FINITE ELEMENT MODAL ANALYSIS OF 4-CYLINDER ENGINE CRANKSHAFT

Sanjay Bhaskar Zope[1], Patil Amol[2] Mahendra Dhere[3]

[1] *Guide, Department of Mechanical Engineering, Savitribai Phule Pune University, Pune, India*
[2] *Co Guide, Department of Mechanical Engineering, Savitribai Phule Pune University, Pune, India*
[3] *Student, Department of Mechanical Engineering, Savitribai Phule Pune University, Pune, India*

ABSTRACT

Abstract— in design and analysis industry, ample time is spent on preparation of models, setting up of instruments and carrying out the testing. In this paper a three-dimension models of diesel engine crankshaft was created using CATIA V6/R2012 software. Meshing is done in Altair Hyperworks 13.0. The finite element solver MSC Nastran 20130 was used to analyse the modal parameters of the crankshaft. The relationship between the frequency by FEA and Experimental were compared to arrive at a conclusion

Keyword: - *Crankshaft, Finite element analysis, solid modelling ….*

I. INTRODUCTION

Crankshaft is very important component of an IC engine. During the analysis, the crankshaft strength is the main factor for the average life and performance of an IC engine. The vibrations generated during the operation of internal combustion engine are having very severe effect on the working of the engine. Vibration analysis can be done by experimental methods which is very critical and it is very impractical obtain actual parameters of boundary conditions.

With the advances in technology, more design of crankshaft has been used finite element method to calculate the various stress of crankshaft. By applying numerical simulation to the design of crankshaft helps engineers to efficiently improve the process progress minimizing the cost and short comes of arranging a database of real parts. Finite element analysis gives a low cost study of different combinations of input parameters including analysis and design.

In the last few years, finite element analysis has become much worthy as compared to actual experimental analysis. The results from finite element method are having about 3.2 % error when compared to experimental results in a study done out by Tianxin Zheng, Tianjian Ji. [1]

The model was prepared using CATIA V6/R2012 software. Then it was imported to the Altair Hyperworks 13.0 and at last it was analysed in solver MSC Nastran 20130.

II. PROBLEM STATEMENT

In industry, the time and resources required for the design analysis by experimental methods is very crucial. Experimental analysis requires special fixtures and accessories for the conduction of experiment. In case of crankshaft analysis, this problem is faced with more effect due to the intricacy of geometry and its working conditions. This thesis suggests the method of doing analysis by finite element method. The obtained results from the FEA are compared with the experimental results.

III. CRANKSHAFT ANALYSIS

Crankshaft Model

It is required to simplify the model in the stress analysis of crankshaft using Altair Hyperworks software. Because the crankshaft structure is symmetrical and the crank throw is identical, one crank throw model, half crank throw model and quarter crank throw model can be utilised to calculate the static strength instead of doing it for whole crankshaft model. The 3 models are the equivalent in computing the static strength of crankshaft, and the quarter crank throw model uses the minimum computer resources. But the amount of large stress and stress gradient is observed at the quarter model section. This case will lead to the inaccurate simulation results. Here, a crank throw model was utilised to calculate the stationary strength of crankshaft

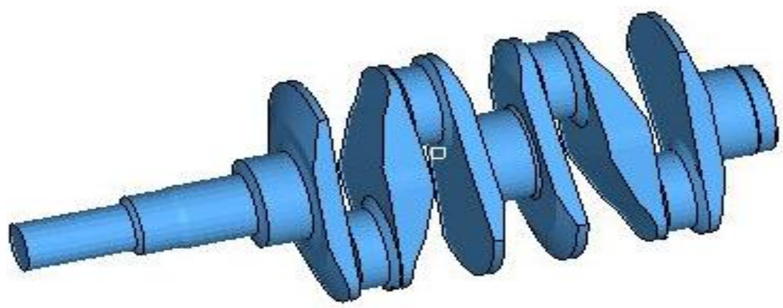

Fig.1. 3-D entity model of crankshaft

Table. 1. Simulation quantities

Physical quantities	Values
Modulus of Elasiticity (GPa)	208
Yield strength (MPa)	950
Notched bar impact strength (J/cm2)	75
Elongation (%)	13
Density (g/cm3)	7.7
Impacting energy (J)	61
Poisson ratio	0.3
Rigidity (HB)	213
Reduction in cross sectional area (%)	6
Tensile strength (MPa)	1000

III. FINITE ELEMENT METHOD

The paper focuses on representing very complex geometry of crankshaft into equivalent simple geometry beam to reduce complexity in modeling and analysis. The static behavior of the crankshaft is represented by cantilever loading condition and used to propose the cross section of equivalent beam. The dynamic behavior of the equivalent beam is verified with original component.

