IJARSCT
ISSN (Online) 2581-9429
IJARSCT
Impact Factor: 7.301
International Journal of Advanced Research in Science, Communication and Technology (IJARSCT)
International Open-Access, Double-Blind, Peer-Reviewed, Refereed, Multidisciplinary Online Journal
Volume 3, Issue 2, July 2023

Dynamic Structural Analysis of Four Stroke Petrol Engine Piston

Mr. Jadhav R. B. [1], **Mr. Vanam V. S.** [2], **Mr. Kaklij S. S.** [3], **Mr. Phaphale D.S.** [4], **Mr. Chaudhari S. N.** [5], **Mr. Godase R. B.** [6]
Professor, Department of Automobile Engineering [1, 2, 3, 4]
Professor, Department of Mechatronics Engineering [5,6]
Amrutvahini Polytechnic, Sangamner, India

Abstract: *Dynamic-structural-analysis is the study of free vibration analysis of the structure, which involves determination of mode -shape, natural-frequencies, transient dynamic response and random dynamic-stress. This paper explains the importance of the structural- dynamic-analysis during designing stage of any structure component. The aim of this paper is to discuss structural-dynamic-analysis by two methods i.e. modal-analysis -method and finite-element-analysis method. Further they are coordinated into an effective diagnostic procedure and it is demonstrated on air cooling petrol engine piston. The modal-analysis is carried on FFT analyzer and FEM is carried on ANSYS software.*

Keywords: Design, Finite element, FFT analyzer Frequency, Structural

I. INTRODUCTION

In automobile industries there is an increasing demand of higher productivity and reducing the lead time of economical design for higher speed for machinery with efficient use of materials though lightweight structure without internal less noise and resonance with mating parts. For fulfilling these demands one parameter must be considered i.e. 'structural dynamics vibration'. The structural dynamic testing is used for rapidly identifying modes shapes and their natural frequencies which are basic dynamic properties of a structure.

Thus the basic structural dynamic data, when obtained accurately from a valid test, also provides a true identification of the structural properties material. It is essential for the design and higher productivity for the development of new structures during the design stage which will reduce the lead time of optimization for master prototype production. In the last few decades, a great revolution has occurred in specialized testing equipment and efficient numerical methods for dynamic modal calculation of structures.

This paper discusses the interrelationship between the two popular methods i.e. modal-analysis and finite-element-analysis. And it concludes with an example of an automobile piston showing how experimental modal analysis and finite element analysis are used for analyzing the dynamic properties of material. The results are obtained and validated with each other. Both of these methods show the weakness of the piston material during testing, which will be helpful for improving material properties. This data is useful for the selection of another material which is worked with piston in single assembly i.e. internal combustion engine. This structure dynamic analysis for the piston in design stage is made beneficial before application for avoiding resonance, internal noise and failure of piston and cylinder assembly in running condition.

1.1 Advantages of Finite Element Analysis

Finite element analysis in conjunction with the high-speed digital computer permits the efficient solution of large, complex-structural-dynamics problems. As the majority of structural dynamics problems are linear they can be solved in the frequency domain using modal transformation. Many efficient and comprehensive finite-element computer-codes are now available to perform structural-dynamics-analysis calculations involving harmonic-response, transient-response, and random-response of complex structures. Provision is made in many large codes for storing specific solutions on tape and using these solutions as input to a second related problem, involving the same structure. For example dynamics problems where high temperatures cause changes in the elastic properties of the structure may be addressed by solving for the temperature distribution to the natural frequency calculations.

1.2 Advantages of Modal Analysis

The advantages of modal analysis are; First, it provides the most rapid and effective procedure available for the acquisition of data on the dynamic properties of a structure. Second, it is an effective analytical procedure for the solution of large sets of structural dynamic equations because it reduces coupled matrix equations (which must otherwise be solved by some iterative procedure) to a set of independent linear equations, each with the well-known closed-form solution. Modal solutions can therefore be obtained directly, without further numerical operations. These solutions are then re-combined to form the complete solution to the structural response problem in question. It should here be noted that solutions to harmonic, transient, random forced vibration problems can obtained by modal analytical procedure.

