International Journal of Modern Engineering Research (IJMER)
www.ijmer.com Vol. 2, Issue. 5, Sep.-Oct. 2012 pp-3086-3088 ISSN: 2249-6645

Analysis & Optimization of Crankshaft Using Fem

R. J. Deshbhratar[1], Y. R. Suple[2]
IVth Sem M .Tech Student, Kavikulguru Institute Of Technical And Science, Ramtek
Asst. Prof. Kavikuguru Institute Of Technical & Science, Ramtek

Abstract: *The modal analysis of a 4-cylinder crankshaft is discussed using finite element method in this paper. The analysis is done on two different materials which are based on their composition. Three-dimension models of diesel engine crankshaft was created using Pro/ENGINEER software. The finite element analysis (FEM) software ANSYS was used to analyse the vibration modal of the crankshaft. The maximum stress point and dangerous areas are found by the deformation analysis of crankshaft. The relationship between the frequency and the vibration modal is explained by the modal analysis of crankshaft. The results would provide a valuable theoretical foundation for the optimization and improvement of engine design.*

Keywords: *crankshaft, finite element analysis; optimization; Thermal analysis*

I. Introduction

Crankshaft is one of the most important moving parts in internal combustion engine. It must be strong enough to take the downward force of the power stroked without excessive bending. So the reliability and life of internal combustion engine depend on the strength of the crankshaft largely. And as the engine runs, the power impulses hit the crankshaft in one place and then another. The torsional vibration appears when a power impulse hits a crankpin toward the front of the engine and the power stroke ends. If not controlled, it can break the crankshaft. Strength calculation of crankshaft becomes a key factor to ensure the life of engine. Beam and space frame model were used to calculate the stress of crankshaft usually in the past. But the number of node is limited in these models. With the development of computer, more and more design of crankshaft has been utilized finite element method (FME) to calculate the stress of crankshaft. The application of numerical simulation for the designing crankshaft helped engineers to efficiently improve the process development avoiding the cost and limitations of compiling a database of

real world parts. Finite element analysis allows an inexpensive study of arbitrary combinations of input parameters including design parameters and process conditions to be investigated. Crankshaft is a complicated continuous structure. The vibration performance of crankshaft has important effect to engine. The calculation

II. 3-D entity model of crankshaft

The structure of the crankshaft has more small fillets and fine oil hole. Considering these factors in establishment process, finite element mesh of crankshaft becomes very densely, the number of node equation increase greatly. These factors would extend the solution time, make the unit shape unsatisfactory and amplify the accumulative error. This would lower the simulation accuracy. Hence,

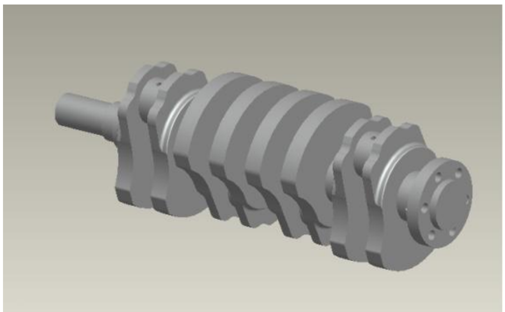
of crankshaft vibration performance is difficult because of the complexity of crankshaft structure, the difficult determinacy of boundary condition. Dynamic matrix method and dynamic substructural method combined with FME were used to calculate the vibration of crankshaft. The method of three-dimensional finite element was carried to analyse dynamical characteristic of diesel crankshaft.
In the paper, 3-D finite element analysis are carried out on the modal analysis of crankshaft and the thermal analysis of crankshaft, And the FME software ANSYS was used to simulate the modal analysis of the crankshaft. The results of natural frequencies and mode shape were obtained. And deformation distributions of crankpin were obtained by using ANSYS software. The results are regarded as a theory basis to optimize the design of crankshaft and thermal analysis of crankshaft.

