

PAPER • OPEN ACCESS

Linear and Non-Linear Stress Analysis for the Prediction of Fracture Toughness for Brittle and Ductile Material using ASTM E399 and ASTM E1290 By ANSYS Program package

To cite this article: A A Nassar and E G Fayyad 2019 *IOP Conf. Ser.: Mater. Sci. Eng.* **579** 012050

View the [article online](#) for updates and enhancements.



IOP | ebooks™

Bringing you innovative digital publishing with leading voices to create your essential collection of books in STEM research.

Start exploring the collection - download the first chapter of every title for free.

Linear and Non-Linear Stress Analysis for the Prediction of Fracture Toughness for Brittle and Ductile Material using ASTM E399 and ASTM E1290 By ANSYS Program package.

A A Nassar^{1#} and E G Fayyad²

¹Basra university /Mechanical Engineering Department/Basra/Iraq

²Basra Oil Company / South Rumelia/design & project Department/ Basra/Iraq

[#]Corresponding author's e-mail address: en.ebtesam@gmail.com

Abstract. Prediction of fracture toughness value of most the metallic materials is a very important issue to be consider in designing of engineering structural components like pipes, flow tanks and pressure vessel where it can be used in the evaluation of critical crack length that is consider very important factor in non-destructive testing because during inspections of these components, the critical crack length is being compared with the minimum allowable flow size. Fracture toughness value can be predicted by using two different types of experimental tests such as ASTM-E399 (which is used to predict fracture toughness under plane strain condition for brittle materials which undergo to small plastic deformation) and ASTM-E1290 (which is used to predict fracture toughness for ductile materials which undergo to large plastic deformation). These experimental tests utilize specimens such as compact tension specimen CT and three-point bend specimen SENB which must be pre-cracked. However, manufacturing of pre-cracked in metallic structural components before testing is expensive and time-consuming process, thus in the current study, fracture toughness will be predicted by using economized method in both time and cost that is the finite elements method by using ANSYS PROGRAM Where, a three-dimensional model has been designed by using two different types of elements (plane-82) and (solid-95). Firstly, two-dimensional model mesh will be idealized by using element type (plane-82) due to ANSYS PROGRAM can pick-up singularity for two dimensional model only, but in order to obtain more accurate results, a three dimensional model will be designed by utilizing Sweep Model with restriction with ANSYS abilities in the treatment of fracture mechanics problems for three dimensional model analysis where the element length must be ranging from 1to4 in all directions. In elastic region, the fracture toughness is been predicted directly from ANSYS PROGRAM using the specification in ASTM-E-399 for compact tension specimen for practical design problems. But in elastic-plastic region, the fracture toughness is been predicted by using the crack tip opening displacement model (CTOD-Model) using the specification in ASTM-E-1290 for three-point bend specimen for practical design problem. The critical value for crack tip opening displacement will be calculated after extracting load-displacement data from ANSYS PROGRAM then using these data in (CTOD-Model) which is separated into two components, elastic and plastic, the elastic component has been estimated using Dug Dual-Model, while the plastic component will be estimated using Plastic Hinge-Model. However, the fracture toughness value that has been predicted by finite element method from non –linear elastic analyses and from non –linear elastic-plastic analyses gives excellent results which are very close to the experimental results with error ratio ranging between (10% to 14%).



1. Introduction

Every structure contains defects such as small flaws or cracks of sizes and distribution are dependent on the material and its manufacturing processing. These may vary from nonmetallic inclusions and micro-voids to weld defects, grinding cracks, quench cracks, surface laps etc. The Presence of these defects in the components creates problems that can adversely affect the integrity of the structural components and shortens their service life. Therefore, Fracture Mechanics methodology has been developed to figure out if these small flaws might grow into larger cracks causing the catastrophic failure of these structural components. The analysis of Fracture Mechanics is become very important for all engineers are researching about the reasons of failure of structural components in different scientific fields. The determination of the stress distribution near cracks especially at crack tip region can be used for proper predictions regarding the consequences of a crack which could lead to rapid fracturing [1, 2].

The basic principal of Linear Elastic Fracture Mechanic theory is “the fracture occurs when the stress intensity factor is equal or greater than plane strain fracture toughness or the fracture occurs when the stress intensity factor is equal or greater than plane stress fracture toughness” i.e. crack propagation occurs when $K_I > K_{IC}$ or $K_I > K_{IC}$. Consequently, using (LEFM) theory which is limited with (plane strain condition & Small-Scale Yielding) engineers can design a structure with the stress intensity factors corresponding to different fracture modes. while, the stress intensity factor does not represent the true fracture mode beyond the elastic region ;therefore, after this region the Non-Linear Elastic Plastic Fracture Mechanics theory must be used and as shown in (Figure 1) where a is a crack length and F is an applied force. There are two major branches in Elastic Plastic Fracture Mechanics, the CTOD-Model and the J-Integral Model which have been developed to understand and solve the non-linear behavior of cracked structures and characterizing crack tip toughness [3].

