International Journal of Enhanced Research in Science, Technology & Engineering
ISSN: 2319-7463, Vol. 12 Issue 8, August-2023, Impact Factor: 7.957

Mesh convergence study of high pressure die cast CT specimen in mode-I through simulation modeling

Chitragupt Swaroop Chitransh[1*], Sanjeev Saxena[2]

[1]IMS Engineering College, Ghaziabad-201015, India
[2] CSIR-Advanced Materials and Processes Research Institute, Bhopal- 462026, India

Corresponding Author: Chitragupt Swaroop Chitransh, IMS Engineering College, Ghaziabad-201015, India, *E-mail address*: chitranshnitbhopal@gmail.com

---

ABSTRACT

The mesh convergence study aims to present the significance of mesh convergence while analyzing compact tension models. A convergence study is performed to define the relationship between the numbers of elements and the analysis accuracy. Mesh quality and its density always affect the solution accuracy, so it is necessary to optimize the mesh quality and its density to find out the optimal solutions. A mesh convergence study is based on a CT model followed by the ASTM E-1820 standard. The generated meshes are used to analyses the CT model in thickness, radial, and circumferential direction and to find out the best combination of meshes by evaluating *J*-integral with the total number of elements. Analysis results are compared to propose and conclude the outcomes for the identification of best mesh combinations and to ensure that the results of the finite element analysis will be optimum and effective with this mesh combination. This paper is an attempt to provide the best meshing combination that can be used in future work to analyze CT models followed by ASTM standard at a reasonable computational cost.

Keywords: Mesh convergence analysis, finite element analysis, simulation modeling, CT model.

Nomenclature

FEA	finite element analysis
FEM	finite element method
C3D8R	linear reduced integrated element
CT specimen	compact tension specimen
CT model	compact tension model
ASTM	American society of testing and materials
HPDC	high pressures die casting
LLDs	load line displacements
MMC	metal matrix composite
ADC	aluminium die casting
J- Integral	contour path *J*-integral value

INTRODUCTION

FEM based study is widely adopted due to its advantages over the traditional analytical approach. Also there are many complex problems that cannot be solved by analytical methods but can be easily solved by finite element analysis [1]. In finite element modeling, for accurate solutions a finer mesh is generally preferred. To get the right mesh it is necessary to balance accuracy and computational resources, which can be achieved by proper mesh convergence study [2].

Two types of refinements are generally used in finite element analysis: one is for the reduction in element sizes and the second is for increasing the order of elements. Sometimes, it is very important to differentiate between the geometric effect and mesh convergence. Particularly when meshing a surface near the crack tip using linear element, require more elements to capture the boundary exactly [3]. The most critical point in the simulation is the choice of the minimum element size in the area of the crack tip. Smaller elements make it possible, but at a higher computational cost [4].

In the case of the CT model, one of the most critical issues is the choice of mesh near the crack tip and the minimum element size close to the crack tip. The use of smaller elements increases the number of elements of the model and the number of cycles needed to reach the desired final crack length [5], but in our work cycle time is comparatively less. The choice of element type and the level of refinement of the element mesh are the most fundamental to construct an accurate finite element model. The mesh refinement requirements are necessary to achieve accurate results especially in high plastic strain regions [6]. It should be strictly applied otherwise elements will be distorted in those areas and it has been considered in our present work. However, if the mesh used is too coarse, inaccurate results may be obtained. If the minimum required mesh refinement level when modeling plasticity-induced closure is unclear, then meshing clarity should be necessary before going for analysis [7].

For the calculation of J-integral the nodal forces of crack tip elements should be reduced gradually to avoid numerical oscillation and non-convergence. It is necessary to understand this at the time of analysis otherwise, it affects adversely by increasing the time of analysis [8]. The calculations at regions where fine mesh are used provide more accurate results as compared to coarse mesh region but there is two-division of boundary conditions in this work [9]. In our work a single symmetric boundary condition has been used with fine mesh.

Numerical instability occurs due to several modifications in boundary conditions. A reasonable compromise seems to be the damage controlled deactivation of elements [10]. This type of analysis gives results with non-converged mesh. In our work there is no need to do so.

The iterative method has been suggested in the reference [11] to see the vertical deflections according to mesh size. In the present work the same approach has been used to see the vertical deflections between J-integral and the total number of elements.

