International Journal of Advances in Engineering and Management (IJAEM)
Volume 5, Issue 9 Sep 2023, pp: 472-483 www.ijaem.net ISSN: 2395-5252

Fracture Mechanics Assessment of a Compact Tension Specimen Using Abaqus CAE

Ali Mustapha Alibe[1]*, A.A Janga[1], and I.M Alibe[3]

[1]*School of Engineering Technology, Department of Mechanical Engineering, Federal Polytechnic Damaturu, 620221, Yobe State, Nigeria*

[3]*Department of Mechanical Engineering, Nigerian Army University Biu, Borno State, Nigeria*

Fracture Mechanics Assessmentof a Compact Tension Specimen Using Abaqus CAE

Date of Submission: 20-09-2023 Date of Acceptance: 30-09-2023

ABSTRACT

Abaqus/CAE covers both modelling and post-processing functionalities that are specifically designed for conducting fracture mechanics assessments. The aforementioned characteristics provide interactive utilization of the contour integrated fracture mechanics technology within the Abaqus/Standard software. Survey has shown various tools specifically designed for fractures, including those used for generating seam cracks, delineating singularities, selecting the crack front and crack tip, specifying q-vectors or normals to the crack front, and constructing focussed meshes. These tools enable the creation of models that may be utilised for the estimation of J-integrals, stress intensity factors, and crack propagation directions. This research work focuses on a model of a standardised compact tension specimen, and compares the J-integral results obtained from this model with those obtained from relevant American Society for Testing and Materials (ASTM) standards and a laboratory testing procedure. The findings demonstrate a high degree of agreement between the results obtained via Abaqus and the experimental data available in literature.

I. INTRODUCTION

Fracture mechanics encompasses a collection of theoretical frameworks that elucidate the behaviour of structures with geometric discontinuities. This field integrates the examination of mechanical properties with the analysis of bodies exhibiting cracks [1]. The initial development of fracture mechanics theory, which pertains to crack propagation and is based on the principle of energy balance, was pioneered by Griffith [2]. The Griffith theory of fracture strength was not widely regarded as credible until the period encompassing and following World War II. This shift in perception was prompted by the occurrence of significant failures in many structures, such as welded liberty ships, oil storage tanks, gas gearbox lines, bridges, and pressurised cabin planes [3]. The Griffith hypothesis was then revised by Orowan [4] and Irwin [5] at separate instances in order to incorporate plastic deformation in materials and gain insights into the reasons of structural failure. Irwin [3, 6] established the stress intensity factor and energy release rate as parameters to characterise fracture behaviour in materials exhibiting small scale yielding. Following that, Wells [7] introduced the crack tip opening displacement model, while Rice [8] proposed the J-integral model, both of which were developed to characterise fracture behaviour in materials exhibiting large scale yielding. Cracks have been observed to originate and propagate from geometric discontinuities, such as flaws, cut-outs, edges, and holes, in loaded structures. However, the size and pace of crack propagation are influenced by the geometry of the discontinuity and the specific type of load applied [9]. The detrimental impact of cracks in steel on the dependability of components and structures in service has been widely recognised [10, 11]. Therefore, the presence of a fracture when subjected to a load results in the propagation of fatigue cracks, ultimately leading to a decrease in the reliability of the structure and subsequent failures [11, 12]. Fatigue has emerged as a prominent factor contributing to the degradation of gas turbine engine components in contemporary military aircraft, thereby leading to an escalation in maintenance expenses for these aircraft [13, 14]. The accurate prediction of fatigue fracture formation under variable amplitude loading in aviation and engineering structures poses a significant problem due to the influence of loading sequence [15]. The evaluation of integrity in structures, pressure vessels, and piping systems within the nuclear, oil, and gas industries is imperative due to the potential for significant failures that could result in irreversible financial and human losses [16]. As a result, the issue of crack and fracture persisted as a significant concern among fracture mechanics researchers [16]. Hence, the enhanced comprehension and integration of fracture mechanics principles into design, prediction, and integrity evaluation methodologies have the potential to mitigate failure rates and maintenance requirements associated with engineering materials.

Engineers frequently employ finite element analysis (FEA), a computer-based numerical technique, to address issues with stress/strain analysis, fluid dynamics, physical transport phenomena, etc. One of the practical benefits of FEA is its capacity to solve issues for which there is no established formula. It has become standard practise for many advanced technical applications over the last few decades [17]. In order to perform stress and strain analysis on a particular specimen geometry, engineers and researchers regularly used finite element analysis (FEA). The numerical findings obtained are typically compared with the analytical one in order to make validation. The results can then be applied to forecast the stress/strain and fracture mechanics behaviour of more complex structures and components [18] if validation has been accomplished. Fracture analysis of compact tension specimens is a critical aspect of understanding the behavior and mechanics of materials when subjected to crack propagation. [1] Abaqus CAE, a widely used finite element analysis (FEA) software, provides powerful tools for simulating and analyzing fracture behavior in various materials.

