SIMULATION AND EXPERIMENTAL WORK OF SINGLE LAP BOLTED JOINT TESTED IN BENDING

Aidy Ali*, Ting Wei Yao, Nuraini Abdul Aziz, Muhammad Yunin Hassan and Barkawi Sahari

Received: Jun 13, 2007; Revised: Nov 29, 2007; Accepted: Nov 30, 2007

Abstract

This paper presents the simulation and experiment work on the prediction of stress analysis in a single lap bolted joint under bending loads. A three-dimensional finite element model of a bolted joint has been developed using MSC Patran and MSC Nastran FEM commercial package. In the simulation, different methods in modelling the contact between the joint, which affects the efficiency of the models were detailed. Experimental work was then conducted to measure strains and deformations of the specimens for validation of the developed numerical model. A four-point bending load type of testing was used in both the simulation and experiment works. The results from both simulation and experiment were then compared and show good agreement. Several factors that potentially influenced the variation of the results were noted. Finally, critical areas were identified and confirmed with the stress distribution results from simulation.

Keywords: Bolted joints, FEM, MSC Nastran, MSC Patran

Introduction

A bolted joint is one of the joining techniques employed to hold two or more parts together to form an assembly in mechanical structures. One of the advantages of a bolted joint over other joint types, such as welded and riveted joints, is that they are capable of being dismantled. A bolted joint is considered as an important element to be employed in civil, mechanical and aeronautic structures.

The previous studies on the behaviour of a single lap bolted joint were mostly dedicated to tension load at the end of plate. An early study was performed by Ireman (1998) in which a three-dimensional finite element model of bolted composite joints was developed to determine the non-uniform stress distribution through the thickness of composite laminates in the vicinity of a bolt hole. The single lap bolted joint was loaded in tension. A similar study was carried out by McCarthy *et al.* (2005) where the effects of bolt-hole clearance on the mechanical behaviour of a bolted composite (graphite/ epoxy) joint were investigated. Furthermore, Ju *et al.* (2004) studied the nominal applied

Fatigue and Fracture Research Group, Department of Mechanical and Manufacturing Engineering, University of Putra Malaysia, 43400 Serdang, Selangor, Malaysia. E-mail: aidy@eng.upm.edu.my
* *Corresponding author*

force of the steel bolted connection. Relatively few studies have dealt with the behaviour of a bolted joint in bending. For instance, Krishnamurthy (1980) investigated the behaviour of a steel bolted connection where the connection transfers the bending moment from the beam to a column or to the cross section of another beam.

Kovács *et al.* (2004) carried out an experimental study on the behaviour of bolted composite joints. The composite base columns were investigated under cyclic loading. Su and Siu (2004) analysed the nonlinear response of a bolt group under in-plane loading using the numerical method. In order to predict the physical behaviours of the structure with a bolted joint, simulation with three dimensional finite element models is desirable. With the recent increase in computing power, three-dimensional finite element modelling of a bolted joint in bending has become feasible.

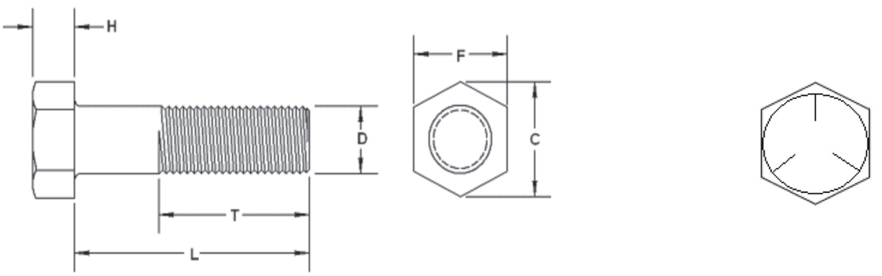
Many structural applications are subjected to bending forces which effectively result in combined tension and shear loads acting simultaneously on the fastener. This commonly occurs in building frames and bridge deck systems when the connections are required to transmit moments to ensure continuous structural action. The study of a single lap bolted joint in bending is an important topic especially in civil and mechanical structures application. Although much research and testing has been accomplished by applying shear loads, there is still limited bending load analysis for many alternative details. Research should be pursued until cost effective and attractive solutions are available for all subjected force types. Therefore, the objectives of this work are: (1) to develop a finite element model of a single lap bolted joint in bending, and (2) to analyse the stress distribution in the bolt and the plate under bending loads.

Materials and Methods

A single bolted lap joint consists of a bolt, a nut, washers and two plates. Most often bolts used in machining are made to SAE standard J429 (Unified Engineering, 2006). The bolt and nut selected is a hex bolt of SAE Grade 5, 0.5 inches in diameter. Figure 1 illustrates the geometry and marking of the standard SAE Grade 5 hex bolt. The mechanical properties and dimensions of the bolts are shown in Table 1. Washers are not used in the design of the specimen because of its influence on the accuracy of torque controlling. The plate material properties used is based on mild steel which is shown in Table 2.

Simulation Work

The simulation work for this study was carried out using the finite element method. In this work, a finite element software program named MSC PATRAN and MSC NASTRAN was used to simulate a single lap bolted joint.



