International Journal of Engineering Research and Development
e-ISSN: 2278-067X, p-ISSN: 2278-800X, www.ijerd.com
Volume 12, Issue 10 (October 2016), PP. 58-66

Effect of Design Modifications of Piston Crown on Modal and Harmonic Response Analysis of Bajaj Kawasaki Four Strokes, Air Cooled, Diesel Engine Piston Using Finite Element Analysis

Nachiket Bhalwankar[1], Yadvendra Aslekar[2], Shardul Rautmare[3]
[1]*Marathwada Mitra Mandal's College Of Engineering, Pune, India.*
[2]*Marathwada Mitra Mandal's College Of Engineering, Pune, India.*
[3]*Pune Vidyarthi Griha's College Of Engineering And Technology, Pune, India.*

Abstract:- Modal analysis in FEA software ANSYS Workbench finds the natural frequencies and mode shapes of a structure and at each frequency. This work intends to analyse the modal and harmonic response analysis on the Bajaj Kawasaki four strokes, air cooled diesel engine pistons. Study also considers the design, analysis and evaluation of three new possible four stroke diesel engine piston designs with the aim of presenting a comparative analysis between them. The results achieved after the design and analysis were very satisfactory.
Keywords: - FEA; piston crown; ANSYS; modal analysis; harmonic response.

I. INTRODUCTION

The motive behind the research on this paper as, there is continuously increasing need for high power density; low emissions and high fuel efficiency impose restrictions on engine component design. Hence design and analysis of engine components has become more complex. The piston of a diesel engine is subjected to periodically changing mechanical loads. Vibrations are generated due to varying mechanical loads on piston during combustion process. The stress fields induced onto the piston due to vibrations are difficult to analytically determine, however using finite element analysis methods it possible to study and analyse the strength of pistons and other complex components and structures. The main requirements of the piston are that it should contain all the fluids above and below the piston assembly during the cycle and that it should transfer the work done during combustion process to crankshaft via the connecting rod with minimal mechanical losses. To meet this major requirement, the piston should have vibration resistance, high strength to weight ratio and. Thus new possible designs were suggested and modal, harmonic response analyses were carried out on actual Bajaj Kawasaki diesel engine piston as well as on new possible designs using FEA software ANSYS Workbench. In structural engineering, modal analysis uses the overall mass and stiffness of a structure to find the various periods at which it will naturally resonate. Vibration analysis, often called dynamic analysis. Modal analysis allows the design to avoid resonant vibrations or to vibrate at a specified frequency and gives engineers an idea of how the design will respond to different types of dynamic loads. The objective of this study is to determine the natural frequencies of pistons, study the mode shapes and subject the pistons to a harmonic loading varying in frequency from 0 to 5000 Hz and uniform pressure of 18 MPa on Piston Crown and study its response in terms of total deformation and equivalent Von-Misses stress.

II. IC ENGINE PISTON

Engine pistons are one of the most complex components among all automotive and other industry field components. The engine can be called the heart of a vehicle and the piston may be considered the most important part of an engine. There are lots of research works proposing, for engine pistons, new geometries, materials and manufacturing techniques, and this evolution has undergone with a continuous improvement over the last decades and required thorough examination of the smallest details. Piston damage is mainly due to wearing, temperature, and fatigue related. The fatigue related piston damages play a dominant role mainly due to thermal and mechanical fatigue, either at room or at high temperature. Piston helps to convert the chemical energy obtained by the combustion of fuel into useful (work) mechanical power. The purpose of the piston is to provide a means of conveying the expansion of gases to the crankshaft via connecting rod, without loss of gas from above or oil from below. Piston is essentially a cylindrical plug that moves up and down in the cylinder. It is equipped with piston rings to provide a good seal between the cylinder wall and piston[1].

A. Material Selection

The engine used for this work is a four cylinder four stroke air cooled type Bajaj Kawasaki diesel engine (C.I. Engine). Aluminium silicon alloys (predominantly eutectic) are widely employed to produce pistons due to their low density, high thermal conductivity, good cast ability and workability, good machinability and sound high temperature strength in this study the piston material is AlSi12CuMgNi cast alloy with eutectic microstructure. As all the engine components around the combustion chamber experience significantly high temperatures and temperature gradients, temperature dependent material properties have to be used. The material used for this piston is KS1275[2].