II. LITERATURE SURVEY

A vast amount of literature related to finite element analysis (FEA) exists. Many research publications, journals, reference manuals, newspaper articles, handbooks are available of national and international editions dealing with basic concepts of FEA. Many other publications indicate the success story of implementation of FEA on various components. The literature review presented here considers the major development in the implementation of FEA.

P. Ghodake and K. N. Patil [1] presented the method for piston design and analysis by CAE tools. In their work, piston design was carried out with the help of CAE tools and various stresses such as maximum principal stresses, minimum principal stresses, Von-Mises stresses and total deflection occurred during working condition were evaluated. The parameter used for the simulation are operating gas pressure and material properties of piston. The simulation results showed that the stresses occurred in piston are within the permissible limit of the piston material and deflection is well within the tolerance limit as provided by IS standard.

F. S. Silva [2] presented a linear static and thermal stress analysis of piston during the combustion using 'cosmos works' software and determined the stress distribution. In this paper, pistons of petrol and diesel engines of automobiles were analyzed and it was deduced that damage initiated at the piston crown, ring grooves, pin holes and skirt at some particular regions. The results indicated that for reducing damage of pistons, improved changes in design, material and processing technologies are to be used which are available.

V. Esfahanian, A. Javaheri and M. Ghaffarpour [3] carried out thermal analysis of S.I. engine piston using different combustion boundary condition. In this study, the heat transfer to an engine piston crown was calculated. Three different methods for the combustion boundary conditions were used. The results of different combustion side boundary condition treatments were compared and their effects on the thermal behaviour of the piston were investigated.

H. Okamoto, N. Anno and T. Itoh [4] presented a new computational and experimental stress analysis for determining the stress in the piston which is an improvement over finite element method, it is called boundary element method (BEM). It can calculate stress on the piston merely by meshing the piston surface shape in much less time. It also have developed methods for thermal load and fatigue load tests to verify the result of stress analysis and have designed optimized piston configurations to reduce crack fomiation as quickly as possible.

2.1 Material Specification of Piston

Chemical Combination of Material Aluminium Alloy AC 360-T7

Cu	Pb	Sn	Si	Mg	Zn	Fe	Mn	Ni	Ti	Al
2.99	0.23	0.42	14.98	1.49	0.46	0.76	0.046	0.91	0.032	Rest

2.2 Structural Dynamic Analysis By Theoretical Method

Finite-element-analysis is a computerized procedure for the analysis of structures. Rapid engineering analyses can be performed because the structure is represented (modeled) using the known properties of standard geometric shapes, i.e., finite elements. Efficient, large, general-purpose computer codes now exist with appropriate matrix assembler routines and equation solvers for calculation of the following structural properties:

IJARSCT
ISSN (Online) 2581-9429
International Journal of Advanced Research in Science, Communication and Technology (IJARSCT)
International Open-Access, Double-Blind, Peer-Reviewed, Refereed, Multidisciplinary Online Journal
Impact Factor: 7.301
Volume 3, Issue 2, July 2023

a) Static displacement and static stress.
b) Natural frequencies and mode shapes.
c) Forced harmonic response amplitude and dynamic stress
d) Transient dynamic response and transient stress.
e) Random forced response, random dynamic stress.

Finite element analysis used in this manner provides the dynamic properties of structures, including mode shapes and corresponding natural frequencies.

General purpose finite element codes such as NASTRAN, ANSYS, SAP, ADINA, etc., are programmed to develop and solve the matrix equation of motion for the structure as following.

If we disturb any elastic structure in an appropriate manner initially at time $t = 0$ (i.e., by imposing properly selected initial displacements and then releasing these constraints), the structure can be made to oscillate harmonically. This oscillatory motion is a characteristic property of the structure and it depends on the distribution of mass and stiffness in the structure. If damping is present, the amplitudes of oscillations will decay progressively and if the magnitude of damping exceeds a certain critical value, the oscillatory character of the motion will cease altogether. On the other hand, if damping is absent, the oscillatory motion will continue indefinitely, with the amplitudes of oscillations depending on the initially imposed disturbance or displacement. The oscillatory motion occurs at certain frequencies known as natural frequencies or characteristic values, and it follows well-defined deformation patterns known as mode shapes or characteristic modes. The study of such free vibrations (free because the structure vibrates with no external forces after $t = 0$) is very important in finding the dynamic response of the elastic structure.

