Fracture mechanics comparison studies on an edge cracked specimen and compact tension specimen using ABAQUS

Vishwas B Patil
Machine Design
B.M.S. College of Engineering
Bengaluru, India
vishwasb.mmd19@bmsce.ac.in

Dr. H. K. Rangavittal
Dept of Mechanical Engineering
B.M.S. College of Engineering
Bengaluru, India
hkrv.mech@bmsce.ac.in

Kiran Sundara Rajan (Supervisor)
CAE Team
Molex India Business Services Pvt. Ltd
Bengaluru, India
Kiran.SundaraRajan@molex.com

***Abstract*— The present research paper focusses on the stationary crack simulation of an edge cracked specimen and compact tension specimen to obtain the stress intensity factor, energy release rate and the stress field around the crack tip for both 2D and 3D cases. The simulation is carried out on stainless steel considering only the elastic properties. The implementation of both conventional finite element method and extended finite element method is carried out and the results are compared. Extended finite element method is being used predominantly in the field of design engineering to reduce time for the creation of mesh specific to the crack geometry. The aim of this paper is to give helpful insights into the advantages of using XFEM and to arrive at a conclusion for the accuracy and CPU time for the simulation results.**

Keywords— Crack, Stress intensity factor, J integral, Conventional Finite Element Method, Extended Finite Element Method

I. INTRODUCTION

Fracture mechanics is a damage tolerant approach implemented to understand the structural integrity of the component having a flaw. These flaws also called as cracks can be existing in the component due to manufacturing defect or initiated during the service under the influence of loading. The assessment of the component with the crack becomes essential to increase the service life of the component and to save material costs. The crack can be subjecting to loading under three different modes [1] as shown in the Fig 1.

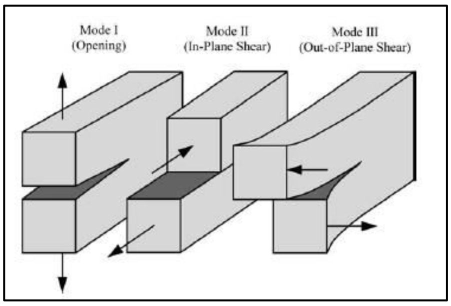

Fig. 1. Different modes of fracture

The stress, strain and displacement fields near the crack tip are cumulatively represented by a parameter called as Stress Intensity Factor (SIF). When the Stress intensity factor crosses the threshold value called Fracture Toughness (Critical Stress Intensity Factor) which is a material property, the crack starts to propagate. The stress field around the crack tip is asymptotic i.e. theoretically the stress is infinite at the crack tip. Stress intensity factor can be used to evaluate the structural integrity of the component under the given loading and boundary conditions. The critical stress intensity factor for metals can be obtained from the test mentioned in ASTM E399 standard. Another parameter Energy Release Rate (ERR) is used to estimate the energy required for the crack faces to separate and crack growth to occur.

ABAQUS software provides both Conventional Finite Element Method (CFEM) and Extended Finite Element Method (XFEM) for the evaluation of Stress Intensity Factor and Energy Release Rate for different modes.

It is observed that CFEM procedure becomes cumbersome and time consuming to setup the preprocessing model. This situation is mitigated by XFEM. The theoretical background behind the two methods is explored in the next few sections.

II. THEORITICAL BACKGROUND

A. *Conventional Finite Element Method(CFEM)*

A crack existing in a component has singularity at the crack tip i.e. the stress and the strain values are infinite in theory. However, in order to capture the same, the type of elements and mesh used around the crack tip is changed. In the case of two dimensional(2D) problems, a degenerated tria element from a quadrilateral element and for 3D problems, a degenerated wedge element from a hexahedral element is used as shown in Fig 2

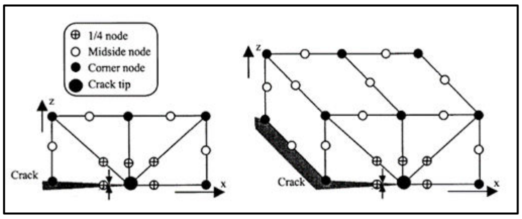

Fig 2. Degenerated elements in 2D and 3D [2]