Specimen and Finite Element Model

The specimen utilised for the evaluation of the J-integral is commonly referred to as a compact tension (C(T)) specimen. This specimen, as depicted in Figure 1, has a width of 50 mm and is widely recognised as a standard specimen for evaluating the fracture resistance of various materials [19]. The two-dimensional finite element model of the specimen assumes plane strain conditions.

A Compact Tension (CT) specimen is an ASTM standardized and widely used test specimen in materials engineering and fracture mechanics to assess the resistance of materials to crack propagation. It is specifically designed to measure the fracture toughness of a material under controlled conditions.

The CT specimen typically consists of a rectangular or square-shaped plateand its dimensions follow international standards (e.g., ASTM E399, ASTM E1820) with a centrally located crack or notch along its length. The crack is precisely machined or pre-cracked to ensure its accurate size and shape. The dimensions of the specimen and the crack size are carefully chosen to create a stable crack growth during testing.

The geometry of the specimen is usually flat and thin, with a uniform thickness. The standard size of the specimen may vary depending on the testing standards, but it typically ranges from a few centimeters to a few inches in length and width. Central crack is created in the specimen, positioned at the center, symmetrically about the mid-length of the specimen. The crack is essential for inducing controlled fracture behavior during testing.

CT specimen is subjected to tensile (opening) loading at its ends. The load is applied parallel to the crack's axis, through the loading pin hole created, which induces a tensile stress that promotes crack growth. This configuration ensures a mode I (opening mode) crack propagation, which is the most common mode of crack growth in many materials.

International Journal of Advances in Engineering and Management (IJAEM)
Volume 5, Issue 9 Sep 2023, pp: 472-483 www.ijaem.net ISSN: 2395-5252

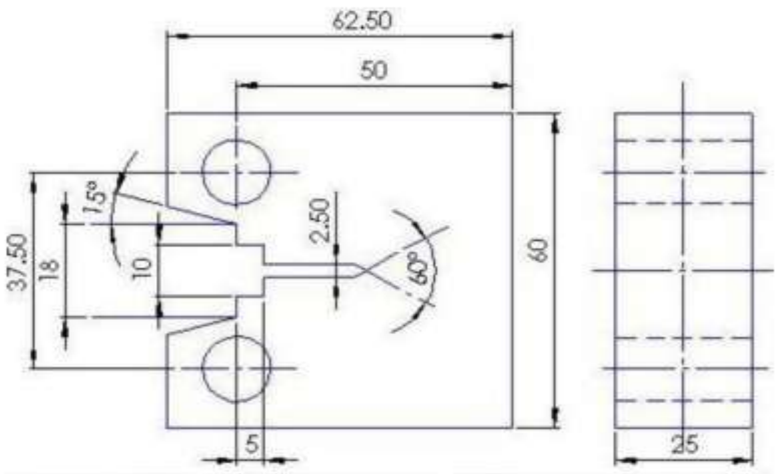

Figure 1:Compact tension specimen layout

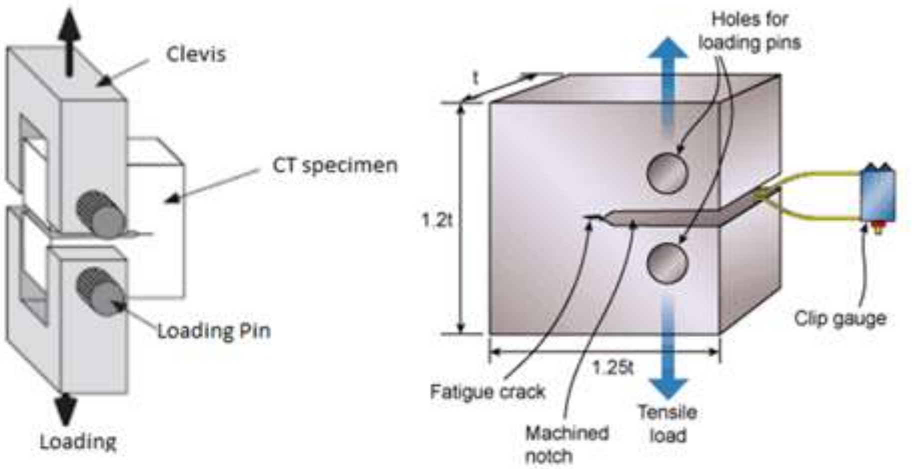

Figure 2:Apparatus set-up for CT specimen testing

Specimen can be classified based on parent material it is produced. CT specimens can be made from a variety of materials, including metals, ceramics, polymers, and composites. The dimensions and crack size are adjusted based on the material's properties and the specific testing requirements.

A typical application of this specimen is the fracture toughness testing. The CT specimen is used to perform fracture toughness tests, based on guidelines provided by the standards such as ASTM E399 or ISO 12135. These tests involve applying a controlled load to the specimen while measuring the crack's extension during the test. The data obtained from these tests help in characterizing a material's resistance to crack propagation and its ability to resist brittle fracture.