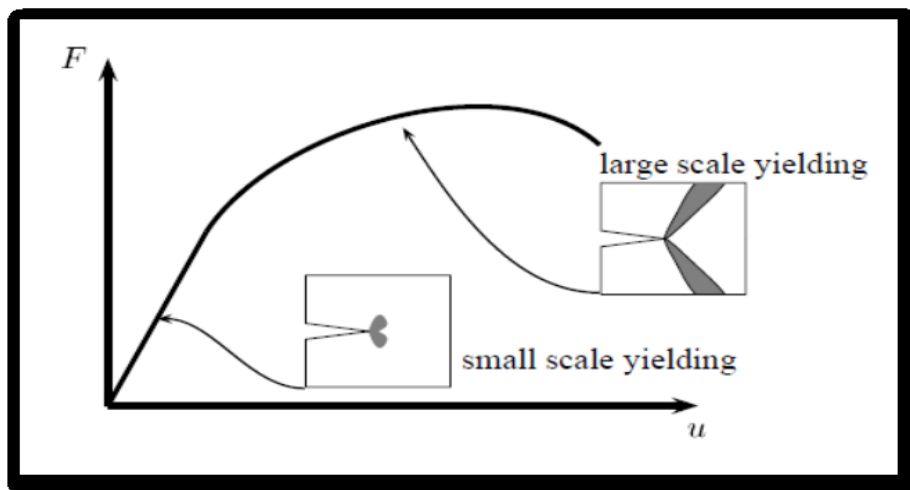


Figure 1. The EC experimental setup.

2. Fracture Toughness Concept

In physical science, Fracture toughness is an indication of the resistance of a material to physical separation by a process of unstable macro-crack propagation. Conceptually, it is an intrinsic material parameter that should not vary with changes in specimen size, speed of loading, temperature, etc. In material science, Fracture toughness is an empirical material property that is determined by one or more of several standard fracture toughness test methods and is one of the most important parameters to evaluate the mechanical properties of any materials for virtually all design applications [4,5].

3. Fracture Toughness Parameters

There are two fracture toughness parameters commonly obtained from testing [6]: -

3-1 Stress Intensity Factor (K_I)

The stress intensity factor can be considered as a stress based-estimate of fracture toughness. Depending on this parameter, plane strain fracture toughness under plane strain and small-scale yielding condition for most metallic materials can be predicted according to (ASTME-399) specification. Experimental international data for practical design problems will be used to predict plane strain fracture toughness while keeping the validity conditions according to recommendation in (ASTM E-399) being very important in choosing of the suitable thickness of the design structural components.

3-2 Crack Tip Opening Displacement (CTOD)

Linear elastic analysis is used to predict the value of plane strain fracture toughness of high strength materials according to (ASTM E-399) that is constrained with small plastic deformation zones that being involved with the creation of large plastic deformation zones and of the non-linear elastic-plastic behavior for most low-to-medium strength materials. Therefore, another standard and model have been utilized to predict fracture toughness value ;that is the (ASTM E-1290) (Standard Test Method for Crack-Tip Opening Displacement CTOD-Fracture Toughness Measurement).

The crack-tip opening displacement CTOD can be considered as a strain-based estimate of the fracture toughness where it is one of a family of fracture mechanics tests that measures material resistance to growing crack; Similar tests (KIC and JIC) can be used to determine fracture resistance of a material. CTOD is used to characterize the crack tip field at extensive plastic deformation as in ductile materials such as steel structures which is used in the constructions of pipes, flow tank, pressure vessel and ships. The critical value of crack-tip opening displacement CTOD can be separated into elastic and plastic components. The elastic component of CTOD can be estimated using Dug Dual Model while the plastic component is only obtained from Plastic hinge model by converting the crack mouth opening displacement CMOD into crack tip opening displacement CTOD using the relationships of similar triangles. These two models will be used in the current study to predict fracture toughness value.

The specimens which have been used to predict and calculate of the fracture toughness values through testing in the laboratory the structural components of metallic materials should be pre-cracked as shown in (Figs. 2 and 3). The presence of sharp pre-crack in notched specimen is a major requirement in fracture toughness testing procedure that has been executed by employing servo-hydraulic machine, the generation of a pre-crack specimen is an expensive and time-consuming process [7].