A spider web mesh shown in the Fig 3 shows correct depiction of the mesh needed to obtain accurate results [3]

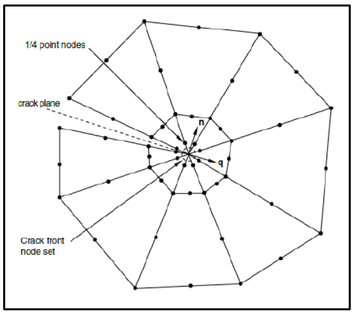

Fig 3. Spider-web mesh around the crack tip

This type of mesh can be used to create the singularity in small strain analysis given by the equations in the following three cases [3]

$$\epsilon \ \alpha \ \frac{1}{\sqrt{r}} \quad (1)$$

$$\epsilon \ \alpha \ \frac{1}{r} \quad (2)$$

$$\epsilon \ \alpha \ \frac{1}{r^{\frac{n}{n+1}}} \quad (3)$$

Eqn 1, 2 and 3 represent linear elastic, elastic plastic and power law hardening material behavior. For linear elastic case as described in the Eqn 1, the mid side nodes of the elements are moved to 25% distance from the tip as depicted in Fig 2.

This approach consumes more time for complex geometries which serves as a disadvantage for FEA Engineers since the simulations are governed by strict deadlines to meet a particular requirement. Hence XFEM helps in resolving the issues faced by Conventional Finite Element Method.

B. *Extended Finite Element Method (XFEM)*

Conventional finite element method requires the mesh to align with the crack tip and to have sufficient refinement in order to conform to the required crack geometry. This leads to further difficulties for the case of crack propagation which requires continual updating of the mesh. XFEM is a robust method to simulate both stationary cracks and crack propagation. It provides special enriched functions along with additional degrees of freedom to the nodes around the discontinuities by making use of concept of partition of unity(Eqn 4)[4]. A sufficiently coarse or fine mesh based on the requirement of the simulation can be used for different crack size and location [4]

$$u = \sum_{I=1}^{N} N_I(x)[u_I + H(x)a_I + \sum_{\alpha=1}^{4} F_\alpha(x)b_I^\alpha] \qquad (4)$$

$N_I(x)$ is the nodal shape function and u_I is the displacement vector used in conventional FEM and is implemented on all the available nodes in the mesh. $H(x)$ is the enrichment function used for the nodes across the crack faces. a_I and b_I^α are the nodal enriched degree of freedom vectors, the former being for the nodes associated with elements cut by the crack interior and later for nodes at the crack tip. F_α is the function responsible for inducing asymptotic behavior at the crack-tip.

A general example of enrichment in XFEM is shown below

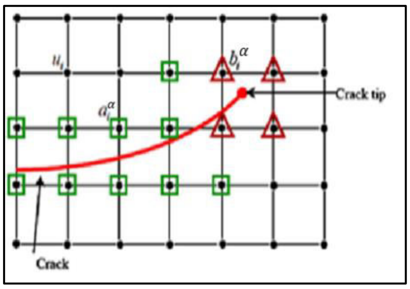

Fig 4. Enrichment in the mesh [4]

C. *Stress Intensity Factor(SIF)*

Stress intensity factor is a measure of the crack tip driving force. When SIF reaches an extreme value exceeding the fracture toughness of the material, it would lead to unstable crack growth and fracture of the component.

