"I hereby declare that I have read this thesis and in my opinion this thesis is sufficient in terms of scope and quality for the award of the degree of Bachelor Mechanical Engineering (Structure & materials)"

Signature : ....
Name of Supervisor : Lee Yuk Choi
Date : 23/05/2006

ANALYSIS OF THE RIVETED JOINT USING FINITE ELEMENT METHOD

NIK MOHD ZULFALAH BIN NIK HUSSIN

A project report submitted in partial fulfillment of the requirement for the award of
the Degree of Bachelor Mechanical Engineering (Structure & Materials)

Faculty of Mechanical Engineering
Kolej Universiti Teknikal Kebangsaan Malaysia

May 2006

"I hereby declare that this thesis is my own work except the ideas and summaries which I have clarified their sources"

Signature :..............................
Author : Nik Mohd Zulfalah b. Nik Hussin
Date : 23/05/2006

Specially dedicated to my family, supervisor, friends and companion

ACKNOWLEDGEMENT

Thanks to ALLAH, for giving permission to complete my thesis. Many people have made contributions to this thesis. Firstly, I wish to acknowledge my supervisors, Mr. Lee Yuk Choi for his guidance, supervision and encouragement. He also one of the person to share his own idea to give me inspiration to develop this simulation. Secondly, I would like to thanks to all my lectures on KUTKM for teaching me, especially Mr. Ahmad Rivai. Also thanks to Mr. Lee Kwan Yuen (trainer for MSC Software) for his time spent for teaching and guiding me in this thesis.

My most heartfelt thanks go to my family for their support and providing an environment in which writing the paper was a pleasurable challenge.

Lastly, I would like to express my special thanks to my friends and those who have contributed their efforts and help in assisting me to complete this system. With the assistant, guidance, and support from all related stake holder have make the project complete successfully and achieve the objective project.

ABSTRACT

Within the industry today, companies save significant amounts of money through computer simulations. Mathematics through numerical methods becomes more important and therefore the investigation is done with the Finite Element Method (FEM). This work presents an analysis of riveted joint which is to define and explore the distribution of the stress of the riveted joint process. The effect of the clamping pressure was simulated and the distribution of the stress was conducted. The analysis was conducted using the available commercial finite element packages MSC.Nastran - Patran and MSC Marc using Solution 600 (implicit nonlinear). A 2-D axisymmetric model is used to describe the problem. The simulation was modeled using four different types of material (aluminum, silver, brass and iron). All materials and geometric non-linearity as well as non linear contact boundary condition is used in this model.

ABSTRAK

Pada masa kini, banyak syarikat telah menggunakan simulasi berkomputer bagi menjimatkan kos dalam proses analisis dan pembuatan. Kaedah matematik melalui kaedah berangka sangat penting pada masa kini, begitu juga dengan tesis ini yang menggunakan kaedah unsur terhingga (Finite Element Method – FEM). Tesis ini adalah bertujuan mengkaji/menganalisa parameter yang mungkin menyebabkan perubahan pada bentuk atau pengagihan tegangan pada penyambungan rivet. Tujuan utama tesis ini adalah untuk melihat dan mengkaji kesan-kesan dari kerja-kerja merivet. Bagi menganalisa masalah ini, simulasi kaedah unsur terhingga (Finite Element Method) dengan perisian MSC.Nastran – Patran dan MSC.Marc. telah digunakan. Untuk menghasilkan simulasi tersebut, model simetri 2-D digunakan. Parameter yang berlainan telah dibangunkan bertujuan membuat perbandingan antara satu sama lain. Perbandingan dengan kajian yang telah di buat juga dilakukan dalam kajian ini.

TABLE OF CONTENTS

CHAPTER I INTRODUCTION

CHAPTER III THEORY

CHAPTER IV MSC NASTRAN-PATRAN & MSC.MARC

CHAPTER V METHODOLOGY

CHAPTER VI RESULTS AND DISCUSSION

CHAPTER VII **CONCLUSION**

LIST OF TABLES

LIST OF FIGURES

LIST OF CHARTS

CHAPTER I

INTRODUCTION

1.1 Overview

Riveted joint has been used in many large scale industry including shipbuilding, boilers, pressure vessels, bridges, building, automotive etc. In recent years, there has been a progressive move from riveted joints to welded, bonded and even bolted joint.

A riveted joint in larger quantity is sometime cheaper than other option like welded joint or bolted joint, but its required high skill levels to install the joint. Nowadays a great number of types and sizes of rivet are available, and so are many joints which they are used.

Many structural analysis tools exist that have capability to conduct fracture mechanics and crack propagation analyses. Two software codes are particularly suitable for analyzing riveted lap joint. There are;

i. A crack propagation analysis tool, AFGROW, which estimates the fatigue crack growth life of a structural component based on specified loading condition and initial crack geometry.

ii. Finite element Method/Analysis tool, which simulates crack growth in layered structures.