Fracture toughness values obtained from CT tests are critical in engineering applications, such as the design and safety assessment of structures subjected to potential crack initiation and growth. The controlled testing conditions and accurate measurement techniques make the Compact Tension specimen a valuable tool for evaluating materials' resistance to cracking and promoting safer and more reliable engineering practices.[18, 20]

It has been observed that older versions of this software that existed before the version 6.5, the fracture mechanics aspect of Abaqus/Standard were not included interpedently, the addition of fracture enabling tools into Abaqus/CAE created room for enhancement of fracture mechanics models. This research article is concerned in describing the procedural activities of fracture mechanics characterization and analysis of a typical structural steel CT specimen using the Abaqus/CAE and Abaqus/Standard. Several values of J-integrals were calculated, and validated by comparison of FEA results to those obtained by analytical procedures.

Figure 1 & 4 displays the specimen's dimensions used for this study. The original crack measures 5 mm in length (not visible). The specimen material has an elastic modulus of 213 GPa, and the Poisson's ratio is 0.3. The real stress vs logarithmic strain curve for this material is plotted as shown in Figure 3. The yield stress for this material is around 715 MPa. In Abaqus, a plane-strain model in two dimensions is examined. Rigid bodies are used to model the loading pins.

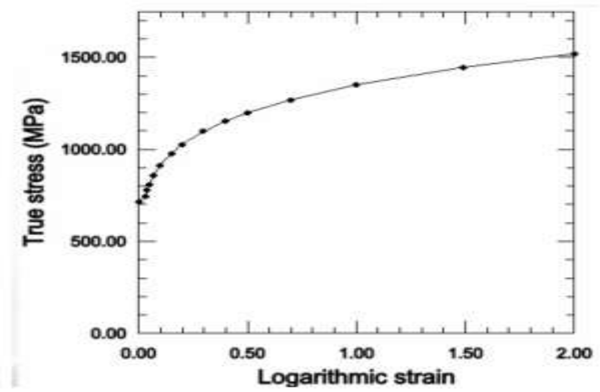

Figure 3:the real vs logarithmic strain curve

The pins are displaced vertically in order to load the specimen; all other motions of the pin are restricted. A finite-sliding concept is used to define surface-to-surface contact between the pins and the specimen. There are two analytical steps. In the first phase, contact is made between the pins and the specimen by applying a slight vertical displacement (1x105 mm). The pins are subjected to controlled displacement loading in the second step.

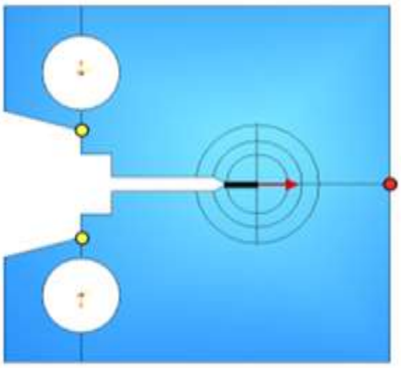

Figure 4:Partitioned 2D CT Specimen

Definition of the model in Abaqus/CAE

Figure 4 depicts the model's partitioned geometry. The load line displacement is assessed at the locations denoted by yellow dots since it is necessary for post-processing. A bolded black line in Figure 4 highlights the specimen crack. A seam from Abaqus/CAE's Interaction module is used to inject the fracture into the model. An edge (in a two-dimensional component) or a face (in a solid part) that is initially closed but may open throughout an analysis is specified as a seam in the model. When the mesh is produced, Abaqus/CAE creates overlapping duplicate nodes along a seam.

The virtual crack extension direction is determined using the q-vector from the crack editor in Figure 5. The resulting q-vector is displayed in red and is defined in the current model with the starting point at the crack tip and the end point at the red dot in Figure 4. The normal to the crack plane can also be used to specify the direction of the crack expansion.

International Journal of Advances in Engineering and Management (IJAEM)
Volume 5, Issue 9 Sep 2023, pp: 472-483 www.ijaem.net ISSN: 2395-5252

The strain field for a severe crack becomes solitary at the crack tip. The J-integral, the stress intensity factors, and the stress and strain calculations are more precise when the singularity at the crack tip is taken into account for a small-strain study.

The circular lines centred on the crack tip serve as the boundaries for the geometry's division (Figure 4), which makes it easier to create a focussed mesh. A ring of collapsed quadratic quadrilateral pieces is used to mesh the fracture tip. The most common method for obtaining a mesh singularity at the crack tip is to use second-order elements.

The definition of the singularity is also specified using the crack editor, as seen in Figure 5. Different singularity types can be produced by defining the midside node parameter and the crack-tip element degeneracy. The midside node parameter is set at 0.25 in the current analysis. The midside nodes on the element sides abutting the collapsed edge are moved to the 1/4 points in this description. The element sides collapse with single-node-type degenerate element control at the crack tip. There is a singularity in strain as a result of these circumstances.

