Mesh Sensitivity Assessment on 2D and 3D Elastic Finite Element Analysis on a Compact Tension Specimen Geometry Using Abaqus/CAE Software

H. Abdulsalam
Ibrahim Shehu Shema Centre for Renewable Energy Research, Umaru Musa Yar’adua University, Katsina, Katsina State, Nigeria.

Email: hassandaura@gmail.com; Phone number: +2348060015946

Abstract. Finite element analysis has been a tool mostly employed by engineers to conduct stress and strain analysis of a given specimen geometry, after which the numerical and analytical results are compared for validation. The validated results can then be used to predict the stress/strain and fracture mechanics behavior of more complicated structures/components. In this study, Abaqus/CAE software was employed to conduct 2- and 3-dimensional (2D and 3D) elastic finite element analysis on compact tension C(T) specimen geometry, given a specimen width of 50mm, thickness of 25mm, initial crack length of 25mm and total applied load of 1000N. Comparisons of the predicted stress distributions along both X and Y directions (S11 and S22) from the 3D model and those of 2D plane stress/strain obtained from mesh sensitivity results were made. The highest maximum stress values along X and Y directions for 2D plane stress/strain were found to be 297.144Pa and 413.51Pa respectively, while the lowest maximum stress values of 121.681Pa and 166.614Pa for 2D plane stress, 118.121Pa and 165.049Pa for 2D plane strain in the X and Y directions respectively were recorded. Similarly, the highest and lowest maximum stress values for 3D plane stress along X and Y directions were 333.8588Pa and 474.417Pa, and 124.458Pa and 173.422Pa respectively. The stress values were found to increase as the mesh get finer, and also convergence start to initiate as the mesh refinement continued.

Keywords: Abaqus, Elastic, Finite Element, Stress, Strain

1. Introduction

Finite element analysis (FEA) is a computer-based numerical technique popularly used by engineers to solve problems related to stress/strain analysis, fluid dynamics, physical transport phenomena etc. One of the practical advantages of FEA is its ability to deal with problems that lack any standard formula. Over the last decades it has become current practice to many advanced engineering applications [1]. Engineers and researchers frequently employed FEA to perform stress and strain analysis of a given specimen geometry, the numerical results obtained are usually compared with the analytical one to make validation. Once the validation is achieved, then, the results can be used to predict the stress/strain and fracture mechanics behavior of more complicated structures and components [2]. Pinho et al. [3] conducted a study using compact tension geometry to measure the fracture toughness of tensile and compression fiber failure mode and revealed the values of 91.6 kJ/m² and 133 kJ/m² respectively as the initiation and propagation values of the tensile fiber failure critical energy release rate. Similarly, Joes et al. [4] carried out finite element analysis to predict laminated fracture toughness by comparing the results obtained from their study with the one reported for isotropic data reduction of The ASTM standard E399 [5]. They found a result that differs
substantially with that of ASTM. While Mohammed et al.[6] have conducted a detailed numerical investigation of the fracture mechanics via compact tension test specimen (CT) with the objective to standardize the test for composite structure. They found finite element method to be a superb technique to predict fracture action during the crack propagation to failure of composite laminates. This finite element method succeeded in predicting about 97% energy release rate for both initiation and propagation stages and also the fracture history in the laminates which appeared by finite element method contours. In this paper, mesh sensitivity analysis was studied using ABAQUS/CAE, [7], where 2 and 3-dimensional elastic finite element analysis was conducted on a compact tension specimen with a given geometry of 50mm of width, 25mm of thickness, initial crack length of 25mm, a total applied load of 1000N and elastic properties of $E = 140,000,000,000$Pa and Poisson’s ratio value of 0.3.

A model was first set up, loading and boundary conditions were then applied, mesh sensitivity analysis was conducted and results were obtained. Stresses along both X and Y directions were compared at various mesh element number for 2D plane stress, 2D plane strain and 3D plane stress.

2. Methodology

3D Plane stress procedure

Model Set Up

The design parameters given were used to design the compact tension specimen using ABAQUS/CAE as follows;

Creating parts - Parts are the essential blocks of Abaqus/CAE model. In the Abaqus environment, rectangle geometry was created using a length of 62.5mm, height of 30mm and thickness of 25mm [8].

Creating material- Here, two elastic materials were created with elastic young’s modulus of $140,000,000,000$Pa (infinitely large) and $100,000,000$Pa for material 1 and 2 respectively.

Sections- Two sections were created for the two material types and 25mm plane stress/strain thickness was assigned in each of the two material types, and the section with higher young’s modulus was associated with a small area of the total geometry and the other section was assigned to the remaining part of the geometry [9].

Assembly- In a model each part has its own axis orientation, but they can be assembled as one entity, in this model an independent instance type was created, and from Stepmodule, General Static General Basic procedure type was chosen and period was kept at 1 and Nlgeom function switched off [10].

