THREE-DIMENSIONAL FINITE ELEMENT ANALYSIS OF SINGLE BOLTED ALUMINIUM LAP JOINT AND VALIDATION BY XRD TECHNIQUE

1SREESANKAR K. K., 2AMAL HISHAM E. SHEREEF, 3ADARSH DINESH, 4DHANUSH V., 5RASIL RAJ P. V.

1,2,3,4Department of Mechanical Engineering, College of Engineering Thalassery
E-mail: sreesankarkk@gmail.com, amalhishames@hotmail.com, adarshdinesh92@gmail.com, dhanushv2030@gmail.com, rasilraipv@gmail.com

Abstract- In structural applications such as aircraft, spacecraft and civil engineering structures, composite components are often fastened to other structural members by bolted joints. Bolted joints being very often the critical part of the structure, it is therefore important to design them safely. In this study, Three-dimensional finite element model have been developed. Investigations on the effect of failure criteria including the behavior of bolted joints in aircraft using aluminium laminates are examined. The joint type studied is single - bolt, single - lap joint. Load bolt displacement curves and stress around the hole is analyzed. The model is generated in Finite Element Software, FEAST\textsuperscript{SMT} and attempts are made to validate it by X-ray diffraction technique.

Keywords— Bolted Joints, Aircraft, Spacecraft, Finite element analysis, Stress concentration, Failure criteria, X-ray diffraction.

I. INTRODUCTION

Bolted lap joints are often used in structures and machines to join several elements to one another and to form a structural assembly. Bolted joints are one of the main factors to determine the structural answer to dynamic requirements. Bolted joints have large scale application in aeronautical and aerospace industries as the joints can be dismantled any time for maintenance. Bolted joints being very often the critical part of the structure, it is therefore important to design them safely. Since bolted and riveted parts are stress raisers due to geometrical discontinuities at holes, fatigue cracks often initiate at riveted and bolted joints which result in unexpected failures. Bolted joints require hole to be drilled in the structure and hence large stress concentration tends to develop around the hole, which can severely reduce the overall strength of the structure. Joints represent potential weak points in the structure. Hence, the design of the joints can have a large influence over the structural integrity and load carrying capacity of the overall structure. Due to factors such as bolt bending and tilting, bolt preload due to torqueing, stress and strains in bolted joints vary three – dimensionally. In addition, the stress field near the hole is due to the presence of preloading.

Majority of finite element studies carried out before are based on two-dimensional approach as model development time and processing power with three dimensional analyses is quiet high. In this study, Three-dimensional finite element model of a single bolted lap joint have been developed. Investigations on the effect of failure criteria including the behavior of bolted joints in aircraft using aluminium laminates are examined. Load bolt displacement curves and stress around the hole is analyzed. The model is generated in Finite Element Software, FEAST\textsuperscript{SMT} and attempts are made to validate it by design calculation.

The objective of the study is to model the single – bolt, single – lap joint in finite element analysis software and investigate and predict the stress concentration pattern around the hole of the bolted joint. The study of stress distribution pattern is important for the safe and reliable design of bolted joints. Pre-loading is analogous to compressing a spring. Plates acts as spring. In a bolted joint during pre-loading, the shank of the bolt will be in tension and plates will be in compression. For analyzing using software, similar condition shall be reproduced. This paper presents an analysis of a bolted lap joint, subjected to a relative displacement after applying a pre-load on the bolt in order to characterize the joint behavior. Pre-loading was given to the bolt joint by rigid link method. Accuracy of the model is critically examined by comparing results with design calculations. Attempts are made to improve the model through a series of mesh refinements, increases in element order and modifications to boundary conditions, material modeling and analysis type. The single - bolt, single - lap joint was chosen as it provides a suitable test case for three-dimensional modeling since it involves three-dimensional variations in stress and strain.

The report is organized as follows. In section 2, the problem has been defined. Section 3 describes the finite element model of the bolted lap joint which includes meshing method, contact element description, constraints and application of force and material property and explains the result obtained from FEM. The model validation by X-ray diffraction
technique has been explained in section 4. Short conclusion ends the paper.

![Figure I. Specimen geometry](image)

**II. PROBLEM DEFINITION**

An aluminium lap joint of length, L and width, W with hole diameter, D is considered. The hole is at a distance of e from the free edge of the plate. The thickness of the plate is H. Two aluminium plates of Elastic Constant, \(E = 70,000 \text{ N/mm}^2\) and poison ratio, \(\nu = 0.3\) is studied. The bolt considered is of steel and has an Elastic Constant of \(E = 210,000 \text{ N/mm}^2\) and poison ratio, \(\nu = 0.3\). The lap joint considered for study is shown in Fig. I. The plate considered has a length of, \(L = 100\text{mm}\), width, \(W = 50\text{mm}\) and thickness, \(t = 10\text{mm}\). The bolt considered has a head diameter, \(D_h\) of 16mm and shank diameter, \(d\) of 8mm. The diameter of the hole, \(D\) is 9.6mm.

