



**PERFORMANCE AND ANALYSIS OF NEWLY
DESIGNED PUMP SEATING FOR
CENTRIFUGAL PUMP OPERATION**

NABILAH BINTI HAMIDAN

(2014642812)

**BACHELOR OF MECHANICAL ENGINEERING
(MANUFACTURING) (HONS.)**

UNIVERSITI TEKNOLOGI MARA (UiTM)

JULY 2017

“I declare that the content present in this thesis are my own work which was done at Universiti Teknologi MARA (UiTM) unless stated otherwise. The thesis has not been previously submitted for any other degree.”

Sign :

Date :

Nabilah Binti Hamidan

UiTM No : 2014642812

ACKNOWLEDGMENT

Alhamdulillah, all praise to Allah S.W.T, by His grace and His kindness, I can complete this undergraduate research project successfully. First and foremost, I would like dedicate my deepest gratitude to Universiti Teknologi Mara (UiTM), Faculty of Mechanical and Manufacturing Engineering, for providing the facilities and opportunity for me to do this project.

I would like to express sincere appreciation to my supervisor, Dr Ahmad Sufian Bin Abdullah, for invaluable encouragement and guidance during the preparation of this thesis. It is indeed a true honour and privilege for being able to work under the supervision of such a dedicated and enthusiastic lecturer. Thank you.

Finally, I also wish to express my deep appreciation and gratitude to the members of my family, especially my parents for love, sacrifice, motivation, and support throughout this project. Not forgetting the friends and everyone who helped directly or indirectly to the success of this project. Only Allah S.W.T could bestow all of you with His bless and kindness, handsomely reward for your contributions.

ABSTRACT

The main purpose of pump seating is to place the pump while maintaining the function of the pump. This study is concerned with the development of an efficient method for the analysis of performance pump seating. The objectives are to developed a reliable finite element model of pump seating under static and vibration loadings and to analyze the performance and behavior of the design under static and vibration loading for pump weighted 235 kg. The result can be obtained by analyse it using two methods which are analytical calculations and finite element method (FEM). This project is done by using finite element analysis which is Abaqus software. The model is designed according to the industries that are related with the pump seating in work. The model is designed in 3D cad software modelling which is Solidworks. Subsequently, the model is imported into Abaqus software for finite element analysis (FEA). The performance of the pump seating under various loading can be analyze by simulate the model using Abaqus software. The reliability of the finite element model is analyzed by comparing the analytical/theoretical calculations obtained with the simulation using Abaqus software. The analytical/theoretical calculations correlated well with simulation analysis results where both proved that the design of pump seating can hold the pump in static and vibration loading

TABLE OF CONTENTS

ACKNOWLEDGMENT	i
ABSTRACT	ii
LIST OF TABLES	v
LIST OF FIGURES	vi
LIST OF SYMBOLS	viii
LIST OF ABBREVIATION.....	ix
CHAPTER 1	1
1.1 Project Background.....	1
1.2 Problem Statement	2
1.3 Objective of Study.....	3
1.4 Scope of Work.....	4
CHAPTER 2	5
2.1 Introduction to Pump	5
2.2 Pump Seating – an overview.....	6
2.2.1 Other analysis on pump seating.....	6
2.3 Material	7
2.3.1 Steel ASTM A36.....	7
2.3.2 Material properties and material models	8
2.4 Loadings.....	12
2.4.1 Static loading	12
2.4.2 Vibration loading.....	12
CHAPTER 3	14