Table I: piston material (ks 1275) chemical composition

Element	Quantity
Si	11-13.5%
Cu	0.8-1.5%
Ni	0.7-1.3%
Mg	0.7-1.2 %
Fe	0.7% Max
Mn	0.35% Max
Al	Rest%

Table II: mechanical properties

Element	Quantity
Thermal conductivity	100 W/mk
Cu	0.8-1.5%
Coefficient of thermal expansion at 3000C	22.0 x 10-6 Kelvin
Hardness	60~70 HRB
Material of Piston	Cast aluminium alloy 201.0
Young's Modulus [E]	71 GPa
Poisson's ratio	0.33
Yield strength	435 MPa
Shear strength	290 MPa
Elongation	7 % [4-5]

Table III: bajaj kawasaki diesel engine specifications

Element	Quantity
Engine Type	Air cooled, Four stroke, diesel engine
Induction	TCIC
Number of cylinders	4 cylinders
Bore	74 mm
Stroke	70 mm
Length of connecting rod	97.6 mm
Displacement volume	99.27 cm3
Compression ratio	16
Maximum power	21.6 KW at 7000 rpm
Maximum Torque	86 Nm at 3500 rpm

B. Finite Element Analysis

FEA tool is the mathematical idealization of the real system. It is a computer based method that breaks geometry into elements joined by nodes and a series of equation to each element are formed, and then solved simultaneously to evaluate the behaviour of the entire system. An analytical solution is a mathematical expression that gives the values of desired unknown quantity at any location in a body structure and as a consequence, it is valid for an in finite number of locations in the body/structure. However, analytical solutions can be obtained only for simple engineering problems. It is extremely di cult to obtain exact analytical mathematical solutions for many complex engineering problems. In such cases, the technique known as finite element method (FEM) is used. In mathematics, the finite element method (FEM) is a numerical technique for finding approximate solutions to boundary value problems for partial differential equations. It uses subdivision of a whole problem domain into simpler parts, called finite elements, and variation methods from the calculus of variations to solve the problem by minimizing an associated error function. The subdivision of a whole domain into simpler parts has several advantages [3]

1. Accurate representation of complex geometry
2. Inclusion of dissimilar material properties
3. Easy representation of the total solution
4. Capture of local effects.

C. Meshing

In Finite Element Method, the body is divided into finite number of smaller units known as elements. This process of dividing the body is called as discretization. The elements are considered interconnected at joints which are known as nodes. In FEM the amount of data can be handled is dependent upon the number of elements into which the original body is divided. Mesh generation is the practice of generating a polygonal or polyhedral mesh that approximates a geometric do-main. The term grid generation[4] is often used interchangeably. Three dimensional meshes created for finite element analysis need to consist of tetrahedral, pyramids, prisms or hexahedra. In this work, the mesh generation is done with the help of ANSYS workbench. The mesh size or the element size is given as 2 mm in all the analysis. The method of mesh element or the type of mesh is selected as tetrahedron because of less CPU time requirement or less computational time and accuracy of results.

D. Modal Analysis

The goal of modal analysis in structural mechanics is to determine the natural mode shapes and frequencies of an object or structure during free vibration. It is common to use the finite element method (FEM) to perform this analysis because, like other calculations using the FEM, the object being analysed can have arbitrary shape and the results of the calculations are acceptable.

Modal analysis also called frequency analysis, modal analysis finds the natural or resonant frequencies of a structure and the shape of the structure at each frequency (also called the mode of vibration or the eigenvector.) [5] Modal analysis assumes the structure vibrates in the absence of any excitation and damping. This state is known as free undamped vibration. The FEA software ANSYS Workbench highlights areas in the first few modes (the first mode represents the lowest number of cycles/sec) as being predisposed to structural failure because it is these modes where the structure tends to maximize the kinetic energy. We carried out the modal analysis in ANSYS Workbench to determine natural frequencies, no of modes and mode shapes of 4 pistons subjected to varying frequency, ranging from 0 to 5000 Hz.

