International Journal of Technical Research and Applications e-ISSN: 2320-8163,
www.ijtra.com Volume 4, Issue 3 (May-June, 2016), PP. 391-393

A REVIEW: VIBRATION RESPONSE OF RECIPROCATING ENGINE CRANKSHAFT

Dipak P. Gulhane[1], Dr. Girish D. Mehta[2], Prof. S. M. Awatade[3]
Associate Professor[2], Assistant Professor[3]
Department of Mechanical Engineering
Priyadarshini College of Engineering, Nagpur
dipakgulhane82@gmail.com

Abstract— **Crankshaft is one of the main component in I.C-.Engine which converts reciprocating motion into rotation motion. The crankshaft has to bear many vibration and stresses and load of combustion in the engine. The most probable area of failure in crankshaft is near the crankpin. This paper present a survey of different types of dynamics analysis done performed using ANSYS ,FEA Software which gives a complete information about the stresses generated, stress variation ,load generation and point of failure in the engine crankshaft . The main objective of this paper is to show the research work done by different author on vibration analysis of crankshaft.**

Index terms- **IC-engine, Crankshaft, Vibration Response, Ansys, FEA .**

I. INTRODUCTION

Now a day's most of the automobile and marine equipment's are driven by reciprocating engines. Most of automobile industries are focussing on the comfort zone of the passenger therefore the study says that the comfort to passenger can be improved by minimizing the vibrations induced in automobile. It is postulated that engine is the main sources of vibration I the automobile .Many researches do focus on engine vibration phenomenon in detail.

Therefore the present work is planned taking this as an epic centre. The efficiency of I.C-engine primarily depends on the crankshaft which converts the reciprocating motion into rotation motion. The main and important part of engine is crankshaft which propel the vehicle.

II. RELATED STUDY

A. *"Modal Analysis of a 2-cylinder crankshaft using ANSYS" by Basavaraj Talikoti, Dr. S. N. Kurbet, Dr. V.V.Kuppast, Aevind M. Yadwad*

In this paper author has research model analysis providing a mean by which deformation in structure can be viewed while designing a crankshaft .Model analysis which done in ANSYS gives a vibration characteristics of structure since vibration is dynamics, it is helpful for dynamic analysis of crankshaft author has also focused on behaviour of vibration, the structure is at high risk when vibration are maximum.

It also shows the analysis result in different shapes of the structure at different frequencies of vibration .It also shows deflection of structure and distortion in geometry when the structure in under vibration of particular range.

The conclusion obtained here gives the relation of frequency and vibration characteristics of crankshaft.

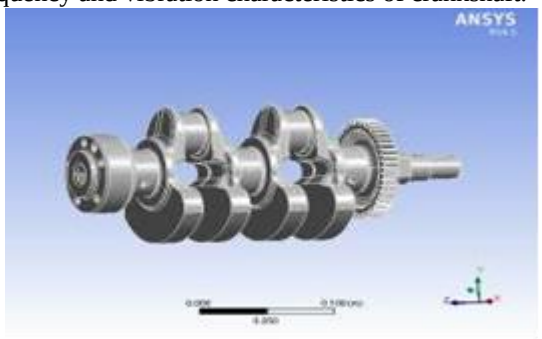

Fig. 1 The Geometry of Crankshaft imported in ANSYS.

B. *"Dynamic & Statics Analyses of Model of Engine Crankshaft" by R. B. Sharma & Vikas Sharma*

Author described the couple modes including couple torsional flexural vibration and coupled longitudinal flexural vibration in FEM of these structure where used are in two categories beam element & solid element .The author has investigation on the solid element category the model of crankshaft has been design Pro-e Wiidfire 4.0 with suitable dimension and after dynamics & statics analysis has been performed using ANSYS 14.0 And the result has where specified from various modes of result obtained for dynamic analysis and considering specific load and boundary condition for statics analysis it gives a detail knowledge of dynamic behaviour & statics behaviour of engine crankshaft under different respective conditions.

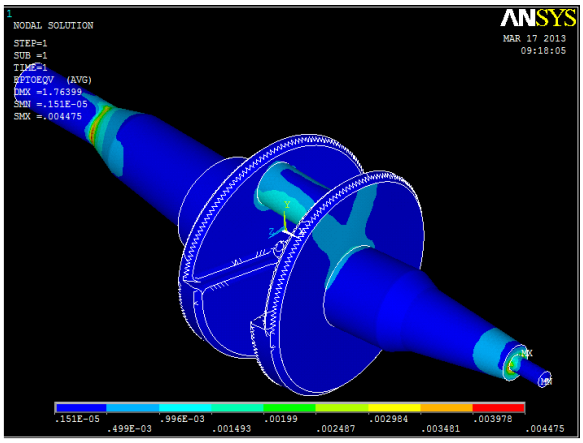

Fig.2 Total Von Mises mechanical strain

C. "Finite Element analysis of Diesel engine using FEM" by Jian Meng, Yongqi Liu

In this paper analysed crankshaft model and crank through were created by Pro-e software and then imported to ANSYS software. The crankshaft deformation was mainly bending deformation under the lower frequency. And the maximum deformation was located at the link between main bearing journal, crankpin and crank cheeks.

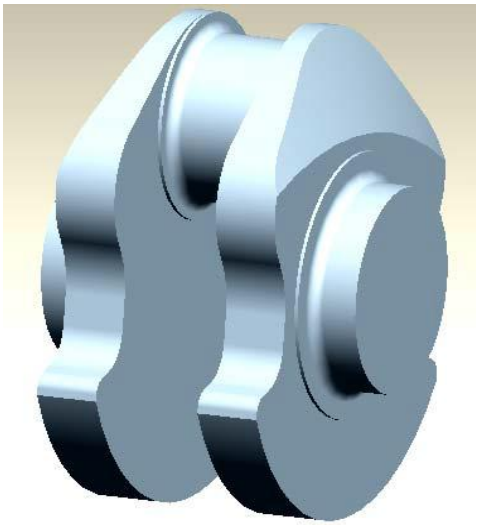

Fig.3 Crankthrow mode in PROE model

The crankshaft deformation has mainly bending deformation of lower frequency.