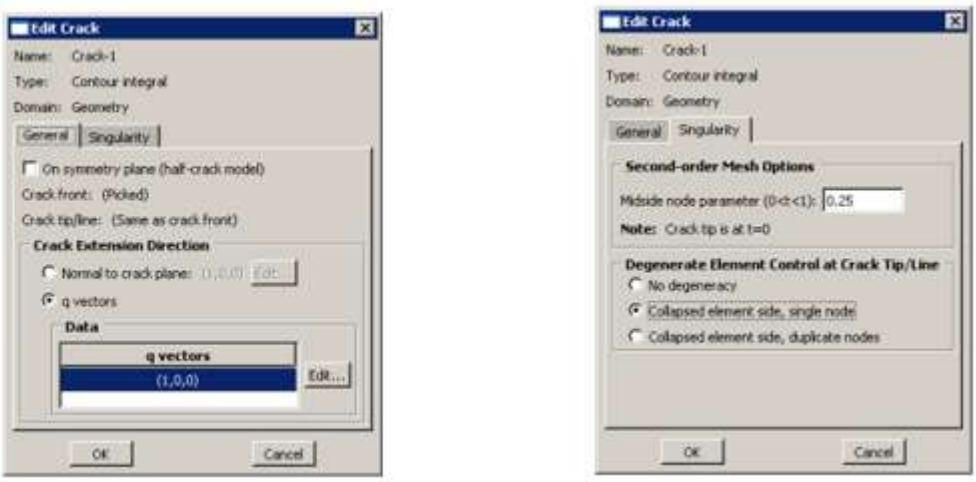

Figure 5Crack editor used for defining crack singularity

The swept meshing technique is used to mesh the circularly partitioned sections, allowing for a focused and regular mesh. When the seam and singularity definitions are present, Abaqus/CAE automatically generates collapsed elements with the appropriate connectivity definitions. The "medial axis" meshing algorithm is used to free mesh the remaining portion of the model. The creation of a targeted mesh around the crack tip is made easier by the edge-based tools for mesh seeding specification.

Figure 6 depicts the mesh that was employed in the current analyses. The "single node" degeneracy approach was applied to the components at the crack tip, as defined in the crack editor. This method uses repeating nodes in the element connectivity to define the compressed element edges.

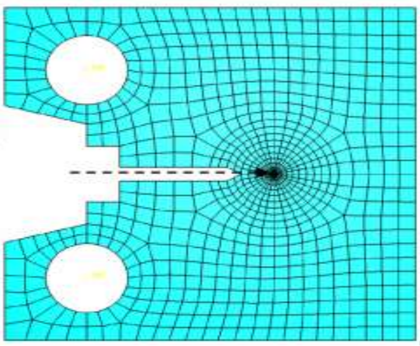

Figure 6:Mesh applied for the test

International Journal of Advances in Engineering and Management (IJAEM)
Volume 5, Issue 9 Sep 2023, pp: 472-483 www.ijaem.net ISSN: 2395-5252

II. METHODOLOGY

Fracture analysis involves studying how cracks propagate and influence the structural integrity of materials. It helps engineers and researchers assess the ability of a material to withstand crack initiation and propagation under different loading conditions. Abaqus CAE, a widely used finite element analysis (FEA) software, provides powerful tools for simulating and analyzing fracture behavior in various materials.[18, 28]

Software:

Abaqus CAE is part of the Abaqus/Standard and Abaqus/Explicit software suite offered by Dassault Systèmes SIMULIA. It provides a user-friendly, graphical interface for modeling, simulating, and post-processing finite element analyses. Abaqus CAE supports various simulation types, including fracture analysis, and enables users to create complex models with ease.

Model Development:

Parts Creation: In Abaqus CAE, geometry definitionis very essential, the first step was to create the compact tension specimens, a three dimensional (3D) geometry or alternatively import it from a CAD file. A rectangular solid geometry was created according to the standard [30,31]The modelling of only one half of the specimen is sufficient due to the presence of symmetry. This symmetry is produced by imposing boundary conditions, specifically zero displacements in the y-direction, on the nodes located in the ligament. Figure 7 displays the mesh, which comprises 376 eight-noded elements commonly referred to as "quadratic" elements.

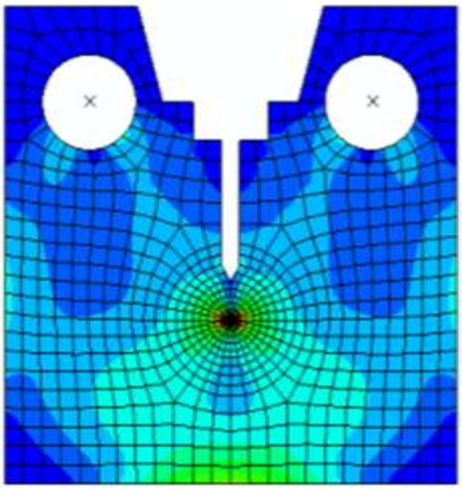

Figure 7:quadratic element mesh

Creating material: Assigning appropriate material properties is essential for an accurate analysis. Abaqus CAE allows users to define elastic, plastic, and fracture properties, including critical values like fracture toughness (KIC) and critical stress intensity factor (KICc). Essentially, two elastic materials were created with elastic Young's modulus of 140GPa and 100MPa for material one and two respectively.

