy may obtained if value of GIC is known.

D M Kulkarni, Ravi Prakash ,"The effect of specimen thickness on the experimental and finite element characterization of CTOD in extra deep drawn steel sheets" (Sadhana, Vol. 29, Part 4, pp. 365–380,2004)

The results of this study showed that the 'CTOD' parameter can be successfully characterized for increasing thickness of EDD (0·06%C) steel sheets. The stress–strain field around the crack-tip is described with the help of two parameters: J and CTOD. It was exhibited that CTOD increases with increase in specimen thickness.

III. OBJECTIVE

I. To determine stress intensity factors for different materials that are steel, aluminium, concrete, glass and Glass Fibre Reinforced Polymer (GFRP) for distinct specimen thicknesses through CT specimen in finite element method-based software "Abaqus".
II. To evaluate J-integral for CT specimen with varying geometrical parameter (i.e. thickness) for aluminium as well as steel along with glass, GFRP and concrete by "Abaqus".
III. Finite element software-based results will be validated by analytical or theoretical solutions along with comparative graphical representation.
IV. Whether "Abaqus" as a finite element analytical tool can be employed to find fracture-based properties accurately or not will be verified.

IV. METHODOLOGY

In this research number of CT specimens for different thicknesses i.e. 16mm , 20mm and 25 mm of different materials aluminium , steel , concrete , glass and GFRP were constructed in finite element method based software "*ABAQUS"* in part module as described in fig 1. CT specimens constructed in part module were assigned different values of modulus of elasticity in N/mm^2 and distinct values of poison's ratios in property module as described in fig 2(a) and fig 2(b) respectively as a demonstration of property module.