(a) (b)

Figure 1. (a) Geometry and (b) marking of SAE Grade 5 Hex bolt

Geometry Modelling

In simulation, the geometry scale factor is determined as 1,000 (millimeter). Generally, the unit used in MSC Patran modelling is listed in Table 3 below.

The geometry of the model is based on two solid plates (170 × 74 × 9 mm), one support plate (40 × 74 × 9 mm), and a unit consisting of a bolt and a nut. The bolt hole, of radius 6.25 mm, is edited in the upper and lower plate by Boolean method and the substrate. The diameter of the bolt shank is assumed to fully fit in the bolt hole. Figure 2 shows the geometry of the FEM model. As illustrated, the position of origin is pointed by the arrow; direction represents the length of the plate, y direction represents the width of the plate, and z direction represents the thickness of the plate.

Table 1. Properties and dimension of the bolt and nut

Material type	Medium carbon steel,Quenched and tempered
Modulus of elasticity, E	200 GPa
Poisson's ratio, Å	0.29
Proof strength	586 MPa
Minimum tensile yield strength	634 MPa
Minimum tensile ultimate strength	827 MPa
Nominal length, L	38.1 mm
Nominal diameter, D	12.3 mm
Height of bolt, H	7.0 mm
Width across flat, F	18.5 mm
Width across corners, C	21.5 mm
Height of nut	11.0 mm

Table 2. Properties and dimension of the plates

Material type	Mild steel
Modulus of elasticity, E	200 GPa
Poisson's ratio, Å	0.29
Thickness	9.0 mm
Width	74.0 mm

Table 3. Unit and parameter used in MSC Patran Modelling

Parameter	Unit (SI)
Length	mm
Force	N
Mass	Tonnes
Time	Second
Stress	MPa (N/mm^2)
Density	Tonnes/mm

In order to assist the analysis, the stresses at the regimes of interest were analysed as labelled in Figure 3. The stress distribution along the middle of surfaces of upper plate symmetry is taken from Profile (a) and Profile (b). Profile (c) and Profile (d) are designed to compare the stress distribution along the bolt hole in different locations. Profile (e) is taken along the middle of the bolt.

Mesh Generation

The element used on the bolt and plates is the same with SOLID type Tet10. The total elements in the model are 6,665 and the numbers of nodes are 11,954. The action of "Equivalent" is applied on the mesh of the model to delete the nodes which are duplicated between the surface contacts. Finally, the mesh was verified to test the failure of aspect ratio, edge angle, face skew, collapse, normal offset, tangent offset, Jacobian ratio and Jacobian zero. The verification summary (Figure 4) showed the number of failures the elements involved. Zero number of failures is desired to minimize error in the analysis process.

A contact element was introduced through multi-point constraint (MPC). MPC is employed in the finite element model to solve the problem of the more detailed free body load in the overall model. MPC is A linear combination of displacement. Generally, one displacement is dependent on the remaining independent displacement (Brown, 1987). In this model, three types of MPC were introduced in different situation. The first MPC is created between the bolt and the two plates. The MPC type used is Rigid (fixed). A single point constraint is used to prevent rigid body motion. As shown in Figure 5, Node 10824 in the middle of the bolt shank was defined as an independent term and the nodes on the bolt hole of the plates are defined as dependent terms. The second MPC, RBE2 was employed on the overlap region between the upper plate and support plate as illustrated in Figure 6.

Finally, the third MPC Explicit was used to define the contact condition between two plates. In this technique, the dependent term is on the edge of the lower plate while the independent terms are the nodes of the upper plate in the overlap region. The coefficient of friction is input and 6 degrees of freedom is selected. The value of coefficient of friction is assumed as 0.74 (Beardmore, 2007).

Boundary Condition

In this section, displacement is created to constrain the model from translating when a bending load is applied. The ideal displacement is applied in a line/curve to uniform the displacement distribution. The locations of displacement are 25 mm and 225 mm in x direction from the origin. However, in this model, the application

