Review on Failure Analysis of Single Cylinder Diesel Engines Crankshaft

Mr. Patil S.P.[1], Mr. Pimpale R.S.[2], Mr.Avhad R. V.[3], Mr.Shirsat B. S.[4], Prof. Jawale P.P.[5]
[1, 2,3,4,5] Department of Mechanical Engineering
[1, 2,3,4,5] Sinhgad Institute of Technology, Lonavala

Abstract- *The crankshaft is an important and large component of an engine. The function of crankshaft is to convert reciprocating displacement of the piston into a rotary motion. The objective of this study is to review failure analysis of single cylinder diesel engine crankshaft to determine critical location in the crankshaft. The modeling of the crankshaft is created by using 3-D modeling software's like CREO & Pro-E etc. Finite element analysis (FEA) is performed to obtain the variation of the stress at critical location of the crankshaft using ANSYS software. The analysis of crankshaft is based on according to the engine condition, boundary condition and design specification of crankshaft. It has been observed that high level stresses are appears on critical areas like web fillets therefore crankpin web fillet region is the critical region where fatigue failure of crankshaft occurs.*

I. INTRODUCTION

The crankshaft is an important and large component of an engine. The function of crankshaft is to convert reciprocating displacement of the piston into a rotary motion. Crankshafts are made from materials which can be readily shaped, machined, and heat-treated. Traditionally, these shafts were forged, which gives all the necessary properties. Evolution of the nodular cast irons and improvements in foundry techniques, cast crankshaft are used for moderate loads. Only for heavy-duty applications do forged shafts used.

The crankshaft also drives the camshaft and some other elements via a chain or a belt drive system. Crankshaft is one of the critical components of an IC engine. Failure of crankshaft makes engine useless and repair cost is also high. It has complicated geometry and during operation experiences complex loading pattern. In IC engines, the transient load of cylinder gas pressure which is dynamic in nature with respect to magnitude and direction transmitted to crankshaft through connecting rod. However, the piston along with connecting rod and crankshaft makes respective reciprocating and rotating system of components. The dynamic load and rotating system exerts repeated bending and shear stress due to torsion. Which are common stresses acting on crankshaft and mostly responsible for crankshaft fatigue failure. Hence, fatigue strength plays an important role in crankshaft development and its parts considering its safety and reliable operation. This paper is a study of failure analysis of single cylinder diesel engine crankshaft.

In this presents study detail overview of failure analysis process including theoretical methods and result integration for predicting life of components as compared to life estimation by means any suitable software. In this paper efforts have been made to present various cost effective reliable analytical, numerical and experimental techniques developed by various researchers for fatigue failure analysis of crankshaft. In this paper the effect of various parameters like fatigue failure, life cycle of crankshaft, bearing subjected to vibration for different loading condition have been reviewed.

II. LITERATURE REVIEW

Bhumesh J. et al. Identifies and solves the problem by using the modeling and simulation techniques. Model was created by Pro-E Wildfire 4.0 software. Then the model was imported to ANSYS software. The analysis of the crankshaft is done using five different materials. Static Structural Analysis and fatigue analysis of crankshaft was performed on ANSYS software and the deformation and stresses were compared. Analysis has been performed on existing material of Crankshaft and four alternate materials also considered for crank shaft. Analysis describes the critical portion where stresses acting are maximum and the chances of crack formation are maximum. The stresses induced are minimum for SAE 1137 material of crank shaft as compare to other materials. The fatigue 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 they say that FEA is a good tool to reduce time consuming theoretical work.

Jaimin Brahmbhatt have been analyzed crankshaft model was created by Solid works 2009 software. Then, the model created by Solid works was imported to ANSYS software. After that FEA Results Conformal matches with the theoretical calculation so they can say that FEA is a good tool to reduce time consuming theoretical Work. The maximum deformation appears at the center of crankpin neck 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 Value of Von-Misses Stresses that comes out from the analysis is far less than material yield stress so their design is safe and they should go for optimization to reduce the material and cost. After Performing Static Analysis they performed Dynamic analysis of the crankshaft which results are more realistic whereas static analysis provides an overestimate results. Accurate stresses and deformation are critical input to fatigue analysis and optimization of the crankshaft. After Analysis Results, they can Say that Dynamic FEA is a good tool to reduce Costly experimental work.

V.Vijayakumar and T.Gopalkrishnan created the crankshaft model by ABAQUS software. They conformal match the FEA Results with the theoretical calculations. They can say that FEA is a good tool to reduce time consuming theoretical Work. The maximum deformation appears at the center of crankpin neck 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 value of Von-Misses stresses that comes out from the analysis is far less than material yield stress so design is safe and go for optimization to reduce the material and cost.