CRANKSHAFT MODELLING

In the present research, diesel engine crankshaft was studied. The crankshaft has four crankthrows, three rod journals and two main journals, and the mainly dimension parameters are considered while preparing model in Pro/E.According to complicated structure of crankshaft, the integral crankshaft should be applied when performing finite element model analysis. In view of the structure shape characteristics of crankshaft, reducing the solution time and enhancing the simulation accuracy, the crankshaft model was simplified in establishment process. The premise of simplification is that it does not influence the dynamics characteristics of crankshaft. The crankshaft model was created by Pro/ENGINEER software.
model was meshed by 8 unit solid45. the meshing accuracy is 3 grade. After automatic meshing, in order to enhance the simulation accuracy and avoid generating the macrocephalic mesh, the crankshaft dangerous areas were further tessellated. The 3-D crankshaft finite element grid model is shown in Figure. The two materials different materials for crankshaft is used. The physical parameters used in the crankshaft simulation were list in Tables 01 and Table 02.
Then the model was imported to the ANSYS software. According to the structure of crankshaft, the crankshaft diameter less than 12mm were ignored. The model of four-cylinder crankshaft is shown in Fig.

in the paper, the real crankshaft was represented by a simplified model. In this simplified model, the chamfers which radius less than 5mm and the oil holes which

Meshing of 3-D Entity



FINITE ELEMENT METHOD

The finite element method is numerical analysis technique for obtaining approximate solutions to a wide variety of engineering problems. Because of its diversity and flexibility as an analysis tool, it is receiving much attention in engineering schools and industries. In more and more engineering situations today, we find that it is necessary to obtain approximate solutions to problems rather than exact closed form solution. It is not possible to obtain analytical mathematical solutions for many engineering problems. An analytical solutions is a mathematical expression that gives the values of the desired unknown quantity at any location in the body, as consequence it is valid for infinite number of location in the body. For problems involving complex material properties and boundary conditions, the engineer resorts to numerical methods that provide approximate, but acceptable solutions.

The finite element method has become a powerful tool for the numerical solutions of a wide range of engineering problems. It has developed simultaneously with the increasing use of the high-speed electronic digital computers and with the growing emphasis on numerical methods for engineering analysis. This method started as a generalization of the structural idea to some problems of elastic continuum problem, started in terms of different equations or as an extrinum problem

The fundamental areas that have to be learned for working capability of finite element method include:

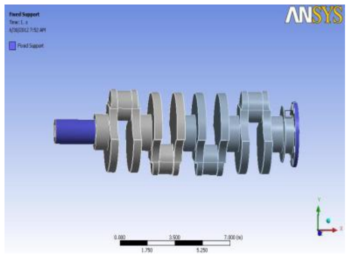
- Matrix algebra.
- Solid mechanics.
- Variation methods.
- Computer skills.

Matrix techniques are definitely most efficient and systematic way to handle algebra of finite element method. Basically matrix algebra provides a scheme by which a large number of equations can be stored and manipulated. Since vast majority of literature on the finite element method treats problems in structural and continuum mechanics, including soil and rock mechanics, the know-ledge of these fields became necessary. It is useful to consider the finite element procedure basically as a Variation approach. This conception has contributed significantly to the convenience of formulating the method and to its generality.

The term “finite element” distinguishes the technique from the use of infinitesimal “differential elements” used in calculus, differential equations. The method is also distinguished from finite difference equations, for which although the steps in to which space is divided into finite elements are finite in size; there is a little freedom in the shapes that the discrete steps can take. F.E.A is a way to deal with structures that are more complex than dealt with analytically using the partial differential equations. F.E.A deals with complex boundaries better than finite difference equations and gives answers to the „real world‟ structural problems.

BOUNDARY CONDITION

The crankshaft bears the constraints of main journals and longitudinal thrust bearing. Because of the effect of load, crankshaft main journals appear bend deformation between the lower main-bearing half and upper main-bearing half. And the longitudinal thrust bearing can prevent effectively the crankshaft axial movement and ensure the piston-and-connecting-rod assembly normally work. Five surface radial symmetry constrains were exerted on the five main journals surface respectively, axial displacement constrains were exerted on the two end face of crankshaft. Then the modal analysis was carried out using the ANSYS software.