The equations for stress intensity factor for the edge cracked specimen and compact tension specimen [5] shown in the Fig 4 and 5 are given below

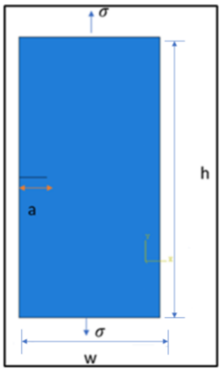

Fig 5. Edge cracked specimen

Mode I stress intensity factor for edge cracked specimen is given by

$$K_I = \sigma\sqrt{\pi a}\, f\left(\frac{a}{w}\right) \quad (5)$$

$$f\left(\frac{a}{w}\right) = 1.122 - 0.231\left(\frac{a}{w}\right) + 10.55\left(\frac{a}{w}\right)^2 - 21.710\left(\frac{a}{w}\right)^3 + 30.382\left(\frac{a}{w}\right)^4 \quad (6)$$

Where ‘a’ is the crack length, ‘w’ is the width and ‘σ’ is the stress applied

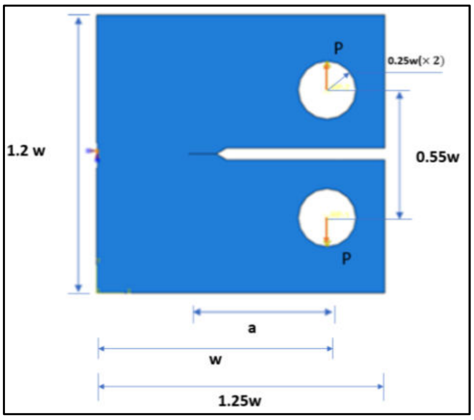

Fig 6. Compact tension specimen specimen

Mode I stress intensity factor for compact tension specimen is given by

$$K_I = \frac{P}{\sqrt{B}\,w} f\left(\frac{a}{w}\right) \tag{7}$$

$$f\left(\frac{a}{w}\right) = \frac{\left(2+\frac{a}{w}\right)\left(0.886+4.64\frac{a}{w}-13.32\left(\frac{a}{w}\right)^2+14.72\left(\frac{a}{w}\right)^3-5.6\left(\frac{a}{w}\right)^4\right)}{\left(1-\frac{a}{w}\right)^{1.5}} \tag{8}$$

where ‘P’ is the load applied, ‘w’ is width till the center of the through holes and ‘a’ is the crack length

D. *J- integral*

J- integral is the parameter used to determine the behavior of elastic-plastic materials which characterizes the energy release required for crack growth. It is essentially the integral value of a path around the crack tip given in Fig 7. However, for linear elastic material, it becomes equal to energy release rate (J=G) [6]

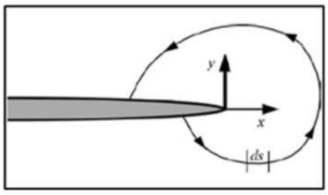

Fig 7. Contour around the crack tip[6]

J- integral is given by the formula

$$J = \int \left(w dy - T_i \frac{\partial u_i}{\partial x} ds\right) \tag{9}$$

Where
w = strain energy density
T_i = traction vector
u_i = displacement vector
ds = small increment length around the path

The energy release rate(G) in plane stress and plane strain is given by the Eqn 10 and 11

$$G = \frac{K_I^2}{E} \tag{10}$$

$$G = \frac{K_I^2}{E}(1-\vartheta^2) \tag{11}$$

Where
K_I = mode I stress intensity factor
ϑ = Poisson’s ratio
E = young’s modulus

E. *Stress Field*

The stress field near the crack tip is asymptotic in nature. Theoretically the stress is infinite at the crack tip. The value of stress near the crack (Fig 8) is given by the following equations [6]

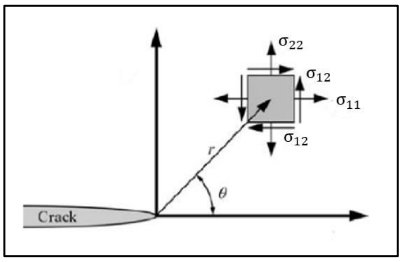

Fig 8. Stress field near the crack tip[6]

$$\sigma_{11} = \frac{K_I}{\sqrt{2\pi r}}\cos\frac{\theta}{2}\left(1-\sin\frac{\theta}{2}\sin\frac{3\theta}{2}\right) \tag{12}$$

$$\sigma_{22} = \frac{K_I}{\sqrt{2\pi r}}\cos\frac{\theta}{2}\left(1+\sin\frac{\theta}{2}\sin\frac{3\theta}{2}\right) \tag{13}$$

$$\sigma_{12} = \frac{K_I}{\sqrt{2\pi r}}\cos\frac{\theta}{2}\sin\frac{\theta}{2}\sin\frac{3\theta}{2} \tag{14}$$

