FINITE ELEMENT ANALYSIS OF 3D STRUCTURE WING BOX TEST RIG

HASBOLAH BIN MOHD SAID

(2007270978)

A thesis submitted in partial fulfillment of the requirements for the award of Bachelor Engineering (Hons) (Mechanical)

Faculty of Mechanical Engineering

Universiti Teknologi MARA (UiTM)

MAY 2010

"I declared that this thesis is the results of my own work except the ideas and summaries which I have clarified their sources. The thesis has not been accepted for any degree and is not concurrently submitted in candidature of any degree."

Signed: ....

Date: 24/5/2016

Hasbolah Bin Mohd Said
UiTM No: 2007270978

ACKNOWLEDGEMENT

All Praises to Allah s.w.t, Lord of the universe. We praise Him, with His help allowed me to complete my Final Year Project report on "Finite Element Analysis of 3D Structure Wingbox Test Rig". I owe a great deal of thanks to those who have helped me and give a lot of knowledge along my study to complete this project.

My supervisor, Mr. Ramzyzan Ramly has been invaluable for guiding, advising and teach the many thousands of knowledge regarding to this study about a wingbox test rig. Not forgotten to all my friends who had helped me to complete this final year project report either directly or indirectly. Especially for the team member that involves with this project.

Finally I owe a great thanks to those who steadfastly supported and encouraged me to complete my final year project report. Last word from me, we ask Allah s.w.t to accept this report, and give us for our mistakes and make this report a valuable asset for the future references.

ABSTRACT

Wing box test rig is designed to hold the wing in place and keep them cantilevered so that the lift on the wing doesn't deform the test rig during experimental session. In order to conduct a test that involve high load, the design of the wing box test rig structure need to be constructed. The purposes of this project are to study on designing and conducting finite element analysis of the wing box test rig for static test experiment. There are several consideration to be emphasizes such as load requirement, structural stability and using finite element modeling software, the test rig is designed, simulated and analyzed in the computer. The finite element analysis modeling software that is used in this project is ANYSY/ED 12.1. During the simulation, the structural analysis data such as deformation of beam and the stress distribution were determined. The material used for this project is mild steel A36 and two different beams were selected then will be compared. Results obtained from study using ANSYS software were then compared with the result obtained from the theoretical analysis. Finally, the design alteration needs to be done to counter any problem in analysis. From this study, it is expected that this project can help another student as reference to manufacture the wing box test rig in a future.

TABLE OF CONTENTS