E. Harmonic Response

A harmonic, or frequency-response, analysis considers loading at one frequency only. Loads may be out-of-phase with one another, but the excitation is at a known frequency. A harmonic solution using the Mode Superposition method will automatically perform a modal analysis first. ANSYS automatically determines the number of modes 'n' necessary for an accurate solution although a free vibration analysis is performed first; the harmonic analysis portion is very quick and efficient. In a harmonic analysis, the peak response will correspond with the natural frequencies of the structure. Harmonic response analysis can be used to determine the stability response of the supporting structure under harmonic changing load with time. The responses of supporting structure under two or more kinds of frequencies will be calculated and the response peak will be found after deformation response curve with frequency is obtained. Because of the combustion process, the piston is influenced by a harmonic load with the same frequency as the force on piston. In harmonic response analysis using ANSYS Workbench the piston head/crown is subjected to uniformly distributed pressure of 18 MPa at a frequency ranging from 0 to 5000 Hz. The piston pin area is considered to be fixed support during all the analysis

III. PRESENT BAJAJ KAWASAKI PISTON

A. Piston after rendering process in Solid Works and piston after mesh generation

Fig. 1. (a) CAD Model (b) Mesh Model

B. Modal Analysis of Present Bajaj Kawasaki Piston

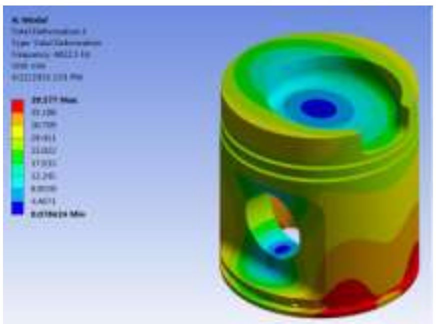
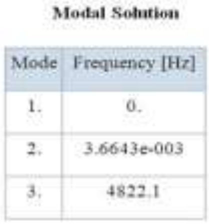


Fig. 2. Total deformation in modal analysis



Modal Solution

Mode	Frequency [Hz]
1.	0.
2.	3.6643e-003
3.	4822.1

Modal Solution Results

Object Name	Total Deformation	Total Deformation 2	Total Deformation 3
Definition			
Type	Total Deformation		
Mode	1.	2.	3.
Results			
Minimum	15.805 mm	18.17 mm	7.8424e-002 mm
Maximum	34.29 mm	31.013 mm	39.577 mm
Information			
Frequency	0. Hz	3.6643e-003 Hz	4822.1 Hz

Fig. 3. Image of Modal Solution Result Table generated by ANSYS

C. Harmonic Response Analysis of Present Bajaj Kawasaki Piston

Fig. 4. Total deformation and Equivalent stress in harmonic response analysis

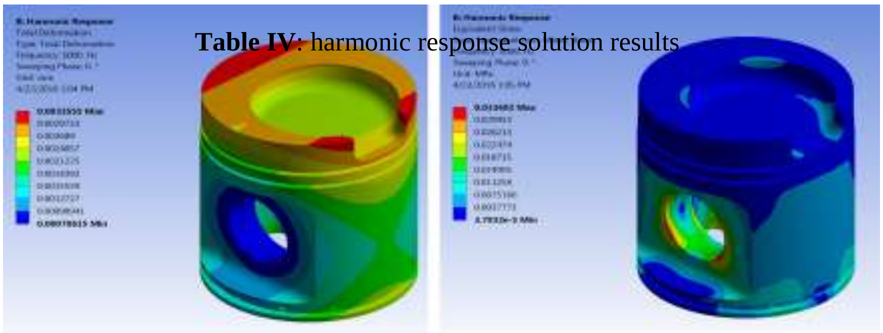


Table IV: harmonic response solution results:

Natural Frequency(Hz)	Total deformation(mm)	Equivalent Von-Misses stress (Mpa)
4822.1	0.003255	0.33692