III. FINITE ELEMENT ANALYSIS

Finite element analysis is performed on the edge crack specimen and compact tension specimen. The analysis is restricted to only stationary crack. Stress intensity factor, energy release rate and variation of stress near the crack tip are obtained from FEA and compared with the analytical values. The evaluation of stress intensity factor and energy release rate has been done in [7] for a semi- elliptical crack by comparing both CFEM and XFEM. A fine mesh implemented in XFEM throughout the geometry would lead to more number of nodes and greater computational time [8]. In the present paper, the mesh size is kept as 1mm near the crack region and 3mm away from it, which is uniform for all the components and provides better insights for the comparison.

The stress of 100MPa is applied on edge cracked specimen and 200N applied on compact tension specimen.

A. Dimensions

The dimensions of edge crack specimen are:

- crack length (a) = 10 mm
- width(w) = 50 mm
- height(h)= 100mm
- thickness(t) = 1mm

The dimensions of compact tension specimen are:

- crack length (a) = 30 mm
- width(w) = 50 mm
- thickness(t) = 1mm (2D) and 25mm (3D)

B. Material properties[9]

Only elastic properties have been considered for the analysis

- E (Young's modulus) = 170 GPa
- ϑ(Poisson's ratio) = 0.305

C. Type of mesh

The FEA has been carried out on stationary crack for both 2D and 3D cases.

For 2D cases, second order Plane Stress Elements (CPS8) are used in CFEM and first order hexahedral elements(C3D8) in Extended Finite Element Method. In XFEM 1,2 and 3 layers of elements are used across the thickness and the results are compared. Currently ABAQUS XFEM supports only 1^{st} and 2^{nd} order tetra elements and 1^{st} order brick elements for obtaining SIF and ERR [3]

For 3D cases, second order solid elements(C3D20) are used in CFEM and first order hexahedral elements(C3D8), first order tetra elements (C3D4) and second order tetra elements(C3D10) in XFEM

For mesh created in CFEM, it is observed that degenerated elements have been used as explained in section II.

The mesh used in Conventional Finite Element Method (CFEM) and Extended Finite element method (XFEM) for 2D and 3D are shown in the Figures 9 -15

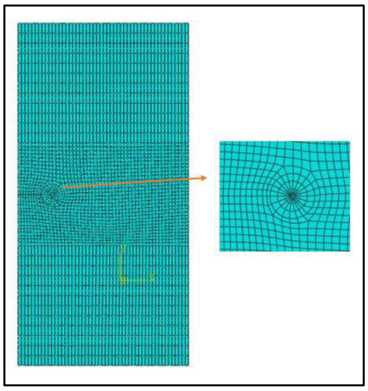

Fig 9. Edge crack specimen mesh (CFEM)

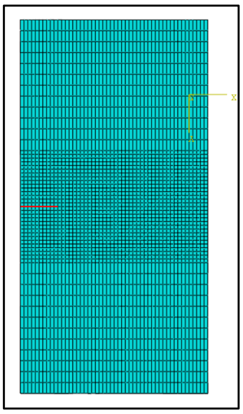

Fig 10. Edge crack specimen mesh(XFEM)

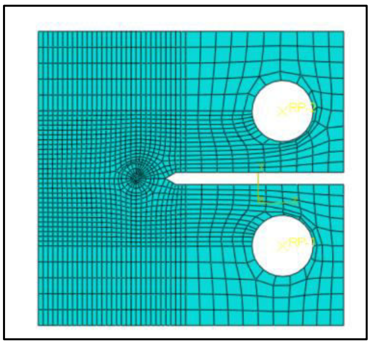

Fig 11. 2D CT specimen mesh (CFEM)