The geometry modeling is done in CATIA V6/R2012. Finite element model is prepared in Altair Hyperworks 13.0. This finite element modeling is done with tetragonal 3d elements. The element size used in this procedure is 5 mm to capture all the features of crankshaft and finally it was analyzed in solver MSC Nastran 20130.

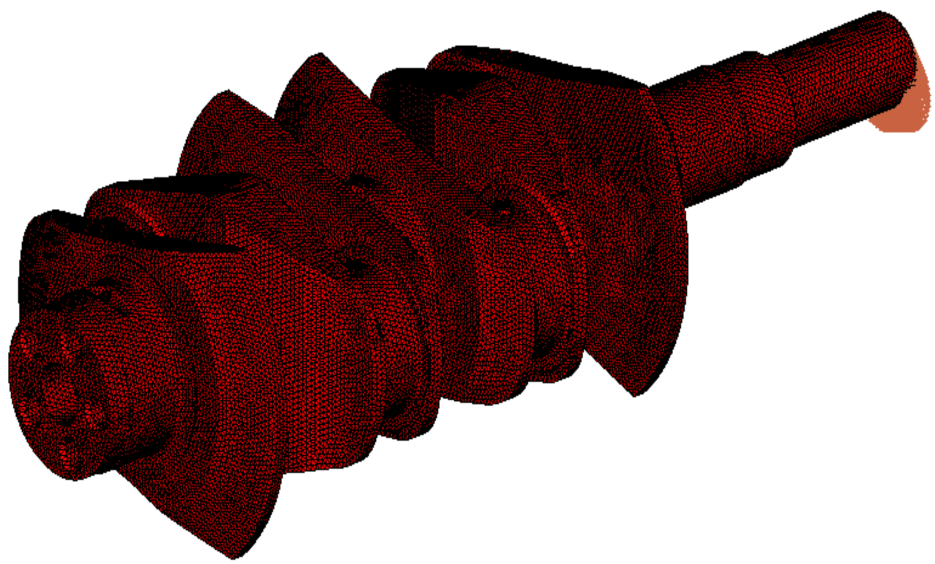

Fig.2. Crankshaft FE 3D mesh

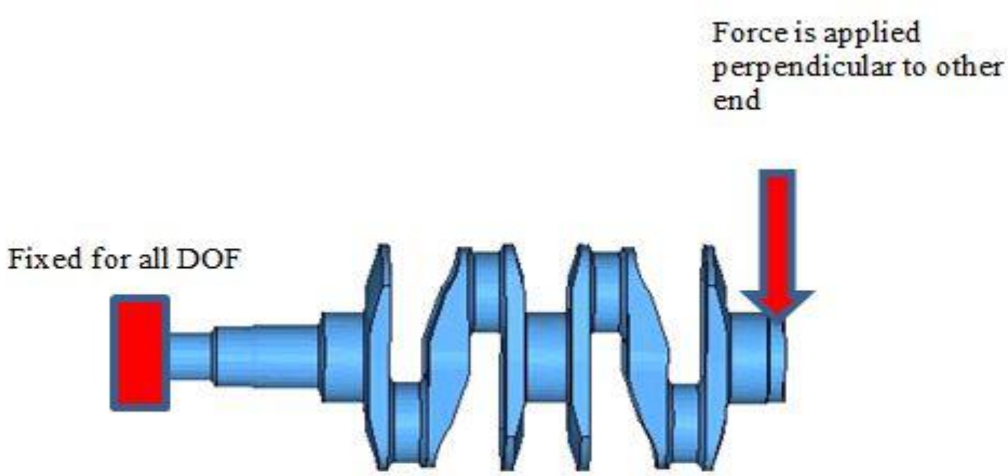

Fig.3. Loading conditions of crankshaft

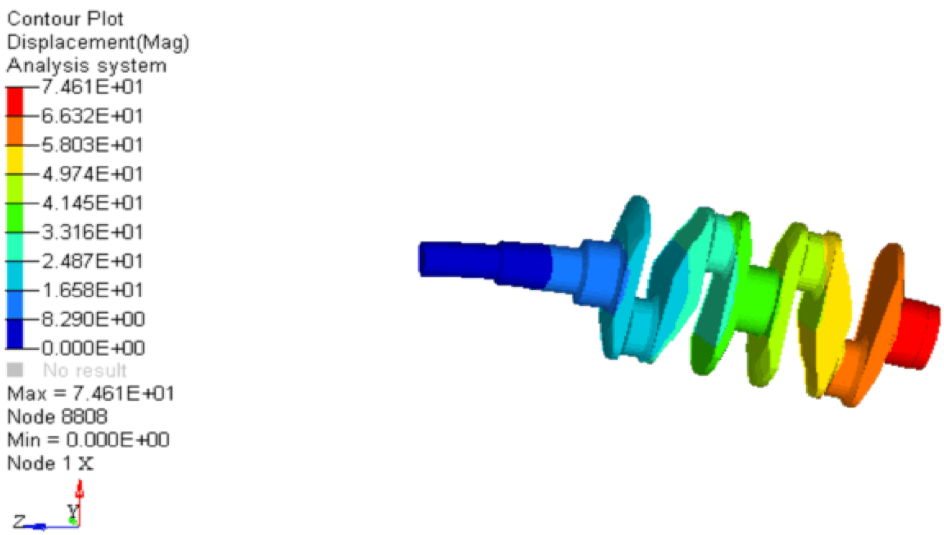

Fig.4. Crankshaft bending characteristics

V. EXPERIMENTAL ANALYSIS

The experimental analysis was carried out at facility centre of Pravara Rural Engineering College, Loni Ahmednagar. The tests are done with accelerometer OROS NV Gate Version 5. This analysis is carried out to find out the natural frequencies of the crankshaft. Free - Free position analysis is carried out on the crankshaft to obtain the modal parameters which means that constraints in the all directions were kept zero.[7] The obtained results are given in the below figures. The values obtained are tabulated in Table 1.