K. Thriveni and Dr.B.Jaya Chandraiah has been analyzed crankshaft model and crank was created by CATIA-V5 software and finite element analysis is performed by ANSYS software. The result show that the maximum deformation appears at the centre of the crankpin neck surface. The maximum stress appears at the fillet areas between the crankshaft journal and crank cheeks and near the central point journal. The value of von-misses stresses that comes out from the analysis is far less than material yield stress so our design is safe.

K. Thriveni and Dr.B.JayaChandraiah have been analyzed crankshaft model and crank were created by CATIA-V5 software and finite element analysis performed by ANSYS. There are generally two categories for the vibrations the free frequency vibrations and frequency vibrations, free vibrations occur when the system is under the action of oscillating systems and their inherent forces external forces there are controversial. In free frequency case there is no boundary conditions are applied in the crankshaft. In natural free-frequency the crankshaft should not be vibrating but some period of time vibrations are occurred because self-weight of the crankshaft. The frequency occurred in 7th node. This frequency is known as resonance frequency. In free frequency case the resonance frequency is 1150.967Hz at 7th mode. When the engine was running at the high speed the driving frequency is merely 100Hz. As the lowest natural frequency is far higher than driving frequency, possibility of resonance is rare. Infrequency case the minimum frequency occurred at 1st mode is890.735Hz, the maximum frequency occurred at 10th node is5539.023Hz.

Amit solanki and Jaydeepsinh Dodiya in his paper conducted simulation on a crankshaft from a single cylinder 4-stroke diesel engine. Finite element analysis (FEA) is performed to obtain the variation of stress magnitude at critical locations of crankshaft. Simulation inputs are taken from the engine specification chart. Model was created by Pro-E Software. Then, the model created by Pro-E was imported to ANSYS software. FEA Results Conformal matches with the theoretical calculation so they say that FEA is a good tool to reduce time consuming theoretical Work. The maximum deformation appears at the center of crankpin neck 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 Value of Von-Misses Stresses that comes out from the analysis is far less than material yield stress so design is safe and goes for optimization to reduce the material and cost. After Performing Static Analysis they performed Dynamic analysis of the crankshaft which results are more realistic whereas static analysis provides an overestimate results. Accurate stresses and deformation are critical input to fatigue analysis and optimization of the crankshaft. After Analysis Results, they can Say that Dynamic FEA is a good tool to reduce Costly experimental work.

Rinkle Garg &Sunil Baghla. conducted static analysis on a cast iron crankshaft from a single cylinder four stroke engine. Finite element analysis was performed to obtain the variation of the stress magnitude at critical locations. Three dimensional model of the crankshaft was created in Pro-E software. The load was then applied to the FE model and boundary conditions were applied as per the mounting conditions of the engine in the ANSYS. Results obtained from the analysis were then used in optimization of the cast iron crankshaft. This requires the stress range not to exceed the magnitude of the stress range in the original crankshaft. The optimization process included geometry changes without changing connecting rod and engine block.

III. CASE STUDY

Crankshafts are the components which experiences complex dynamic loadings due to rotating, bending and torsion on main journals and bending on crankpins. The aim of this work is to determine critical regions where crankshafts frequently fails. In this case study a dynamic simulation is conducted on a crankshaft from a single cylinder 4- stroke diesel engine. This analysis is done for finding critical location in crankshaft, stress variation over the engine cycle and the effect of torsion and bending load. Von-misses stress is calculated using theoretically and FEA software ANSYS. The relationship between the frequency and the vibration modal is explained by the modal and harmonic analysis of crankshaft using FEA software ANSYS.Fig-1 shows the crankshaft model in ANSYS and Fig-2 shows meshed model of crankshaft. Tetrahedrons element is used for meshing.