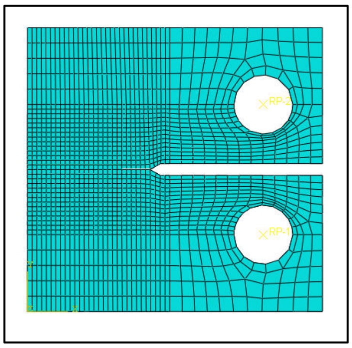

Fig 12. 2D CT specimen mesh (XFEM)

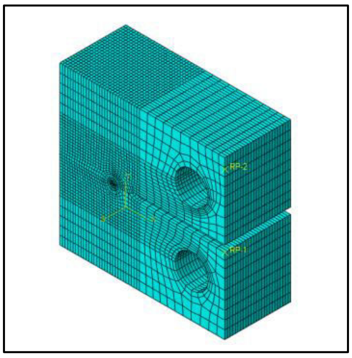

Fig 13. 3D CT specimen mesh (CFEM)

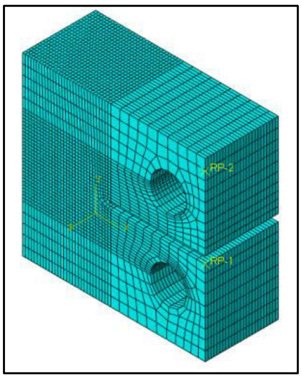

Fig 14. 3D CT specimen mesh (XFEM) with hexahedral mesh

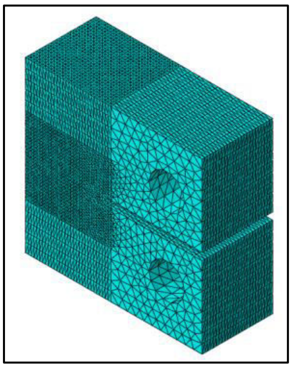

Fig 15. 3D CT specimen mesh (XFEM) with tetrahedral mesh

For XFEM, a separate crack part is modeled and assembled to the geometry as explained in [10]

D. Results and discussion

a) Stress intensity factor and J integral Comparison

Stress intensity factor (SIF) and energy release rate(ERR) obtained from analytical formulae given in Eqn 5-8 and 10-11 are given in Table I

TABLE I : SIF(Analytical values)

	***SIF*(*MPa*$\sqrt{mm}$)**	***ERR*(*KJ*/m^2)**
2D EC	48.59	13.90
2D CT	12.21	0.878
3D CT	0.4885	1.28×10^{-3}

For 2D edge cracked specimen, stress intensity factor and energy release rate obtained from FEA compared with analytical results along with the CPU time is given in Table lII

TABLE II : SIF (2D Edge crack specimen)

	SIF(FEA) (MPa $\sqrt{mm}$)	*% Error*	*ERR(KJ/ m^2)*	*%Error*	*No of nodes*	*CPU time(sec)*
CFEM	48.44	0.3	13.80	0.7	8579	3.5
XFEM 1 layer	49.60	2	13.87	0.2	5610	15.4
XFEM 2 layer	50.0	2.9	13.95	0.35	8415	19.5
XFEM 3 layer	50.36	3.51	13.99	0.65	11220	25.7

CFEM results are nearer to the analytical values compared to XFEM results. It is observed that, in XFEM the percentage error increases for SIF with the increase in the no. of layers. This is due to evaluation of these parameters on the nodes across the thickness as observed in [11]. Excluding the values of SIF at the ends of the thickness for the calculation is suggested, to reduce the % error and get more accurate depiction of the crack. However, in the present paper, even including those values has not increased % error beyond 3-4%.The CPU time increases for the XFEM simulations due to the consideration of enrichment functions.

For 2D compact tension specimen, stress intensity factor and energy release rate obtained from FEA compared with analytical results along with the CPU time is given in Table III

TABLE III : SIF (2D Compact tension specimen)

	SIF(FEA) (MPa $\sqrt{mm}$)	*% Error*	*ERR(KJ/ m^2)*	*%Error*	*No of nodes*	*CPU time(sec)*
CFEM	12.17	0.32	0.87	1.13	4857	4.2
XFEM 1 layer	12.55	2.7	0.90	2.27	3414	5.5
XFEM 2 layer	12.67	3.7	0.89	1.14	5121	7.6
XFEM 3 layer	12.72	4.17	0.90	2.27	6828	8.7

Results for 2D Compact tension specimen are similar to edge crack specimen. The CPU time increases with increasing layers of elements across the thickness.

