FINITE ELEMENT ANALYSIS OF A 2D STRUCTURAL FRAME OF A WING BOX TEST STAND

SITI SYAHIDAH BT MOHD RUM

(2005646290)

BACHELOR ENGINEERING (HONS) (MECHANICAL)

UNIVERSITI TEKNOLOGI MARA (UiTM)

MAY 2009

ACKNOWLEDGEMENT

Alhamdullillah, in the name of Allah, the Beneficent, and the Merciful. It is with most humbleness and gratitude that this work is completed with His Blessings. My heartfelt gratitude is directed to my supervisor, Mr Ramzyzan Ramly for guiding and advising me through this work from the beginning till end. Profuse thanks to him for being very patient, understanding and for keeping me focused on my work. Without his criticism, comments, timely aid and intervention this work may not have materialised.

Special thanks and appreciation to all lecturers in Mechanical Engineering programme for advices and kind helps, which made the completion of this study a reality. Not forgetting to all laboratory staffs for their technical assistances and valuable advice.

Last but not least, my special thanks owing to my lovely parents Mohd Rum Bin Anuar and Nani Herwani Bt Oeloeng Kamaloeddin for their moral support and also to my friends who had helped me to complete this final year project paper either directly or indirectly. Thank you.

Siti Syahidah Bt Mohd Rum

ABSTRACT

Test stands are designed and built to test the performance of the structures against the targets set in the design specification. This study reports on the finite element analysis of a 2D structural frame of a wing box test stand. In this study, two types of structural analyses of the test stand are conducted which are static analysis and buckling analysis. The static analysis is conducted to determine displacements, stresses, and deflection under static loading, whereas the buckling analysis is done to determine buckling loads and the buckling mode shape. The finite element analysis is accomplished using ANSYS/ED and ANSYS 11.0. Results from the finite element analysis were then compared with the results obtained by formulation. The material used for this project is a hot rolled tube mild steel (A36). It is expected that this study can contribute to practical research for mechanical engineering student

TABLE OF CONTENTS

CONTENTS **PAGE**