FINITE ELEMENT ANALYSIS OF AN UPPER MOTORCYCLE PISTON

AZROL BIN AROF

A dissertation submitted in partial fulfillment of the requirements
for the award of the degree of
Bachelor of Mechanical Engineering with Automotive Engineering

Faculty of Mechanical Engineering
UNIVERSITI MALAYSIA PAHANG

NOVEMBER 2009

SUPERVISOR'S DECLARATION

I hereby declare that I have checked this thesis and in my opinion this thesis is satisfactory in terms of scope and quality for the award of the degree of Bachelor of Mechanical Engineering.

Signature:
Name of Supervisor: MR. MOHD RASHIDI BIN MAAROF
Position: LECTURER
Date:

STUDENT'S DECLARATION

I hereby declare that the work in this thesis is my own except for quotations and summaries which have been duly acknowledged. The thesis has not been accepted for any degree and is not concurrently submitted in candidate of any other degree.

Signature:
Name: AZROL BIN AROF
ID Number: MH06066
Date:

DEDICATION

Al-Fatihah to my late mother and father

Mrs. Lasiah Binti Marjuki

Mr. Arof Bin Midi

ACKNOWLEDGEMENT

In the name of Allah, the Most Benevolet, the Most Merciful. In particular, I wish to express my sincere appreciation to my main thesis supervisor, Mr. Mohd Rashidi Bin Maarof, for encouragement, guidance, critics and advise, without his continuous support and interest, I would not have been able to complete this final year project successfully.

I am also indebted to Universiti Malaysia Pahang (UMP) for providing internet facility. Librarians at UMP also deserve special thanks for their assistance in supplying the relevant literatures. Their views and tips are useful indeed

My sincere thanks go to my best friend, Nur Farah Bazilah, my classmate, my roommate, my teamwork for their excellent co-operation and staff of Mechanical Engineering Department who helped me in many ways and made my stay at UMP pleasant and unforgettable.

ABSTRACT

This dissertation describes the stress distribution of the upper piston with using finite element analysis. The finite element analysis is performed by using computer aided engineering (CAE) software. The main objectives of this project are to investigate and analyze the stress distribution of upper piston at the real engine condition during combustion process. The dissertation describes the mesh optimization with using finite element analysis technique to predict the higher stress and critical region on the component. The upper piston is implemented in the six stroke engine of 110 cc Modenas motorcycle. Aluminum 356-T7 is selected as an upper piston material. Despite all the stresses experience by the upper piston does not damage the upper piston due to high tensile strength but the upper piston may fail under fatigue loading. Thus, it is important to determine the critical area of concentrated stress for appropriate modification. With using computer aided design (CAD) which is SOLIDWORK, the structural model of an upper piston is developed. Furthermore, the finite element analysis performed with using MSC PATRAN and MSC NASTRAN. The finite element analysis is performed by using linear static stress method. The result of the analysis shows that mesh type of TET 10 give more accurate result compare to TET 4 at its each mesh convergence point. The stress analysis results are significant to improve the component design at the early developing stage. The result can also significantly reduce the cost and time to manufactured the component and the most important to satisfy customer needs.

ABSTRAK

Disertasi ini menggambarkan serakan tekanan terhadap piston atas dengan menggunakan analisis elemen hingga. Analisis elemen terhingga dilakukan dengan menggunakan perisian kejuruteraan bantuan computer (CAE). Objektif utama projek ini adalah untuk mengkaji dan menganalisis serakan tekanan terhadap piston atas dalam keadaan sebenar enjin semasa proses pembakaran. Disertasi menggambarkan pengoptimuman jala dengan menggunakan teknik analisis elemen terhingga untuk menjangka tekanan yang lebih tinggi dan kawasan kritikal pada komponen. Piston atas digunakan pada enjin motorsikal Modenas enam lejang 110 cc. Aluminum 356-T7 dipilih sebagai bahan piston atas. Meskipun semua tekanan yang dialami oleh piston atas tidak merosakkan piston tetapi piston atas mungkin gagal apabila daya lesu dikenakan. Oleh demikian, sangat penting untuk menentukan kawasan kritikal yang ditumpu oleh tekanan untuk pengubahsuaian yang sesuai dapat dilakukan. Dengan menggunakan perisian lukisan bantuan komputer (CAD) iaitu solidwork, model struktur piston atas dapat dihasilkan. Seterusnya, analisis elemen terhingga dilakukan dengan menggunakan PATRAN MSC dan MSC NASTRAN. Analisis elemen terhingga dilakukan dengan menggunakan kaedah tekanan pegun linear. Keputusan analisis menunjukkan bahawa jenis jala TET 10 memberikan keputusan yang lebih tepat berbanding dengan TET 4 pada titik menumpu bagi setiap jala. Keputusan analisis tekanan amat bermakna untuk memperbaiki reka bentuk komponen pada tahap awa pengahasilan. Keputusannya juga dapat mengurangkan kos dan masa untuk menghasilkan komponen dan yang paling penting untuk memuaskan kehendak pelanggan.

