FINITE ELEMENT ANALYSIS OF SINGLE CYLINDER ENGINE CRANK SHAFT

Bhumesh J. Bagde[1], Laukik P. Raut[2]

[1]M.Tech. Student, G.H. Raisoni College of Engineering, Nagpur, India

[2]Guide and Assistant Professor, G.H. Raisoni College of Engineering, Nagpur, India

ABSTRACT

This paper deals with, the problem occurred in single cylinder engine crank shaft. It consist of static structural and fatigue analysis of single cylinder engine crank shaft. It identifies and solves the problem by using the modeling and simulation techniques. The topic was chosen because of increasing interest in higher payloads, lower weight, higher efficiency and shorter load cycles in crankshaft. The main work was to model the crank shaft with dimensions and then simulate the crank shaft for static structural and fatigue analysis. The modeling software used is PRO-E wildfire 4.0 for modeling the crank shaft. The analysis software ANSYS will be used for structural and fatigue analysis of crank shaft for future work. The material for crank shaft is EN9 and other alternate materials on which analysis will be done are SAE 1045, SAE 1137, SAE 3140, and Nickel Cast Iron. The objectives involves modeling and analysis of crank shaft, so as to identify the effect of stresses on crank shaft, to compare various materials and to provide possible solution.

KEYWORDS **-** *Crankshaft, FEM.*

I. INTRODUCTION

Crankshaft is a large component with a complex geometry in the engine, which converts the reciprocating displacement of the piston to a rotary motion with a four link mechanism. Since the crankshaft experiences a large number of load cycles during its service life, fatigue performance and durability of this component has to be considered in the design process. Design developments have always been an important issue in the crankshaft production industry, in order to manufacture a less expensive component with the minimum weight possible and proper fatigue strength and other functional requirements. These improvements result in lighter and smaller engines with better fuel efficiency and higher power output. Crankshaft must be strong enough to take the downward force of the power stroke without excessive bending. So the reliability and life of the 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. This study is conduct on a single cylinder engine crank shaft. The modeling of single cylinder engine crank shaft is done by using PRO-E wildfire 4.0 software. The finite element analysis will be perform on crankshaft in order to optimize the weight and manufacturing cost. The material for crank shaft is EN9. Other alternate materials on which analysis will be done are SAE 1045, SAE 1137, SAE 3140, Nickel Cast Iron.

II. LITERATURE REVIEW

Yingkui and Zhibo [1] established three dimensional model of a diesel engine crankshaft by using Pro E software. Using ANSYS analysis tool, the finite element analysis for the crankshaft was conducted under extreme operation conditions and stress distribution of the crankshaft was presented. The crank stress change model and the crank stress biggest hazard point were found by using finite element analysis, and the improvement method for the crankshaft structure design was given. This shows that the high stress region mainly concentrates in the Knuckles of the crank arm & the main journal, and the crank arm and the connecting rod journal, which is the area most easily broken.

Jian et al. [4] analyzed three dimensional model of 380 diesel engine crankshaft. They used ProE and ANSYS as FEA tools. First of all, the 380 diesel engine entity crankshaft model was created by Pro E software. Next, the model was imported to ANSYS software. Material properties, constraints boundary conditions and mechanical boundary conditions of the 380 diesel engine crankshaft were determined. Finally, the strain and the stress figures of the 380 diesel crankshaft were calculated combined with maximum stress point and dangerous area. This article checked the crankshaft's static strength and fatigue evaluations. That provided theoretical foundation for the optimization and improvement of engine design. The maximum deformation occurs in the end of the second cylinder balance weight.

Gu Yingkui et.al. [3] researched a three-dimensional model of a diesel engine crankshaft was established by using PRO/E software. Using ANSYS analysis tool, it shows that the high stress region mainly concentrates in the knuckles of the crank arm & the main journal and the crank arm & connecting rod journal ,which is the area most easily broken.

Jian Meng et al. [2] analyzed crankshaft model and crank throw were created by Pro/ENGINEER 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.

Xiaorong Zhou et al. [5] described the stress concentration in static analysis of the crankshaft model. The stress concentration is mainly occurred in the fillet of spindle neck and the stress of the crankpin fillet is also relatively large. Based on the stress analysis, calculating the fatigue strength of the crankshaft will be able to achieve the design requirements. From the natural frequencies values, it is known that the chance of crankshaft resonant is unlike. This paper deals with the dynamic analysis of the whole crankshaft. Farzin H. Montazersadgh et al.

III. OBJECTIVES

The problem occurred in the single cylinder engine crank shaft was formation of cracks after certain time period. The objective of the project is to find out the cause of crack generation and provide the possible solution. The objectives involved are:-

a. To model single cylinder engine crank shaft using modeling software Pro-E 4.0
b. Analysis of single cylinder engine crank shaft using Ansys 11.0 software.
c. To identify the area where the possibility of crack generation is maximum and provide the possible solutions.