Ch.V.Sushma et. al (2013) [12] performed convergence analysis for the various loading conditions to see the variations of J-integral corresponding to load line displacements. But in the present study only a single loading condition has been used and for convergence, J-integral vs. total numbers of elements have been plotted to reduce the computational cost.
There is a suggestion drawn for mesh convergence in reference [13] and [14] to emphasize the significance of mesh size, local mesh, and singularity but how to predict crack is nowhere mentioned. In the present work all these assumptions have been considered along with the crack while doing convergence study.

To maximize the accuracy in results and to reduce analysis time, it is necessary to find out optimum mesh combination while performing FEA analysis that leads to low computational cost. In the present work all the above statements are kept in mind at the time of convergence study. In convergence study mesh combinations have been used in radial, circumferential, and in the thickness direction to find out the optimal mesh combinations. Meshes are converged at 38540 hexahedral C3D8R linear elements and the total numbers of nodes used in the converged mesh are 42336. Mesh convergence study ensures that to obtain an accurate solution with a mesh that is sufficiently dense and not overly demanding of computing resources. These convergence studies will also aid reviewers in evaluating the quality of a finite element model and the accuracy of results.

EXPERIMENTAL TESTED DATA

In the present study to fabricate tensile samples, HPDC ADC-12 MMC has been processed through stir casting process. In this process ADC-12 alloy was heated upto elevated temperature to melt it properly. For grain refinements and to reduce slag formation, TiB_2 rods and flux has been used. SiC particles are poured into the melt and stirrer was used to disperse particles uniformly. This process will be continued until the ADC-12 alloy completely dispersed with reinforcement. Further the melt has been taken into 400 tonnes high pressure die casting and processed into HPDC material. Tensile samples with 25mm gauge length and 6mm gauge width and overall length of 64 mm has been fabricated and tested it on Instron machine to determine stress strain curve at a strain rate of 10^{-3}/s using ASTM standard code E8.

FEM MODELING

Three-dimensional geometry with 12.5mm thickness and 0.5 a/W ratio following the ASTM-1820 standard [15] has been modeled using Abaqus software. FEM modeling can be divided into pre-processing, analysis, and post-processing. Pre-processing includes modeling of CT specimens and applying boundary conditions like constraints, symmetry conditions, and loads. The CT specimen is considered as an isotropic three-dimensional model. The assigned material property as an input is high pressure die-cast metal matrix composite material property determined through tensile test data. The elastic modulus of the assigned material is 70 GPa and Poisson's ratio is 0.3. The offset yield stress of the material is 180 MPa. Taking advantage of symmetry only half of the CT model is taken for the study and Mode-I has been used for analysis under this study. The Y-symmetry boundary condition is assigned and 1mm displacement applied at the loading point. C3D8R type hexahedral linear elements have been used for the meshing of CT model. The C3D8R element is a general-purpose linear brick element with reduced integration, also known by 8-noded elements. In these types of elements stresses, strains are most accurate in the integration points. The integration point of the C3D8R element is located in the middle of the element. Thus, small elements are required to capture a stress concentration at the boundary of a structure. The element has plasticity, creep, swelling, stress stiffening, large deflection, and large strain capabilities. Meshing combinations have been used in radial, circumferential, and thickness direction to find out the optimum solution. The converged CT meshed model is shown in figure 1. The meshed models have been submitted for the analysis with different combinations. In post-processing *J* values are determined with the respective number of elements to identify the optimum solution. The Job module allows creating a job, to submit it for analysis, and to monitor its progress. Results will display on the screen for computational values of *J*-integral at the different numbers of contours. It can be saved in a file to see the results multiple times without any delay.

CONCEPT BEHIND MESH CONVERGENCE STUDY

A mesh convergence study generally verifies that the FEA model has converged for a solution or not. It also provides the information for mesh independence and additional refinement is not necessary after a particular point. Mesh convergence study determines how many elements are required in a model to ensure that the results of an analysis are not affected by changing the size of the mesh. The formal method of mesh convergence requires a curve of a critical result parameter in a specific location, to be plotted against some measure of mesh density. In the present work this approach has been applied to plot curve between J-integral and the total number of mesh elements at 0.25mm, 0.50mm, 0.75mm and 1mm load line displacements. To perform convergence study various mesh combinations have been used in radial direction (i.e. 10 biasing with 4, 8, 10, 12, 15, 20, 25 elements), thickness direction (i.e. 5, 8, 10, 12, 15, 20 elements) and in circumferential direction (i.e. 16, 24, 32, 40 elements), to find out the best mesh combination. Iterative approach has been used with different mesh combinations and plotted graph between *J*-integral vs. total number of finite elements until the best mesh combination is achieved.