TABLE OF CONTENTS

LIST OF TABLES

LIST OF FIGURES

LIST OF SYMBOLS

ρ	Density
k	Thermal conductivity
T	Temperature
E	Modulus of elasticity
σ_{UTS}	Ultimate Tensile strength
P	Pressure
mm	Millimetre
E	Exponent
Pa	Pascal
MPa	Megapascal
GPa	Gigapascal

LIST OF ABBREVIATIONS

Al	Aluminium
CAD	Computer-aided Design
CAE	Computer-aided Engineering
FE	Finite Element
FEM	Finite Element Modeling
FEA	Finite Element Analysis
FVM	Finite Volume Method
FDM	Finite Different Method
2D	Two Dimension
3D	Three Dimension
TET	Tetrahedral
cc	Centimetre Cubic
CPU	Central Processing Unit
MPC	Multi Point Constraints
SAE	Society of Automotive Engineers
ASME	American Society of Magazine Editors
Max	Maximum
Min	Minimum

CHAPTER 1

INTRODUCTION

1.1 PROJECT BACKGROUND

In recent years, there are many kind or development of vehicle engine especially car and motorcycle engine. Each automotive company tried to develop their own engine to compete for new technology or invention. Internal combustion engine is one type of automotive engine which is fuels that run the mechanism burned internally or burned inside the engine. There are two types in internal combustion engine which is reciprocating and rotary engine. The type of engine that usually been used for motorcycle are two stroke and four stroke engine.

Based on the development of automotive engine, the new technology of six-stroke engine is existed for motorcycle engine. The six-stroke engine was developed based on two-stroke and four-stroke motorcycle engine. The operation of the six stroke engine is almost same compare to others types of engine which is have the common cycle of spark ignition engine cycle. The cycle is includes the four stages which are intake, compress, combustion and exhaust. In six-stroke engine, there are two pistons in one cylinder which is worked together in the same time. The first piston is the main piston in the original engine which is for four-stroke. The additional two-stroke is generated by the second piston which is called the upper piston.

In internal combustion engine, piston is one of the important components parts defined as cylindrical components that moves up and down in the cylinder bore by force produce during the combustion process. The features of the piston are piston head, piston pin bore, piston pin, skirt, ring groove, ring land and piston ring. The shape of the piston is usually depending on the engine design and piston usually different at the crown design. The upper piston of the six stroke engine is one type of the internal combustion piston. The feature of the upper piston is almost same as the general piston.

Finite Element Analysis is a simulation technique which evaluates the behavior of components, equipment and structures for various loading conditions including applied forces, pressures and temperatures. Thus, a complex engineering problem with non-standard shape and geometry can be solved using finite element analysis where a closed form solution is not available. The finite element analysis methods result in the stress distribution, displacements and reaction loads at supports for the model. Finite element analysis techniques can be used for a number of scenarios as example mesh optimization, design optimization, material weight minimization, shape optimization and code compliance.

Finite element analysis also has the capability to perform FEA for aircraft components, automotive components, engine components and other mechanical / structural components. The design for various components is checked for compliance against the ASME Code or other appropriate code. Finite Element Analysis is performed for both design and analysis / evaluation situations. Two-dimensional and three-dimensional FEA problems are addressed for structural, thermal, and thermal stress evaluations.

1.2 PROBLEMS STATEMENT

The piston is one of the most stressed components of an entire vehicle because it is placed in the cylinder that has combustion process. Therefore, it is must be designed to withstand from damage that caused of the extreme heat and pressure of combustion process. There are many damages or failures for piston due to high pressure and heat such as piston skirt seizing, piston head seizing, piston ring damage and cylinder damage. The value of stress that obtained the damages can be determined by using finite element analysis. With finite element analysis, the stresses value that act to the piston is determined by simulation. Thus, it can reduce the cost and time due to manufacturing the components and the same time it can increased the quality of the product. To ensure the stresses value are accurate value, there is a process which is called mesh optimization. Mesh optimization process is a process of numerical method and approximation solution are required to solve most of partial differential equation for a component.