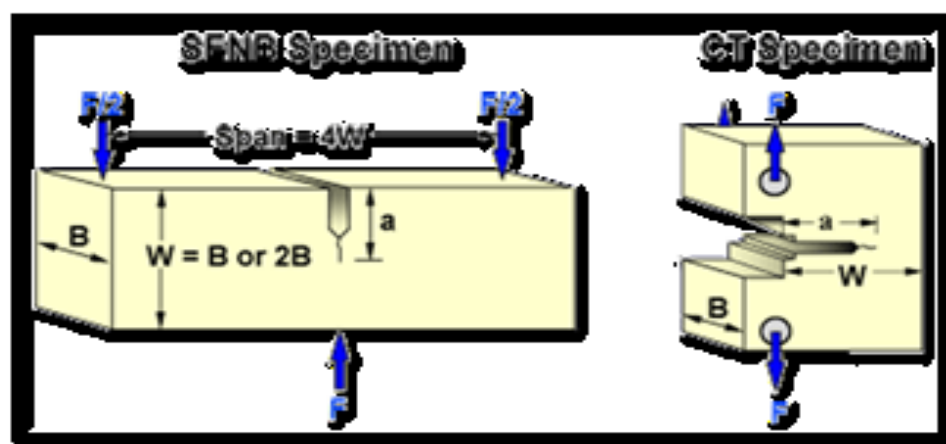


Figure 2. Fracture Toughness Test Specimens Types

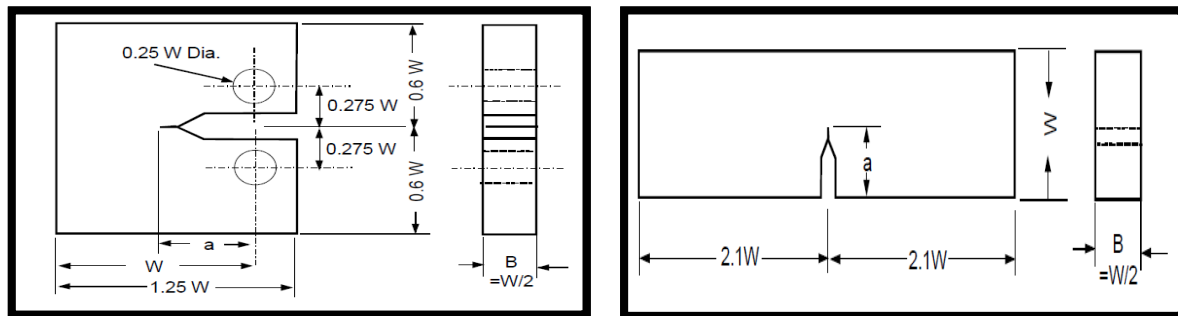


Figure 3. Pre-Crack in Fracture Toughness test Specimens

4. Modeling and Meshing of Test Specimens

A three-dimensional finite element model of the CT and SENB fracture test specimen will be utilized to predict the value of stress intensity factor at critical load i.e. fracture toughness near the crack tip in two cases. The first case is small scale yielding where the dimension of model will be prepared according to ASTM E-399 and the second case is large scale yielding where the dimension of model will be prepared according to ASTM E-1290. Firstly, two-dimensional continuum finite element model will be created by using element type (plane-82) under plane strain condition. This element is defined by eight nodes, one at each corner and each mid-side. Then, generation of three-dimensional crack elements by defining element type (Solid-95) through Sweep/Extrusion Techniques as shown in fig. 4a and fig. 4b.

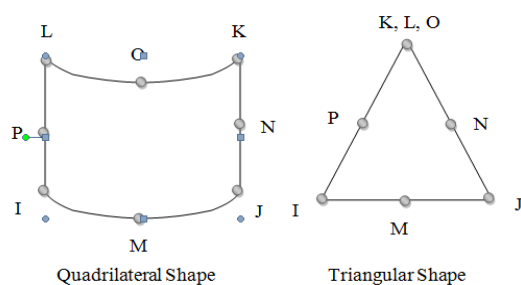


Figure 4.a (plane-8) 8-nodes structural solid

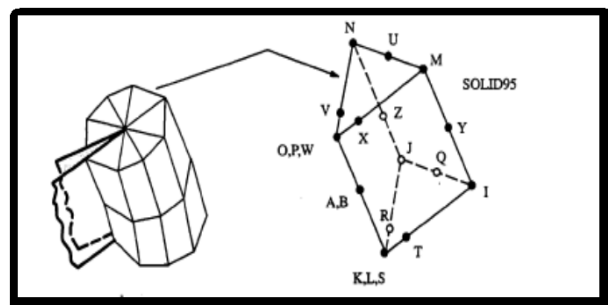


Figure 4.b (Solid-95) 20-nodes structural solid

5. Results and Discussion

The fracture toughness will be predicted for brittle and ductile materials by performing the non-linear three-dimensional finite elements model with using the specification in ASTM E-399 and ASTM E-1290. According to ASTM E-399 that has used to predict fracture toughness under plane strain condition for brittle materials, the non-linear three-dimensional finite elements model will be designed for CT specimen and according to ASTM E-1290 that has used to predict fracture toughness for ductile materials, another non-linear three-dimensional finite element model will be designed for SENB specimen for practical design problems.