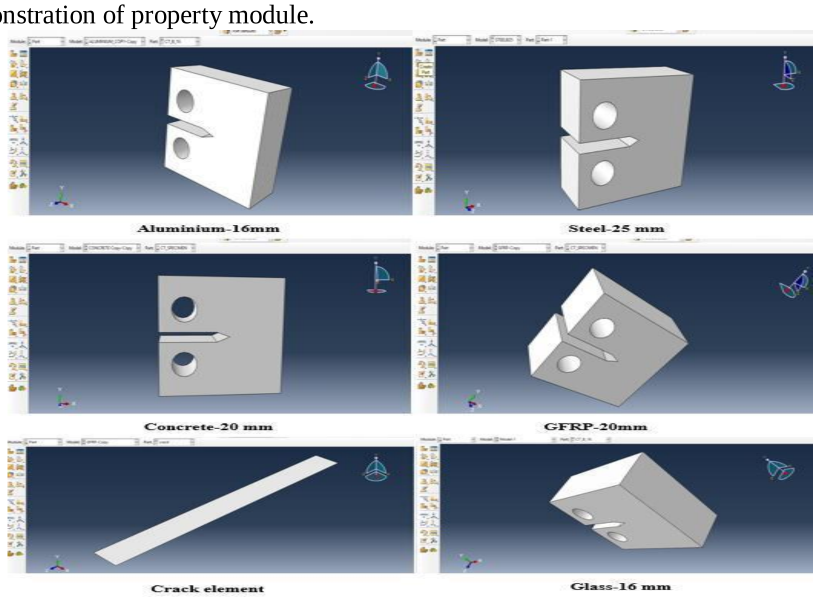

Figure 1. CT specimens of various thickness and crack element

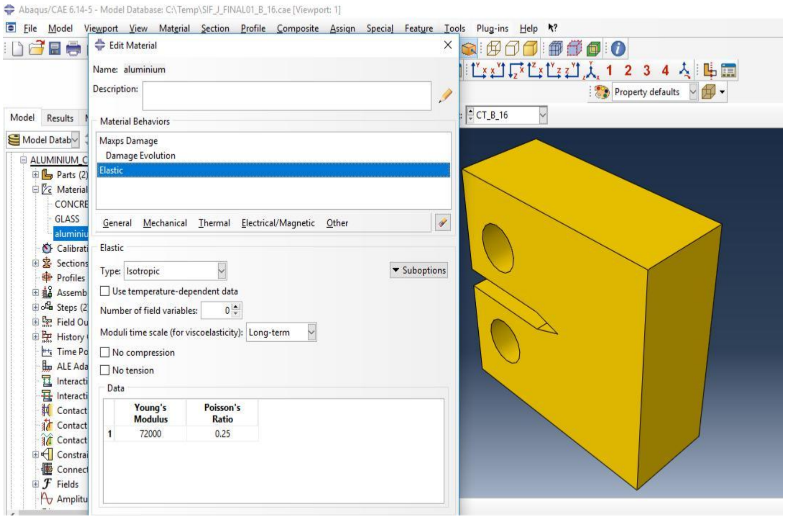

Figure 2(a). Aluminium properties

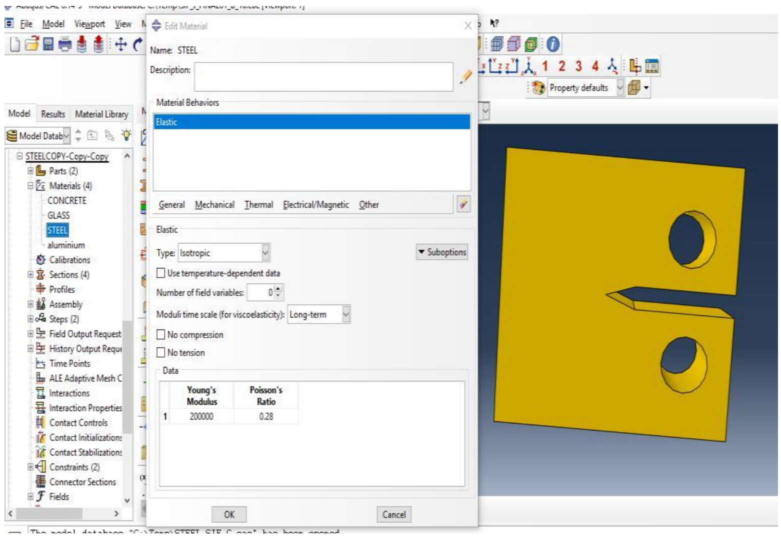

Figure 2(b). Steel properties

Assembly module is utilised to create instances of parts and to position the instances relative to each other in a global coordinate system, thus creating the assembly. Fig 3 illustrates one typical assembly of CT specimen with crack.

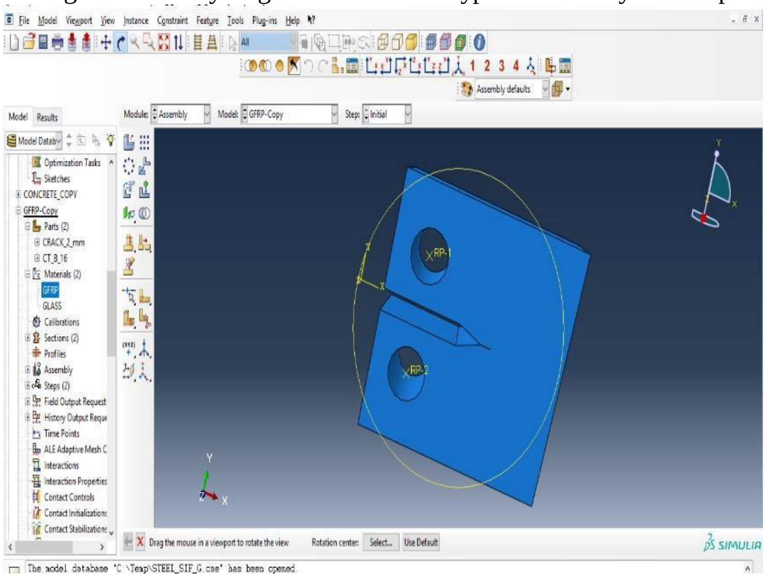

Figure 3. Assembly module

The Step module is used to perform analysis steps. In interaction module two reference points named RP1 & RP2 were assigned for hole in CT specimen and two constraints were added. Fig 4, 5 and 6 illustrates load set up and boundary conditions.