1.3 OBJECTIVES

There three objectives for this dissertation which is focus on the mesh optimization of an upper piston for six stroke engine based on the finite element analysis. The objectives are:

(i) To design a 3D model of upper piston for six stroke engine

(ii) To develop a Finite Element Model for the mesh optimization an upper piston

(iii)To analyze an upper piston using linear static stress method.

1.4 SCOPE OF STUDY

There are three scope of study for this dissertation of finite element analysis of an upper piston for six stroke engine. There are:

(i) Modeling the 3D of upper piston with using CAD software

(ii) Finite Element Modeling and mesh optimization using MSC PATRAN and MSC NASTRAN

(iii)Using linear static stress analysis under static loading method

1.5 FLOW CHART

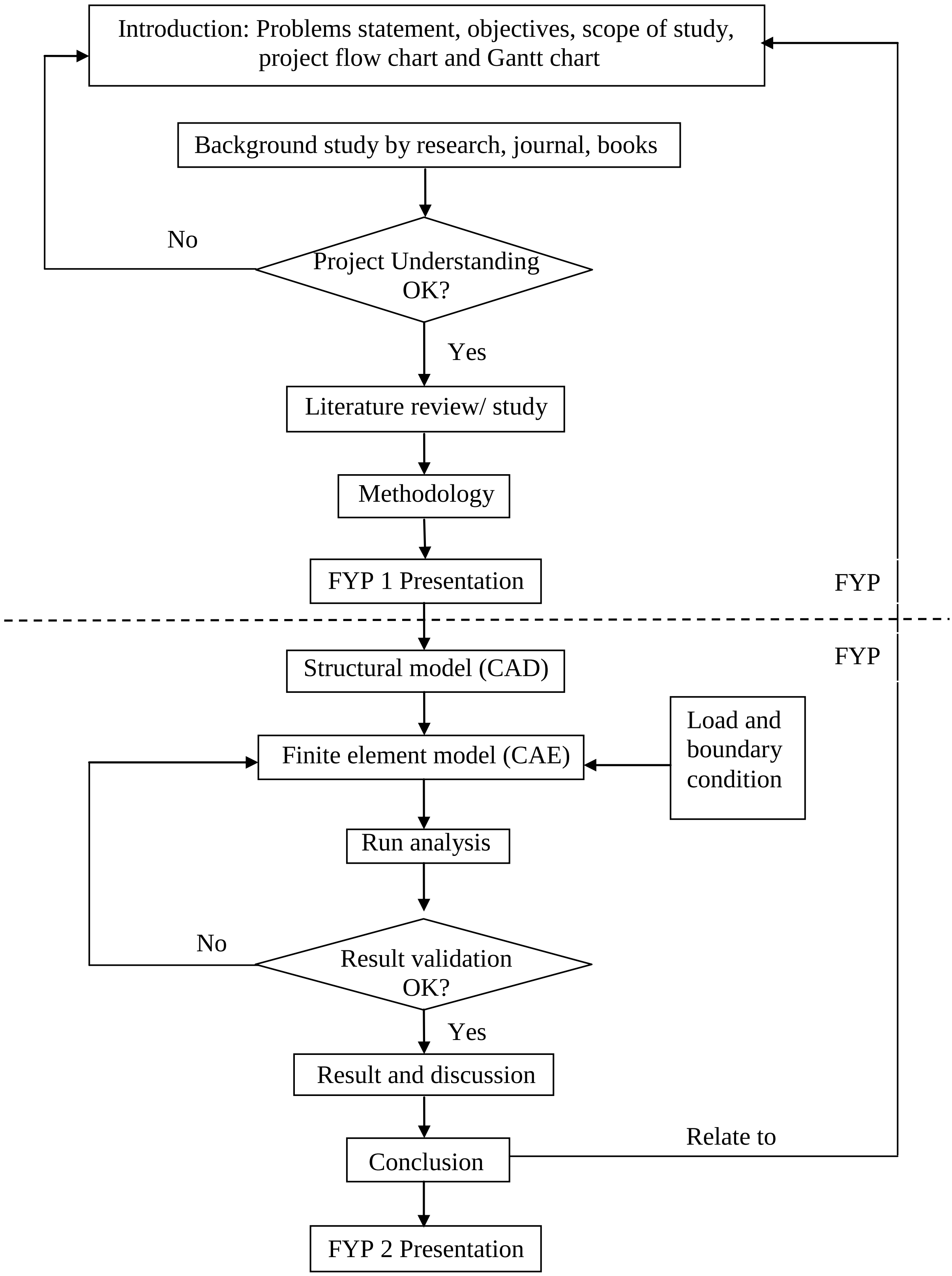

Figure 1.1: Project's flow chart

1.6 GANTT CHART

Table 1.1: Gantt chart for final year project 1

	TASK	SEMESTER 2 2008/2009															
		WEEK															
		1	**2**	**3**	**4**	**5**	**6**	**7**	**8**	**9**	**10**	**11**	**12**	**13**	**14**	**15**	**16**
1	Discuss the title of the FYP																
2	Project understanding																
3	Work planning																
4	Determine objectives, problems statement and scope																
5	Literature study related to the title of FYP																
6	Forming of research methodology																
7	Result expectation																
8	Discuss the information from literature study																
9	Project proposal approval & submission																
10	Presentation 1 preparation																
11	Presentation project 1																

Table 1.2: Gantt chart for final year project 2

	TASK	SEMESTER 1 2009/2010															
		WEEK															
		1	2	3	4	5	6	7	8	9	10	11	12	13	14	15	16
1	Literature study																
2	Introduction to CAD and CAE																
3	Structural modeling																
4	Finite element modeling																
5	Analysis																
6	Result and discussion																
7	Conclusion																
8	Presentation 2 preparation																
9	Project presentation 2																
10	Final report preparation																
11	Final report approval and submission																

1.5 OVERVIEW OF THE DISSERTATION

Chapter 1 gives the brief the content and background of the project. The problem statement, scope of study and objectives are also discussed in this chapter.

Chapter 2 discusses about the six stroke engine, piston, material, finite element analysis, mesh optimization and all software which is used for this dissertation.