6. Numerical Model Analysis for the Prediction of Fracture Toughness According To ASTM E-399 Specification

The practical design problems have been selected to predict fracture toughness numerically. The dimensions of CT specimen are listed in (Table 1), elastic and plastic material properties are listed in (Table 2), Experimental load and fracture toughness data are listed in (Table 3) for design problems A:

Table 1. Specimen Dimensions for Design Problem A

Design problem	Crack size	Thickness	width
A	38.1mm	46mm	101.6mm

Table 2. Material properties for Design Problem A

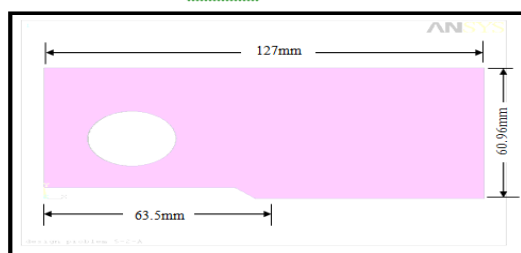
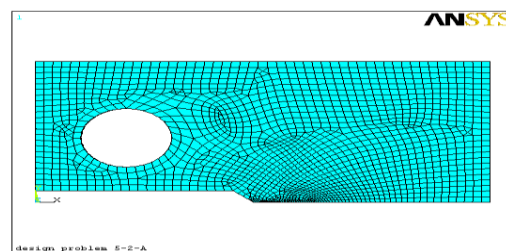
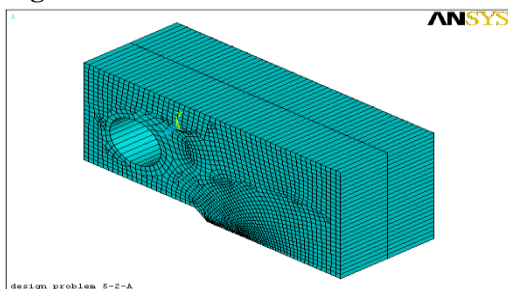
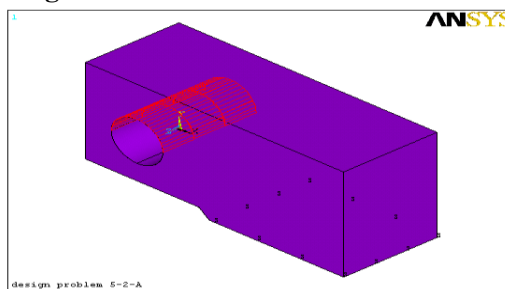
Design problem	Yield Strength MP	Tangent Modulus Gp	Poisons ratio	Elasticity Modulus GP
A	1172.15	200	0.3	210

Table 3. Experimental load and KIC data

Design problem	Experimental maximum load	Experimental critical load	Experimental Fracture Toughness
A	245.351KN	231.296 KN	109.9MPa√m

7. Numerical Analysis Procedure

The half model as shown in (Figure 5) has used during this analysis, 2-D finite elements mesh idealization for half model of this specimen is shown in (Figure 6) which has performed using element type (plane-82) where the maximum elements numbers were 1621 and the maximum nodes numbers were 5001, and the element length is 2.5. Then, by defining element type (solid-95) and giving two divisions through thickness, the Sweep Model will be generating for this problem. 3-D finite elements mesh for half model of this specimen will become as shown in (Figure 7) where the maximum elements numbers are 4863 and the maximum nodes number are 18383. The boundary condition has been applied as shown in (Figure 8).

**Figure 5.** half model idealization for CT**Figure 6.** 2-D finite elements mesh idealization**Figure 7.** 3-D finite elements mesh idealization**Figure 8.** Application of B.C.