IV. MODELING OF CRANK SHAFT USING PRO-E WILDFIRE 4.0

Pro-E Wildfire 4.0 has been developed by Parametric Technology Corporation (PTC) of U.S.A. This is CAD/CAM/CAE software but we are using this for only 3-D part modeling (CAD). This CAD includes.

1. Sketcher
2. Part Modeling (part design)
3. Advanced Part Design
4. Surface Design
5. Assembly Design

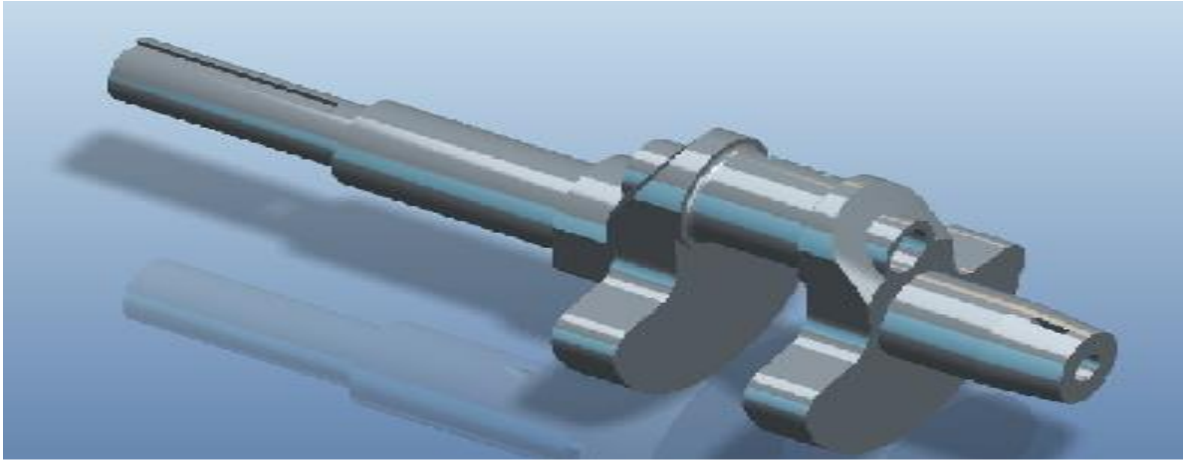

Fig 1: 3-D model of Crankshaft using Pro-E

4.1. Basic Steps in F.E.M.

Basically there six steps in F.E.M. which are as follows.
i) Discretization (mesh generation)
ii) Selecting the displacement function
iii) Develop the element matrices and equations
iv) Assemble element matrices
v) To find the unknowns
vi) Interpretation of the results.

4.2. Need of Finite Element Method

There are number of needs of finite element method. But we are considering some basic needs.
i) To reduce the amount of prototype testing.
ii) To simulate design that is not suitable for prototype testing.
iii) Cost saving.
iv) Time saving.

4.3. Element Properties

The discretization of the structure or body into Finite Elements forms the basic first step in the analysis of a complicated structural system.
Rules for Discretization of the Structure into Elements:

1. Sub-division of a body or structure into Finite Elements should satisfy the following requirements:
2. Two distinct elements can have common points only on their common boundaries if such boundaries exist. No overlapping is allowed. Common boundaries can be points, lines or surfaces.
3. The assembled element should leave no holes within the two elements and approximate the geometry of the real body or structure as closely as possible to do.
4. When the boundary of a structure or body cannot be exactly represented by the elements selected, an error cannot be avoided. Such error is called Geometric Discretization Error and it can be decreased by reducing the size of the elements or by using elements allowing boundaries to become curved.

4.4. Finite Element Analysis of crankshaft using ANSYS 11.0 (workbench): Crank shaft

Model Analysis of Crankshaft will be done using following steps:-

a. Click on new simulation file.
b. Give name as EN9.
c. Import geometry (crank. prt) from folder of Pro-E files.
d. Meshing the model using the option generate mesh.

e. Select new analysis, Select static structural.
f. Apply boundary conditions.
g. Select the area for fixed support.
h. Find the total deformation for each mode by selecting solve option.
i. Find von- mises stress selecting von-mises option.
j. Generate report preview.
k. Save the file.