Chapter 3 presents the development of methodology, structural modeling using CAD, finite element modeling and analysis and mesh optimization.

Chapter 4 discusses the result and discussion of the project. The discussion aims is to determine the predicted result can withstand the real pressure and stress in the engine condition.

Chapter 5 presents the conclusions of the project. Suggestions and recommendations for the future work about this dissertation.

CHAPTER 2

LITERATURE REVIEW

2.1 SIX-STROKE ENGINE

Blair (1990) studied about two-stroke engine which is the piston controlled the intake opening and closing that combine the intake and compression stroke in 180˚ crank rotation. For four-stroke engine, the cycle is more fuel-efficient, clean burning and higher engine power output compare to the two-stroke engine due to higher volumetric efficiency, higher combustion efficiency and low sensitivity to pressure losses in exhaust system. A single-cylinder two-stroke engine produces power every crankshaft revolution, while a single-cylinder four-stroke engine produces power once every two revolutions (Rajput, 2005). The two-stroke engine operational concept is applied to the original existing four-stroke engine cylinder head. The combination of this two engine operation is called six-stroke engine. There will have two pistons which are upper piston and the original four-stroke piston at the bottom in one cylinder of engine head. The ratio of stroke between upper piston and original four-stroke piston is 4:2 (Beare, 1998).

2.2 UPPER PISTON

Based on Challen & Baranescu (2003) studied, a piston in an engine is a component of reciprocating engines, pumps and gas compressors. It is located in a cylinder and is made gas-tight by piston rings. In an engine, its purpose is to transfer force from expanding gas in the cylinder to the crankshaft via a piston rod and/or connecting rod. Chov (2004) stated that in some engines, the piston also acts as a valve by covering and uncovering ports in the cylinder wall and it can be a carrier for the gas and oil sealing elements (piston rings). The duty cycle, the thermal and mechanical loading, and the requirements of long life and high reliability of the piston assembly represent one of the most arduous sets of conditions for any mechanical component in engineering today. Mechanical and thermal loading are the two type of loading which is applied to pistons during engine operation. Mechanical loading is due to the gas pressure in the cylinder and the reaction from the gudgeon pin and the cylinder wall.