After running the program, the deform shape for this specimen is shown in (Figure 9) which appears that the maximum displacement which occur in the y-direction is about 0.49423mm, the nodal solution

is shown in (Figs. 10 and 11) which appears that the maximum displacement occurring in the y-direction at node No.5323 is about 0.447412mm. The element solution shows that the maximum stress occurring in the y-direction is about 11046MPa as shown in (Figure 12). Thus, the maximum failure load can be concluded from reaction solution in y-direction FY which is about 230.930KN at last step and about 217.345KN at step number 16 which represents the critical load. Consequently, fracture toughness can be extracted directly from ANSYS which is about 96.39MPa.m^{1/2} case.

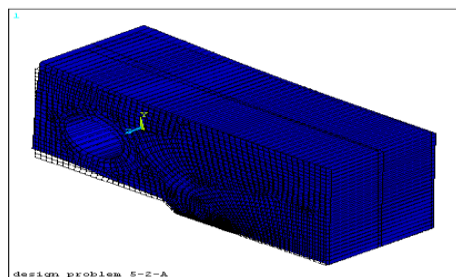


Figure 9. Deforms shape of the CT specimen

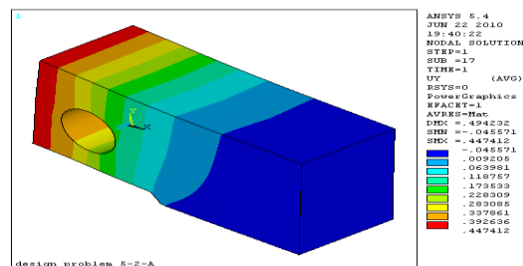


Figure 10. Maximum displacement in y-direction

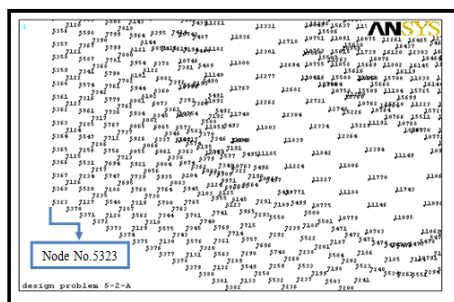


Figure 11. Node have max. Displacement

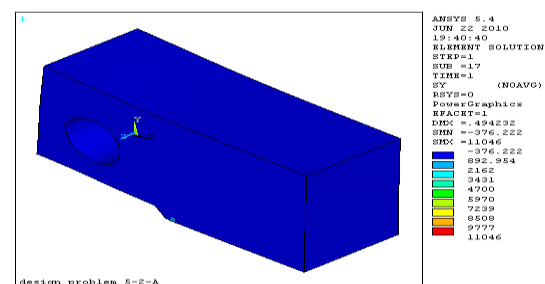


Figure 12. Maximum stress in the y-direction

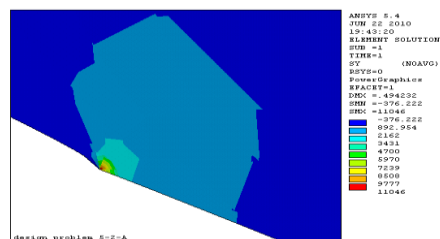


Figure 13. maximum stress at crack tip region

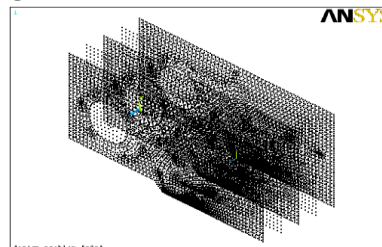
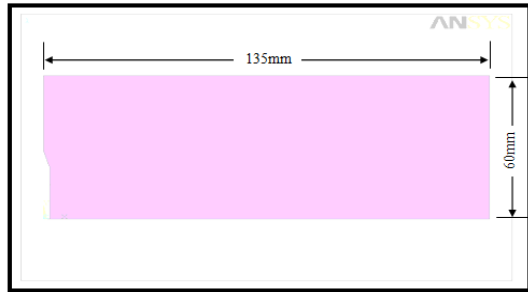
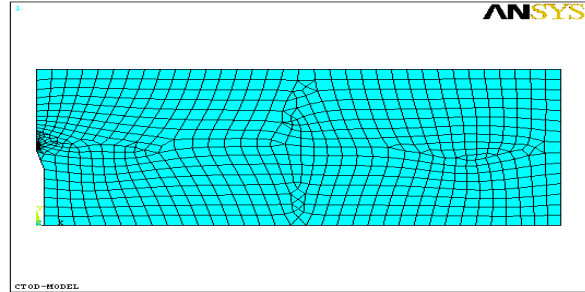
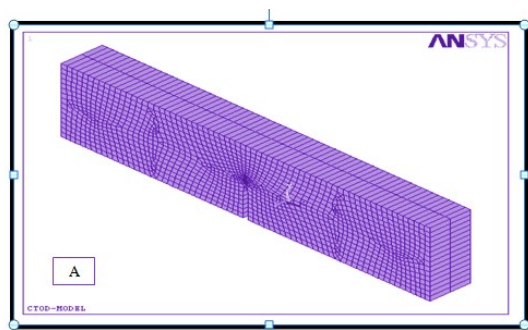
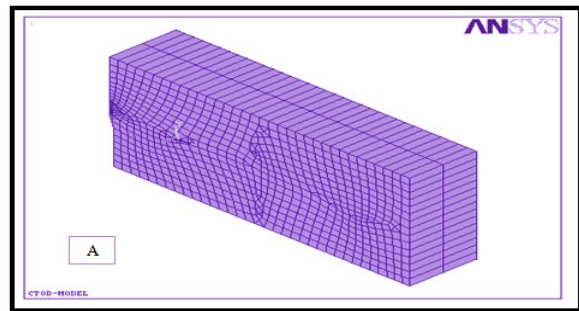