3. Loading and Boundary Conditions

In Abaqus, steps determine the loads and boundary conditions, meaning steps in which the loads and boundary conditions are operating must have to be specified. In this simulation, having created the step, three boundary conditions were applied to the regions where displacement and rotation are specific. [1].

Load- In this study, concentrated mechanical load of magnitude 1000 was enforced on the geometry. The load was applied per node and 0 time/frequency for 0 amplitude, and 1 time/frequency for 1 amplitude were selected [11].
4. Meshing and Job Creation
Discretization of the model was conducted based on the procedure employed by Mehmanparast [2] to develop mesh by partitioning the model geometry into portions which are connected together at node points.
Job was created following the procedure of Chandrupatla [7] by clicking on “create job” button in the abaqus environment and naming the job appropriately, the job manager was then used to submit the job for analysis. After few minutes the analysis was finished and the results were obtained in odb file. Then the stress and the strain values ahead of the crack tip along X and Y axis were obtained for further analysis.

2D PLANE STRESS PROCEDURE
The procedure for 2D plane stress simulation is similar to that of 3D plane stress with the following changes; the modeling space in the creating part has 2D instead of 3D, and the Z-axis has zero value while creating geometry in 2D. The creating material for 2D has elastic young's modulus of 100,000,000Pa (infinitely large) for the small portion of the geometry, while 140,000Pa for the remaining part of the geometry. The interaction for 2D has initial and final coordinates as (0,0) and (1,0). The boundary conditions for 2D were two as against three for 3D simulation. In meshing the 2D geometry, the element shape was created by plane stress, quad-dominated and structured technique [2].

2D PLANE STRAIN PROCEDURE
The 2D plane strain simulation was performed after obtaining the results for 2D plane stress. It was done by selecting the mesh module and clicking on the “assign element type” button, and then selecting the entire geometry and clicking on “done” button. Plane strain was chosen in the pop-up menu, and finally the job was submitted for analysis [2].

5. Mesh Sensitivity Analysis
Mesh sensitivity analysis was conducted to see how the simulation results vary with mesh element size. The analysis was done by changing the element size, for 2D plain stress/strain analysis, mesh size control of 0.05-0.5, 0.03-0.3, 0.02-0.2, 0.01-0.1 and 0.008-0.08 were individually tested, and gave out the following element numbers; 2434, 2754, 3170, 4386, and 5026 respectively, while for 3D plane stress the size control of 0.05-0.5, 0.03-0.3, 0.01-0.1,0.008-0.08 and 0.007-0.08 were employed and generated the mesh element numbers of 52384, 112848, 154560,200384 and 229104 respectively. Then maximum stress values along both X and Y directions from the “contour plot option” were gotten for the 2D plane stress/strain and 3D at each and every mesh element number, and plots relating stress along X and Y direction and element number were produced for 2D plane stress/strain and 3D plane stress [2][12].

Similarly, to perform mesh sensitivity analysis to see the variation of stress and strain distributions ahead of the crack tip, the following procedure was followed; initially, “mesh module” and “option seed edge” were chosen, and then the lines in the semi-circle were highlighted individually, then it was done and in the pop-up menu that appears the mesh control size of 0.03 and 0.3 were applied for the minimum and maximum values respectively, then mesh part instance was clicked upon and “yes” was chosen, then from the job module and under job manager, the new meshed geometry obtained was submitted for job analysis, and after some time slightly larger than the first mesh size, the job analysis was finished, and the stress/ strain values ahead of the crack tip were extracted along both X and Y directions
The same procedure was performed for the individual mesh sizing control of 0.02-0.2, 0.01-0.1, 0.008-0.08 and 0.007-0.07.

6. Results and Discussions

Table 1: 2D Plane Stress/Strain Mesh Sensitivity Analysis Results

| MESH QUALITY | 2D PLANE STRESS | 2D PLANE STRAIN |
|--------------|-----------------|-----------------|
| MESH SIZE    | ELEMENT NUMBER  | S11(Maximum Stress) | S22(Maximum Stress) | S11(Maximum Stress) | S22(Maximum Stress) |
| 0.05-0.5     | 2434            | 121.681          | 166.614          | 118.121          | 165.049           |
| 0.03-0.3     | 2754            | 155.54           | 214.237          | 150.966          | 212.19            |
| 0.02-0.2     | 3170            | 195.437          | 267.893          | 183.842          | 259.321           |
| 0.01-0.1     | 4386            | 275.691          | 380.67           | 273.262          | 381.311           |
| 0.008-0.08   | 5026            | 297.114          | 413.51           | 297.114          | 413.51            |