By assuming the external force vector P to be zero and the displacements to be harmonic as $= Q.e^{iwt}$ gives the following free vibration equation:

$$[K]-\omega^2[M]\ Q=0 \qquad (1)$$

Where Q represents the amplitudes of the displacements Q (called the mode shape or eigenvector), and ω denotes the natural frequency of vibration. Equ. (1) is called a "linear" algebraic eigen value problem since neither $[K]$nor $[M]$is a function of the circular frequency ω, and it will have a nonzero solution for Q provided that the determinant of the coefficient matrix $([K] — \omega^2 [M])$ is zero, that is

$$[K]-\omega^2[M] = 0 \qquad (2)$$

In general, all the Eigen values of Eq. 2 will be different, and hence the structure will have ndifferent natural frequencies. Only for these natural frequencies, a nonzero solution can be obtained for Q from Eq. 1 We designate the eigenvector (mode shape) corresponding to the j^{th} natural frequency (ω_J) as Q_j.

It was assumed that the rigid body degrees of freedom were eliminated in deriving Eq.1. If rigid body degrees of freedom are not eliminated in deriving the matrices [K] and [M], some of the natural frequencies ω would be zero. In such a case, for a general three-dimensional structure, there will be six rigid body degrees of freedom and hence six zero frequencies. It can be easily seen whyω = 0 is a solution of Eq.1.

For ω=0, Q = Q = constant vector in Eq.1and Eq. 2 gives

$$[K]\ Q\ Rigid\ Body\ = 0$$

This is obviously satisfied due to the fact that rigid body displacements alone do not produce any elastic restoring forces in the structure. The rigid body degrees of freedom in dynamic analysis can be eliminated by deleting the rows and columns corresponding to these degrees of freedom from the matrices $[K]$and $[M]$ and by deleting the corresponding elements from displacement $(Q)$and load (P)vectors.

2.2 Procedure for a FEM Analysis

It consists of four main steps
1. Geomentry modeling of piston
2. Apply loads and obtain the solution
3. Expand the modes
4. Review the results.

IJARSCT
ISSN (Online) 2581-9429
International Journal of Advanced Research in Science, Communication and Technology (IJARSCT)
International Open-Access, Double-Blind, Peer-Reviewed, Refereed, Multidisciplinary Online Journal
Impact Factor: 7.301
Volume 3, Issue 2, July 2023

A. Geometry modeling of piston

The piston of an engine includes irregular parts and shapes (such as pin hole, ribs, skirt, concave land slots on piston top surface etc), so it is very difficult to measure perfect 2-D dimensions of the piston with geometrical and dimensional-tolerances. A few experimental methods are in practice such as digitization with co-ordinate measuring machine for measuring the dimensions of complicated parts of an automobile engine. However, these methods are very expensive and time consuming. [52] Hence the 2-D dimensions of the piston are measured manually with the help of transitional measuring devices like verniercaliper, height gauge etc by reverse-engineering-method.[52] The values of geometrical and dimensional tolerances are taken from manufacturer catalogs and after discussion with R& D persons involved in the piston design area.

Surface and solid modeling of the piston is done in CATIA V9 with help of the manually measured dimensions, and then the solid model is imported in ANSYS 10.

B. Apply loads and obtain the solution.

After the building the piston model, the next process is to apply load. This load is as per design constraint. In this step, we use the solution processor to define the analysis type and analysis option, apply load, specify load step, option and initiate the finite element solution. The word loads, as used in this manual includes boundary conditions as well another externally and internally applied loads. The most of these loads either on solid model or finite element model. Another term in load applied is load step and set up. A load step is simply a configuration of loads for which you obtain a solution. After applying load obtained a solution by using ANSYS 10 software.

C. Expand Modes.

Expand means reduced solution is usually termed as Degree of freedom. The term expansion to mean writing a model shaped to results files, expanding the model applies not just to reduced mode shapes from the reduced mode shaped from the reduced mode extraction method, but to full mode shape from the other mode extraction method as well. Thus, if we want to review mode shape in post processor, we must expand them.