Figure 14. Drawing of two division through thickness

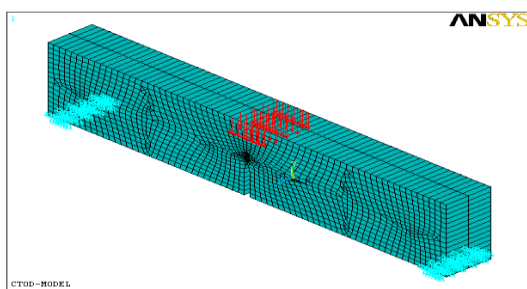
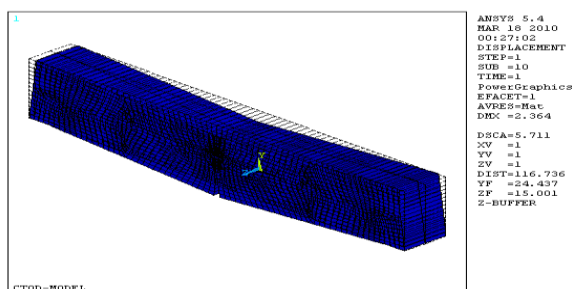
8. Numerical Analysis of prediction Fracture Toughness Using CTOD

A three-point bend specimen with $S = 250$ mm, $W = 60$ mm, $a = 30$ mm, and $B = 30$ mm is used to determine the critical crack tip opening displacement of steel plate. The load versus crack mouth displacement (P-V) of the test shows that the Maximum load is 31.600 KN. The elastic material property which will be used as an elastic input data for ANSYS to estimate the elasticity response of the material, also the plastic material properties (tangent modulus and effective yield strength) will be used as plastic input data to estimate the plasticity response of the material.

The half model of geometry of three-point bend specimen is shown in (Figure 15), the 2-D finite elements mesh idealization for half model is shown in (Figure 16) which has been also designed using element type plane-82 where the maximum elements numbers were 740 and the maximum nodes number were 2285. By defining element type solid-95 and giving two divisions through thickness, the sweep model will be generated, as shown in (Figure 17) for full model and the 3-D finite elements mesh idealization for half model of this specimen will become as shown in (Figure 18) where the maximum elements numbers were 2220 and the maximum nodes number were 8401.

**Figure 15.** half model idealization**Figure 16.** 2-D finite elements mesh idealization**Figure 17.** 3-D Finite elements mesh idealization**Figure 18.** 3-D Finite elements mesh idealization

Then, the boundary condition will be applied as shown in (Figure 19), After running the program and obtaining solution, the deform shape for this specimen is shown in (Figure 20) which shows that the maximum displacement that occurring in the y-direction is about 2.364 mm, Then, the maximum failure load can be concluded from reaction solution in y-direction FY which is about 34.589 KN at last step.

**Figure 19.** Application of B.C.**Figure 20.** Deform shape for bending specimen

```

ANSYS 5.4
MAR 18 2010
00:27:02
DISPLACEMENT
STEP=1
SUB =10
TIME=1
PowerGraphics
EPACET=1
AVRES=Max
DMX =2.364
DSCA=5.711
NV =1
YV =1
ZV =1
DIST=116.736
YF =24.437
ZF =15.001
Z-BUFFER

```

9. Numerical Results Discussion of compact specimen

The non-linear three-dimensional finite elements model according to ASTM E-399 specification provides excellent results predicting of the fracture toughness directly from ANSYS as shown in (Table 4), it also provides excellent results for maximum loading from ANSYS as shown in (Table 5). Adding to that, it provides excellent results for critical loading from ANSYS as shown in (Table 6).