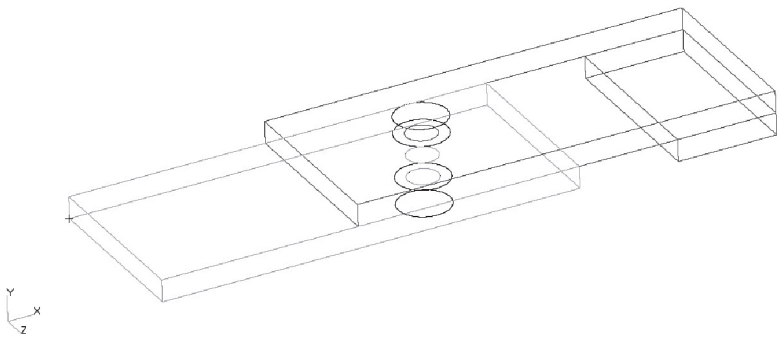


Figure 2. Geometry modelling and position of origin

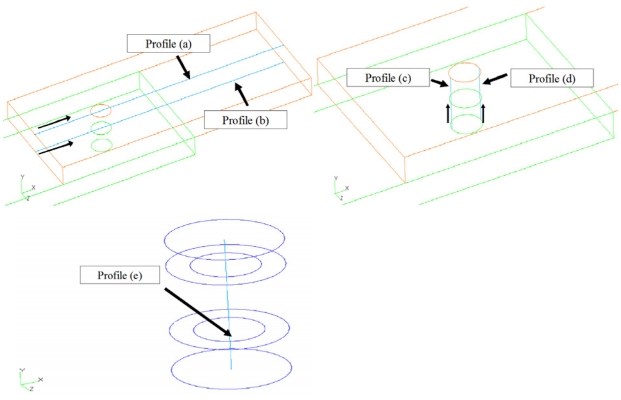


Figure 3. Profile (a) to (e) of interest in stress distribution

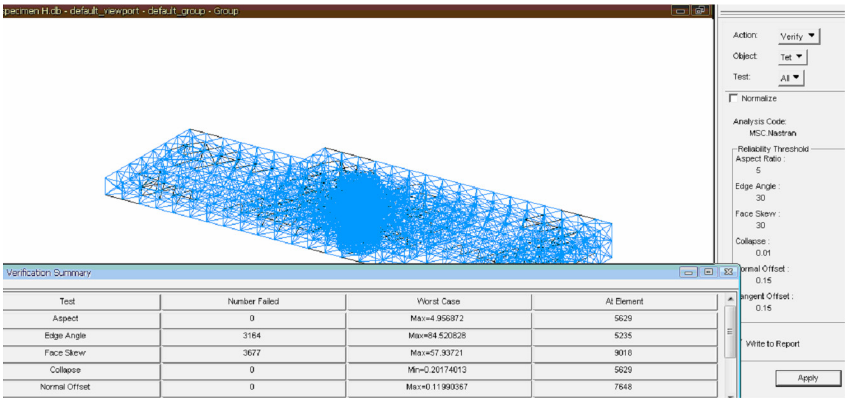


Figure 4. Verification of mesh

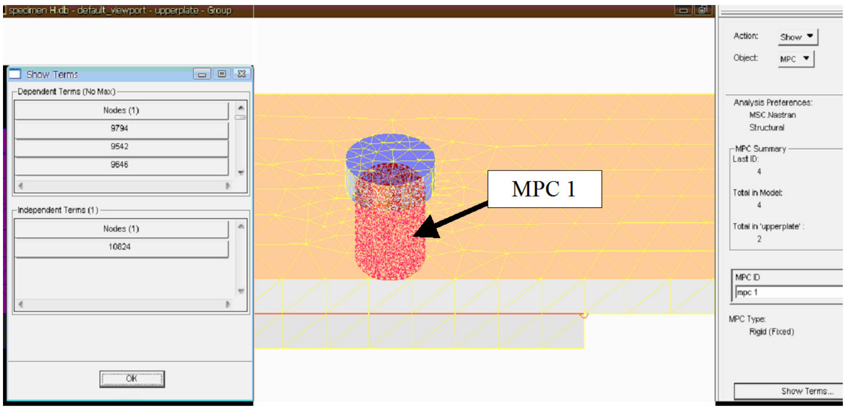


Figure 5. Location and terms of MPC 1

region is determined by those nodes which are on the desired lines (Figure 7). These nodes are constrained by fixing the degrees of freedom; translations are in x, y, and z directions.

Loadings

In this step, two types of loads were considered which are the clamping force and bending force. The clamping force is applied by uniform pressure onto the overlap area between the bolt nut and the plates (Figure 8). A clamping force of 100 N is modelled by using a negative pressure value. The bending force is applied onto the upper plate by selecting those nodes which are close to the ideal location as shown in Figure 9. The load locations applied is 85 mm and 165 mm in direction x in origin.

Experimental Procedures

The experimental work was carried out in the Strength Laboratory at UPM to test four-point-bending on a single lap bolted joint. The results obtained in the experimental measurements are load-displacement and load-strain data. The experimental load-displacement curves were