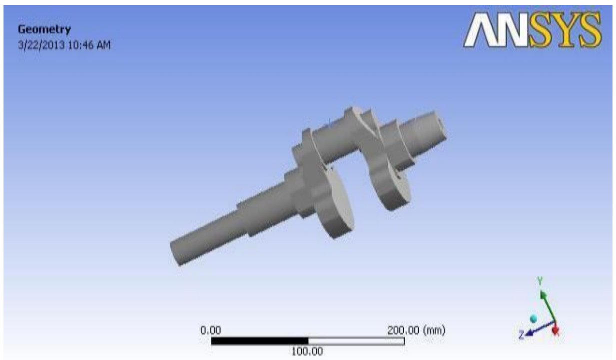

Fig-1 :-Crank Shaft in Ansys

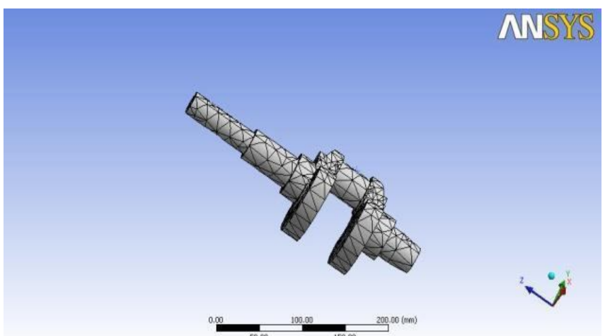

Fig-2 :- Meshed model

Define boundary condition for analysis, after applying the Load, Run the analysis. Von misses stresses are calculated and as shown in Fig-3. Applying the maximum force at phase angle 3550, corresponding Deformation is measured as shown in Fig-4

Fig:-3: Von misses stress

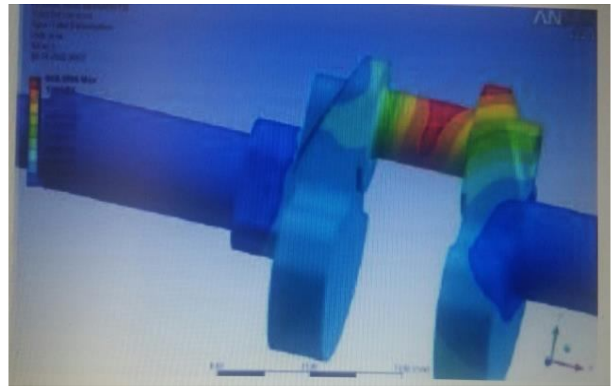

Fig:-4 Deformation angle at 3550

Result Table: 1- FEA results

Sr no	Type of stresses	Theoretical	FEA analysis
1	Von-misses stresses (N/mm^2)	112.25	110.3
2	Shear stresses (N/mm^2)	50.15	59.89

Above Results Shows that FEA Results matches with the theoretical calculation. From fig-4,we can say that the maximum deformation appears at the centre of crankpin neck 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 Value of Von- Misses Stresses that comes out from the analysis is less than material yield stress, so our design is safe and we should go for optimization to reduce the material and cost.

IV. CONCLUSION

Crankshaft experiences a large number of load cycles during its service life, fatigue performance and durability of the component has to be considered in the design process. It has been observed that by making the changes in materials, life of crankshaft can be increased. The maximum deformation appears at the center of crankpin neck 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. FEA Results Conformal matches with the theoretical calculation so FEA is a good tool to reduce time consuming theoretical Work and also reduce costly experimental work.

REFECRENCES

[1] Bhumesh J. Bagde, Laukik P. Raut, “Finite Element Structural and Fatigue Analysis of Single Cylinder Engine

Crankshaft”; International Journal of Engineering Research & Technology (IJERT),Vol. 2, (2013), pp:981-986.

[2] Jaimin Brahmbhatt, Prof. Abhishek choubey, "Design and Analysis of Crankshaft for Single Cylinder 4-Stroke Diesel Engine"; International Journal of Advanced Engineering Research and Studies, Vol.I, (2012) , pp:88-90.

[3] V.Vijayakumar and T.Gopalkrishnan, "design and contact analysis of crankshaft using abaqus"; international journal of research in aeronautical and mechanical engineering, vol.2, 2014. pp: 57-63.

[4] K. Thriveni, Dr. B.Jaya Chandraiah, "Modal Analysis of A Single Cylinder 4-Stroke Engine Crankshaft", International Journal of Scientific and Research Publications, Volume 3, (2013)

[5] K.Thriveni1 Dr.B.Jaya Chandraiah, "Modeling and Analysis of the Crankshaft Using Ansys Software", International Journal of Computational Engineering Research, Vol. 03, pp: 84-89.

[6] Amit Solanki, Jaydeepsinh Dodiya, "Design and Stress Analysis of Crankshaft for Single Cylinder 4-Stroke Diesel Engine"; international journal for research in applied science and engineering technology (IJRASET),Vol-2, (2014), pp:320

[7] Rinkle Garg & Sunil Baghla “Finite element analysis and optimization of crankshaft design”; International Journal of Engineering and Management Research,vol-2,(2012),pp:26-31