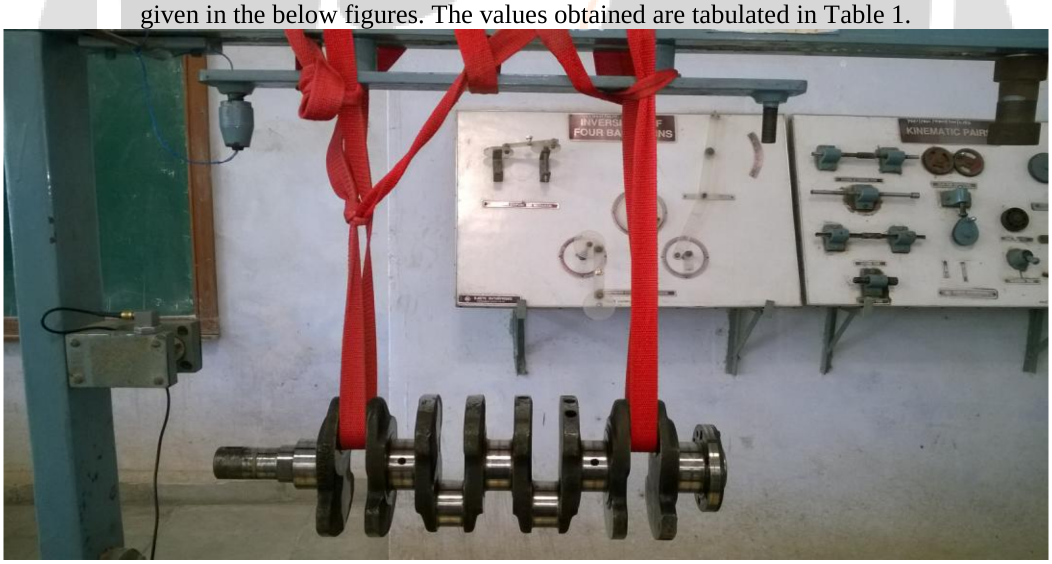

Fig.3. Experimental set up for analysis

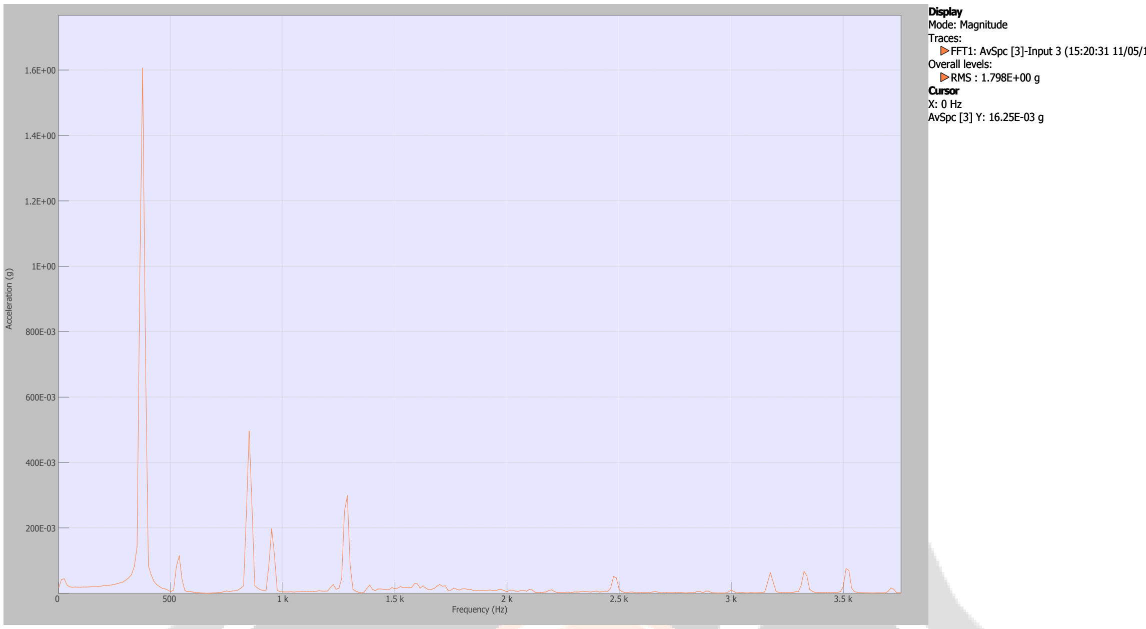

Fig.5 Frequency-Acceleration Plot

VI. RESULTS AND DISCUSSIONS

The results obtained are tabulated with respect to the computational time required for the analysis of crankshaft.

Table. 3. Comparison of results

Mode	FEM	Experimental
1	363.5	375
2	504	537.5
3	843.5	850
4	958	950
5	1307	1287

VII. CONCLUSIONS

Comparing the results of the Finite element method and experimental, the time required to perform the analysis by existing method is minimized by proposed method. The obtained results from finite element method show good match within the limits with experimental results.

REFERENCES

[1] .Tianxin Zheng, Tianjian Ji, Equivalent representations of beams with periodically variable cross-sections, Engineering Structures 2011; vol. 33, pp. 706–719

[2]. Yoonhwan Woo. Abstraction of mid-surface from solid model of thin walled parts a divide & conquer approach. Computer Aided Design 2014; vol. 41: pp. 1-11.

[3]. F.Cluni, M.Gioffre, V.Gusella. Dynamic response of tall buildings to wind loads by reduced order Equivalent shear Beam Models. Journal of wind engineering and industrial aerodynamics 2013; vol. 123: pp. 339-348.

[4]. J.G.Kim, J.K.Lee. A modified Beam Modeling method considering dynamic characteristics for curved flexible pipes. International Journal of pressure vessel and piping 2011; vol. 88: pp. 262-267.

[5]. Jian Meng, Yongqi Liu, Ruixiang Liu, Finite Element Analysis of 4-Cylinder Diesel Crankshaft, International Journal of Image, Graphics and Signal Processing, 2011, vol. 5, pp. 22-29.

[6]. M.Maleki, H.Sereshteh M.Mousivand M.Bayat. An Equivalent Beam model for the analysis tunnel building interaction. International Journal of Tunneling and underground space technology 2011; vol. 26, pp. 524-533.

[7]. R. G. Desavale , A. M. Patil, Theoretical and Experimental Analysis of Torsional and Bending Effect on Four Cylinders Engine Crankshafts by Using Finite Element Approach, International Journal of Engineering Research 2013; vol. 2, pp : 379-385.