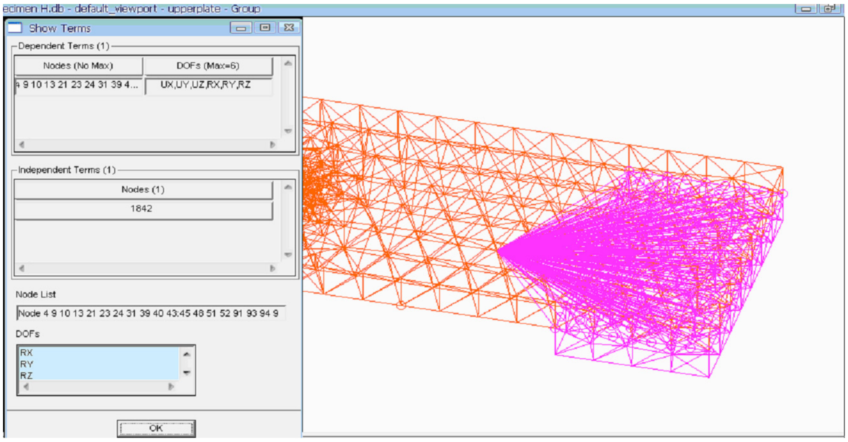


Figure 6. MPC 2 between upper plate and support plate

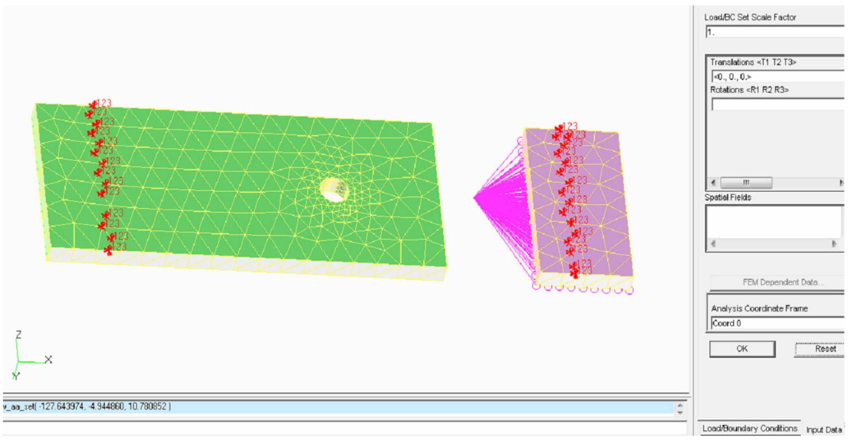


Figure 7. Boundary condition of the joint

obtained directly from the testing machine. Strains at selected points on the joint surface were measured while bending forces were controlled manually by the Instron Universal Testing Machine. The surface of the lower plate was strain-gauged in the axial direction. Figure 10 shows the position of the unidirectional strain gauge, which had a 5 mm gauge length. Axial strain in the plate is observed for every increment of 1 kN. The specimen and jigs were mounted on the universal testing machine as illustrated in Figure 11. The bending load was applied by the lower load cell vertically, while the top load cell was fixed. The application of the load was controlled manually to obtain the strains measurement for every increasing load level.

Results and Discussion

Joint Deformation

The deformed shape of the finite element model is shown together with the actual deformation in Figure 12. The experimental load-deflection curves were found to be essentially

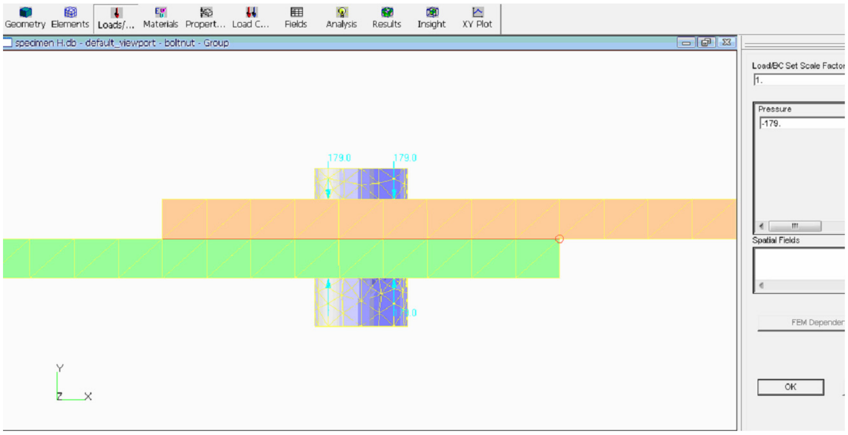


Figure 8. Clamping force in uniform pressure

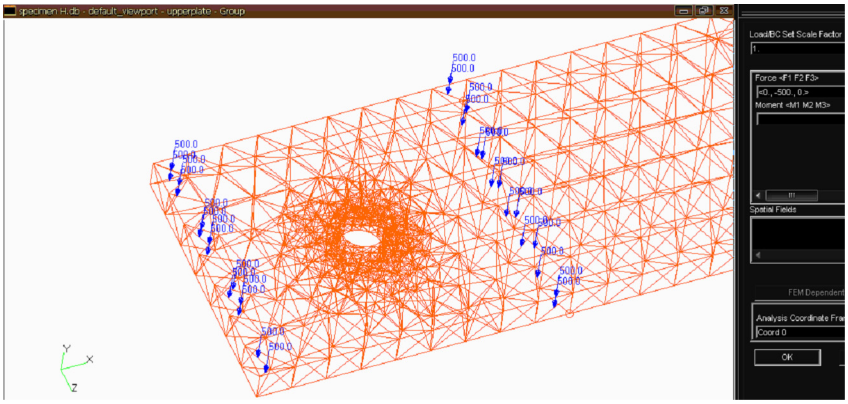
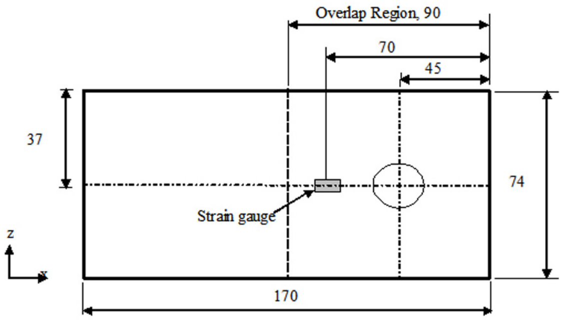
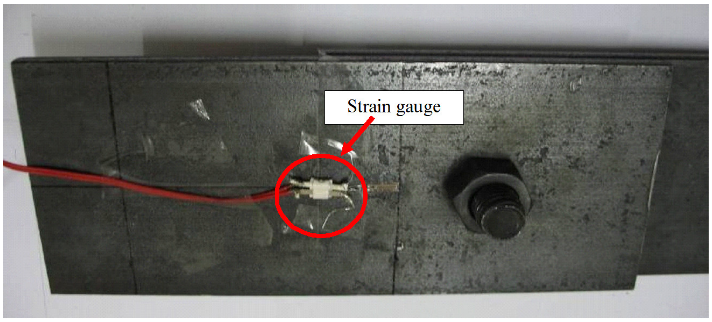