The SIF values can be obtained accurately with more no. of layers and excluding the SIF values at the ends of the thickness. However, this procedure should be restricted to 2D cases where the thickness is very small.

For 3D compact tension specimen, stress intensity factor and energy release rate obtained from FEA compared with analytical results along with the CPU time is given in Table IV

TABLE IV : SIF (3D Compact tension specimen)

	SIF(FEA) $(MPa\sqrt{mm})$	*%Error*	*ERR(KJ/* m^2*)*	*%Error*	*No of Nodes*	*CP time*
CFEM	0.49	2.08	1.31×10^{-3}	2.34	169617	188
XFEM-Hex elements	0.5028	2.92	1.35×10^{-3}	3.05	42276	78.3
XFEM-1st order tetra elements	0.5091	4.22	1.36×10^{-3}	6.25	42625	118
XFEM-2nd order Tetra elements	0.478	2.149	1.34×10^{-3}	4.68	328397	990

Hexahedral elements have been used in [12] for the case of crack propagation. However, there was a need to obtain SIF for standard specimens having mesh with same element size which would give better understanding of the accuracy. From Table IV, it is evident that CFEM and XFEM (hexahedral elements and 2nd order tetra elements) provide similar results for SIF but with more no of nodes leading to higher computational time.

b) Variation of stress near the crack tip

The stress intensity factor and energy release rate give an assessment of the critical condition of the crack. However, the stress field around the crack tip gives a better understanding of the local region around the crack tip.

The variation of stress has been studied in [13] and [14] for a center cracked plate. However, the mesh size considered in both the cases is very small compared to that used in the applications and implemented only in 2D cases. The present paper gives details into 2D and 3D cases, considering tetra hedral mesh as well, which is not observed in many cases in the literature of stationary crack.

The present paper considers evaluation of stress component given by Eqn 13 along the path parallel (0 Deg) and perpendicular (90 Deg) to the crack.

Variation of stress in edge cracked specimen is given in Fig 16 and 17.