Sections and Assembly: ideally, Sections- For each of the two material types, two sections with a 25 mm plane stress/strain thickness were produced. The section with the greater young's modulus was connected to a tiny portion of the overall geometry, while the other part was given to the remaining geometry's portion [9]. In assembling, each part in a model has its own axis orientation, but they can be put together in a variety of ways. One entity was generated as a separate instance type in this model, and from Stepmodule, broad Static the General Basic process type was chosen, and the period and Nlgeom values were left at 1. function was disabled [10]. The Nlgeom setting for a step determines whether Abaqus will account for geometric nonlinearity in that step

Loading and Boundary Conditions: were specified, as simulating fracture behavior requires applying realistic loading and boundary conditions. In the case of a compact tension specimen, a load was typically applied in a controlled manner to induce crack propagation. Abaqus CAE provides various loading options to simulate tension, bending, or mixed-mode conditions.The process of loading is achieved through a predetermined

International Journal of Advances in Engineering and Management (IJAEM)
Volume 5, Issue 9 Sep 2023, pp: 472-483 www.ijaem.net ISSN: 2395-5252

movement of the central node located within the pin hole. This node is coupled to the specimen through a quarter circle arrangement of elastic elements. The resulting response force is then estimated using the finite element (FE) programme. The displacement of the load line, denoted as VLL, is assessed at the location situated below the load point, which is represented by a diamond symbol in Figure 8The loads and boundary conditions in Abaqus are determined by the steps, thus the steps in which the loads and boundary conditions are in operation must be provided. After creating the step in this simulation, three boundary conditions were applied to the locations where rotation and displacement are particular. [1].

The geometry in this investigation was subjected to a concentrated mechanical load of magnitude 1000. 0 time/frequency for 0 amplitude and 1 time/frequency for 1 amplitude were chosen [11] and the load was applied per node.

Meshing and job Creation: After defining the geometry, the specimen was discretized into small elements using meshing techniques. Proper meshing is crucial to capture the crack and material behavior accurately. Thus, the model was discretized in accordance with the method used by [31] to create mesh by dividing the model geometry into sections that are joined together at node positions. The mesh surrounding the crack tip has been refined, with each element having a length of 50 µm, as depicted in Figure 8. The mesh under consideration, which has been suggested for an ESIS numerical round robin on cleavage fracture predictions, deviates from the standard approach used in ductile fracture analyses, as discussed in reference [6]. However, in the results, it will be demonstrated that the J-integral findings obtained from this mesh exhibit satisfactory accuracy. Nevertheless, the accurate depiction of the stress singularity at the fracture tip is unattainable using the current mesh configuration.

By selecting the "create job" button in the abaqus environment and giving the job the proper name, the job was created in accordance with [7] instructions. The job manager was then utilised to submit the job for analysis. The analysis was completed and the findings were obtained in an **odb**file after a short while. Then, for further examination, the stress and strain values along the X and Y axes that are prior to the fracture tip were collected.

Crack Modeling is the central activity of the whole processes. Modeling the crack accurately is therefore crucial. Abaqus CAE offers several techniques like the cohesive zone model (CZM) or extended finite element method (XFEM) to simulate crack initiation and propagation.Defining a suitable fracture criterion is essential to predict crack growth and failure. Common fracture criteria include critical stress intensity factors (KIC, KICc), J-integral, or energy-based criteria.

The last aspect of the procedures was the analysis and post-processing. As the model was set up, the analysis was performed using Abaqus CAE's solver. After completion, post-processing tools visualized and analyzed the results, including crack growth, stress distribution, and displacement fields.

Fracture analysis of compact tension specimens using Abaqus CAE plays a significant role in material testing and design, especially in industries where structural integrity and safety are critical factors, such as aerospace, automotive, and civil engineering. The software's capabilities and ease of use make it a popular choice for researchers and engineers conducting fracture-related studies.

III. RESULT AND DISCUSSION

The calculations have been conducted using both small and large deformations, denoted by the parameter NLGEOM. The selection of the deformation theory has an impact on the stresses and strains in the area of the crack tip, while the overall behaviour remains unaffected. Hence, the load versus loadline-displacement (VLL) curve seen in Figure 8 remains consistent throughout both calculations.

International Journal of Advances in Engineering and Management (IJAEM)
Volume 5, Issue 9 Sep 2023, pp: 472-483 www.ijaem.net ISSN: 2395-5252