In this study, the Finite Element Method is used as a method. The finite element method of analysis is not a new technique. In the late 1940s, the method started as a structural analysis tool that was used for helping aerospace engineers design better aircraft structures. Since then, aided by the rapid increase of computer power, the method has continually developed until it became a very sophisticated generic tool for accomplishing a wide array of engineering tasks. Its development and success is not paralleled by any other numerical analysis technique.

The technique is based on the premise that an approximate solution to any complex engineering problem can be reached by subdividing the problem into smaller more manageable (finite) elements. Using finite elements, solving complex partial differential equations that describe the behavior of certain structures can be reduced to a set of linear equations that can easily be solved using the standard techniques of matrix algebra.

The finite element method is being used in virtually every engineering discipline. Aerospace, automotive, biomedical, geotechnical, electrical, hydraulic, and nuclear engineering applications have become standard objects for finite element analysis. In addition, it is not only used for analyzing classical static structural problems, but also for such diverse areas as mass transport, heat transfer, dynamics, stability, and radiation problems.

1.2 Objectives of the project

The objectives of this project are as follows:-

1. To analyze the riveted joint using Finite Element Method.
2. To investigate the modeling technique of riveted joint using MSC.PATRAN and MSC.NASTRAN software.
3. To obtain the optimization analysis for different materials.

1.3 Scopes of the project

In order to achieve the objective, the scopes of the project must be fulfilled. The scopes of the project are listed below:

1. Simulation of riveted joint using MSC.PATRAN, MSC.NASTRAN and MSC.MARC software.
2. Compare related results with available analytical solution or theoretical from previous work.

1.4 Gantt Chart

The progress of the project was shown in Chart 1.1 and 1.2.

Activities	Week														
	1	2	3	4	5	6	7	8	9	10	11	12	13	14	15
Select title of PSM	▨	▨	▨												
Confirmation of scope and objectives			▨	▨											
Literature review					▨	▨	▨	▨	▨	▨	▨	▨	▨		
Understand and familiarize with MSC.Patran & Nastran	▨	▨	▨	▨	▨	▨	▨	▨	▨	▨	▨	▨	▨		
First draft												▨	▨	▨	▨
Power point preparation														▨	▨
Presentation															▨

Chart 1.1: Progress of the PSM 1

Activities	Week														
	1	2	3	4	5	6	7	8	9	10	11	12	13	14	15
Running MSC.Patran & Nastran	▨	▨	▨	▨	▨	▨	▨	▨							
Data collection				▨	▨	▨	▨	▨							
Data analysis & discussion								▨	▨	▨	▨	▨	▨		
Second draft												▨	▨	▨	
Power pont preparation													▨	▨	▨
Final presentation															▨
Project report preparation								▨	▨	▨	▨	▨	▨	▨	▨

Chart 1.2: Progress of the PSM 2

CHAPTER II

LITERATURE REVIEW

B. Langrand et al. (2001) used a numerous rivets to simulate the behavior of aeronautical frames under crash loading conditions. Until now this type of bonding modeling has not been judged satisfactory. The strength of a riveted joint under dynamic loading and the characterization of a simplified rivet element, in particular, were sources of questions. In this paper, they did solid finite element modeling and carried out experiments to measure the dynamics loading conditions. The authors also determined the elastic-plastic and damage mechanical properties for sheet metal plates and rivet materials. From this study, it's finds that it was essential to develop a new kind of rivet element taking material non-linearity into account. Experiments and solid "finite element modeling of an adapted Arcan test procedure were then conducted and pure shear and tensile non-linear responses as well as parameters of a macroscopic criterion were identified. The use of this new rivet element was found to improve prediction of the dynamic behavior for a frame assembled with 700 rivets.

M.P. Szolwinski and T.N. Farris (1997) presented the role of rivet installation in fretting crack nucleation and Finite Element Model of force-controlled rivet installation which covered a model features and assumption and modeling challenges. This study finds that:

i. Nucleation of cracks due to fretting in structural joints is a prevalent problem
ii. Rivet installation and resulting load transfer characteristics of joint are related intimately

iii. Modeling accurately the yielding behavior of plate and rivet material for loading & unloading cases is critical to accurate calculation of residual stress field

iv. Mesh refinement and re-meshing are being investigated currently to better predict clamping pressure

William Bowen (1998) investigated the need for structural analysis can be triggered by several events. These can be broadly classified as design of a new structure or product; investigating use of an existing design for other uses; or investigation of failure of an existing design. How well a finite element model represents the real structure is dependent on the time, skill, and motivation applied by the analyst or analysis team. A good analysis based on a finite element model must be planned from the beginning, well before any computer model is generated. Often there are unrealistic or divergent expectations for a finite element analysis.