Figure 9. Application of bending load



All dimensions in *mm*

(a)



(b)

Figure 10. Strain-gauge (a) location (b) set up

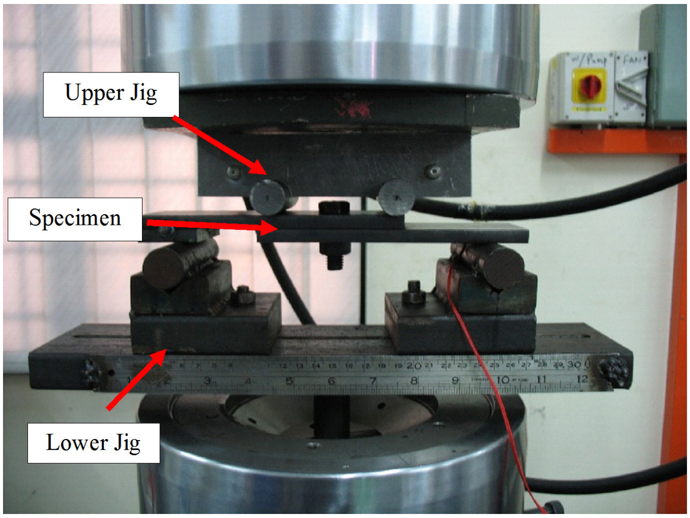


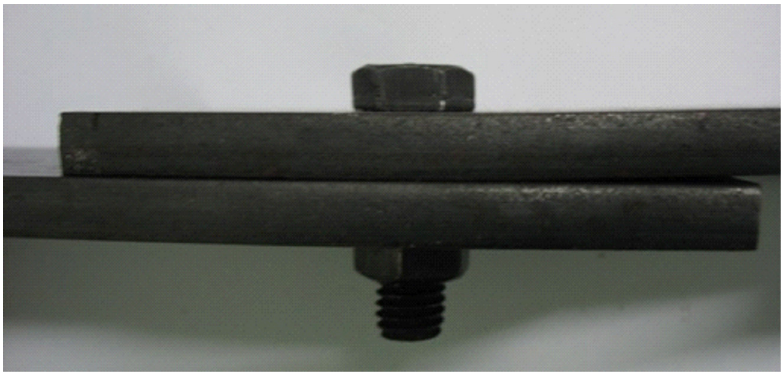
Figure 11. Experimental configuration

linear up to and including 7 kN. Since the experimental specimen deformation becomes non-linear plastic behavior after 7 kN, the shape of the experimental and finite element model with applied load 7 kN is shown for comparison.

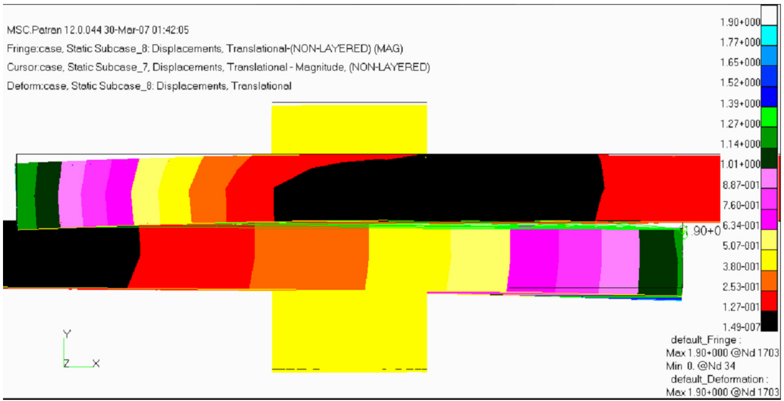
It can be seen that the finite element model displays similar deformation shape to the experiment. From Figure 12(b), penetration of the master line (upper plate) into slave nodes (lower plate) shows the effect of contact between two deformable bodies. It is seen from Figure 12(a) and (b) that the agreement of the presence of a gap between the upper plate and lower plate due to the effect of the bending load is generally good. The experimental load-displacement curves were found to be essentially linear until an applied load of 7 kN, so the displacements of the experimental and finite element model were measured over this range. Figure 13 shows the calibration curves of load-displacement of the experimental and finite element model under elastic region of three tested samples.