![Figure II. Rotscher's cone.](image)

During preloading of the bolt joint, the compressive stress distribution in the vicinity of hole is assumed to follow a conical pattern i.e. the pressure falls off further away from the bolt. Thus it suggests the use of Rotscher’s pressure-cone method for stiffness calculations with a variable cone angle. This method is quite complicated, and so simpler approach using a fixed cone angle is used.

Fig. II illustrates the general cone geometry using a half apex angle \(\alpha\). An angle \(\alpha = 45^\circ\) has been used, but this overestimates the clamping stiffness. When loading is restricted to a washer-face annulus (hardened steel, cast iron, or aluminium), the proper apex angle is smaller. Osgood reports a range of \(25^\circ \leq \alpha \leq 33^\circ\) for most combinations. In this analysis, \(\alpha\) is taken as \(30^\circ\), except in cases in which the material is insufficient to allow the frusta to exist.

\[
d\delta = \frac{pdx}{EA} \quad (1)
\]

The area element,

\[
A = \pi\left(r_0^2 - r_1^2\right) = \left[\left(\frac{x\tan(\alpha)}{2} + \frac{D_h}{2}\right)^2 - \left(\frac{D_h}{2}\right)^2\right]
\]

Substituting Eq. (2) in Eq. (1) and integrating gives the total contraction,

\[
\delta = \frac{p}{\pi E} \int_0^L \frac{dx}{\left[\left(\frac{x\tan(\alpha)}{2} + \frac{D_h}{2}\right)^2 - \left(\frac{D_h}{2}\right)^2\right]}
\]

Thus the spring rate or stiffness of this frustum \(k_p\) is,

\[
k_p = \frac{p}{\delta} = \frac{\pi E \tan(\alpha)}{\ln\left(\frac{2\left(\tan(\alpha) + D_h/2\right)}{2\left(\tan(\alpha) + D_h/2\right) - D_h}\right)}
\]

Stiffness of the bolt \(k_b\) can be calculated by the equation;

\[
k_b = \frac{D_2 E_2}{l} \quad (5)
\]

**III. FINITE ELEMENT MODEL**

A. Meshing method

The deformation behavior of the single bolted lap joint is predicted using a three-dimensional finite element model developed in the software FEAST. Equivalent model of bolted joint is discretized by meshing. Mesh density is made higher near the holes, where high strain gradient exist. Moving radially out, the bias factor is made greater than one so that meshing becomes uneven. Nodes are defined at locations where changes of geometry or loading occur. Changes in geometry relate to thickness, material and/or curvature. The course and biased meshing helps to decrease the number of elements and in turn decrease the degrees of freedom (dof) without affecting the quality of meshing. Lesser degree of freedom means less time required for the solver to compute and less chance of crushing due to garbage value. Care should be taken when discretizing to avoid developing finite elements of high aspect ratio. Aspect ratio is the ratio between the largest and smallest dimension of a 2-D or 3-D element. As a rough...
guideline, elements with aspect ratios exceeding 3 should be viewed with caution and those exceeding 10 with alarm. Fig. III shows the finite element model of the plate.

B. Contact elements

![Figure IV. Contact elements.](image)

The method requires the definition of “contact bodies”, i.e. bodies that potentially come in contact with each other. In the Finite Element model the parts which are in contact are, two laps which forms the lap joint, bolt head and first lap, nut and second lap. Diameter of the shank is made sufficiently small compared to the hole diameter so that the curved surface area of the shank doesn’t come in contact with the hole. Gap element is provided between nodes which comes in contact which each other. Fig. IV shows the contact elements between the two plates. They are two-node elements formulated in three dimensional spaces. Stiffness of the gap element is given very high so that it won’t behave as an elastic body.

The contact bodies should have identical meshing so that the corresponding node of one part is right under the other part.

C. Force application and constraints

The objective is to reproduce the equivalent effect in a lap joint tightened by nut and bolt arrangement. In the practical case the shank of the bolt will be in tension and head of the bolt as well as nut will be in compression when the nut is being tightened. In the Finite Element model, attempts are made to generate the same condition by providing a rigid link. A portion of the shank from the center (say a split of 1mm) is removed by deleting elements. At the ends of remaining split shank slave nodes of rigid links are linked and the master node is given a force in Z-axis. The shank is split into two halves, for convenience say upper shank and lower shank. The end nodes near the split of lower shank are connected to slave nodes of rigid link whose master node is in negative Z direction downward) and vice versa. In this analysis, the materials of the structures used in the finite element model are assumed to be linearly elastic. The geometric stiffness, nonlinear strain, of the element and the friction force between two surfaces due to sliding are not considered. Therefore the loading and unloading do not dissipate energy. The stress and strain of the structure is completely defined by the final deformed geometry which is independent of loading history. Fig. V shows the method of application of force by using rigid link.