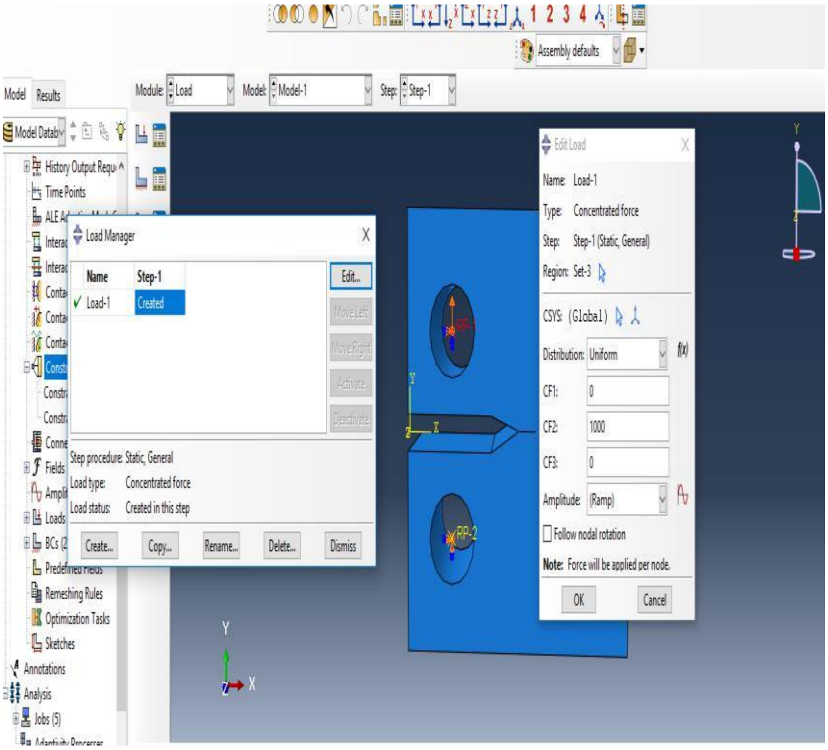

Figure 4. Load set up

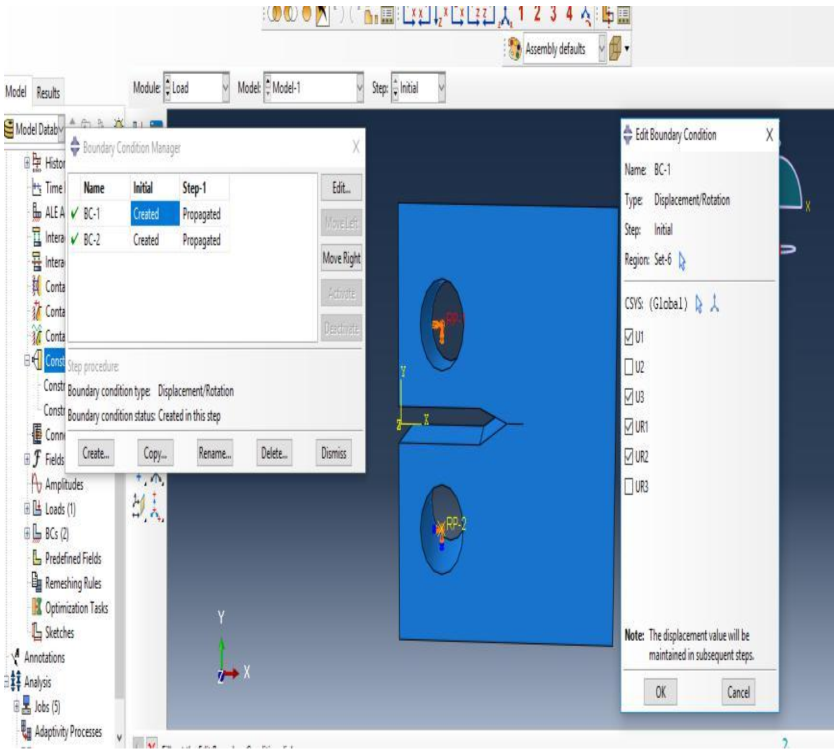

Figure 5. boundary condition for RP1

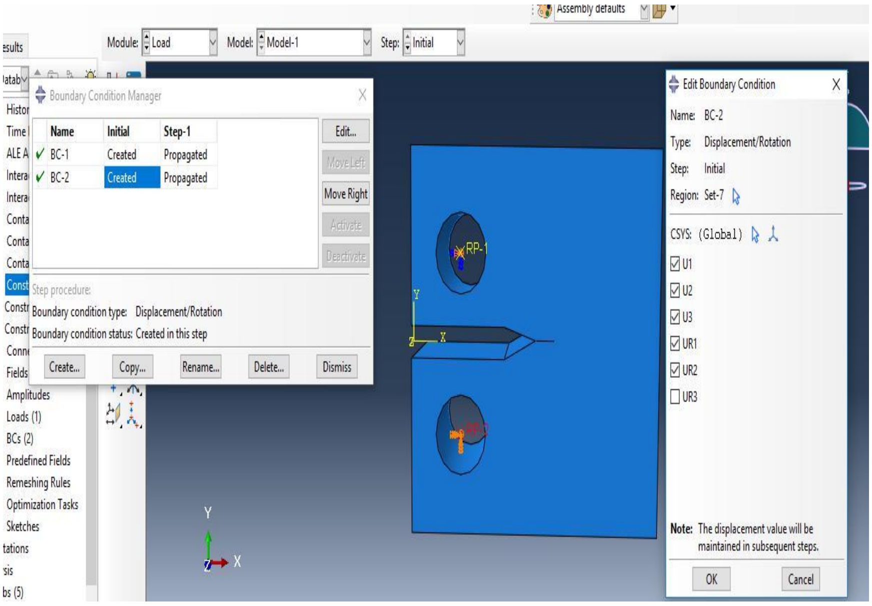

Figure 6. boundary condition for RP2

Once anyone has finished all of the tasks involved in defining a model (such as defining the geometry of the model, assigning section properties, defining contact, mesh assignment), Job module can be employed to analyse model. The Job module allows to create a job, to submit it for analysis, and to monitor its progress. Monitor window will represent the computational values for stress intensity factor as well as J-integral.

V. OBSERVATIONS