4.5. Meshed model of crankshaft

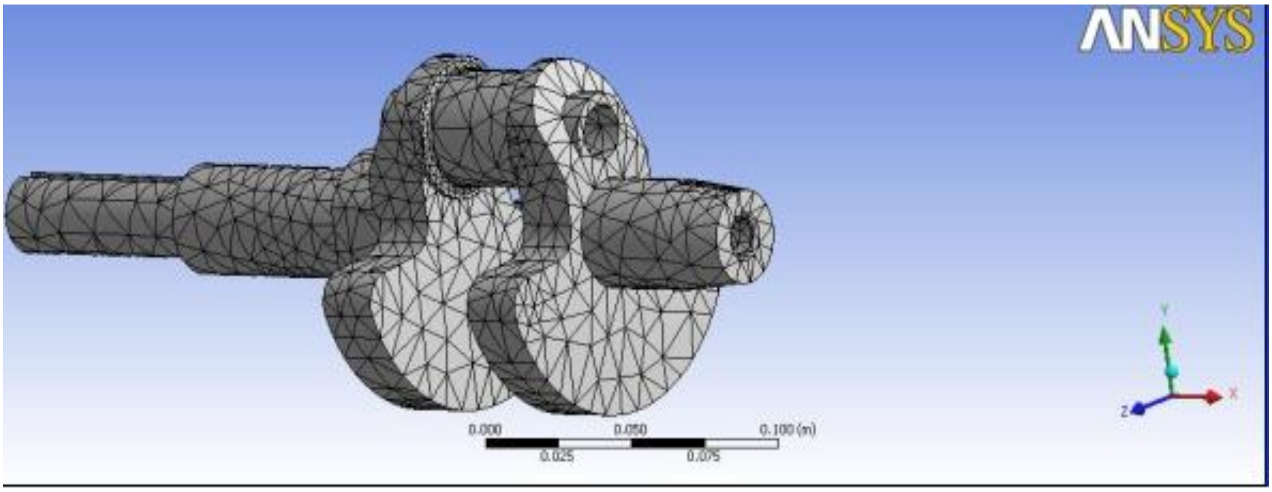

Fig 2: Meshed model of crankshaft

The fig.2 shows the meshed model of crank shaft. The Descretization (Mesh generation) is the first step of Finite Element Method. In this step the component or part is divided into number of small parts. In discretization the no. of nodes formed are 136492 and no of elements are 7339.The effect of force on each portion of the component is not same. The purpose of descretization is to perform the analysis on each small division separately.

4.6. Loading and Boundary Conditions

Crankshaft is a constraint with a ball bearing from one side and with a journal on the other side. The ball bearing is press fit to the crankshaft and does not allow the crankshaft to have any motion other than rotation about its main axis. Since only 180 degrees of the bearing surfaces facing the load direction constraint the motion of the crankshaft, this constraint is defined as a fixed semicircular surface as wide as ball bearing width. The other side of the crankshaft is journal bearing. Therefore this side was modeled as a semicircular edge facing the load at the bottom of the fillet radius fixed in a plane perpendicular to the central axis and free to move along central axis direction. The distribution of load over the connecting rod bearing is uniform pressure on 120 degree of contact area. Since the crankshaft is in interaction with the connecting rod, the same loading distribution will be transmitted to the crankshaft. In this study a pressure of 3.5 MPa is applied at the crankpin at top dead center position of piston.