Table 4. Differences between experiment and finite elements result for prediction of fracture toughness

Design problem	Predictive of Fracture Toughness by experiment	Predictive of Fracture Toughness by FEA	Differences between experiment and FEA
A	109.9MPa√m	96.39MPa√m	14 %

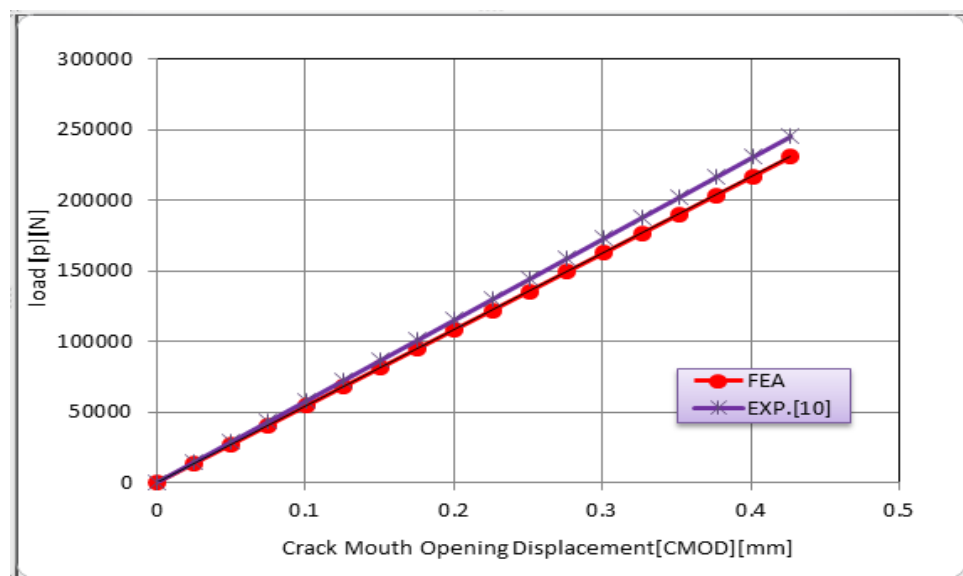
Table 5. Differences between experiment and finite elements result for maximum loading

Design problem	Maximum load by experiment	Maximum load by FEA	Differences between experiment and FEA
A	245.351KN	230.930KN	6.24%

Table 6. Differences between experiment and FE results for Critical loading

Design problem	Critical load by experiment	Critical load by FEA	Differences between experiment and FEA
A	231.296KN	217.345KN	6.41%

Difference between FEA and experiment results for compact specimen as shown in (Figure 21): -

**Figure 21.** Difference between FEA and Experiment results for compact specimen

10. Numerical Results Discussion of bending specimen

The differences between experimental and finite element results of three-point bend specimen, for maximum loading are shown in (Table 7) and in (Figure 22), clip gage displacement (V_p) can be concluded after drawing a straight line which is equal to linear elastic portion which was about (0.85mm). The differences between the experimental and finite element value for (V_p) is shown in (Table 8), the plastic part of crack tip opening displacement ($(\delta_t)_{pl}$) can be estimated from plastic hinge model which was about (0.242mm). The differences between the experimental and finite element value for ($(\delta_t)_{pl}$) are shown in (Table 9), the elastic part of crack tip opening displacement ($(\delta_t)_{el}$) can be estimated from dug dual model which was about (0.007mm). The differences between the experimental and finite element value for ($(\delta_t)_{el}$) as shown in (Table 10), critical value of crack tip opening displacement can be concluded by adding the elastic part to the plastic part which was about (0.249mm) then fracture toughness value is about (311.636MPa $\sqrt{m}$).

Table 7. Differences between experiment and finite elements result for maximum loading

Design problem	Maximum load by experiment	Maximum load by FEA	Differences between experiment and FEA
SENB	31.6 KN	34.589KN	9.45%

Table 8. Differences between experiment and finite elements result for clip gage displacement

Design problem	(V_p) by experiment	(V_p) by FEA	Differences between experiment and FEA
SENB	1mm	0.85mm	17.64%

Table 9. Differences between experiment and finite elements result for plastic part of CTOD

Design problem	($(\delta_t)_{pl}$) by experiment	($(\delta_t)_{pl}$) by FEA	Differences between experiment and FEA
SENB	0.286mm	0.242mm	18.18%

Table 10. Differences between experiment and finite elements result for elastic part of CTOD

Design Problem	($(\delta_t)_{el}$) by experiment	($(\delta_t)_{el}$) by FEA	Differences between experiment and FEA
SENB	0.006mm	0.007mm	16.66%