Progressive Stress Distribution at Different Load Level

In this section, stress distribution of the joint is observed at different load levels to track the stress distribution in each parts of the joint as shown in Figure 14. This technique possesses accurate prediction of stress distribution. The applied load levels used in the analysis are 0 kN, 1 kN, 4 Kn, and 7 kN. The



(a)



(b)

Figure 12. Joint deformation of (a) experimental specimen and (b) FEM model

relative magnitude of the stress distribution on the bolt head decreases with the increasing load level when compared with the stress on the plates. It can be observed that the bolt nut starts to loosen when bending force is applied. As expected, higher stress is distributed at the bending and constraint area as illustrated in Figure 15 when the specimen experienced 15 kN bending load. Non-symmetrical stress distribution along the x-axis is due to the non-uniform mesh in the model.

Stress Distribution of Interest Regimes

Stress distribution along a path is studied by developing several regimes of interest in order to assist the analysis. Profile (a) and Profile (b) given in Figure 16 show the comparison of stress distribution on both surfaces of the upper plate. The purpose of mapping these profiles is to observe the tension and compressive stress of bending. Along the path of both profiles, the stress increases gradually to the edge of the bolt hole, no stress exists in the hole, and the stress decreases smoothly to the end of the plate. The upper and lower stress distributions observed were similar; these represent the maximum tension and maximum compressive stress having the same value as well as established in text books. The highest stress is found at the edge of the bolt hole which is caused by the combination of clamping force and bending force.

In order to measure the stress distribution along the bolt hole surface of both plates, Profile (c) and Profile (d) were developed. The stress in this section is the same as the stress in the bolt for no clearance between the hole and bolt. The paths of both profiles begin from the edge of the lower plate bolt hole along y axis. The results of the stresses distribution of these two profiles are shown in Figure 17. The trend of stress along the path shows that the maximum stress occurs between the contact of the upper and lower plates. This phenomenon is caused by the combination of contact stress, clamping force and bending force. Obvious differences between the stress of Profile (c) and Profile (d) can be seen. This occurs when heavier contact stress between the bolt shank and bolt hole occurs due to bending. Then again, higher stress at the edge of the bolt hole shows agreement with the clamping force between the bolt nut and plates.

Finally, the stress distribution in the bolt is measured by observing Profile (e) as shown in Figure 18. It is interesting to note that the

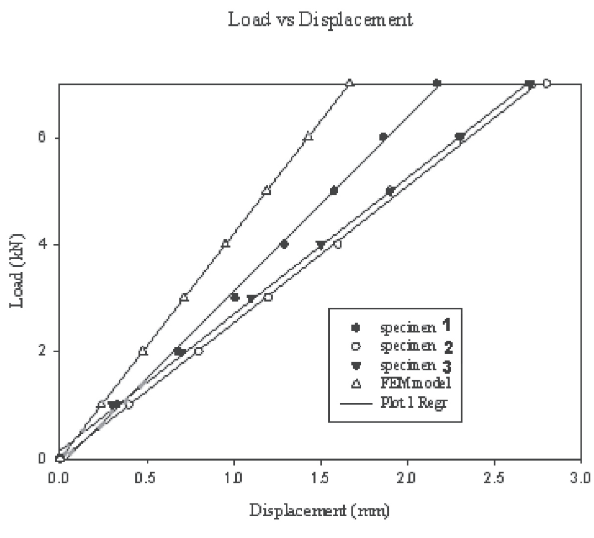
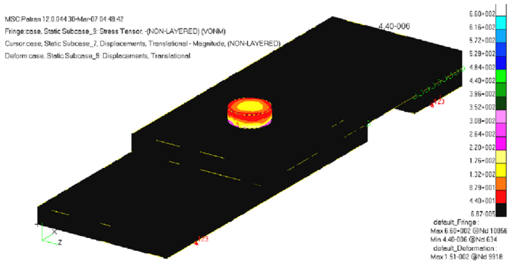
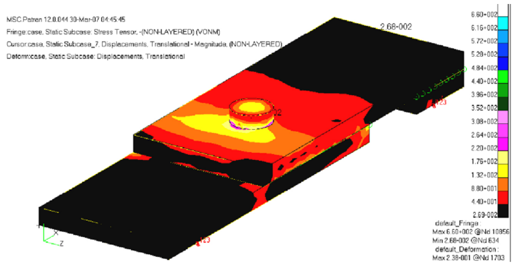


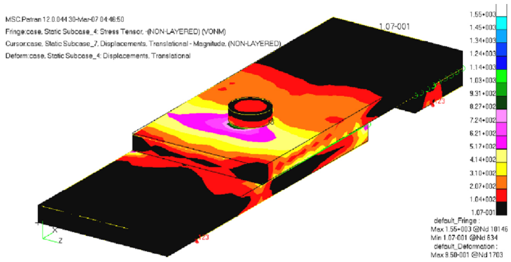
Figure 13. Comparison of load-displacement curve between the experiment and FEM model