METHODOLOGY FOR MESH CONVERGENCE STUDY

In the present study to calculate *J*-integral, different mesh combinations have been used. Load line displacements (LLDs) are applied at the loading point and for different LLDs, *J*-integral values were determined with reference to total number of finite elements for different mesh combinations. The discretization of mesh and size of elements generally influence finite element solutions, especially when applies fracture mechanics approach to determine *J*-integral. In fracture mechanics problems, the mesh must be fine enough to capture the severe stress concentration at the crack tip. Extensive mesh refinements lead to large computational time and are responsible to increase process time. A balance between these two considerations is required to avoid the degradation in reliability of results. In the present investigation, following variations in finite element discretization has been applied to study the influence of element size and number of elements on the *J*-integral prediction.

- Variations in *J*-integral with element variation in thickness direction
- Variations in *J*-integral with element variation in radial direction
- Variations in *J*-integral with element variation in circumferential direction

Total 17 meshing variations have been used in the present study as shown in table 1.

Table.1 Variations considered in mesh convergence study

Model	Number of elements		
	Thickness elements	Radial (elements × biasing)	Circumferential elements
1	5	25 × 10	40
2	8	25 × 10	40
3	10	25 × 10	40
4	12	25 × 10	40
5	15	25 × 10	40
6	20	25 × 10	40
7	20	4 × 10	40
8	20	8 × 10	40
9	20	10 × 10	40
10	20	12 × 10	40
11	20	15 × 10	40
12	20	20 × 10	40
13	20	25 × 10	40
14	20	25 × 10	16
15	20	25 × 10	24
16	20	25 × 10	32
17	20	25 × 10	40

Variations in *J*-integral with element variation in thickness direction

Six different mesh combinations were used in this study, as shown in Fig.1,2, which shows the values of *J*-integral varying from 30.27 N/mm to 21.38 N/mm respectively at 9635 and 38540 mesh elements. Different iteration has been carried out by changing the number of elements i.e. 5, 8, 10, 12, 15, 20 elements in the thickness direction, and found that *J*-integral value decreases when the number of mesh elements increases. The meshing is converged at 38540 elements. Reduced *J*-integral value shows the finest mesh accuracy is much more than coarse mesh. The percentage variation in the *J*-integral value is about 30% with thickness direction mesh elements.

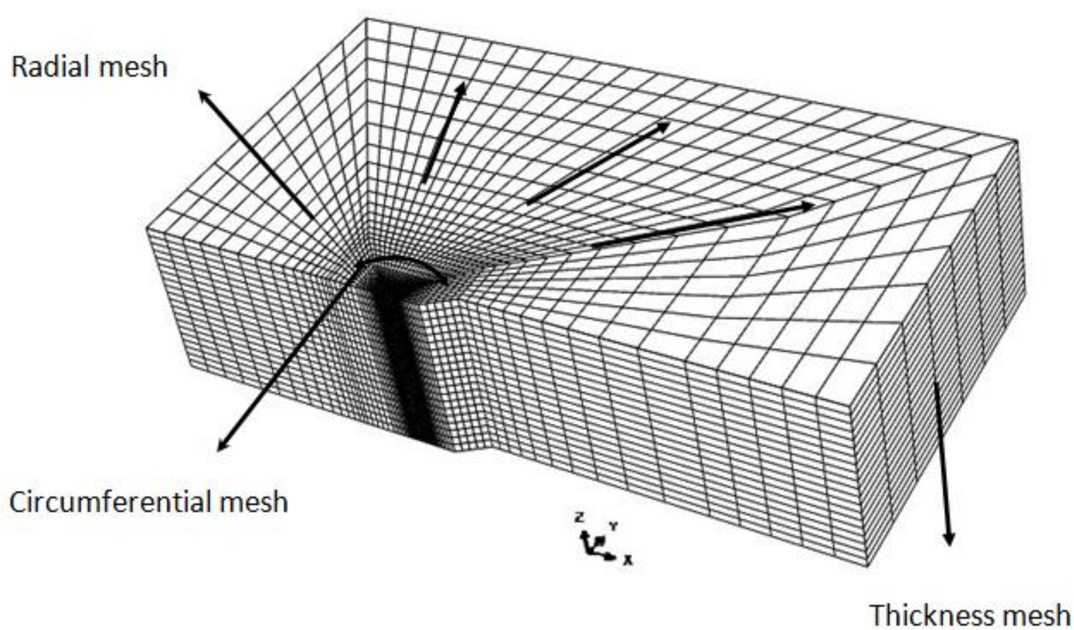

Fig 1. Half CT specimen with final mesh in thickness, radial & circumferential direction