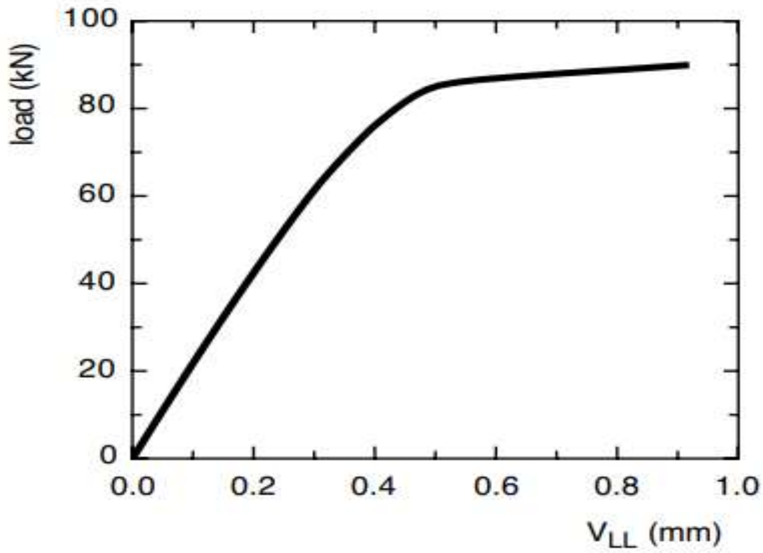

Figure 8:Load Line Displacement curve

In the second part of the analysis, the output of the contour integral is sought for a total of ten contours. In the context of the Interaction module, it is imperative to explicitly provide both the crack front and the crack tip. The first contour integral is determined by considering all the elements located within the crack front, as well as one layer of elements situated outside the fracture front. When computing the additional contour integrals, Abaqus automatically includes a singular layer of components to the group of elements that were used in the calculation of the preceding contour integral.

The J-integral findings produced using Abaqus software are compared with the results computed using ASTM standard techniques and the laboratory testing method described in Reference 1.

Both of the aforementioned strategies necessitate the availability of the historical data pertaining to the relationship between pin response force and load line movement. It is necessary to compute the integral of the curve for each incremental value of the recorded load line displacement. The curves depicting the relationship between reaction force and load line displacement, as well as the computations of their respective areas, can be conveniently performed by utilising the X-Y data tools available in the Visualisation module of Abaqus/CAE.

J contour estimation based on ASTM standard

The J-integral estimations generated from Abaqus software are initially compared to the values obtained using the methodology given in ASTM standards E1737-96 and E1820-01 (References 25 and 30).

The equations employed in these standards are discussed in details below

For compact tension specimen, [30] suggested additional modification to consider tensile component of the applied load on the test specimen for accurate valuation of J. Since the total displacement Δ can be expressed separately as elastic and plastic component, as in $\Delta = \Delta_{el} + \Delta_{pl}$, the total J-integral was also correspondingly written as two separate parts:

$J = J_{el} + J_{pl}$ ……………………………………Equation 1

The elastic component J_{el} can be regarded as elastic strain energy rate, G and is most simply calculated from the stress intensity factor K_I

$J_{el} = \frac{K_I^2}{E'}$………………………………Equation 2

Where $E' = E/1 - v^2$ for plain strain while K is obtained from the load relation specified in [24].Stress intensity factor solutions for various fracture specimens were documented in the work of [22]

$J_{el} = \frac{K_I^2(1-v^2)}{E}$……………………………… Equation 3

$J_{pl} = \frac{\eta A_{pl}}{B(W-a)}$ ……………………………… Equation 4

$J = \frac{K_I^2(1-v^2)}{E} + \frac{\eta A_{pl}}{B(W-a)}$………………………… Equation 5

International Journal of Advances in Engineering and Management (IJAEM)
Volume 5, Issue 9 Sep 2023, pp: 472-483 www.ijaem.net ISSN: 2395-5252

$K = \frac{P}{B\sqrt{W}} f(\frac{a}{W})$... Equation 6

$f(\frac{a}{W}) = \frac{2+a/W}{(1-\frac{a}{W})^{3/2}}\left[0.886 + 4.64\left(\frac{a}{W}\right) - 13.32(\frac{a}{W})^2 + \right.$ *14.72(aW)3−5.60(aW)4* ....Equation 7

[24] applied plastic analysis to provide a more reliable J estimate for C(T) specimens:
$\eta = 2 + 0.522Xb/W$
$b = W - a$
A_{pl} = plastic work, found from the load versus load line displacement curve
W =distance between the point of application of load to the point marked by a red dot in Figure 4
a =distance between the load line and the crack tip
B = thickness of the specimen

The pin reaction force is graphically represented versus the displacement of the load line in Figure 8. Figure 9 is derived by performing calculations to determine the cumulative area beneath the curve seen in Figure 8, corresponding to various load line displacements. A total of 10 values are computed for this purpose. The curve depicted in Figure 9 represents the cumulative amount of work expended in the process of opening the crack. This value is obtained by combining the contributions from both elastic and plastic work.
The determination of plastic work is a necessary step in the ASTM calculation. This can be accomplished by subtracting the elastic work component from the overall work value.

Laboratory estimation of J for small strain

With this method, the Abaqus analysis must be repeated for seam cracks of various lengths. The investigation is performed in the current work for additional crack lengths of 3 and 7 mm. The area under the reaction force/load line displacement curve is calculated and displayed for each unit of load line displacement for each of the models (3, 5, and 7 mm crack lengths); these curves are shown in Figure 10. The curves are then differentiated at a seam fracture length of 5 mm to determine the J-integral values. These calculations are performed using the X-Y data operations in Abaqus/CAE's Visualisation module.