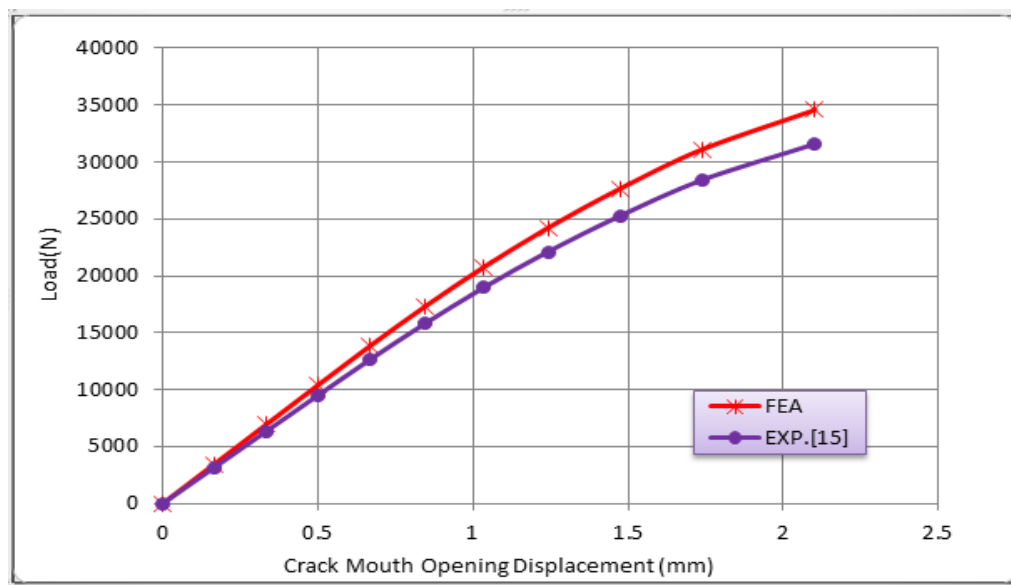


Figure 22. Differences between experimental and finite elements results for bending specimen

11. Conclusions

The following conclusions from the present study can be drawn: -

- ✚ The analysis of the non-linear three-dimensional finite elements model according to ASTM E-399, which has been used to predict fracture toughness directly from ANSYS for compact tension specimens, it provides excellent results for output load-displacement data from ANSYS where it has high accuracy. Thus, the fracture toughness value according to this type of analysis is of high accuracy.
- ✚ The analysis of non-linear three-dimensional finite elements model according to ASTM E-1290, is used to predict fracture toughness by using CTOD-Model for three-point bend specimen, it provides excellent results for the output load-displacement data from ANSYS where it also has high accuracy. The critical value, the elastic and the plastic part for crack tip opening displacement and clip gage displacement according to this type of analysis are accurate.
- ✚ Although, the non-linear analyses have been used for the prediction of fracture toughness according to ASTM E-399, but the relation between load and displacement is linear; this means that no or little plastic deformation occurs in the elements which have been utilized in this program because the behavior of fracture being from the type of brittle mode.

Acknowledgment

First and foremost, all sincere praises and thanks are due to our God. Special appreciation is extended to Assist. Prof. Dr. Abdul-Kareem and Assist. Prof. Dr. Ouday for the sincere and untiring efforts by them in preparing the program and set-ups utilized in this study. Finally, the words are not enough to dedicate thanks to My Family and My Faithful Friends and Colleagues who have never forgotten me.

References

- [1] Hedia H S, Shabara MA, Fattah A A and Helal M M 2005 Effect of Crack Configuration and Pre-Crack Length on Stress Intensity Factors *Journal of Material prüfung* **47** 10 2–7
- [2] Hedia H S, Shabara M A , Fattah A A and Helal M M 2005 Effect of Notch Configuration and Pre-Crack Length on Stress Intensity Factors *Journal of Material prüfung* **47** 10 2–8
- [3] Perez N 2004 *Fracture Mechanics* 1st Ed. (Boston: Kluwer Academic Publishers) pp 47–96
- [4] Sih G C 1976 *Fracture Toughness Concept* America Society For Testing & Materials Special Technical Publication-605 **ASTM STP-605** pp 3–15
- [5] Standard Test Method for Linear-Elastic Plane-Strain Fracture Toughness K_{IC} of Metallic Materials 2008 **ASTM E-399**
- [6] Tornblom S 2007 *Under Matching Butt Weldes in High Strength Steel* **MSc thesis** Department of Civil & Environmental Engineering Lulea University of Tecnology
- [7] Hedia HS and Shabara MA 2002 Effect of Notch Tip Sharpness and Pre-Crack on the Stress Intensity Factor *MP* **44**