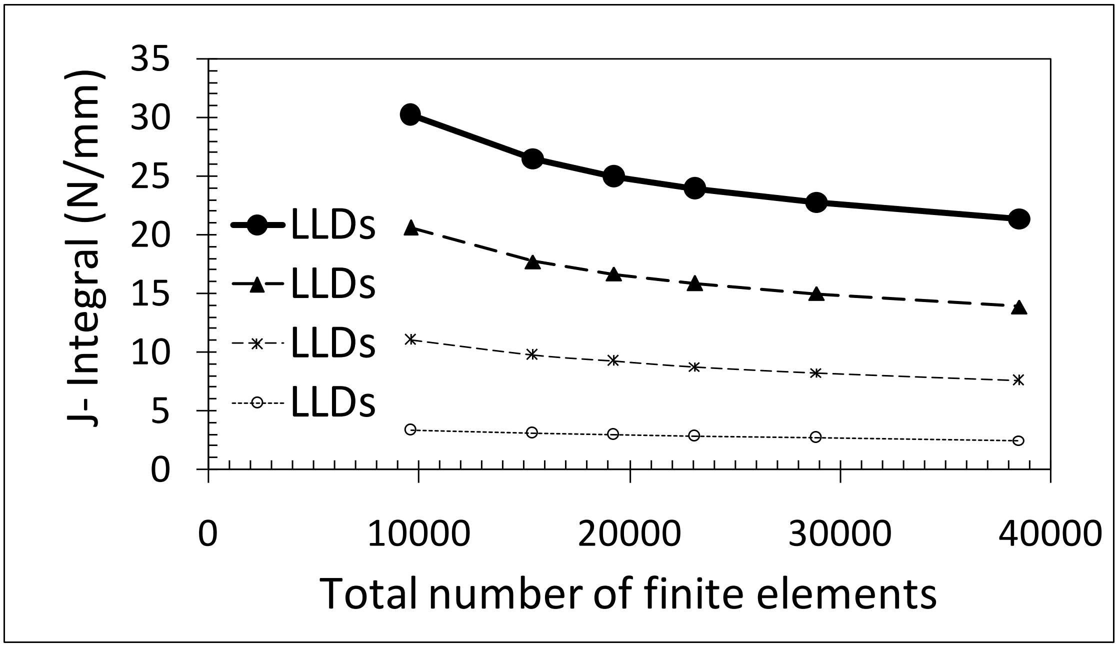

Fig.2 variation of J-integral with element variation in thickness direction

Variations in *J*-integral with element variation in radial direction

According to Fig. 3, which indicates the values of the J-integral ranging from 31.71 N/mm to 21.38 N/mm respectively at 24600 and 38540 mesh elements, seven different mesh combinations have been used. Different iteration has been carried out by changing the number of elements with fixed biasing in the radial direction i.e. 4, 8, 10, 12, 15, 20, and 25 elements with 10 biasing respectively. The maximum value of J-integral is seen when the number of mesh elements reached at 24600 and minimum *J*-integral value at 38540 elements. The meshing is converged at 38540 elements. It has been found by comparing fig. 3 & 4 that variations in *J*-integral value with radial direction mesh elements are maximum and minimum with circumferential direction mesh. The percentage variation in the *J*-integral value is about 33% with radial direction mesh elements.