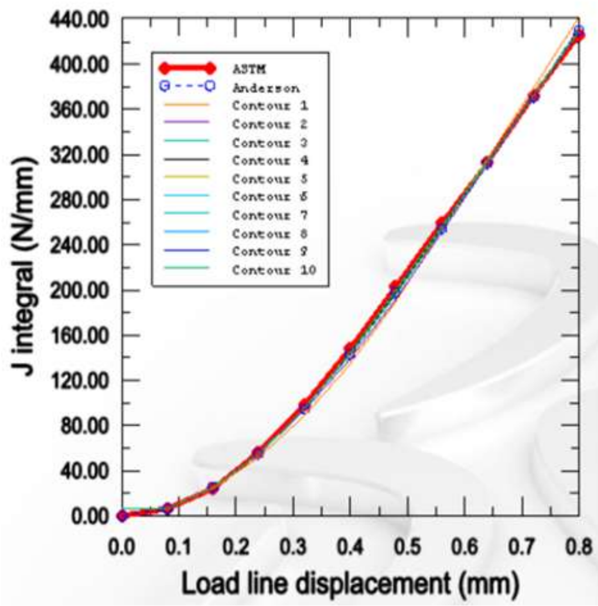

Figure 9:Small strain analysis

Result Comparison of Abaqus small-strain analysis against the ASTM standards and the laboratorytesting method

The findings derived by Abaqus, as presented in Table 1, demonstrate a high level of agreement with the results acquired through the use of ASTM standards and the methodology outlined in Reference 1.

International Journal of Advances in Engineering and Management (IJAEM)
Volume 5, Issue 9 Sep 2023, pp: 472-483 www.ijaem.net ISSN: 2395-5252

Table 1: Summary of result

LLD (mm)	J Integral values (N/mm)		
	Abaqus CAE	ASTM (E1820-01)	Anderson [1]
0	0	0	0
0.0798	6.333	6.200	6.296
0.159	25.000	23.877	24.828
0.239	55.029	56.530	54.696
0.319	95.240	98.889	94.690
0.399	144.030	148.19	143.194
0.479	198.78	203.19	197.580
0.559	255.69	259.01	254.581
0.639	313.299	312.7	312.430
0.719	371.448	371.86	370.956
0.8	430.050	425.78	430.050

Figure 9 displays the displayed J-integral results for all analysis methods. The J-integral for the initial contour is commonly disregarded because to numerical inadequacies in the stress and strain values at the fracture tip. The impact of the inaccuracy is comparatively more subtle in small-strain situations as opposed to finite-strain problems.

The current analysis examines a material exhibiting both elastic and plastic behaviour.

As demonstrated, it can be observed that the material does not possess full plasticity, but rather has hardening behaviour. The crack-tip singularity for this particular material is between that of a linear elastic material, which demonstrates a singularity, and that of a completely plastic material, which also demonstrates a singularity.

The sensitivity of two-dimensional analyses to the strength of the singularity can be minimised by employing a thin mesh around the fracture tip and evaluating an adequate number of contours. The presence of a singularity does not hinder the accurate determination of the J-integral's far-field value, as demonstrated by the strong correlation seen between the findings produced from Abaqus simulations and the analytical calculations.

Result comparison between Abaqus finite-strain analysis and ASTM standards

A subsequent round of analyses was performed, taking into account finite strains and selecting a fracture front region that exceeded the plastic zone surrounding the crack tip. The initial contour region for the crack in the Interaction module was chosen to be the circular partitioned zone directly surrounding the fracture tip. The comparison of the Abaqus results is limited to the ASTM calculation, as the latter solely relies on the load-displacement characteristics of the specimen and does not consider the magnitudes of strain at the crack tip.

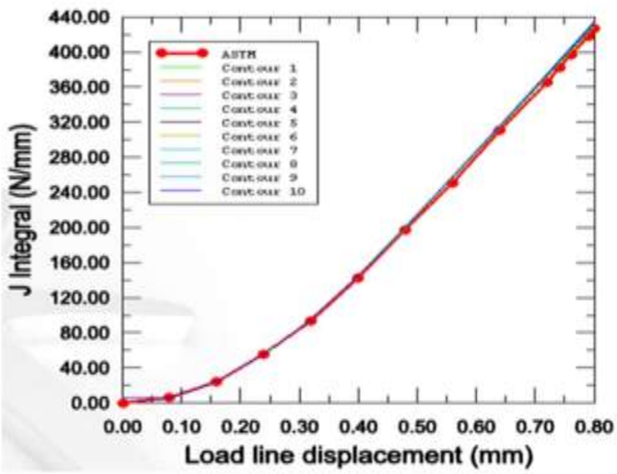

Figure 10:Finite strain analysis

International Journal of Advances in Engineering and Management (IJAEM)
Volume 5, Issue 9 Sep 2023, pp: 472-483 www.ijaem.net ISSN: 2395-5252

The findings derived from the subsequent analysis set are visually presented in Figure 10. The Abaqus results exhibit a high degree of conformity with the results obtained from the ASTM standard. The incorporation of finite-strain effects has a minimal impact on the outcomes of this analysis, as the nonlinearity in the analysis is predominantly concentrated at the crack tip and does not significantly influence the overall behaviour.