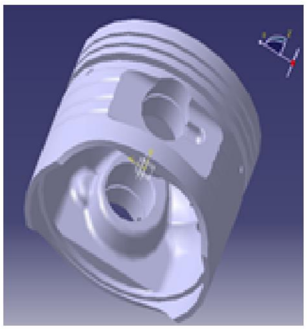

Figure 2. Imported model in Catia V9

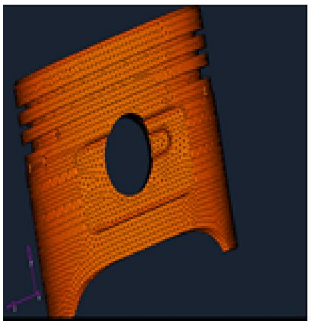

Figure 3. Vibration Analysis Model of piston

D. Review the Result

Once the solutions are calculated, use the ANSYS10 postprocessor. The general postprocessor, to review the result at one sub step over the entire piston model or selected portion of model. By obtains contour displace, deformed shape and tabular listing to review and interpret the result of analysis, it also include error-estimation, load case combination, calculation among result data and path operation.

III. STRUCTURAL DYNAMIC ANALYSIS BY EXPERIMENT METHOD

The computational model can deliver relatively accurate results in analysis only if model exactly match with actual field model in term of the geometry. So, obtaining higher accuracy in the thermo-mechanical analysis 3-D CATIA model is validated using FFT analyzer method

IJARSCT

ISSN (Online) 2581-9429

International Journal of Advanced Research in Science, Communication and Technology (IJARSCT)

International Open-Access, Double-Blind, Peer-Reviewed, Refereed, Multidisciplinary Online Journal

Impact Factor: 7.301

Volume 3, Issue 2, July 2023

Assumption for the testing [5]

i) Only linear behavior is valid in a Modal Analysis, if we use specify non-linear elements, they are treated as linear.
ii) Material property should be isotropic or orthotropic and constant or temperature dependent

General Test System Configurations

The basic test setup required for making the frequency response measurements depends on a few major factors. These include the type of the structure to be tested and the level of the results desired. Other factors, including the support fixture and the excitation mechanism, also affect the amount of hardware needed to perform the test. Fig.4 shows a diagram of a basic test system configuration.

The heart of the test system is the controller, or computer, which is the operator's communication link to the analyzer. It can be configured with various levels of memory, displays and data storage. The modal analysis software usually resides here, as well as any additional analysis capabilities such as structural modification and forced response. The analyzer provides the data acquisition and the signal processing operations. It can be configured with the several input channels, for the force and the response measurements, and with one or more excitation sources for driving shakers. Measurement functions such as windowing, averaging and Fast Fourier Transforms (FFT) computation are usually processed within the analyzer.

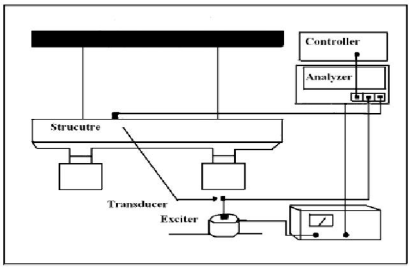

Figure 4. General Test Configuration of FFT

For making measurements on simple structures, the exciter mechanism can be as basic as an instrumented hammer. This mechanism requires a minimum amount of hardware. An electrodynamics shaker may be needed for exciting more complicated structures. This shaker system requires a signal source, a power amplifier and an attachment device. The signal source, as mentioned earlier, may be a component of the analyzer.

Transducers, along with a power supply for signal conditioning, are used to measure the desired force and responses. The piezoelectric types, which measure force and acceleration, are the most widely used for modal testing. The power supply for signal conditioning may be voltage or charge mode and is some-times provided as a component of the analyzer, so care should be taken in setting up and matching this part of the test system.[83]

Excitation the structure: Impact Testing

Another common excitation mechanism in modal testing is an impact device. Although it is a relatively simple technique to implement, it's difficult to obtain consistent results. The convenience of this technique is attractive because it requires very little hardware and provides shorter measurement times. The method of applying the impulse, shown in figure.5 includes a hammer, an electric gun or a suspended mass. The hammer, the most common of these, isused in the following discussion. However, this information also applies to the other types of impact devices

Since the force is an impulse, the amplitude level of the energy applied to the structure is a function of the mass and the velocity of the hammer. This is due to the concept of linear momentum, which is defined, as mass times velocity the linear impulse is equal to the incremental change in the linear momentum. It is difficult though to control the velocity of the hammer, so the force level is usually controlled by varying the mass. Impact hammers are available in weights varying from a few pounces to several pounds. Also, mass can be added to or removed from most hammers, making them useful for testing objects of varying sizes and weights.

