ISSN XXXX XXXX © 2017 IJESC

Research Article Volume 7 Issue No.6

Review on Design Optimization of Engine Crankshaft

Surekha .B. Varpe[1], Ravindra .S. Shelke[2]
Student[1], Associate Professor[2]
Department of Mechanical Engineering
SVIT Engineering College, Chincholi, Savitribai Phule Pune University, India

Abstract:
This work present the study of Optimization of a single cylinder four stroke Engine crankshaft. Finite Element Analysis (FEA) Method to obtain stress variation at critical location. Thus the strength of Crankshaft is to be important parameter to be taken into account while designing. The 3D model is prepared by using Pro-E Creo pre-processing and post-processing is prepared by using Ansys workbench. The load and boundary condition were applied in Ansys as per crankshaft mounting on engine. This process helps in finding the optimized design for the crankshaft in which it has the best performance without any failure and with minimum loads acting on the crankshaft. After implementing optimization, weight of the crankshaft will be reduced. FEA also be carried out on optimized design of the crankshaft to check whether the optimized design is safe or not. FE Analysis has done using model and Experimental tests are done to validate the CAE results.

Keywords: Crankshaft, CAD, FEA, Optimization

I. INTRODUCTION

Crankshaft is one of the most vital complex components in IC engine which converts the reciprocating displacement of the piston to a rotary motion. It required strong enough to take the downward force during power stroked without excessive bending. So the life and reliability of IC engine depend on the strength of the crankshaft. If not controlled, it can break the crankshaft. Crankshaft Strength calculation becomes a key factor to ensure the life of engine. Since the crankshaft subjected to a large number of load cycles during its service life, durability and fatigue performance of this component has to be considered in the design process. These improvements result in lighter and smaller engines with better fuel efficiency and higher power output. The crankpin is like beam with distribute load along with its length that varies crank position subjected bending and twisting.
1. Bending causes tensile and compressive stresses.
2. Twisting causes shear stress.
The main objective of to optimize and finding out effective design of Crankshaft with relevance cost by reducing raw material cost without affecting function The problem under consideration will be modeled through four approaches:

A. CAD Modeling
B. Finite Element Meshing
C. Boundary Conditions
D. Finite Element Analysis
E. Optimization
F. Experimentation

II. LITERATURE REVIEW

JAIMIN BRAHMBHATT, PROF. ABHISHEK CHOUBEY-
In this paper a dynamic simulation is conducted on a crankshaft from a single cylinder 4- stroke diesel engine. Stress variation over the engine cycle and the effect of torsion load in the analysis are investigated.

AMIT SOLANKI, JAYDEEPSINH DODIYA- In this paper a static simulation is conducted 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.

SUJATA SATISH SHENKAR, PROF. NAGRAJ BIRADAR-
The stress analyses of a single-cylinder crankshaft are discussed using finite element method in this paper. The results would provide a valuable theoretical foundation for the optimization and improvement of engine design.

AMIT SOLANKI. , KETAN TAMBOLI. AND M.J.ZINJUWADIA-
The design of the crankshaft considers the dynamic loading and the optimization can lead to a shaft diameter satisfying the requirements of automobile specifications with cost and size effectiveness.

III. FINITE ELEMENT ANALYSIS

Structural Analysis and optimization using finite element method are key parts of the Design & Development of the crankshaft. Here we use Ansys workbench for pre-processing and post-processing of finite element analysis.

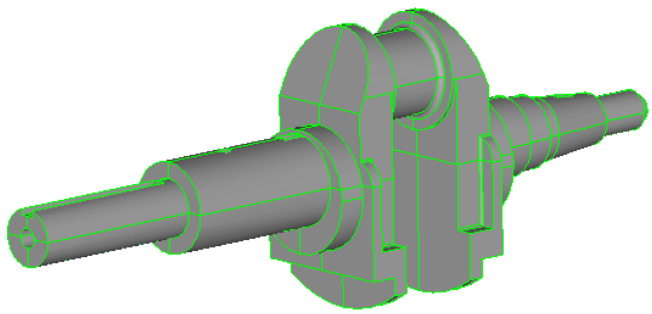

Figure.1. CAD Model

Here CAD geometry of crankshaft as shown in fig 1 is imported as an input in <.stp> form into Ansys.