![Figure V. Force application by rigid link.](image)

D. Material modeling

The plate material considered for the analysis is aluminium. Aluminium is chosen because it is the most widely used material in the design of aircraft structures. Aluminium is remarkable for the metal's low density and for its ability to resist corrosion due to the phenomenon of passivation. Structural components made from aluminium and its alloys are vital to the aerospace industry and are important in other areas of transportation and structural materials. The material of plate is assumed to be linear elastic behavior during clamping. M8 steel bolt is considered for the analysis. The input mechanical properties of the material used in linear elastic finite element analysis for the bolted joint is given in Table I. The plate is modeled by using four layers of brick elements having eight nodes. Each node has three degrees of freedom. The mechanical properties of the bolt material are shown in Table II.

<table>
<thead>
<tr>
<th>Table I. Mechanical properties of Aluminium.</th>
</tr>
</thead>
<tbody>
<tr>
<td>Density (g/cm³)</td>
</tr>
<tr>
<td>-----------------</td>
</tr>
<tr>
<td>2.70</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Table II. Mechanical properties of Steel.</th>
</tr>
</thead>
<tbody>
<tr>
<td>Density (g/cm³)</td>
</tr>
<tr>
<td>-----------------</td>
</tr>
<tr>
<td>7.85</td>
</tr>
</tbody>
</table>

E. Finite element results

a. Resultant deformation

Fig. VI shows the deformation of the upper and lower plate at a magnification factor 10. This effect is characteristics of bending of plate [7, 8]. Appealingly,
the transverse bending changes from concave about half way between the hole and the clamped end to convex near the end of the plate. The concave transverse bending is result of high, localized contact force from the bolt head and nut acting in the direction of thickness. Fig. VII shows the deformed shape of the bolted joint during preloading.

![Figure VII. Deformed shape.](image)

b. Compressive stress distribution

Fig. VIII shows the compressive stress distribution in the plate in z-direction. The compressive stress is due to the contact force exerted by the bolt head and nut in the direction of thickness. The distribution follows a conical pattern rather than a cylindrical stiffness pattern in the z-direction. In most of the standard design problems taking compressive stress distribution along the plates in a bolted joint to be cylindrical is justifiable. When sophisticated structures like aircraft, spacecraft, pipes carrying chemicals in chemical industry etc are considered the former assumption can lead to catastrophic failure of the whole structure by accelerating the bolt failure. Hence in these cases taking stress distribution to be conical is mandatory.

IV. MODEL VALIDATION

In this section, results from the three-dimensional finite element model developed in the previous sections are compared with results from X-Ray diffraction method. Residual stress at any point can be found out using XRD technique. Residual stress by this method relies on the fundamental interactions between the wave front of X-ray beam and the crystal lattice (i.e. Bragg’s law) [9].

\[ n\lambda = 2dsin\theta \]  
\[ (6) \]

Direct measurement of stress is not possible using XRD method. It is only possible to measure the strain induced on the material. To perform strain measurements the specimen is placed in the X-ray diffractometer and it is exposed to an X-ray beam that interacts with the crystal lattice to cause diffraction patterns. By scanning through an arc of radius about the specimen the diffraction peaks can be located and the necessary calculations made.

There is a clear relationship between the diffraction pattern observed when X-rays are diffracted through crystal lattices and the distance between atomic planes (the inter-planar spacing) within the material. Altering the inter-planar spacing, different diffraction patterns will be obtained. The inter-planar spacing of a material that is free from strain will produce a characteristic diffraction pattern for that material. When a material is strained, elongations and contractions are produced within the crystal lattice, which change the inter-planar spacing of the \{hkl\} lattice planes. This induced change in d will cause a shift in the diffraction pattern. By precise measurement of this shift, the change in the inter-planar spacing can be evaluated and thus the strain within the material deduced.

a. Strain measurement

The strain \( \varepsilon_z \) can be measured experimental by measuring the peak position 20, and solving eqn. 7 for a value of \( d_0 \) and unstrained inter-planar spacing \( d_0 \).

\[ \varepsilon_z = \frac{d_n-d_0}{d_0} \]  
\[ (7) \]

![Figure IX. Schematic showing diffraction planes parallel to the surface.](image)