D. Graph of Equivalent Von-misses stress against Frequency Variation

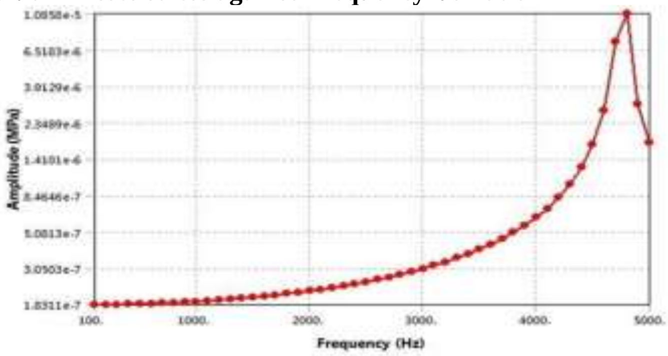


Fig. 5. Graph of frequency vs. Equivalent Von-misses stress

E. Graph of total deformation against Frequency Variation

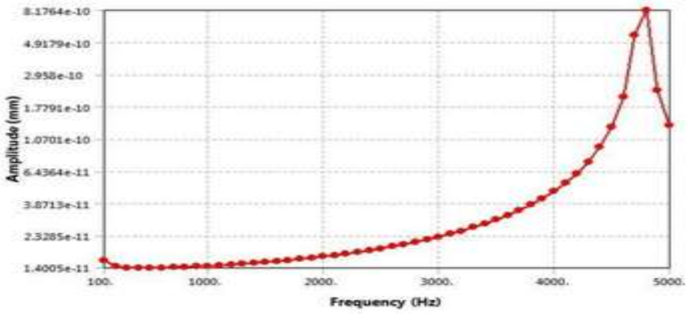


Fig. 6. graph of frequency vs. total deformation

F. Graph of phase angle vs. equivalent stress

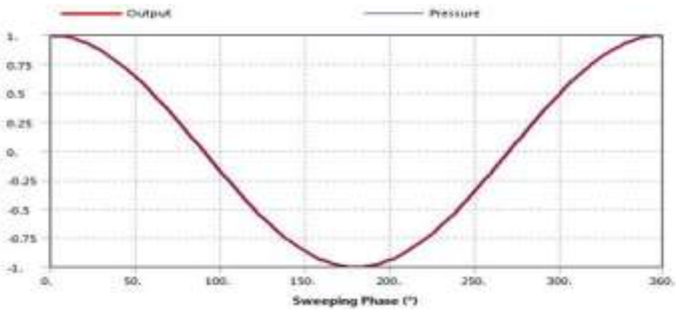


Fig. 7. graph of phase angle vs. equivalent stress

IV. FLAT CROWN PISTON

A. Piston after rendering process in Solidworks and piston after mesh generation

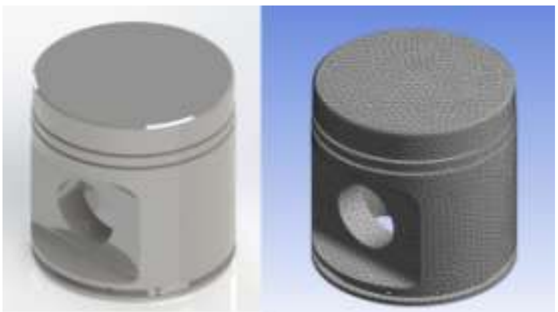


Fig. 8. (a) CAD Model (b) Mesh Model

B. Modal Analysis of Flat Crown Piston

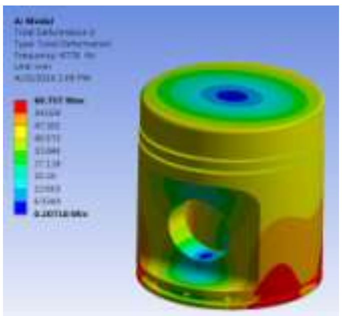


Fig. 9. Total deformation in modal analysis

Modal Solution

Mode	Frequency [Hz]
1.	0.
2.	4.7164e-003
3.	4778.

