"Finite Element Structural and Fatigue 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 main work was to model the crank shaft with dimensions and then simulate the crank shaft for static structural and fatigue analysis. The topic was chosen because of increasing interest in higher payloads, lower weight, higher efficiency and shorter load cycles in crankshaft. The modeling software used is PRO-E wildfire 4.0 for modeling the crank shaft. The analysis software ANSYS is used for structural and fatigue analysis of crank shaft . The objective 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.

Key Words - Crankshaft, FEM, Analysis.

Introduction

Crankshaft is one of the most important moving parts in internal combustion engine. 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 has been performed on crankshaft in order to optimize the weight and manufacturing cost. The material for crank shaft is EN9. Other alternate materials on which analysis has been done are SAE 1045, SAE 1137, SAE 3140, Nickel Cast Iron.

Literature Review

C.M.Balamurugan et.al [1] conduct Computer aided modeling and optimization analysis of crankshaft is to study was to evaluate and compare the fatigue performance of two competing manufacturing technologies for automotive crankshafts, namely forged steel and ductile cast iron. In this study a dynamic simulation was conducted on two crankshafts, cast iron and forged steel, from similar single cylinder four stroke en-gines. Finite element analyses was performed to obtain the variation of stress magnitude at critical locations. The dynamic analysis was done analytically and was verified by simulations in ANSYS. Results achieved from aforementioned analysis were used in optimization of the forged steel crankshaft. Geometry, material and manufacturing processes were optimized considering different constraints, manufacturing feasibility and cost. The optimization process included geometry changes compatible with the current engine, fillet rolling and result in in-creased fatigue strength and reduced cost of the crankshaft, without changing connecting rod and engine block.

Jian et al. [7] 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.

Yingkui and Zhibo [4] 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.

Xiaorong Zhou et al. [6] 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.

Gu Yingkui et al. [5] 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. [3] 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.

Farzin H. Montazersadgh et al.[2] In this study a dynamic simulation was conducted on a forged steel crankshaft from a single cylinder four stroke engine. Finite element analysis was performed to obtain the variation of the stress magnitude at critical locations.The dynamic analysis resulted in the development of the load spectrum applied to the crankpin bearing. This load was then applied to the FE model and boundary conditions were applied according to the engine mounting conditions. Results obtained from the aforementioned analysis were then used in optimization of the forged steel crankshaft. Geometry, material, and manufacturing processes were optimized using different geometric constraints, manufacturing feasibility, and cost. The first step in the optimization process was weight reduction of the component considering dynamic loading. This required the stress range under dynamic loading not to exceed the magnitude of the stress range in the original crankshaft. Possible weight reduction options and their combinations were considered. The optimization and weight reduction were considered in an interactive manner and evaluated by manufacturing feasibility and cost. The optimization process resulted in an 18% weight reduction, increased fatigue strength,and a reduced cost of the crankshaft.

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.

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

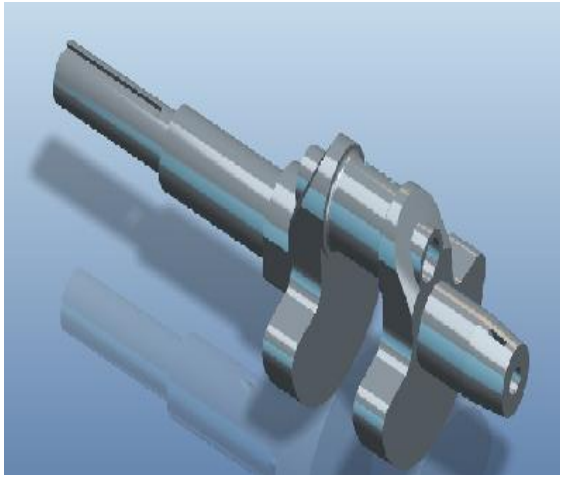

Fig 1: 3-D model of Crankshaft using Pro-E

Meshed model of crankshaft.

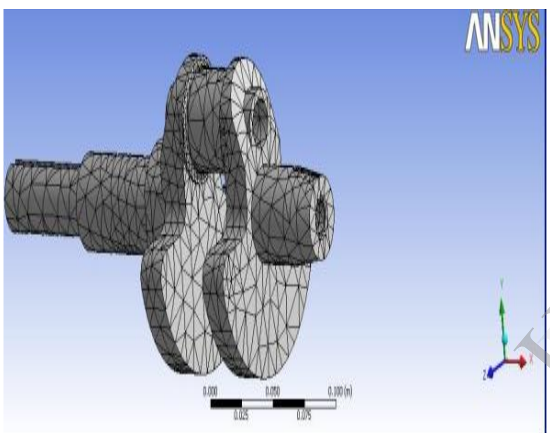

Fig 2: Meshed model of crankshaft

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.