Thus, the strain within the surface of the material can be measured by comparing the unstressed lattice inter-planar spacing with the strained inter-planar spacing. Altering the tilt of the specimen within the diffractometer, measurements of planes at an angle \( \Psi \) can be made as shown in fig. IX and thus the strains along that direction can be calculated using:

\[ \varepsilon_\psi = \frac{d_\psi-d_0}{d_0} \]  
\[ (8) \]

b. Stress determination

The stress on the material can be calculated by using Hook’s law that is,

\[ \sigma_y = E\varepsilon_y, \sigma_x = E\varepsilon_x, \sigma_z = E\varepsilon_z \]  
\[ (9) \]
Because the stress is measured on the top surface of the material, \( \sigma_z = 0 \) and that the stress is biaxial;

\[
\varepsilon_x = - \frac{v}{E} (\sigma_x + \sigma_y) \tag{10}
\]

\[
\frac{\delta \varphi}{\delta_0} = - \frac{v}{E} (\sigma_x + \sigma_y) \tag{11}
\]

Elasticity theory for anisotropic solid shows that the strain along an inclined line is;

\[
\varepsilon_{\varphi\Psi} = \frac{1+v}{E} (\sigma_1 \cos^2 \varphi + \sigma_2 \sin^2 \varphi) \sin^2 \Psi - \frac{v}{E} (\sigma_1 + \sigma_2) \tag{12}
\]

\[
\sigma_{\varphi} = \frac{E}{(1+v) \sin^2 \Psi} \left( \frac{d \varphi - d_0}{d_0} \right) \tag{13}
\]

The experimental set up consists of X-Ray tube, poly-capillaries, mounting table and X-Ray detector. X-Ray tube is operated about 30KV and 7mA. The poly capillaries are the bunch of fiber optic cables which are used to project X-Rays at any angles. The detector is used to observe the diffraction pattern produced. The mounting table is capable of rotating the specimen at any angle for doing the experiment. The method involves varying the parameter \( \Psi \) angle in anti-clock wise direction.

c. Test conditions

<table>
<thead>
<tr>
<th>X-ray target used</th>
<th>Cobalt</th>
</tr>
</thead>
<tbody>
<tr>
<td>Tube voltage/current</td>
<td>30Kv / 7mA</td>
</tr>
<tr>
<td>( \Psi ) angles used</td>
<td>0°, 5°, 10°, 15°, 20°, 25°, 30° &amp; 35°</td>
</tr>
</tbody>
</table>

In positive \( \Psi \) method, firstly the angle \( \theta \) for the unstrained specimen is measured. Then strained specimen is placed in the \( \theta \) direction and the \( \Psi \) angle is gradually increased in positive direction till the same constructive interference pattern is obtained as in case of the unstrained specimen (Fig. X). Using the eqns. 7 and 8 we can deduce the strain in the specimen at the required points. The stress values observed on two points of the sample are (-40 to -44MPa).

d. Analytical stress calculation

The compressive stress can be calculated from the force torque relation. Eqn. 14 helps to find the maximum force acting on the shank.

\[
f = k_{eq} \left( \frac{\tau_0}{180} \right) \tag{14}
\]

\[
\Delta_\beta + \Delta_p = \Delta \tag{15}
\]

\[
k_\beta \text{ and } k_b \text{ can be calculated by eqns. 4 and 5.}
\]

\[
\sigma_z = \frac{f}{\lambda_b} \tag{17}
\]

\[
\Lambda_b = \frac{r}{4} (D_b - D_o) \tag{18}
\]

V. CONCLUSION

The study of stress distribution pattern is important for the safe and reliable design of bolted joints. Bolted joints have large scale application in aeronautical and aerospace industries and its analysis have a crucial role in the safety design and structural integrity. Material parameters can be changed for different materials and can be tested in extreme conditions which may not be economical if done experimentally. Space crafts and satellite launch vehicles use bolted joints for assembly of structures, so its failure cause failure of entire mission. The joints are inaccessible for repairing after launching the space craft, so deep analysis is needed. Joints are the most critical part where chances of failure is very high. So stress distribution in the joints should be the major concern. In this paper, a finite element model of a single-lap, single-bolt joint has been developed, and validated against experimental results and analytical calculation. The joint was modeled using PREWIN. Accuracy of the model is improved by refining the mesh and choosing the apt aspect ratio as well as the type of elements. Analysis of the finite element model gives values for stress and strain in Z-direction.

A number of factors were found to affect the accuracy and efficiency of the solution. The variation of result is due to the factors such as grinding, machining or the use of a wire brush as they introduce additional surface residual stresses into the sample being measured, sample composition and grain Size.

REFERENCES

Three-Dimensional Finite Element Analysis of Single Bolted Aluminium Lap Joint and Validation by XRD Technique


***