C.E Harris et al. (2000) conducted an extensive experimental database that has been assembled from very detailed teardown examination of fatigue cracks found in rivet holes of fuselage structural component. Base on this experimental database, a comprehensive analysis methodology was developed to predict the onset of widespread fatigue damage in lap joints of fuselage structure. A several computer codes were developed with specialized capabilities to conduct the various analysis that makeup the comprehensive methodology. Over the past several years, the authors have interrogated various aspects of the analysis methods to determine the degree of computational rigor required to produce numerical predictions with acceptable engineering accuracy. This study led to the formulation of a practical approach and compares prediction with the result from several experimental studies.

Ron Bolick et al. (2002) presented a comparative study of riveted joints and adhesively bonded joints subjected to static and fatigue loadings. Static and fatigue test were performed at room temperature to obtain the fundamental mechanical properties of the joint such as Ultimate Tensile Strength and Fatigue Strength. Three dimensional finite element models of adhesively bonded specimens were developed to study their failure modes. The study indicated that the failure of adhesively bonded joints was due to normal stress and not third of the cohesive shear strength of the adhesive. The physical testing was similar to the analytical modeling failures in the adhesively bonded joints were of an adhesive-cohesive type. The objective of this research is to provide the following benefits: increase strength and service life, improve distribution stresses and load throughout the joint, reduced weight, reduction of personnel to manufacture and manufacturing time.

SILVA et al. (1999) studied a phenomenon of the Multiple Site Damage (MSD). This paper discussed and exemplified with the behavior of riveted lap-joint using of aluminum alloy 2024-T3. From this study the three following points are treated:

i. Stress distribution with and without the presence of cracks using SPATE infrared technique and strain gauges. The bending inherent to the eccentricity of the lap joint is determined.
ii. Data on the initiation and growth of cracks, and on residual static strength. Cracks growth rates are determined from periodic crack length measurements obtained using a traveling microscope.
iii. Scanning electron microscopy (SEM) analysis of the complete fracture surface. These SEM analyses document the initiation process, the subsequent growth of fatigue cracks and link-up mechanism.
iv. These data presented illustrate characteristics associated to MSD, namely the effects of growth and interaction of multiple cracks.

M.A. McCarthy et al. (2001) have developed a three-dimensional finite element models in order to study the effects of bolt–hole clearance on the mechanical behavior of bolted composite (graphite/epoxy) joints. The joint type studied was single-bolt, single-lap, which is a standard test configuration in both a civilian and a military standard for composite joints. In this paper the model was constructed in the non-linear finite element code MSC.Marc and attempts are made to validate it by comparing results with experiments and other finite element solutions generated in a European project on composite bolted joints. Issues in modeling the contact between the joint parts, which affect the accuracy and efficiency of the model are presented. Experimental measurements of surface strains and joint stiffness are compared with results from a finite element parameter study involving variations in mesh density, element order, boundary conditions, analysis type and material modeling.

Billy Kelly and Colm Costello (2004) performed a simulation of the setting of a blind rivet, using an axisymmetric model in the MSC.MARC FEA package. A 6.4mm diameter aluminum (5056 alloy) blind rivet was used to join two plates, each 4mm thick. Initial material properties were used to simulate the H2 condition of the material after manufacture. Simulation of setting of the rivet was developed in the FE model. An optical profile projector was used to map the profile of the partially set and set rivet shapes. The FE model provided a good prediction of actual set shape. The forming forces involved in the setting of the blind rivet were also investigated. Actual rivets did not always set symmetrically. A relationship between setting force and the asymmetry of the set rivet was observed. The simulation of the pull test was based on BS EN ISO 14589:2001 [Blind rivets-mechanical testing]. FE models were developed to test the rivet to failure in tension using a 2D-axisymmetric model. Riveted joints were physically tested to failure in tension, using a uniaxial tensile testing machine with appropriate test jigs. The FE model results were compared with the physical test results.

X. Deng and J. W Hutchinson (1998) presented an analysis of a model of the process of cold-driving a rivet is carried out with the primary aim of identifying the origin and magnitude of the resulting clamping force exerted by the rivet on the lap joint." The development of high hydrostatic pressure in the rivet shank in the first stages of the driving process, and its subsequent release as the rivet head is squashed, is the key to understanding the origin of the clamping force". The average clamping stress in the rivet shank is on the order of $\sigma_y/4$, where σ_y is the yield stress of the rivet. The model is used to explore the role of some riveting variables, including differing yield stresses in the rivet and sheet materials, strain hardening, and head size of rivet. This paper finds that the resulting average compressive stress in the shank prior to unloading has to drop below σ_y if the residual clamping stress, is existing. The effect is all the all the more subtle because the residual clamping stress, $\approx \sigma_y/4$, is small compared to the maximum compressive stress which develops in the shank during the riveted process, $\approx 2\sigma_y$. The model enables the clamping stress to be compute as a function of the defining geometric and material parameters.

Christian Westerberg (1999) performed a numerical simulation by using the finite element software ABAQUS/Explicit, and involves dynamic inertia effects. The