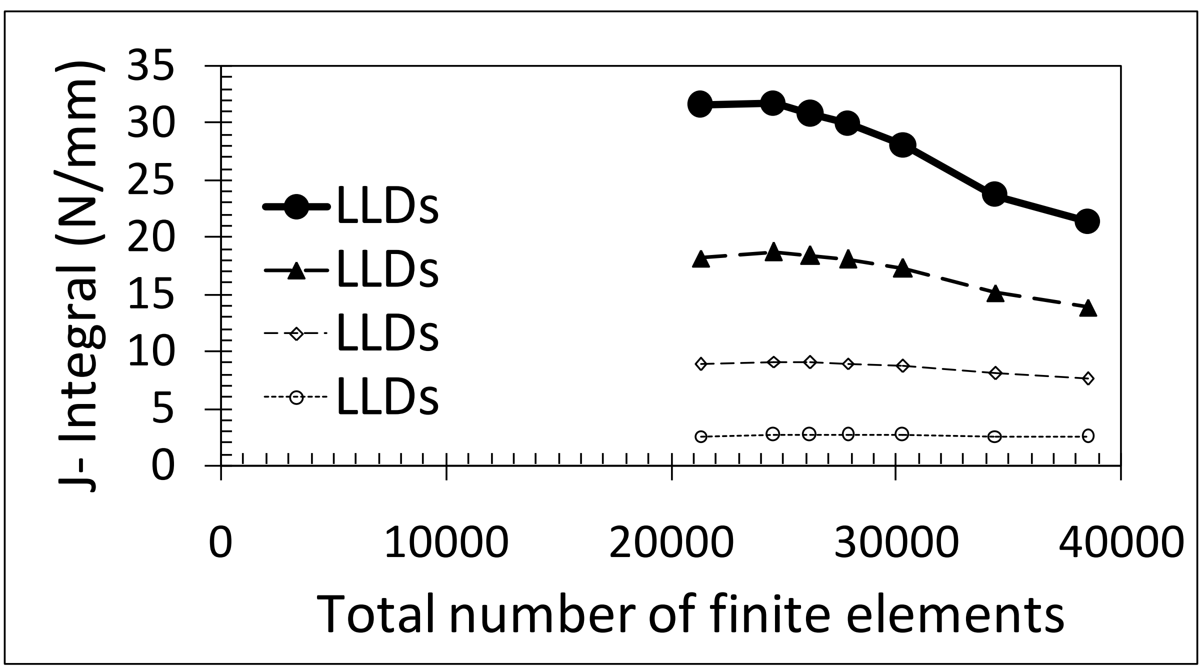

Fig.3 variation of J-integral with element variation in radial direction

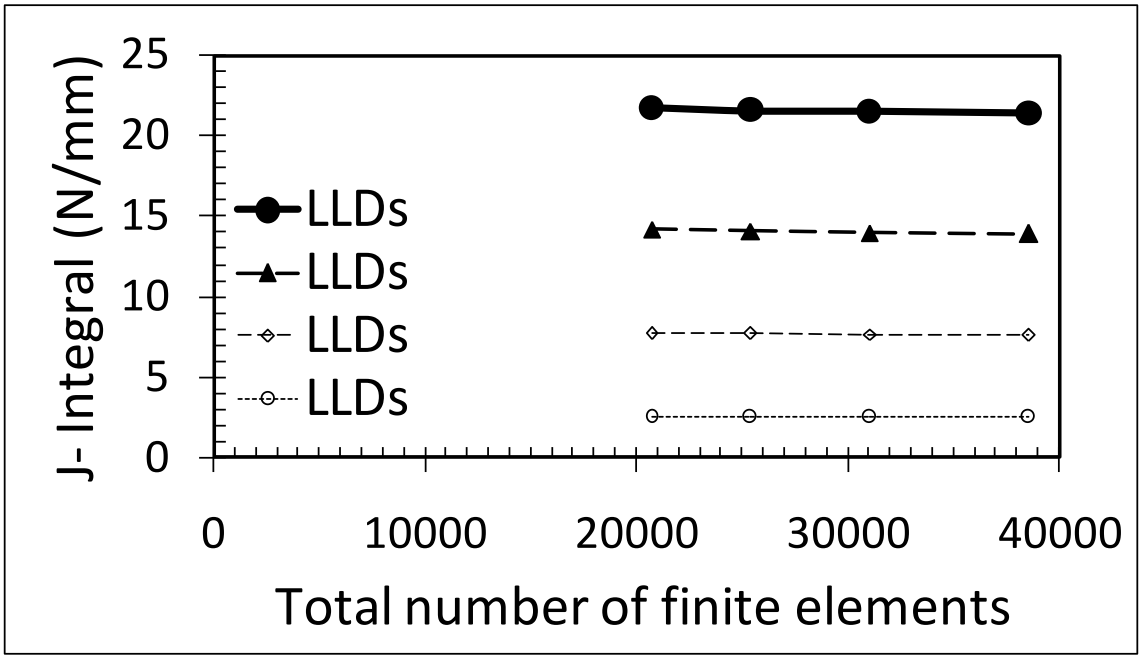

Fig.4 variation of J-integral with element variation in circumferential direction