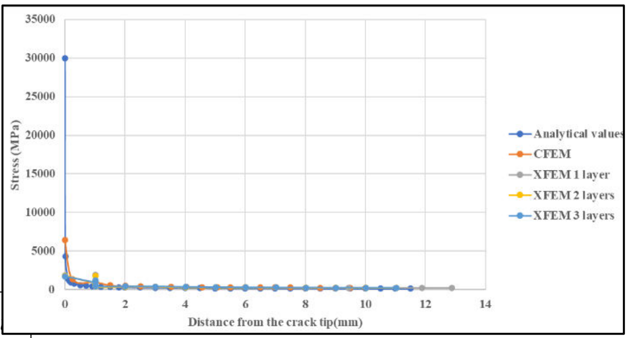

Fig 16. Stress in edge crack specimen parallel to crack

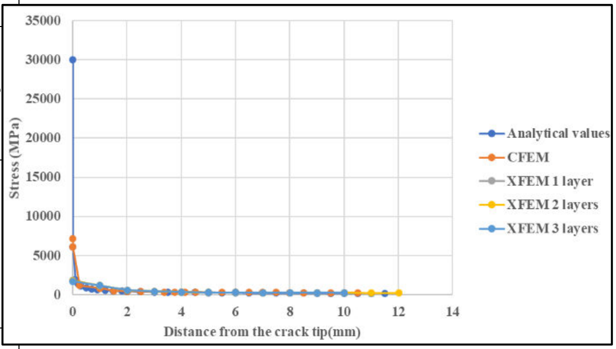

Fig 17. Stress in edge crack specimen perpendicular to crack

Variation of stress in 2D Compact tension specimen is given in Fig 18 and 19

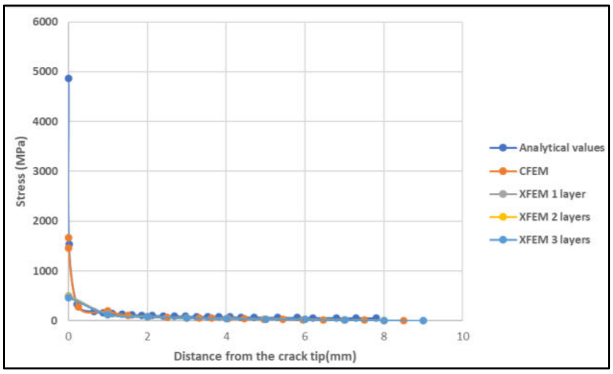

Fig 18. Stress in 2D compact tension specimen parallel to crack

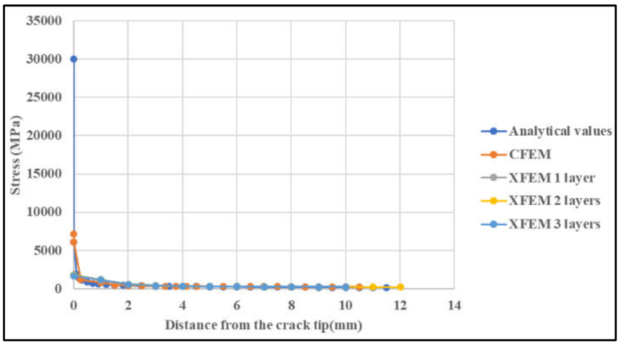

Fig 19. Stress in 2D compact tension specimen perpendicular to crack

Variation of stress in 3D Compact tension specimen is given in Fig 20 and 21

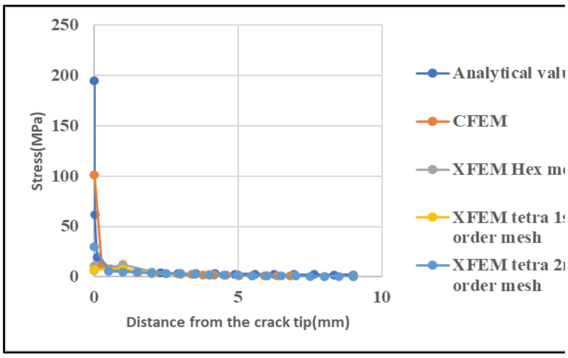

Fig 20. Stress in 3D compact tension specimen parallel to crack

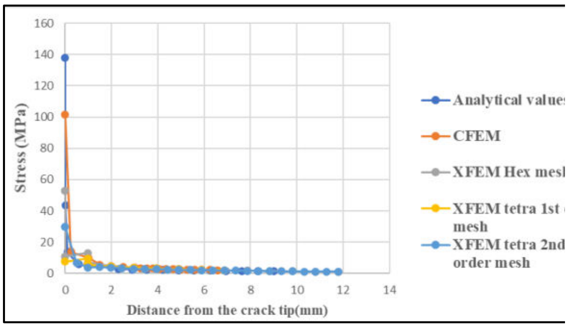

Fig 21. Stress in 3D compact tension specimen perpendicular to crack

Mesh used in CFEM has 2nd order elements and due to the degenerated elements used to create crack tip singularity, the stress values are captured accurately. For the same mesh element size, XFEM captures the stress with less accuracy than CFEM, however after certain distance from the crack tip, all the values are in good agreement

Fluctuations in the values of stress at certain distance are observed due to the averaging of stresses carried out by ABAQUS software postprocessing module. This can be reduced with a finer mesh but at a higher computational cost[14]

In both CFEM and XFEM cases, a finer mesh will lead to better stress capture around the crack tip.

IV CONCLUSION

The present paper provides useful insights into the implementation of CFEM and XFEM to obtain stress intensity factor, energy release rate and stress field.

It can be concluded that for Plane stress cases XFEM can be used to for the simulation. The no. of nodes and CPU time should be taken into consideration during the simulation.

