



اَللّٰهُمَّ صَلِّ وَسَلِّمْ عَلٰى سَيِّدِنَا مُحَمَّدٍ
UNIVERSITI
TEKNOLOGI
MARA

DESIGN AND SIMULATION OF A 3D WING BOX TEST
RIG FOR STATIC TEST

LEONARD LANGKING

2006133633

BACHELOR OF ENGINEERING (HONS) MECHANICAL
UNIVERSITI TEKNOLOGI MARA (UiTM)

APRIL 2010

“I declared that this thesis is the result 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 accepted for any degree”

Signed : *Leonard*
Date : 19/5/2010

Leonard Langking
Uitm No.: 2006133633

ACKNOWLEDGEMENT

I would like to thank God The Almighty for giving me the opportunity, the strength and also the commitment to understand the nature of this project, to embark and to pursue the project in order to add to the pool of knowledge on this earth.

I would like to thank my supervisor Mr Ramzyzan Ramly for all the guidance throughout this project.

To my lovely parents Langking Ebeh Dian and Christina Eram Ungau and my brothers and sisters for their support in everything that I do throughout my college years.

Also to Faculty of Mechanical Engineering and all people who were involve directly and indirectly in this project.

Abstract

Generally, this project is about designing a 3D Wing Box Test rig for static test using solid modelling computer software. Wing box test rig was invented for testing wing structure. The test rig is to hold the wing in place and keep them cantilevered so that the load on the wings doesn't affect the rig. The test will be done on computer using CAD software. The software used is CATIA. The scope of this project is to design a 3D wing box test rig for static test. The design must meet the requirements given or a strong test rig that can withstand the load of wing. The CATIA software is used to design and do the simulation on the test rig. The design process and simulation in this project is done mostly using the CATIA software. The result will be obtained by improving the initial design and choosing the most suitable design. The project will be complemented with project that is the finite element analysis on the wing box test rig design. This is the analysis of load requirement analysis on the test rig. Besides that for future, student can use this design to fabricate a wing box test rig that will be use for static test. The engineering analysis on materials is done through calculation by referring to the theoretical values and formulas. The parameters taken from this engineering analysis will be applied on the CATIA software in the analysis and simulation of the design. The output of this project is a design and simulation of a 3D wing box test rig for static test that used a material of hollow box and C beam mild steel as the main component. The design will meet the requirements of factor of safety in the range of 1.2 to 2.5 and can withstand the maximum load of 50 KN.

Table of content

Contents	Page
Page Title	i
Acknowledgement	ii
Abstract	iii
Table of content	iv
List of tables	vii
List of figures	viii
Chapter 1: Introduction	1
1.0 Introduction of project	1
1.1 Problem statement	2
1.2 Objectives	2
1.3 Scope of research	3
1.4 Significant of studies	3