Modal Solution Results

Object Name	Total Deformation	Total Deformation 2	Total Deformation 3
Definition			
Type	Total Deformation		
Mode	1.	2.	3.
Results			
Minimum	27.328 mm	26.346 mm	0.20718 mm
Maximum	49.086 mm	50.259 mm	60.757 mm
Information			
Frequency	0. Hz	4.7164e-003 Hz	4778. Hz

Fig. 10. Image of Modal Solution Result Table generated by ANSYS

C. Harmonic Response Analysis of Flat Crown Piston

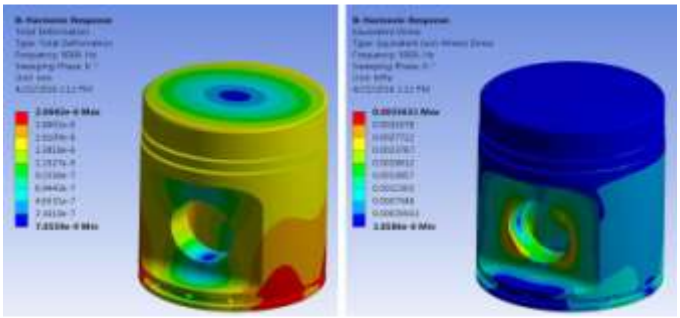


Fig. 11. Total deformation and Equivalent stress in harmonic response analysis

Table V: harmonic response solution results:

NaturalFrequency(Hz)	Total deformation (mm)	Equivalent Von-Misses stress (Mpa)
4778.0	2.0692e-6	0.003563

V. SEMICIRCULAR CAVITY CROWN PISTON

A. Piston after rendering process in Solidworks and piston after mesh generation

Fig. 12. (a) CAD Model (b) Mesh Model

B. Modal Analysis of Semicircular cavity Crown Piston

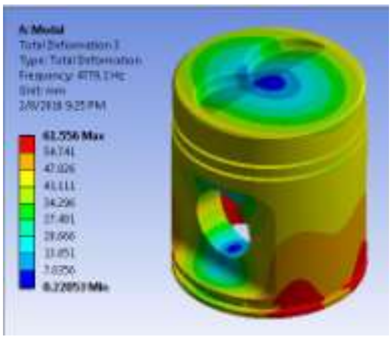


Fig. 13. Total deformation in modal analysis

Mode	Frequency [Hz]
1.	0.
2.	5.459e-003
3.	4779.1

Object Name	Total Deformation	Total Deformation 2	Total Deformation 3
Definition			
Type	Total Deformation		
Mode	1.	2.	3.
Results			
Minimum	26.32 mm	27.701 mm	0.22053 mm
Maximum	51.017 mm	49.34 mm	61.556 mm
Information			
Frequency	0. Hz	5.459e-003 Hz	4779.1 Hz

Fig. 14. Image of Modal Solution Result Table generated by ANSYS

C. Harmonic Response Analysis of Semicircular cavity Crown Piston

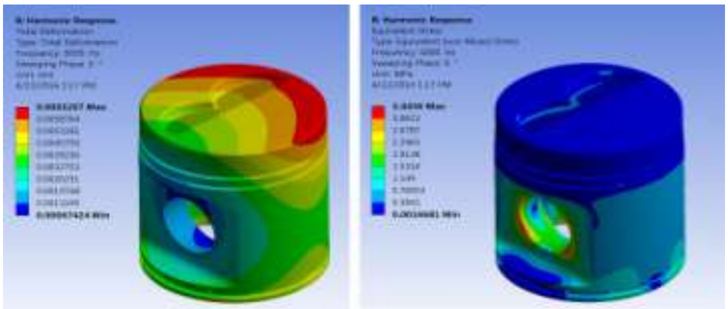


Fig. 15. Total deformation and Equivalent stress in harmonic response analysis

Table VI: harmonic response solution results

NaturalFrequency(Hz)	Total deformation (mm)	Equivalent Von-Misses stress (Mpa)
44779.1	0.0065267	3.4436

VI. CYLINDRICAL CAVITY CROWN PISTON

A**. Piston after rendering process in Solidworks and piston after mesh generation**