Table illustrated below demonstrates comparative results for Stress intensity factor and J integral along with graphical representation for aluminium and GFRP respectively, similarly comparative results are achieved for all other materials too. i.e. steel, concrete and glass.

Table-A- Comparative tabular representation for aluminium (KI)

Thickness (mm)	KI (computational)	KI (analytical)	Error (%)
16	109	106.71	2.1
20	75.92	76.36	0.58
25	61.26	60.13	1.8

Table B comparative tabular representation for aluminium (J)

Thickness (mm)	J (computational)	J (analytical)	Error (%)
16	0.152	0.158	3.7
20	0.076	0.08	5
25	0.0499	0.0505	1.1

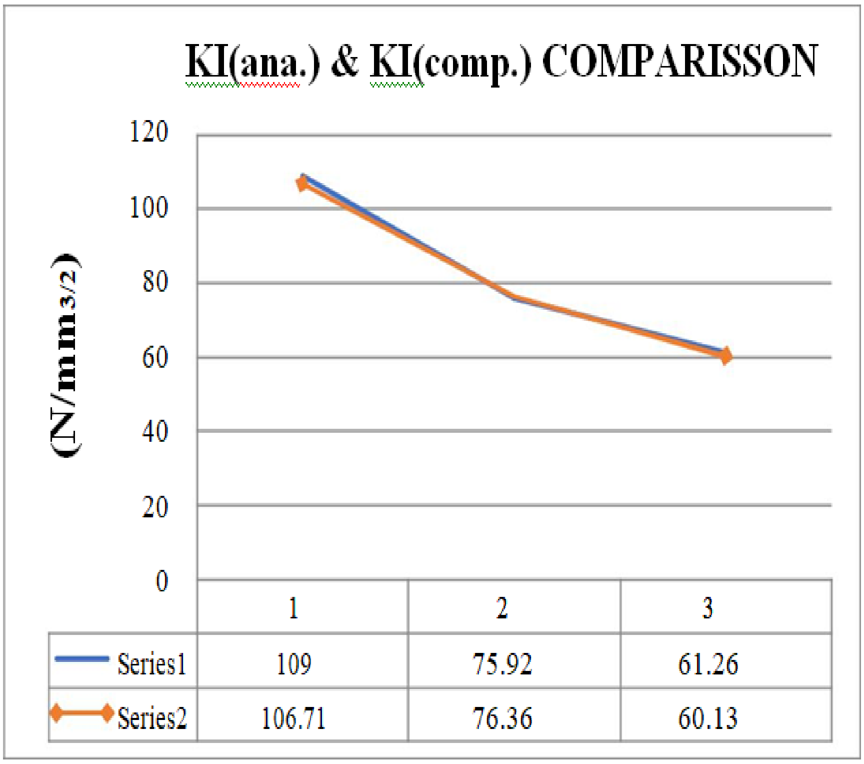

Figure 7. Comparative graphical representation for aluminium (stress intensity factor)