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.

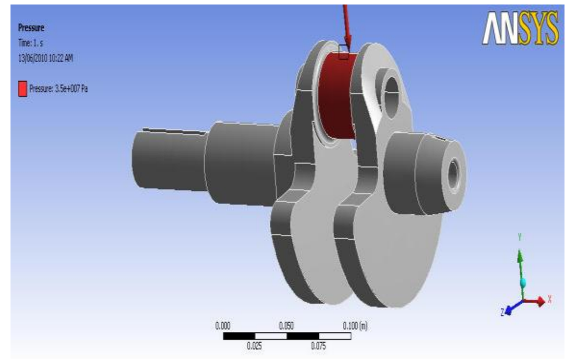

Fig 3: Load applied on crankshaft

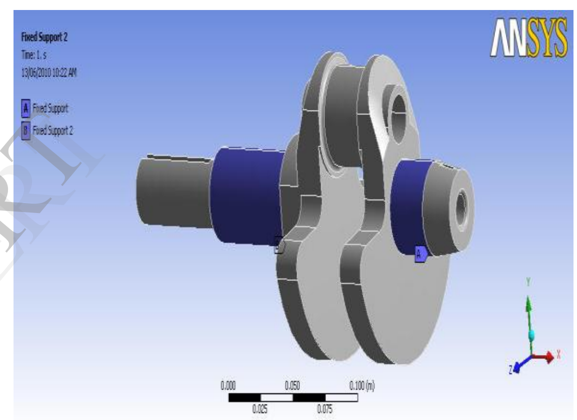

Fig 4: Boundary conditions applied on crankshaft

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 35 MPa is applied at the crankpin at top dead center position of piston.

Structural analysis of crank shaft

After the application of boundary conditions and force, the next step is to perform the structural analysis of crank shaft. In this structural analysis, we are mainly concern with the total deformation and the stresses acting on the crank shaft (von-mises stresses). When the force is applied, the slight deformation and also the stresses takes place in the crank shaft.

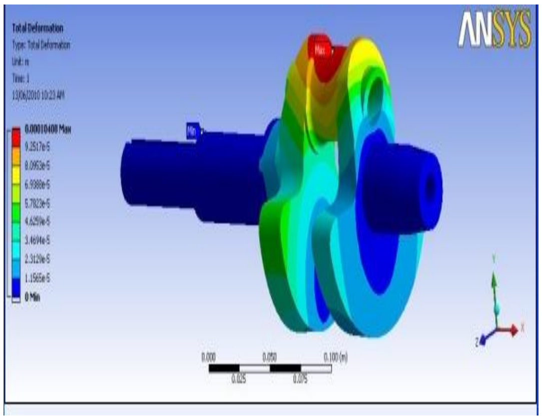

Fig 5: Structural Analysis of crank shaft (Deformation)

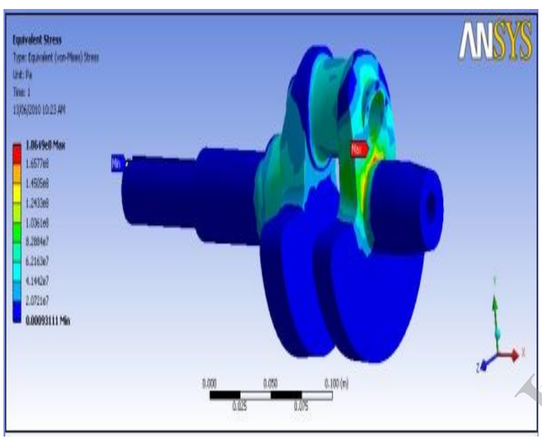

Fig 6: Structural Analysis of crank shaft

(Stresses)

The total deformation of crank shaft is shown in fig.5. The deformation in the crank shaft is not same throughout. The portion in red colour shows that the deformation at that region is maximum and the portion in blue colour shows that the deformation is minimum in that region. The maximum displacement is 0.00022695 m. The stresses acting on the crank shaft is shown in fig.6. The maximum stresses acting on the crank shaft is indicated by the red colour. The maximum stress (von-Mises) is 13.57×10^8 Pa as shown above.

Fatigue Analysis of Crank Shaft using ANSYS Software.

Fatigue failure is the failure occurs when the load is cyclic or repetitive. The fatigue analysis is important in the case of crank shaft. The fig.5.8 shows the fatigue analysis performed on the crank shaft. The figure below shows the probable life of the crank shaft. The red colour portion shows that the life of the crank shaft is minimum at that region and the blue colour portion shows that the fatigue life of the component is maximum at that region. The portion shown by the red colour shows that the fatigue life of the component is minimum and it is the portion where the chances of crack formation are maximum.