The frequency content of the energy applied to the structure is a function of the stiffness of the contacting surfaces and, to a lesser extent, the mass of the hammer. The stiffness of the contacting surfaces affects the shape of the force pulse, which in turn determines the frequency content. It is not feasible to change the stiffness of the test object; therefore the

IJARSCT
ISSN (Online) 2581-9429
International Journal of Advanced Research in Science, Communication and Technology (IJARSCT)
International Open-Access, Double-Blind, Peer-Reviewed, Refereed, Multidisciplinary Online Journal
Impact Factor: 7.301
Volume 3, Issue 2, July 2023

frequency content is controlled by varying the stiffness of the hammer tip. The harder the tip, cause the shorter the pulse duration and thus the higher the frequency content.

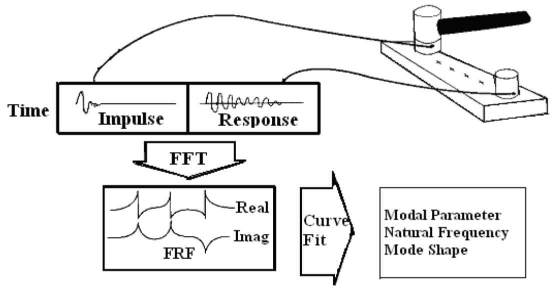

Figure 5. General Impact testing by Hammer

IV. EXPERIMENTAL APPARATUS

4.1 FFT Analyzer: Larson Davis 2009 B

The vibrating signal of a machine running under steady condition in time domain is often called as signature and is generally periodic in nature, since disturbing force may have different frequencies and their harmonics. Rarely one would find the signature to be purely harmonics. Hence the vibratory signal should be properly analyzed in frequency domain. A periodic motion can be broken down into several harmonic motions by using Fourier analysis. Fast Fourier Transformer is an electronic device that is capable of taking the time waveforms of a given signal and converting it into a frequency domain.[1,4,7]

Quite often it is sufficient to know the peak and average, absolute and RMS values of vibratory amplitude to check the condition of machine. These values are obtained by simply over-ranging of the signal commercial vibration meters give ahead out these values directly for a signal in a given frequency range. However to understand the behavior of a machine, it is necessary to analysis its signature by spectrum analysis.

4.2 Vibration measuring devices

The vibrometer is most and widely used, because of its low natural frequency, consequently of its high mass rarely finds its application in practice, particularly to mechanical system. The accelerometer is another device used because of its high natural frequency and consequently very light in its construction, finds immediate use in the measurement of vibration characteristics of a machine. The accelerometer as a measuring device has become more popular advent of sophisticated electronics for integration to determine the velocity as well as displacement amplitudes. [7]

Figure 6.Deltatron Accelerometer

4.3 Model hammer

The model hammer in Fig. 7 exits the structure with a constant force over a frequency range of interest. Three interchange tips are provided which determine the width of the input pulse and thus the bandwidth the hammer structure is acceleration compensated to avoid glitches in the spectrum due to hammer structure resonance. [1, 7]

IJARSCT
ISSN (Online) 2581-9429
International Journal of Advanced Research in Science, Communication and Technology (IJARSCT)
International Open-Access, Double-Blind, Peer-Reviewed, Refereed, Multidisciplinary Online Journal
Impact Factor: 7.301
Volume 3, Issue 2, July 2023

Figure 7. Model Hammer

4.4 Display unit

This is mainly in the form of PC (Laptop) in Fig.8 when the excitation occurs to the structure the signals transferred to the portable PULSE and after conversion comes in graphical form through the software. Mainly the data includes graphs of force Vs time, frequency Vs time resonance frequency data etc.