Fig. 16. (a) CAD Model (b) Mesh Model

A. Modal Analysis of Cylindrical cavity crown piston

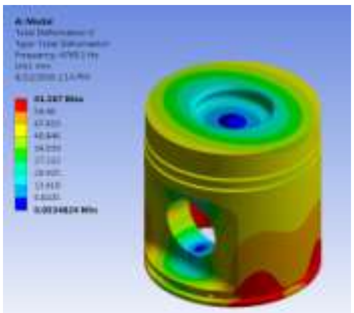


Fig. 17. Total deformation in modal analysis

Modal Solution

Mode	Frequency [Hz]
1.	0.
2.	4.3329e-003
3.	4760.1

Modal Solution Results

Object Name	Total Deformation	Total Deformation 2	Total Deformation 3
Definition			
Type	Total Deformation		
Mode	1.	2.	3.
Results			
Minimum	30.07 mm	24.151 mm	3.4824e-003 mm
Maximum	47.303 mm	54.714 mm	61.267 mm
Information			
Frequency	0. Hz	4.3329e-003 Hz	4760.1 Hz

Fig. 18. Image of Modal Solution Result Table generated by ANSYS

B. Harmonic Response Analysis of Cylindrical cavity crown piston

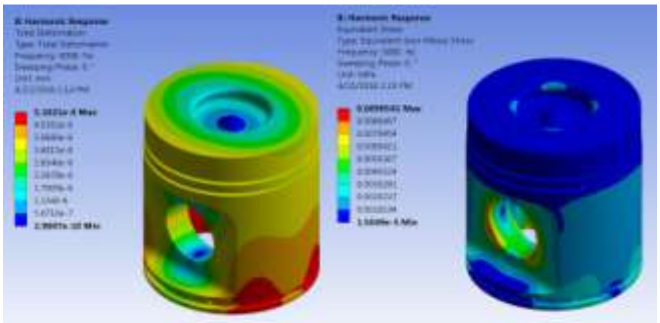


Fig. 19. Total deformation and Equivalent stress in harmonic response analysis

Table VII: harmonic response solution results

Natural Frequency(Hz)	Total deformation (mm)	Equivalent Von-Misses stress (Mpa)
4760.1	5.1021e-6	0.0090541

VII. ANALYSIS OF PISTON MODELS

Table VIII: analysis table of piston models

Piston Name	No. of Modes	Natural Frequency (Hz)	Deformation in Modal (mm)	Deformation in Harmonic (mm)	Equivalent stress (Mpa)
Present Bajaj Kawasaki Piston	3	4822.1	39.577	0.003255	0.33692
Flat crown piston	3	4778.0	60.757	2.0692e-6	0.003563
Semicircular cavity crown piston	3	4779.1	61.556	0.0065267	3.4436
Cylindrical cavity crown piston	3	4760.1	61.267	5.102e-6	0.0090541

VIII. CONCLUSION

These analyses prove to be very helpful in the design of the pistons for dynamic conditions and determining their natural frequencies, mode shapes and number of modes. The frequency response and phase response help in better understanding of vibration response of a component subjected to dynamic loading. In addition, the pistons with flat crown and cylindrical cavity crown have deformation and stress resistant capabilities as compared to presently used piston.

ACKNOWLEDGEMENT

We take this opportunity to thank all those who have directly inspired, directed and helped us towards successful completion of this paper. We express our sincere thanks to Prof. S. N. Satpute, Head of the Dept, Mechanical Engineering, Marathwada Mitra Mandal's College of Engineering, Pune and Prof. S. N. Belsare for their encouragement throughout this study work. We are thankful to Mr. Abhishek A. Keskar of a^2 Innovative Research and Training, Pune, India for his kind support and valuable guidance. We express our sense of gratitude towards our friends and parents for their constant moral support during completion of this paper work.

REFERENCES

[1]. A Textbook of Machine Design by R.S.Khurmi and J.K.Gupta.

[2]. Design and Analysis of Piston by using Finite Element Analysis by Sandeep K. Kourav, Vishnu B. Ghagare. International Journal of Engineering Research and Technology (IJERT) ISSN: 2278-0181 IJERTV4IS090411 Vol. 4 Issue 09, September-2015.

[3]. An Introduction to the Finite Element Method (Third ed.) by J.N. Reddy, (2006).McGraw-Hill. ISBN 9780071267618.

[4]. Geometry and Topology for Mesh Generation by Edelsbrunner, Herbert (2001), Cambridge University Press, ISBN 978- 0-521-79309-4.

[5]. Vibration Analysis for Designers by Paul M. Kurowski and Tom P. Kurowski Design Generator Inc. London, Ontario, Canada Resources.