Table 2: 3D Plane Stress Mesh Sensitivity Analysis Results

| MESH CONTROL SIZING | ELEMENT NUMBER | S11(Maximum Stress) | S22(Maximum Stress) |
|---------------------|----------------|---------------------|---------------------|
| 0.05-0.5            | 52384          | 124.458             | 173.422             |
| 0.03-0.3            | 112848         | 166.028             | 233.156             |
| 0.01-0.1            | 154560         | 277.203             | 392.969             |
| 0.008-0.08          | 200384         | 311.282             | 441.958             |
| 0.007-0.07          | 229104         | 333.858             | 474.417             |

Figure 1: S11 and S22 against Mesh Element Number for 2D plane strain
Figure 2: S11 and S22 against Mesh Element Number for 2D plane stress

Figure 3: S11, S22 against Mesh Element Number for 3D Simulation

Table 1 and 2 present mesh sensitivity analysis result conducted for 2D and 3D simulations, detail procedure for the analysis is given at the beginning of this report. The values of stresses along both X and Y directions (S11 and S22) for both 2D and 3D simulations were the maximum stress values taken from the “contour plot option” at different element numbers. Larger mesh element numbers can be seen in 3D simulation; however, the mesh element number increases as the mesh get finer for both 2D and 3D cases. The highest maximum stress values along X and Y directions for 2D plane stress/strain were found to be 297.144Pa and 413.51Pa respectively at mesh sizing of 0.008-0.08 and mesh element number of 5026, while the lowest maximum stress values of 121.681Pa and 166.614Pa for 2D plane stress, 118.121Pa and 165.049Pa for 2D plan strain in the X and Y directions respectively at mesh...
sizing of 0.05-0.5 and mesh element number of 52384 were recorded. Similarly, the highest and lowest maximum stress values for 3D plane stress along X and Y directions were 333.8588Pa and 474.417Pa, 124.458 Pa and 173.422Pa at mesh sizing of 0.007-0.07 and 0.05-0.5 and mesh element number of 229104 and 52384 respectively. Similarly, Figure 1, 2 and 3 show mesh sensitivity analysis plots for 2D and 3D simulations respectively, all the three figures show increase in stress values along both X and Y directions as the mesh element number increase, also, in each of the three plots S11 and S22 take same curve shape, and there was no convergence when coarse mesh but as the mesh refinement continued, convergence start to initiate. It can be seen that 3D simulation depicts higher convergence than 2D as mesh element number rises; this could be so, considering the fact that the crack tip opening displacement was greater in 3D simulation.

7.0 Conclusions
In conclusion, this paper ordinarily provided an elastic finite element analysis on compact tension specimen geometry for 2 and 3 dimensional cases. Results were gotten from 2D plane stress/strain, and 3D plane stress. Predicted stress distribution along X and Y directions (S11 and S22) from all the cases were compared in tabular and graphical forms. Mostly the results show consistent agreements. The mesh sensitivity analysis reveals the convergence of the stress values along X and Y directions as the mesh element number decreases for all the three scenarios (3D plane stress, 2D plane stress and 2D plane strain). This also indicates that the finer the mesh the less the element number and stress values along X and Y directions and the more the convergence. Similarly, the precision of the results depends upon the degree of the discretization of the specimen geometry. The computational accuracy, time and cost all depend on the mesh size refinement, the finer the mesh size the higher the computational accuracy, time and cost.

References
[1] Bathe K J 1995 Finite element procedures in engineering analysis (Prentice-Hall)
[2] Mehmanparast A 2017 Engineering stress analysis: theory and simulations (Cranfield University Lecture)
[3] Pinho S T, Robinson P and Iannucci L 2006 Fracture toughness of the tensile and compressive fibre failure modes in laminated composites Composites Science and Technology 66 2069-2079
[4] Jose S, Kuma R K, Jana M K and Rao G V 2001 Intralaminar fracture toughness of a cross-ply laminate and its constituent sub-laminates Composites Science and Technology 8 1115–1122
[5] Standard Test Method for Plane-Strain Fracture Toughness of Metallic Materials 1993 ASTM E399-90 Annual Book of ASTM Standards 407–528
[6] Mohammed Y, Hassan M K and Hashem A M 2012 Finite element computational approach of fracture toughness in composite compact tension specimen International Journal of Mechanical & Mechatronics Engineering 12 121504-9797
[7] Chandrupatla T R and Belegundu A D 2002 Introduction to Finite Elements in Engineering (Prentice-Hall)
[8] Grandin H 1986 Fundamentals of the finite element method (Macmillan)
[9] Logan D L 1992 *A First Course in the Finite Element Method 2nd Ed* (PWS Engineering)

[10] Reddy J N 1993 *An introduction to the finite element method* (McGraw-Hill)

[11] Ross C T F 1993 *Finite element methods in engineering science* (Prentice-Hall)

[12] Zienkiewicz O C and Taylor R L 1989 *The finite element method fourth edition,* (McGraw-Hill)