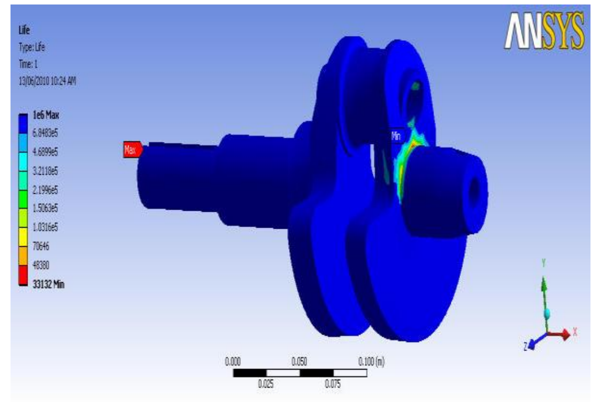

Fig 6: Fatigue Analysis of crank shaft

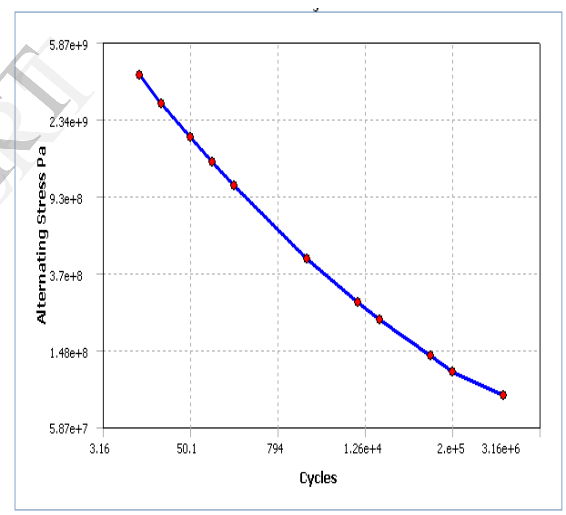

Fig 7 : Alternating stress parameters of crank shaft.

Alternating stress parameters of crank shaft : As the number of cycles per revolutions goes on increasing alternating stress increases proportionally as shown in graph. The maximum alternating stress is between 58.7×10^9 pa. to 23.4×10^9 Pa. Maximum number of cycles is 3.16×10^6.

Results of Total Deformation and Equivalent Stresses

Sr No	Material	Total Deformation		Equivalent Stresses		Fatigue Life
		Max	Min	Max	Min	Min
1	EN-9	6.3662 e-5m	0	1.864e +8pa	9.522 e-4pa	3.3132e4
2	SAE - 1045	6.473e -5m	0	1.976e +8pa	1.193 e-3pa	3.1036e4
3	SAE- 1137	6.174e -5m	0	1.243e +8pa	1.110 e-3pa	3.3132e4
4	SAE- 3140	6.451e -5m	0	1.863e +8pa	1.112 e-3pa	3.1132e4
5	NI – CI	1.040e -4m	0	1.864 9e+8p a	9.311 e-4pa	3.0132e4

For single cylinder engine crank shaft, Static structural analysis & Fatigue analysis performed on crank shaft using finite element analysis. The crank shaft material is EN9 and other alternative materials also considered for analysis which are SAE 1045, SAE1137, SAE 3140 & Nickel Cast Iron. The results obtained are tabulated in table above. The total deformation in all crank shaft materials found to be 0.0006 mm to 0.0010 mm. The von-mises stress in all crank shaft materials found to be 1.863 x 10^8 Pa. to 1.864 x 10^8 Pa. By performing the analysis of crank shaft it has been found that the stresses induced in the crank shaft material is minimum for SAE 1137 material of crank shaft.

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 is done using five different materials. Static Structural Analysis and fatigue analysis of crank shaft was performed on ANSYS software and the deformation and stresses were compared. Analysis has been performed on existing material of Crank shaft and four alternate materials also considered for crank shaft. Analysis shows the critical portion where stress acting are maximum and the chances of crack formation are maximum. The stresses induced is minimum for SAE 1137 material of crank shaft as compare to other materials. The fatigue life life of materials EN9 and SAE 1137 is better as compare to other materials.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]. C.M.Balamurugan et.al,(2011) "Computer Aided Modeling and Optimization of Crankshaft." International Journal of Scientific and Engineering Reseach Volume 2, Issue 8, August-2011, ISSN 2229-5518.

[2]. Farzin H. Montazersadgh et al.2008, "Optimization of a Forged Steel Crankshaft Subject to Dynamic Loading" Paper 2008-01-0432.

[3]. 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

[4]. 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.

[5]. 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

[6]. 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

[7]. 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.

[8]. 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.