Figure 8. Display unit

Figure 9. Piston Specimen with Accelerometer and FFT analyzer

4.5 Setup and Procedure

Larson Davis 2009 B is used to measure the frequency ranges of an automobile piston.

1. The connections of the FFT analyzer, laptop, transducers, and model hammer along with the requisite power connections are made.
2. The accelerometer -4507 type is fixed by beeswax to piston at one of the nodal points.
3. The 2302-5 modal hammer is kept ready to strike the piston at the singular points.
4. Then at each point the modal hammer is struck once and the amplitude Vs frequency graph was obtained from graphical user interface 0.3
5. The FFT analyzer and the accelerometer are the interface to convert the time domain response to frequency domain. Hence the frequency response spectrum H_1 (response, force) is obtained.
6. By moving the cursor to the peaks of the FFT graph ($m/s^2/N$), the cursor values and the resonant frequencies are recorded.
7. At the time of the striking with modal hammer to the singular point precautions were taken whether the striking should have been perpendicular to the piston surface. The above procedure is repeated for all the nodal points.

The values of natural frequencies and resonance frequencies obtained from the FRF spectrums are compared with respect to the FEM analysis as follows.

IJARSCT

ISSN (Online) 2581-9429

International Journal of Advanced Research in Science, Communication and Technology (IJARSCT)

International Open-Access, Double-Blind, Peer-Reviewed, Refereed, Multidisciplinary Online Journal

Impact Factor: 7.301

Volume 3, Issue 2, July 2023

V. RESULT AND DISCUSSION

Ansys Software Gives Output of Natural Frequency and Mode Shape

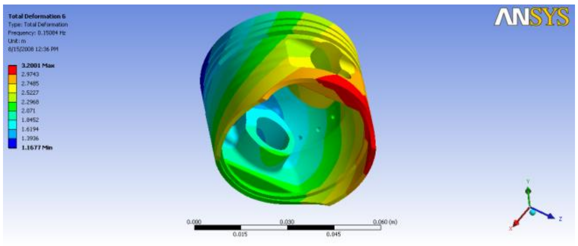

Figure 10. Mode .1st Frequency plots

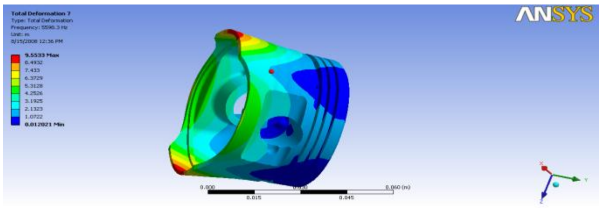

Figure 11. Mode 2nd Frequency Plots

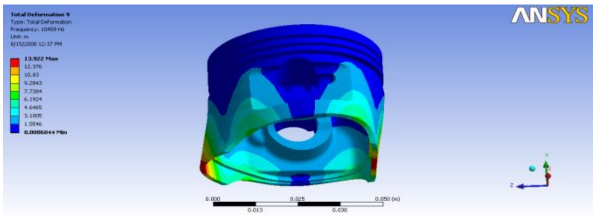

Figure 12.Mode 3rd Frequency Plots

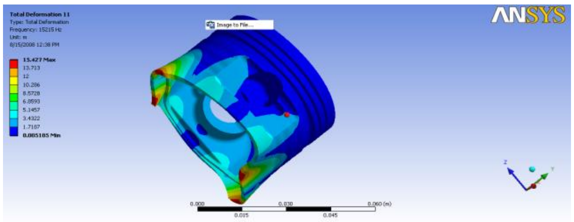

Figure 13. Mode 4th Frequency Plots

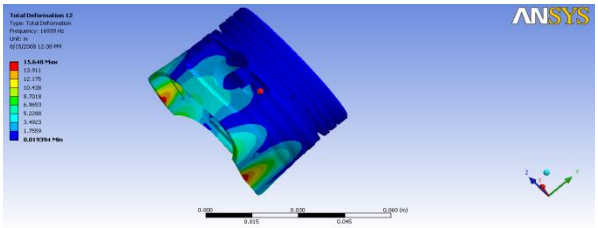

Figure 14. Mode 5th Frequency Plots

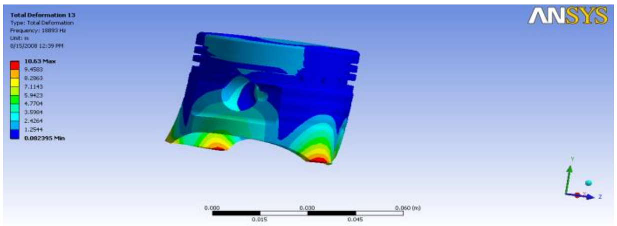

Figure 15. Mode 6th Frequency Plots

IJARSCT

ISSN (Online) 2581-9429

International Journal of Advanced Research in Science, Communication and Technology (IJARSCT)