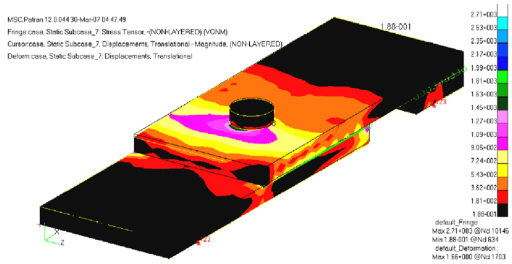
(a)



(b)



(c)



(d)

Figure 14. Progressive stress distribution at different load levels (a) 0 kN, (b) 1 kN, (c) 4 kN, and (d) 7 kN

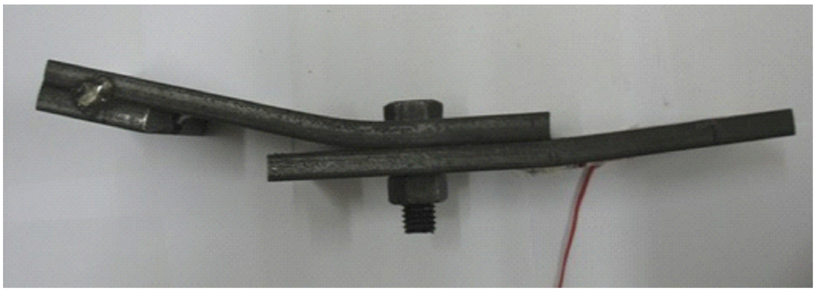


Figure 15. Deformation of specimen after 15 kN bending force

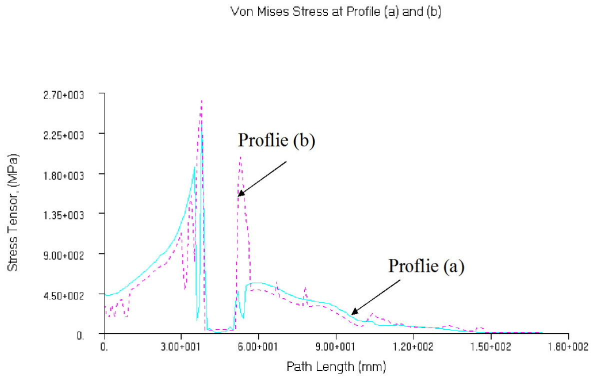


Figure 16. Comparison of Profile (a) and (b) stress distribution along the path

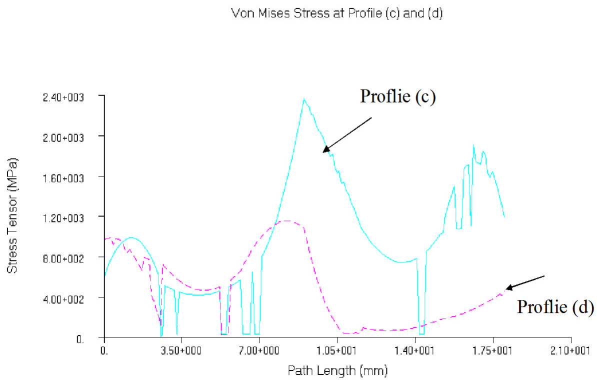


Figure 17. Comparison of Profile (c) and (d): stress distribution along the path

stress changes non-uniformly along the path and increases sharply at the end of the bolt head. This phenomenon is due to the bending force increasing the tension in the bolt head.

In spite of the critical area occuring on the surface of upper plate of the single lap bolted joint as predicted, it is interesting to investigate the critical area of the bolt and nut, since it was the main component in the joint. From the simulation result, it was observed that the critical areas in terms of Von Mises stress, of the bolt and nut were between the clamping surfaces of the bolt head. The critical area of the FEM bolt-nut as well as the experimental specimen bolt is shown in Figure 20. The simulation shows that the bolts will experienced cracks and failures. On the other hand, the plate will experienced deformation due to the bolt penetration will loaded in bending. The joint strength of the bolted joint still relies on the strength of its bolts in both cases, when the tension and bending loads were applied.