Variations in *J*-integral with element variation in circumferential direction

In this study, seven different mesh combinations were used, as shown in Fig.4, which shows the values of J-integral varying from 21.69 N/mm to 21.38 N/mm respectively at 20680 and 38540 mesh elements. It is clear from fig. 2, 3 & 4 that the variation in *J*-integral value is very minor in the case of circumferential direction mesh. Different iteration has been carried out by changing the number of elements at circumferential direction i.e. 16, 24, 32 & 40 elements respectively**.** The meshing is converged at 38540 elements. It has also found that the variation in the *J*-integral is very less among all the incremental changes with circumferential mesh. The percentage variation in *J*-integral value is about 1.5 % only with circumferential direction mesh elements. It indicates that the radial and thickness direction mesh elements are having more weightage in convergence study.

RESULTS AND DISCUSSION

It is clear from figure 2, 3 & 4 that the values of *J*-integral are converged at 38540 C3D8R linear hexahedral linear elements with 42336 nodes. It is also remarkable from fig. 2, 3 & 4 that vertical deflection of elements concerning *J*-integral is maximum in case of radial direction mesh and minimum in circumferential direction mesh. *J*-integral value is higher for coarse mesh and minimum for the finest mesh which indicates that the value of *J*-integral decreases when the number of elements increases. It means the energy release rate per unit area is higher for coarse mesh and lowest for the fine mesh. The choice of element type and the level of mesh element refinement are successfully opted to construct an accurate finite element model. The mesh refinement requirements are completely satisfied in the present mesh convergence study.

CONCLUSIONS

Finally mesh convergence study is come out with an effective outcome. The mesh elements in thickness, radial, and circumferential direction are converged at 38540 elements and 42336 nodes. This combination is best for the CT model followed by the ASTM E-1820 standard to run an analysis with an adequate output without wasting any time at a reasonable computational cost.

REFERENCES

[1]. Hemesh Patil, P V Jeyakarthikeyan Mesh convergence study and estimation of discretization error of hub in the clutch disc with the integration of ANSYS. Materials Science and Engineering 2018 ; 402.

[2]. https://knowledge.autodesk.com/support/simulation-mechanical/learn-xplore/caas/sdf articles/How-to-perform-a-Mesh-Convergence-Study.html

[3]. https://www.simscale.com/blog/2017/01/convergence-finite-element-analysis.

[4]. A. Gonzalez Herrera, J. Zapatero, Influence of minimum element size determine to determine crack closure stress by the finite element method. Engineering Fracture Mechanics. 2005 ; 72: 337– 355.

[5]. D.Camas, J. Garcia-Manrique. Numerical modeling of three-dimensional fatigue crack closure: mesh refinement. International Journal of Fatigue, 2018 .

[6]. Chi-Fung Tso, David P. Molitoris. Mesh Convergence Studies For Hexahedral Elements Developed by the ASME Special Working Group On Computational Modeling. Proceedings of the ASME 2013 Pressure Vessels and Piping Conference, 2013 .

[7]. Kiran Solanki, S.R. Daniewicz, Finite element modeling of plasticity-induced crack closure with emphasis on geometry and mesh refinement effects Engineering Fracture Mechanics 70: 1475–1489. 2003

[8]. Cheng Yan, Yiu-Wing Mai Finite element analysis and experimental evaluation of ductile-brittle transition in compact tension specimen. International Journal of Fracture 87: 345–362. 1997

[9]. Y. Mohammed, Mohamed K. Hassan. Finite Element Computational Approach of Fracture Toughness in Composite Compact Tension Specimen. International Journal of Mechanical & Mechatronics Engineering IJMME-IJENS, 2017 .

[10]. Eberhard Altstadt, Matthias Werner, Institute of Safety Research, qucosa.de pdf.2003

[11]. https://www.xceed-eng.com/finite-element-analysis-convergence-and-mesh-independence/

[12]. Ch.V.Sushma, Dr. P. Ravinder Reddy “Investigation of Fracture Parameters of Compact Tension Specimen By FEA” International Journal of Engineering Research & Technology. 2013

[13]. https://www.nafems.org/publications/knowledge-base/the-importance-of-mesh-convergence -part-1/

[14]. https://www.simscale.com/blog/2017/01/convergence-finite-element-analysis/

[15]. ASTM E-1820. “Standard test method for measurement of fracture toughness” Annual book of ASTM standards metals test methods and analytical procedures.