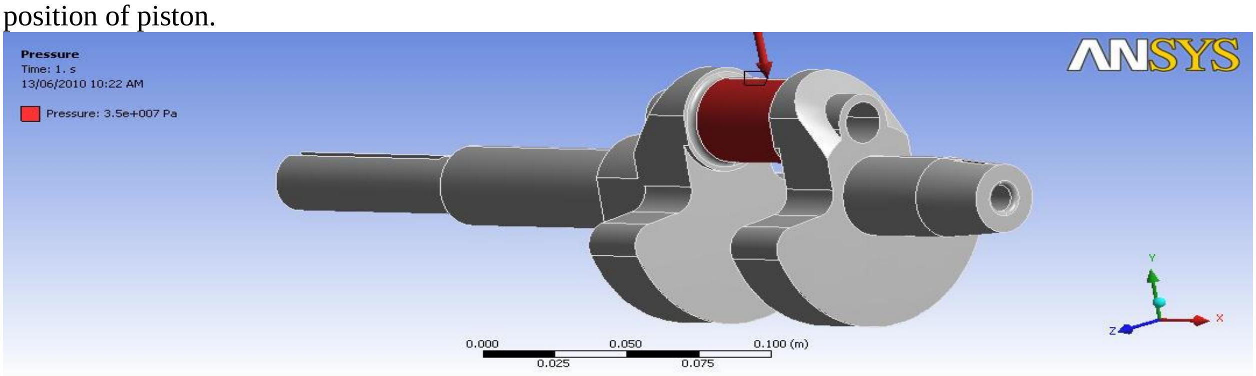

Fig 3: Load applied on crankshaft

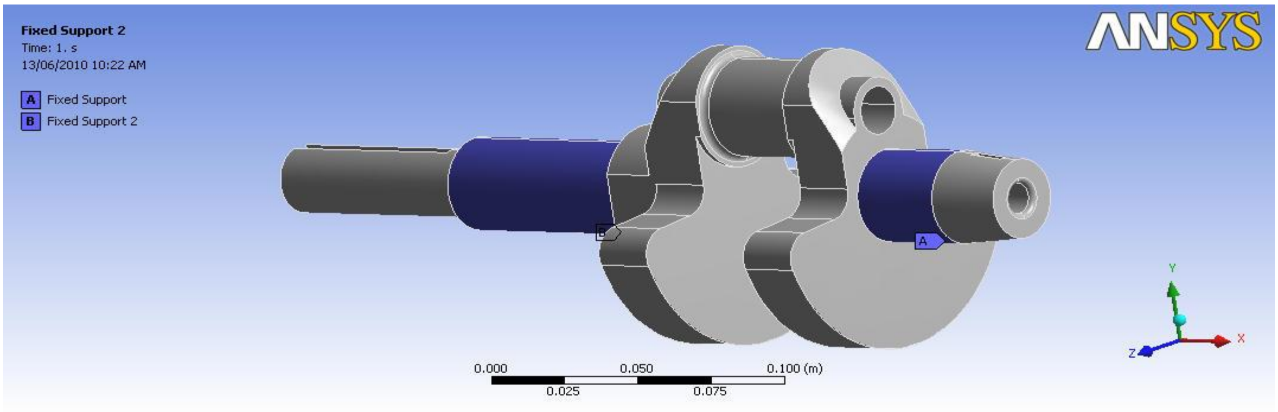

Fig 4: Boundary conditions applied on crankshaft

V. FUTURE WORK

In future static structural analysis and fatigue analysis will be carried out on the crank shaft using the existing material and four alternative materials. This analysis will show the effect of stresses on crank shaft and the portion of crank shaft where chances of crack formation are maximum. Manual calculations using finite element method will be done by considering a bigger portion of crank shaft and the results will be compared with the software results. The comparison of analysis results of all materials will be done.

VI. CONCLUSIONS

In this paper, the crankshaft model was created by Pro-E Wildfire 4.0 software. Then, the model created by Pro-E Wildfire 4.0 was imported to ANSYS software. The analysis of the crank shaft will be done using five different materials. These materials are EN9, SAE 1046, SAE 1137, SAE 3140 & Nickel Cast iron. The comparison of analysis results of all five materials will show the effect of stresses on different materials and this will help to select suitable material. The manual calculations using finite element method considering bigger portion of crank shaft will ensure result validation for software results. The time and efforts required for analysis using software is very less and accuracy is also good. So we can say that FEA is a good tool to reduce time consuming theoretical work.

REFERENCES

[1]. G. Yingkui, Z. Zhibo, (2011), “Strength Analysis of Diesel Engine Crankshaft Based on PRO/E and ANSYS” Third International Conference on Measuring Technology and Mechatronics Automation, pp 362-364.
[2]. Jian Meng., Yongqi Liu., Ruixiang Liu.,2011,“Finite Element Analysis of 4-Cylinder Diesel Crankshaft, ” I.J. Image, Graphics and Signal Processing, 5, 22-29
[3]. Gu Yingkui, Zhou Zhibo.,2011,“Strength Analysis of Diesel Engine Crankshaft Based on PRO/E and ANSYS,” Third International Conference on Measuring Technology and Mechatronics Automation
[4]. M. Jian, L. Yong-qi, L. Rui-xiang, Z. Bin, (2010), “Intension Analysis of 3-D Finite Element Analysis on 380 diesel crankshaft” International Conference on Computational and Information Sciences, pp 1269-1272.
[5]. Xiaorong Zhou., Ganwei Cai., Zhuan Zhang. Zhongqing Cheng., 2009, “Analysis on Dynamic Characteristics of Internal Combustion Engine Crankshaft System,” International Conference on Measuring Technology and Mechatronics Automation
[6]. M.K.Paswan and A.K.Goel,(2008) “Fatigue Testing procedure of 6 Cylinder Diesel Engine Crakkshafts,” ARISER Vol.4 No.3(2008) 144-151.
[7]. S.S.Mohtasebi, H.Afshari and H. Mobli,(2006) “Analysis of Crankshafts Vibrations to compare the Dynamic Behaviour of Steel and Cast Iron Crankshafts.” Journal of Applied Sciences 6 (3): 591-594, 2006.

AUTHORS PROFILE

Bhumesh J. Bagde received his BE degree in Mechanical Engineering from K.D.K College of Engineering Nagpur, Nagpur University, in 2007. Now he is pursuing his M. Tech in CAD/CAM from G.H. Raisoni College of Engineering Nagpur, Nagpur University.