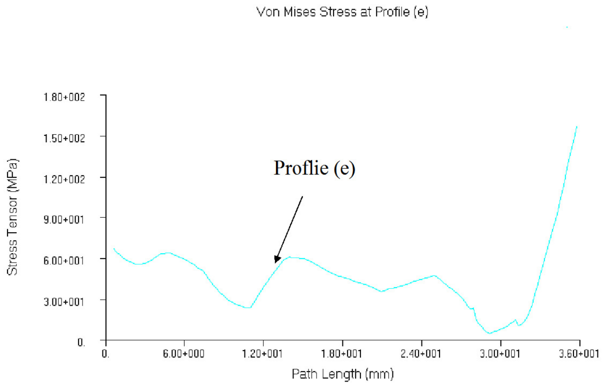
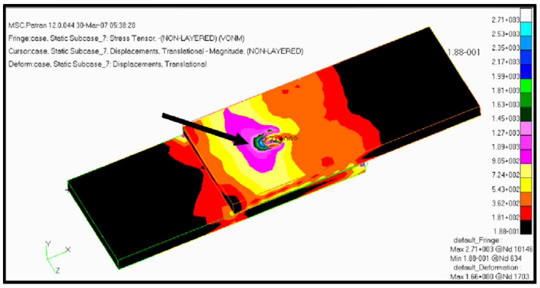
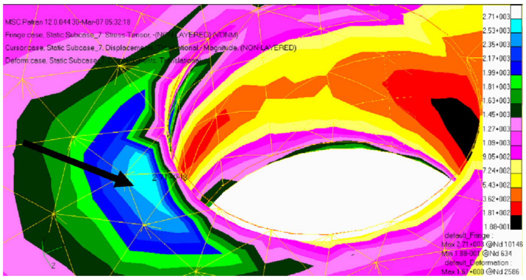


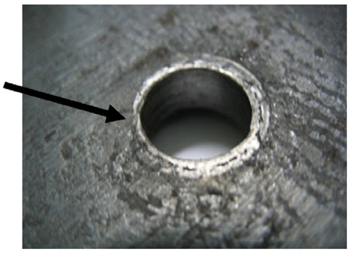
Figure 18. Profile (e) stress distribution along the path



(a)

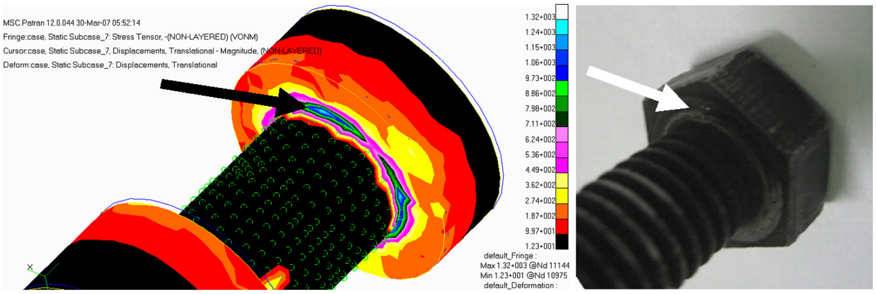


(b)



(c)

Figure19. (a) Critical area in terms of Von Mises criterion in the joint (b) closer view of the critical area (c) experimental specimen. (The arrows points to the critical area of the joint)



(a) (b)

Figure 20. Critical area in terms of Von Mises criterion of the bolt (a) FEM model (b) Experimental specimen (The arrows points to the critical area of the bolt)

Conclusions

A three-dimensional finite element analysis was used to examine the stress distribution of a single lap bolted joint and comparisons were made to the experimental results. These comparisons include surface strain and joint displacement measurement. The result obtained from the simulation analysis shows agreement with experiment analysis and validation of the bolt model was confirmed for an applied load less than 7 kN. A finite element method has been successfully developed and it can be applied for the prediction of other material, load and size of geometry in four-point bending of a single lap bolted joint. The critical area could be predicted from the simulation analysis and it could save cost of carrying out experimental work. The FEM gives good control of experimental techniques, confirming, complementing and refining the specimen design before commencing experiment tests.

Acknowledgement

The authors wish to thank the Department of Mechanical Academic and Technical Staff at University of Putra Malaysia for the support.

References

Beardmore, R. (2007). Friction factors. UK. Available from: http://www.roymech.co.uk/Useful_Tables/Tribology/co_of_frict.htm. Accessed date: Mar 15, 2007.

Brown, J.E. (1987). Using multi-point constraints with MSC/pal2 for detail free body analysis. Los Angeles, CA. Available from: http://www.mscsoftware.com/support/library/conf/wuc87v2/p01087.pdf. Accessed date: Mar 28, 2007.

Ireman, T. (1998). Three-dimensional stress analyses of bolted single-lap composite joints. Composite Structure, 43(3):195-216.

Kovács, N., Calado, L., and Dunai, L. (2004). Behaviour of bolted composite joints: experimental study. Construction Steel Research, 60(3-5):725-738.

Ju, S.H., Fan, C.Y., and Wu, G.H. (2004). Three-dimensional finite elements of steel bolted connections. Engineering Structures, 26(3):403-413.

Krishnamurthy, N. (1980). Modelling and prediction of steel bolted connection behaviour. Computer and Structures, 11(2):75-82.

McCarthy, M.A., McCarthy, C.T., Lawlor, V.P., and Stanley, W.F. (2005). Three-dimensional finite element analysis of single-bolt, single-lap composite bolted joint: Part I – Model development and validation. Composite Structure, 71(2):140-155.

Su, R.K.L. and Siu, W.H. (2004). Nonlinear response of bolt groups under in-plane loading. Engineering Structures, 26(3):403-413.

Unified Engineering. (2006). Hex head bolt makings. Aurora, IL. Available from: Inc. http://www.unified-eng.com/scitech/bolt/boltmarks.html. Accessed date: Apr 26, 2007.