CONSTANT PROPERTIES OF BOTH MATERIAL

Table No.01

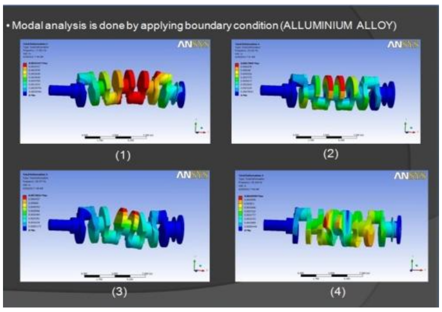
Aluminum Alloy Constant Properties	
Name	**Value**
Compressive Yield Strength	2.8×10^{8} Pa
Density	2,770.0 kg/m³
Poisson's Ratio	0.33
Tensile Yield Strength	2.8×10^{8} Pa
Tensile Ultimate Strength	3.1×10^{8} Pa
Young's Modulus	7.1×10^{10} Pa
Thermal Expansion	2.3×10^{-5} 1/°C
Specific Heat	875.0 J/kg·°C
Relative Permeability	10,000
Resistivity	5.7×10^{-8} Ohm·m

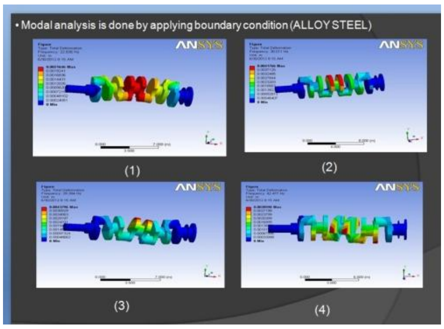
Table No. 02

Alloy Steel properties	**constant**
Name	**Value**
Compressive Yield Strength	2.5×10^{8} Pa
Density	7,850.0 kg/m³
Poisson's Ratio	0.3
Tensile Yield Strength	2.5×10^{8} Pa
Tensile Ultimate Strength	4.6×10^{8} Pa
Young's Modulus	2.0×10^{11} Pa
Thermal Expansion	1.2×10^{-5} 1/°C
Specific Heat	434.0 J/kg·°C
Relative Permeability	10,000.0
Resistivity	1.7×10^{-7} Ohm·m

IV. RESULT AND DISCUSSION

The first four modal frequencies for both the material is calculated from ansys





V. CONCLUSIONS

In this paper, the crankshaft model is created by Pro/ENGINEER software. Then the model created by pro/Engineer was imported to ANSYS software. The maximum deformation appears at the centre of crankshaft surface. The maximum stress appears at the fillets between the crankshaft journal and crank cheeks, and near the central point. journal. The edge of main journal is high stress area. The crankshaft deformation was mainly bending deformation under the lower frequency. And the maximum deformation was located at the link between main bearing journal and crankpin and crank cheeks. So this area prones to appear the bending fatigue crack. Base on the results, we can forecast the possibility of mutual interference between the crankshaft and other parts. The resonance vibration of system can be avoided effectively by Appropriate structure design. The results provide a theoretical basis to optimize the design and fatigue life calculation

ACKNOWLEDGEMENTS

The support extended by the guide (Shri Y.R. Suple) and college authorities is highly appreciated and acknowledged with due respect.

REFERENCES

[1] Amit Solanki, Ketan Tamboli, M.J.Zinjuwadia, 'Crankshaft Design And Optimization- A Review' ,National Conference On Recent Trends In Engineering & Technology.

[2] C.M.Balamurugan, R.Krishnaraj, Dr.M.Sakthivel, K.Kanthavel, Deepan Marudachalam , 'Computer Aided Modelling And Optimization Of Crankshaft' International Journal Of Scientific & Engineering Research Volume 2, Issue 8, August- 2011 Issn 2229-5518.

[3] Vladimir V. Riabov , 'Computational And Analytical Methods In The Engine Crankshaft Design' ,Rivier College, Nashua, New Hampshire 03060, Usa.

[4] Jian Meng, Yongqi Liu, Ruixiang Liu Zibo, Shandong, China 'Finite Element Analysis Of 4-Cylinder Diesel Crankshaft'I.J. Image, Graphics And Signal Processing, 2011, 5, 22-29 Published Online August 2011 In Mecs

[5] ZHOU Xun, YU Xiao-li , 'Reliability Analysis Of Diesel Engine Crankshaft'Based On 2d Stress Strength Interference Model, School Of Mechanical And Energy Engineering, Zhejiang University, Hangzhou 310027, China.

[6] Jenson (1970), ' Dynamic Load Determination And Analysis'

[7] Mehrdad Zoroufi and Ali Fatemi, 'A Literature Review on Durability Evaluation of Crankshafts Including Comparisons of Competing Manufacturing Processes and Cost Analysis', 26th Forging Industry Technical Conference, Chicago, IL, November 2005.