And also it conclude that maximum deformation appears at the centre of the crankpin neck surface.

It maximum stress appears of the fillets between crank cheeks, crankshaft journal as soon as the nearest point of the centre. It is high stress area.

The resonance vibration of system easily avoid effectively by the structural design .and the results provide a theoretical basis to optimize design and its fating life calculation.

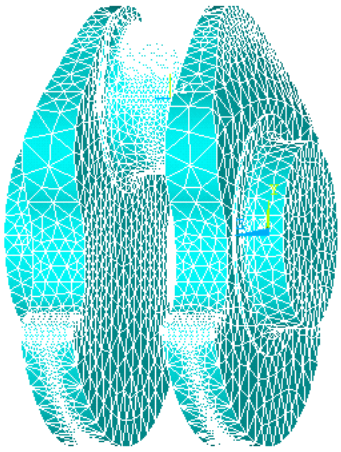

Fig.4 The crankpin finite element mesh in ANSYS

D. "Dynamic Load and stress Analysis of Crankshaft" by Farzin H Montazersadgh and Ali Fathemi

In this paper dynamic simulation of crankshaft of single cylinder four stroke engine was concluded FEA was performed to obtained the magnitude if stress generated at various part of crankshaft . the dynamic analysis was done Analytically and was crosschecked by simulation ADAMS which resulted in load spectrum applied to crankpin ,bearing in further simulation the load was applied to FE model in ABAQUS & boundary condition where applied according to crankshaft mounting conditions. And hence the analysis was done at different engine speed the result obtained from FEA where verified.

E. "Theoretical and Experiment Analysis of Torsional and bending effect on four Cylinders Engine Crankshaft by Using Finite elemt approach" by Prof. R.G. Desavale and A. M.. Patil

The paper aims at identification of stresses in the crankshaft of high speed diesel engine .the author has analyzed it for natural it for natural frequency, rigid body modes shapes by ANSYS and Holzer method. It also focused it on complete simulation of actual boundary condition for journal bearing support, inertia lumping for reciprocation part and bearing stiffness.

The 3D model of crankshaft, flywheel ,pulley where made in Pro-e ,Wildfire meshing of the model was created by tetrahedral solid 45 element in hyper mesh and MPC 184 element was used for Journal bearing .The experimental analysis is carried out using FFT analyzer machine ,then behavior torsional mode, bending mode and combined modes where studied for all system. Dynamic response and critical stresses generated where found at journal bearing and final the final result where obtained in analysis software.

The author result are as follows:-The natural frequencies for above system in Hz are,Table 1

Comparison of results

Mode no.	7	8	9	10	11
Natural frequency by Holzer method	433.2	902.42	993.44	1049.34	1132.12
Natural Frequency by ANSYS	408.87	571.47	937.11	1047.43	1054.34

Above table author found that the ANSYS results are agree with Holzer method.

And experimental result of FFT analyzer used dual channel FFT analyzer 2900 B from Larson & Davis Company Lid.

Fig;5 Experimental setup

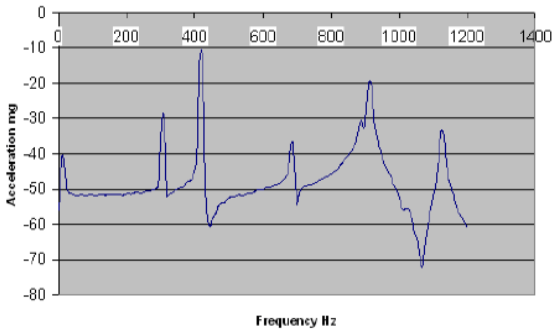

Fig.6 Frequncy vs Magnitude by Normal function

III. Conclusion

1) By studying all these paper it can be concluded that the vibration which affect the crankshaft most is torsional vibration which can be dangerous and can even break the crankshaft.

2) Crankshaft vibration analysis process consist of molding the crankshaft using software like CATIA-V5,CREVO,PROV-E ,Solid Words etc. And ultimately analyzing the model in crankshaft in ANSYS.

All the paper described one thing in common that the area at the center at crankpin is where maximum deformation occur and maximum stress generated, so a serious attention should be given at these particular areas at crankshaft while designing and manufacturing it.

References

[1] BasavarajTalikoti,Dr.S.N.Kurbet, Dr.V.V.Kuppast, Aevind M.Yadwad; "Modal Analysis of a 2-cylinder crankshaft using ANSYS" ,Int.Journal of Engineering Research and application December 2015.

[2] R.B.Sharma & Vikas Sharma;"Dynamic & Static Analyses of Model of Engine Crankshaft" Int.Journal of Engineering research and application ,June2013

[3] Jian Meng, yongqi liu; "Finite Element analysis of diesel engine using finite element method", Published Online August 2011 in MECS..

[4] Ali Fathemi.and Farzin H.Montazersadgh; "Dynamic Load and stress Analysis of crankshaft",SAE International ,2007..

[5] Prof. A.M.Patil and Prof..R.G.Desavale;"Theoretical and Experiment Analysis of Torsional and bending effect on four Cylinders Engine Crankshaft by Using Finite element approach" International Journal of Engineering Research Volume no.2 Issue No.6,pp:379-385, Oct.2013