International Open-Access, Double-Blind, Peer-Reviewed, Refereed, Multidisciplinary Online Journal

Impact Factor: 7.301

Volume 3, Issue 2, July 2023

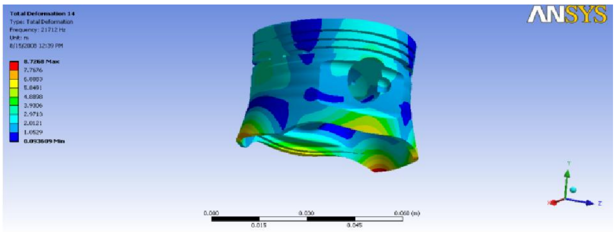

Figure 16.Mode 7th Frequency Plot

VI. CONCLUSION

Mode	ANSYS 10 (Theoretical Method)	FFT (Practical Method)	Percentage of Deviation
1	5598.3	5150	8.0
2	10459	9894	5.4
3	11247	11979	6.11
4	15215	14758	3.0
5	16039	14675	8.5
6	18893	17570	7.0
7	21712	19757	9.0

The structural-dynamic-analysis of a piston holds a lot of significance in its designing and performance over a period of time. FFT test analysis is carried out with free-free condition. It is seen that the results are in good co-ordinance with theoretical values. The lowest frequency was observed in 1stmode. The natural frequency is increasing with each subsequent mode of vibration. The predicted values of natural frequencies for the existing piston model with ANSYS10 software are quite close to natural-frequency, which is obtained experimentally. The average percentage of deviation of natural- frequencies by FFT analyzer and ANSYS10 software are found to be 6.71 %.The reason of deviation is that piston model includes curve surfaces and complex shapes (such as pin hole shape, slots on piston top surface etc.). The deviation is so less that this piston CATIA model is validated for further study. Methodology of this paper can be used for the dynamic-structural- analysis of all types of IC engine piston materials for improving its reliability and service life.

REFERENCES

[1]. Chopra and Anil K., Dynamics of Structures, Prentice Hall, New Jersey, 1995,pp: 24-53

[2]. P. Ghodake and K. N. Patil, "Piston Design and Analysis by CAE Tools", lOSR Journal of Engineering, pp. 33-36.

[3]. F. S. Silva , 2006, "Fatigue on engine pistons - A Compendium of Case Studies", Department of Mechanical Engineering, University of Minho, Portugal, Engineering Failure Analysis 13, 480-492. 3.

[4]. V. Esfahanian, A. Javaheri and M. Ghaffarpour, 2006, "Thermal Analysis of S.I. Engine Piston using Different Combustion Boundary Condition Treatments", Applied Thermal Engineering, pp. 277-287.

[5]. H. Okamoto, N. Anno and T. Itoh, 1992, "New Computational and Experimental Stress Analysis Method for the Design Decision on Optimum Piston Configuration of Production Engine", SAE International Journal.

[6]. Klaus Jurgen Bathe, Finite Element Procedures, Prentice Hall, Uppersaddle River, New Jersey, 1996

[7]. Brian J. Schwarz and Mark H. Richardson,**"**Experimental Modal Analysis" Vibrant Technology, Inc.Jamestown, California 95327 October, 1999

[8]. Patrick Ryan Turner, "Integrating Experimental and Finite Element Method Modal Analysis" ,Schlumberger Technologies, CAD/ CAM Division, Ann Arbor, Michigan, USA 48106,1999.

[9]. Neville F. Rieger, "The Relationship Between Finite element Analysis And Modal Analysis"Stress Technology Incorporated, Rochester, Network, 2007, pp: 1234-1240