UNIVERSITI TEKNOLOGI MARA

CHARACTERIZATION AND MODEL UPDATING OF MODAL PARAMETERS OF A RIVETED JOINTS STRUCTURE

NOR AZELA BINTI TORMODI

MSc

April 2018

ABSTRACT

Thin metal sheets are widely used to fabricate various types of aircraft components because of the flexibility and easily to form any shapes. However, it is really difficult to keep the components permanently attached through the surface contact in the numerical model. Through this study, the simplified model of aircraft wing structure made from thin aluminium sheets was evaluated. This simplified model of wing structure consists of three different types of components which are a plate (skin), spar and rib. There are only five components included in this structure which consist of one plate, one spar and three ribs. All these components are assembled via riveted joints. The riveted joint has become one of common joints applied in aircraft industries because of its easiness to be disassembled, maintained and inspected. It has been investigated that the dynamic behaviour of the joined structures can be significantly affected by its joints due to the mating areas and clamping force. Thus, the main objective of this research is to investigate the modal parameters regarding natural frequencies and mode shapes of the jointed structure through experimentally and numerically. Then, the results from the normal modes analysis are applied in the model updating analysis in order to improve the correlation between experimental and numerically. Lastly, through this study, the accuracy of the developed model of riveted structure was tested. A few types of connector's elements available in the HyperWorks are applied to determine the reliable model of riveted joint such as CGAP, RBE, CFAST, CBEAM and other types of connector elements that are available in FEA. The experimental modal analysis (EMA) was used in conducting the experimental works by applying impact hammer testing. Therefore, the accuracy of the riveted modelling via finite element analysis was discussed based on the numerical and experimental data analysed. The most reliable connector element which was represented as rivet joint in finite element analyses was determined. CFAST element was identified as the most suitable connector element to represent as rivet joint in FEA due to its characteristics. There are a few parameters that can be manipulated, 3-translation, 3-rotational, stiffness and diameter in producing the best model of rivet joint.

CHAPTER ONE
INTRODUCTION

1.1 Research Background

Riveted joints are commonly used in well-established technology in manufacturing industries as a connection method compared to others, such as bolted joint and welding especially in aircraft industry to permanently join the thin-walled sheet-metal parts. This technology of riveted joint is a modest and versatile method in dealing with low thickness sheets especially lightweight alloys [1]. It has been extensively used due to its advantages, such as simple production process, low cost, product efficiency and high connection strength [2]. However, it is difficult to determine and measure the characteristic of riveted joint structure in order to produce an effective modelling [3]. The deformation of rivet and the sheet-metal parts can happen due to riveting process [4]. On top of that, the jointed quality of the structure will be affected by axis deviation when the rivet's axis is not equivalent with the hole axis of the plate which influences the deformation, stress and strain of riveting [5].

Most of the structures will experience vibrational problems that can cause deformation due to the movement of the structures and these affect the jointed structures too. It has been stated that vibrational problems occurred to the most prime movers due to defective design or poor manufacture [6]. The aircraft, spacecraft and automobile structures are among the structures being exposed to vibrational problems [7]-[9]. In addition, when the natural frequency of vibration of a structure coincides with the frequency of the external excitation, resonance would occur. This can cause excessive deflection and failure [6]. Thus, vibration test is conducted to analyse dynamic characteristics of the structures such as natural frequency and mode shapes in order to control vibration of the structures.

It is recognized that finite element analysis (FEA) can be used effectively to evaluate the structures design performance at an early stage without preparing any prototype. As an improvement in finite element technology, it has been declared as the main analysis tools nowadays [6]. Modelling is an effective way to solve the problem related to the structural analysis. However, the existing joint in the structure makes it difficult to get an accurate analysis since the characteristic parameters in the equation of

motion are inaccurate and contribute in giving large error to dynamic analysis [7]. Previous researcher states that it is difficult to deal with detailed rivet models due to time constraint and there are possibilities of overlooking the global-local analysis [8]. There are two methods that are applied to analyse the dynamic characteristics of structures namely experimental approach using experimental modal analysis (EMA) and numerical approach using finite element analysis [7]. However, there may have some differences between the experimental result and finite element analysis due to error that has occurred in both experimental and finite element analysis.

Therefore, this research work concentrates purely on modelling the simplified model of rivet joint in finite element analysis on thin plate structure. Furthermore, it is aimed to determine the dynamic characteristic of the thin plate structure. A previous study has developed the 3D complex models of rivet, but it has considerably discouraged the extension of multi-riveted structures [9]. Thus, it is important to investigate which connector element that can best represent riveted joint structures in finite element model. Besides, the intention in producing the simplified model of rivet joint is to ease analysing complex structure when it includes the joint element. Furthermore, model updating method is applied in order to reduce relative error between the finite element method and experimental results.

1.2 Problem Identification

Complex engineering structures such as BIW, aeroplane and ship are made up of thousand numbers of subcomponent which are assembled through various types of joint. It is important to include the joining in modelling in order to represent as close as possible to the real structure. One of the typical joints that is mainly used in the industries is rivet joint. However, there are still limitations and difficulties in dealing with the modelling of complex engineering structures especially when it involves the rivet joint. In modelling an accurate and reliable rivet joint, it is not an easy task due to the load transfer between the rivet and plate [10]. This is due to the huge effect of the rivet joint to the stiffness value of the whole structure in determining the dynamic characteristic. Moreover, there are a lot of connector element types that are available in the commercial software such as CBUSH for spring element, CWELD for spot weld joint and CFAST for bolted joint. However, a reliable connector element that is believed suitable to represent as the rivet joint still remains unidentified. Furthermore,

in particular cases which involves thin plate structure, it will tend to bend which easily leads to the non-linearity. Besides, the thin plate had experienced a shear force, bending and twisting moments which are also stiffer compared to the beam structure due to the load carried [11]. Therefore, investigating the dynamic characteristics of these structures is very challenging and time-consuming.

Another huge current concern that attracts the engineering community is the reconciliation issues of the result obtained between FEA and EMA. The initial assumptions of the input parameters made in FEA have led to a great discrepancy of the results in comparison with the experimental counterparts [12]-[13]. In addition, even though updated components model are used for the assembly analysis of the whole structure, an invalid parameter of rivet joint model might further introduce a significant error to the result. Thus, it is very important to correctly model the connector element because it can give a great influence to the dynamic characteristics of the whole structure.

In this study, the simplified model of riveted joint aircraft wing structure which is made from the thin plate is used as a tested model structure to investigate the most suitable connector element that can be used to represent a rivet joint in finite element modelling. Finite element model updating method is used to numerically decrease the discrepancies of the results obtained between FEA and EMA [14]. This study has triggered many researchers in working on it and still continuing to determine the best element connector to represent the rivet joint in FEM.

1.3 Research Objectives

The main goal of this study is to investigate the efficient and reliable element connectors, capable of representing rivet joints in complex structure via finite element model updating method. Fours objectives are identified which are as follows:-

I. To develop finite element model of the components and the riveted joint structure.
II. To determine modal parameters of the components and the riveted joint structure numerically and experimentally.
III. To evaluate the finite element model of proposed element connector for updating process with the result obtained from experimental modal analysis.