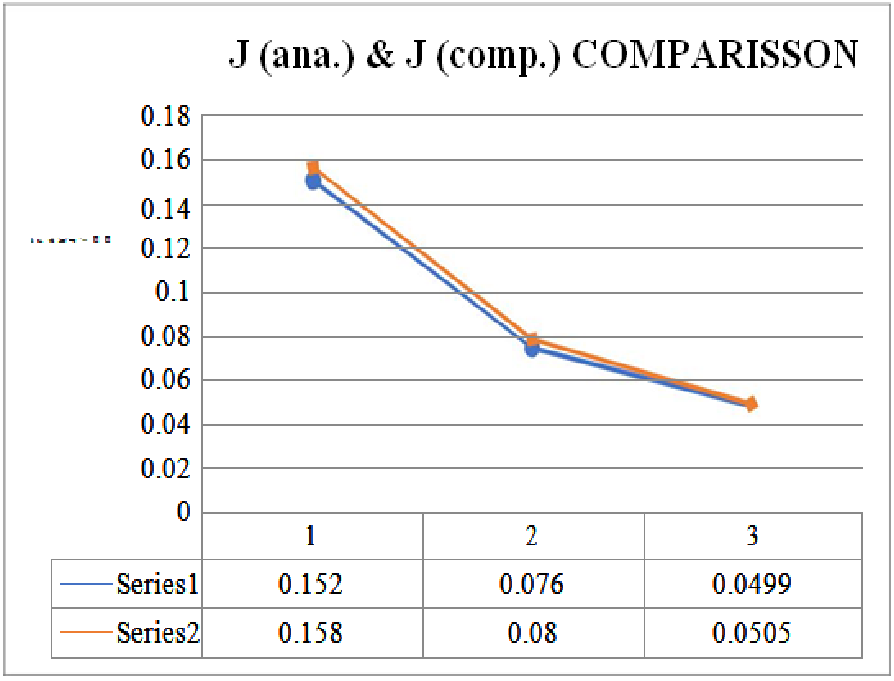

Figure 8. Comparative graphical representation for aluminium (J-integral)

TABLE C COMPARATIVE TABULAR REPRESENTATION FOR GFRP(KI)

Thickness (mm)	KI (computational)	KI (analytical)	Error (%)
16	103.3	106.71	3.3
20	76.2	76.36	0.2
25	60.7	60.13	0.93

TABLE D COMPARATIVE TABULAR REPRESENTATION FOR GI

Thickness (mm)	J (computational)	J (analytical)	Error (%)
16	0.5705	0.6048	5.6
20	0.323	0.338	4.4
25	0.205	0.21	2.3

VI. CONCLUSION

1. Computational results obtained through “Abaqus” are matching to analytical solution available through past literature.
2. Stress intensity factor (KI) increases with increasing thickness of specimen for all materials i.e. aluminium, steel, concrete, glass and GFRP.
3. J integral decreases with decreasing thickness of specimen for all materials i.e. aluminium, steel, concrete, glass and GFRP.
4. The error found in J integral is of 5.5% which is in acceptable range between obtained solution using “Abaqus” with respect to analytical solution.
5. The error found in Stress intensity factor is of 3.3% which is in acceptable range between obtained solution using “Abaqus” with respect to analytical solution.
6. The above fracture mechanics properties are schematically used to evaluate overall behaviour of engineering materials.

VII. FUTURE SCOPE

- This research can be further carried out to evaluate more fracture properties such as CTOD (Crack Tip Opening Displacement) , Energy release rate respectively.
- The same study can be worked out by using other computational software.
- Same study can be extended by varying specimen dimensions respectively.

REFERENCES

[1] en.wikipedia.org/wiki/Fracture mechanics
[2] en.wikipedia.org/wiki/Structural steel
[3] Abaqus user manual: abaqus.software.polimi.it/v6.14/books/usi/default.html
[4] Niharika Sharma, “Influence of specimen thickness on fracture toughness of mild steel” (International Journal of Advance Research in Engineering, Science & Technology, pp-12-13, Volume 3, Issue 9,2016)
[5] Nikhil Gupta and Praveen Pachauri “An Experimental and Computational Investigation of Crack Growth Initiation in Compact Tension (CT)Specimen” ,pp 365-380, volume-2 , issue 8 , August 2012