For thermal loading, it is due to the temperature and heat transfer condition in the cylinder and at other boundaries to the piston. Based on the ignition process in the engine, pistons are subject to high temperatures on one side and relatively cold on the other side. It is necessary to understand the loading because this provides the basis for design, the reason for oil cooling and the cause of structural failure of a piston (Holt, 2004). In addition to the piston design, material is a major element in the piston manufacturing. The common material that always used in the manufacturing of piston is aluminum and cast iron. Nowadays, piston is made using aluminum as the raw material compare to cast iron in the early manufacturing of piston. Earlier low speed engine also had pistons of cast iron to match the material used for the cylinder. With increasing technology of engine speeds, modern pistons are made from an aluminum based on the characteristic (Challen & Baranescu, 2003).

2.3 ALUMINUM 356-T7 (Al 356-T7)

A unique combination of properties makes aluminum one of our most versatile engineering and construction materials. A mere recital of its characteristics is impressive. In term of light weight, aluminum 356-T7 has a density (ρ) of 2.68 Mg/m^3, compared with 7.9 Mg/m^3 for iron. Hence, for the same component, the aluminum will be about one-third of the mass of the iron version and make aluminum piston is more lightweight compare to cast iron piston. Thermal conductivity is the property of a material that indicates its ability to conduct heat and aluminum has high thermal conductivity compare to cast iron. The high thermal conductivity enables heat to be more rapidly conducted away and so result in the piston running at a lower temperature and reduce ability of piston to damage (Philip & Bolton, 2002). Next, aluminum has a great affinity for oxygen and any fresh metal in air rapidly oxidizes to give a thin layer of the oxide on the surface. This surface layer is not penetrated by oxygen and so protects the metal from further attack. This self-protecting characteristic gives aluminum its high resistance to corrosion. Unless exposed to some substance or condition that destroys this protective oxide coating, the metal remains fully protected against corrosion (Kalpakjian, 1995).

At temperature (*T*) up to about 300 ˚C in the engine, the material used must retain its properties (Rajput, 2005). High thermal expansion is however a problem for aluminum. Table 1.0 gives properties data for both materials. With absence of special design features, the different in expansion could result in seizure when hot (leaving enough room for the expansion would mean too loose a fitting piston when cold and result in 'piston slap').Such design feature include the piston being tapered so that there is additional clearance when cold near the top. It is this part which attains the highest temperature when hot and thus expands more than the cooler lower part of the piston (Colton, 2006).

Table 2.1: Properties of Aluminum 356-T7

Material	Aluminum 356-T7
Density, ρ (Mg/m^3)	2.7
Poisson Ratio	0.35
Modulus of Elasticity, E (Gpa)	72.4
Ultimate Tensile Strength, σ_{UTS} (MPa)	172

Source: Budynas & Nisbett (2008)

2.4 FINITE ELEMENT ANALYSIS

The stress-state for a component given the specific loading condition is defined by the finite element analysis results. The most common FE analysis method used in conjunction with fatigue analysis is to apply each load independently as a unit load case. The component geometry with FE model, the boundary conditions, and the loading information are the inputs for the FE analysis. The location and direction of each load input define the loading information. In the finite element method, the component behavior and material properties is assumed linearly elastic. The linear static superposition is used to combine the loading histories and the FE analysis results to calculate the local pseudo-stress history for an element. The local pseudo-stress history is corrected to take into account for any elastic-plastic behavior.

Rahman et al. (2006) stated that the finite element analysis (FEA) provide the foundation for predicting product performance throughout the entire design to manufacturing process and into the hands of customers. The advantages of finite element analysis (simulation) are less time, less cost and easier compare to experiment method. In addition, a dramatic improvement in computing power has made finite element (FE) based fatigue life calculations a routine task. Fatigue life can be estimated for every element in the finite element model and contour plots of life or damage plotted in a similar way to stresses. The finite element results define how an applied load is transformed into a stress or strain at a particular location in the component.(1)

Draper & Malton (2002) explained how the fatigue life can be calculated from finite element analysis. The stress tensor at a node and scales the stresses by a time history of the applied load, producing a time history of the elastically-calculated stress tensor. This stress history is converted into elastic-plastic stresses and strains using a cyclic plasticity model. The process is repeated for each node in the model, and the results are plotted as fatigue life contours. The method can be extended to analyze components which are subjected to many applied loads, applied at different points on the model. In this case, each applied load is analyzed separately in the FEA, and the results written to a stress output file as separate stress solutions.

2.5 MESH OPTIMIZATION

Zienkiewicz and Taylor (2000) stated that numerical method and approximation solution are required to solve most of partial differential equation in science and engineering. The Finite Element (FEM) and the Finite Volume (FVM) methods are among the most popular choices for such numerical approximations. Both the FEM and