For 3D geometries, it is observed that XFEM provides accurate results similar to CFEM. For complex geometries in engineering applications, tetrahedral meshes are used, since creation of such mesh is less time consuming. However, there will be large increase in the computational time which needs to be taken into account. It should be noted that the preprocessing time for CFEM is more compared to XFEM. A suitable approach should be implemented based on the complexity of the geometry and time constraint.

This work can be further progressed to

- Performing simulations with different crack lengths for both the specimens in 2D and 3D cases
- Performing simulations for different thickness and crack lengths for 3D cases

In the above-mentioned cases, the mesh size around the crack tip should be kept the same which is ideal for comparison

ACKNOWLEDGMENT

The authors would like to thank the support rendered by the Principal, BMSCE, the Head of the department Mechanical Engineering, BMSCE and CAE Team at Molex India Business Services Pvt. Ltd

REFERENCES

[1] T.L Anderson, " History and overview," in Fracture Mechanics Fundamentals and Applications,3rd edition..Taylor and Francis Group. Boca Raton .2017

[2] S. Courtin, C. Gardin, G. Bézine, H. Ben Hadj Hamouda, "Advantages of the J-integral approach for calculating stress intensity factors when using the commercial finite element software ABAQUS,"Engineering Fracture Mechanics,Vol 72, Issue 14,pp 2174-2185.2005

[3] ABAQUS 6-14.1,SIMULIA,Providence,USA,2014

[4] Lee, K.L. & Martin, David. "Application of XFEM to Model Stationary Crack and Crack Propagation for Pressure Containing Subsea Equipment." Proceedings of the ASME Pressure Vessels and Piping Conference, Vol 5. 2016.

[5] Tada, H., P. Paris and G. Irwin. "The Stress Analysis of Cracks Handbook", Third Edition. ASME .New York.2000.

[6] T.L Anderson, " Linear Elastic Fracture Mechanics," in Fracture Mechanics Fundamentals and Applications,3rd edition.Boca Raton.Taylor and Francis Group.2017

[7] Fakkoussi, S.E., Moustabchir, H., Elkhalfi, A., & Pruncu, C. "Computation of the stress intensity factor KI for external longitudinal semi-elliptic cracks in the pipelines by FEM and XFEM methods". International Journal on Interactive Design and Manufacturing (IJIDeM),Vol 13, pp 545-555. 2019.

[8] Singh, Krishna & Keswani, Kamal & Vaggar, Mallikarjun. "Crack growth simulation of stiffened fuselage panels using XFEM techniques". Indian Journal of Engineering and Materials Sciences, Vol 21. pp 418-428. 2014.

[9] ASM METALS HANDBOOK, ASM International Vol 2.1990.

[10] Doiphode, Ganesh.Shah,Pooja, "Study of fracture based parameters on ct specimen with thickness variation".IJAERD,Vol 6.Issue 1.Feb 2019

[11] Shim, D, Uddin, M, Kalyanam, S, Brust, F, & Young, B. "Application of Extended Finite Element Method (XFEM) to Stress Intensity Factor Calculations." Proceedings of the ASME Pressure Vessels and Piping Conference: Materials and Fabrication. Boston, Massachusetts, USA. Vol 6A .July 19–23, 2015.

[12] Petrasinovic, Danilo & Rasuo, Bosko & Petrasinovic, Nikola. "Extended finite element method (XFEM) applied to aircraft duralumin spar fatigue life estimation". Tehnicki Vjesnik, Vol 19. pp 557-562.2012

[13] Živković, Jelena & Jovicic, Gordana & Živković, Miroslav & Slavkovic, Radovan, "Accuracy analysis of extended finite element method for tension of plate with a central crack".IMEKO TC15- Experimental Mechanics, Proceedings of the 11th Youth Symposium on Experimental Solid Mechanics.pp 96-101. 2012

[14] Gigliotti, Luigi, " Assessment of the applicability of XFEM in Abaqus for modeling crack growth in rubber",M.S. thesis,Dept.Solid Mechanics.,Royal Institute of Technology,Stockholm,Sweden,2012