IV. CONCLUSIONS

The paper presents a detailed approach of CT specimen testing using Abaqus CAE, steps involved are simplified and the major findings signified the high degree of comformity between Abaqus results and ASTM standard derived results. This work may assist in further understanding of the basics of fracture toughness measurement.

REFERENCES

[1]. Anderson TL (2005) Fracture mechanics—fundamental and application, 3rd edn. CRC Press Publisher, Boca Raton

[2]. Griffith AA (1920) The phenomena of rupture and flow in solid. Philos Trans 221:163–198

[3]. Irwin GR (1957) Analysis of stresses and strains near the end of a crack traversing a plate. J Appl Mech 24:361–364

[4]. Orowan E (1949) Fracture and strength of solids. Rep Prog Phys 12(1):185–232

[5]. Irwin GR (1948) Dynamic of fracture: in fracturing of metals. In: ASM symposium, Cleveland, pp 147–166

[6]. Irwin GR (1960) Structural mechanics. In: Proceedings of the first symposium on Naval structural mechanics. Pergamon Press, New York, pp 557–591.

[7]. Wells AA (1961) Unstable crack propagation in metals: cleavage and fast. In: Crack propagation symposium proceeding, p 84

[8]. Rice JR (1968) A path independent integral and the approximate analysis of strain concentration by notches and cracks. J Appl Mech 35:376–386

[9]. Newman JC (1971) An improved method of collocation for the stress analysis of cracked plates with various shaped boundaries. NASA technical notes, Washington, DC

[10]. Ameh ES, Onyekpe BO (2016) Effect of high strength steel microstructure on cracks tip opening displacement. Am J Eng Res 5:72–78

[11]. Vukelic G, Brinc J (2011) J-integral as possible criterion in material fracture toughness assessment. Eng Rev 2:91–96

[12]. Gopichand A, Srinivas Y, Sharma AVL (2012) Computation of stress intensity factor of brass plate with edge crack using J-integral technique. Int J Res Eng Technol 10:2319–1163

[13]. McDowell DL (1996) Basic issue in the mechanics of high cycle metal fatigue. Int J Fract 80:103–145

[14]. Cowles BA (1996) High cycle fatigue in aircraft gas turbine-an industry perspective. Int J Fract 80:147–163

[15]. Dirik H, Yalcinkaya T (2016) Fatigue crack growth under variable amplitude loading through XFEM. Procedia Struct Integr 2:3073–3080

[16]. Chhibber R, Arora N, Gupta SR, Dutta BK (2006) Use of bimetallic welds in nuclear reactors: associated problems and structural integrity assessment issues. J Mech Eng Sci 220:1121–1133

[17]. K. E. Atkinson, (1989) An Introduction to Numerical Analysis, vol. 125, no. 1–2.

[18]. D. S. Simulia, (2014) Abaqus 6.14 Online Documentation, Rhode island.

[19]. Zhu X (2013) Fracture toughness testing, evaluation and standardization. J Pipeline Eng 12(3):145–155

[20]. Veeranjaneyulu J, Rao RH (2012) Simulation of the crack propagation using fracture mechanics techniques in aero structure. Int J Eng Res Appl 2(5):1168–1173

[21]. Pwei R (2010) Fracture mechanics, integration of mechanics material science and chemistry. Cambridge Press University, Cambridge

[22]. Zhu X, Joyce JA (2012) A review of fracture toughness (g, k, ctod, ctoa), testing and standardization. Eng Fract Mech 85:1–46

[23]. Erdogan F (2000) Fracture mechanics. Int J Solids Struct 13:171–218

[24]. Brocks, W., and I. Scheider, ―Numerical Aspects of the Path-Dependence of the J-integral in Incremental Plasticity—How to calculate reliable J-values in FE analyses, "GKSS FORSCHUNGSZENTRUM internal report—GKSS/ WMS/01/08, 2001.

[25]. ASTM Standards, (1996)―Test Method for J-integral characterization of Fracture Toughness, ASTM E1737-96, American

Society for Testing and Materials, Philadelphia.

[26]. ASTM Standards, (1997)—Standard Test Method for Plane-Strain Fracture Toughness of Metallic Materials, ASTM E399- 97, American Society for Testing and Materials, Philadelphia, 1997.

[27]. T. Belytschko and T. Black, (1999) Elastic crack growth in finite elements with minimal remeshing, Int. J. Numer. Methods Eng., vol. 45, no. 5, pp. 601–620,

[28]. D. S. Simulia, (2014) Mesh-independent Fracture Modeling (XFEM), Modeling Fracture and Failure

[29]. P. Paris and F. Erdogan, A critical analysis of crack propagation laws, J. Basic Eng., vol. 85, no. 4, pp. 528– 533, 1963.

[30]. ASTM E 647-00, 2001 Standard Test Method for Measurement of Fatigue Crack Growth Rates," ASTM Int., vol. 3, pp. 1–43.

[31]. ASTM- E1820-01, 2001. Standard test methods for measurement of fracture toughness. Annual Book of ASTM Standards, American Society for Testing and Materials

[32]. M. Levén and R. Daniel, (2012) Stationary 3D crack analysis with Abaqus XFEM for integrity assessment of subsea equipment.