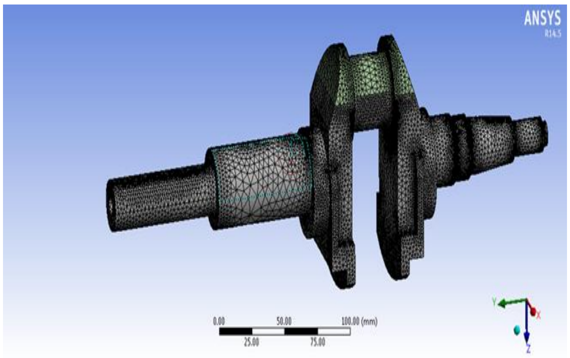

Figure.2. FEA Model

After importing the geometry, modify the geometry data and prepare it for meshing. Any continuous body has infinite degree of freedom hence solving in this form would not be possible. Finite element method performs meshing or discretization to reduces the infinite degree of freedom to finite degree of freedom.

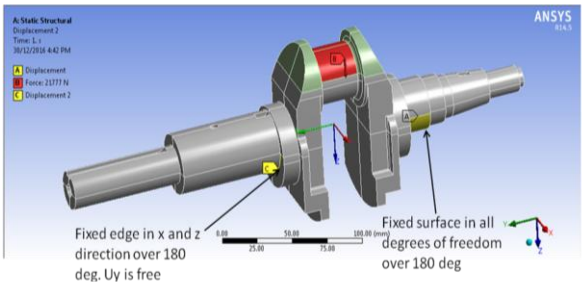

Figure.3. Loading and boundary conditions

a) Loading Condition:

- To check the strength of crankshaft while bending..

- Compressive force on crankshaft 21776.8N is applied over crank pin.

b) Boundary Condition:

-Crankshaft one end fix radially and free along the axis and other end is fixed at all direction.

-Material Steel AISI 1045 is to be used.

Young's modulus = E= 2.1×10-5 MPa

Poisson ratio = μ =0.26

Density of steel = ρ = 7.85×10-9 kg/mm3

Yeild strength=625MPa

Ultimate strength=827MPa

IV. RESULTS AND DISCUSSION

All machine component analysis, a component must be designed such that the stresses occurring during operation will not exceed material limits.. Analysis has to conclude whether the component is safe or fail comparing the max stress value with yield or ultimate stress.

1.1 Static Analysis Results

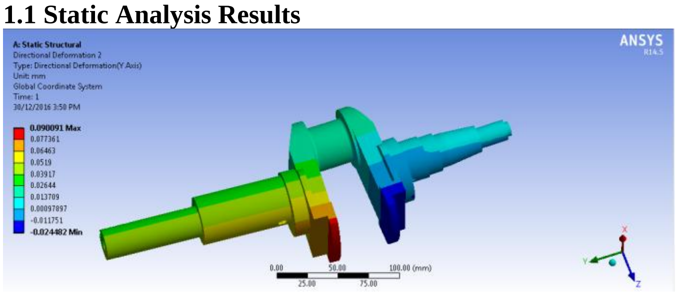

Figure.4. Displacement Contour plot

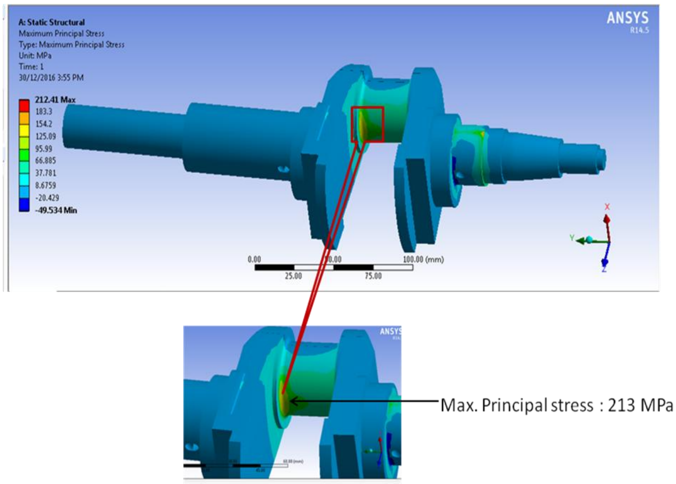

Figure.5. Contour Plot for Max Principal Stress

Results:-

1. Maximum Deflection = 0.09 mm.

2. Max Principal Stress Stress observed = 213MPa.

From Fig. 5 the encircled region shows the high stress region. Here the maximum stress is observed which is less than the yield strength of the material. So design is safe against load.

V. OPTIMIZATION

Optimization is a process of converging onto a final solution amongst a number of possible options, such that a certain requirements are best satisfied. The objective of the optimization problem is often some sort of maximization or minimization, for example minimization of stress or maximization of stiffness. The purpose of this optimization problem is to minimize the weight of crankshaft without exceeding the allowable stress.

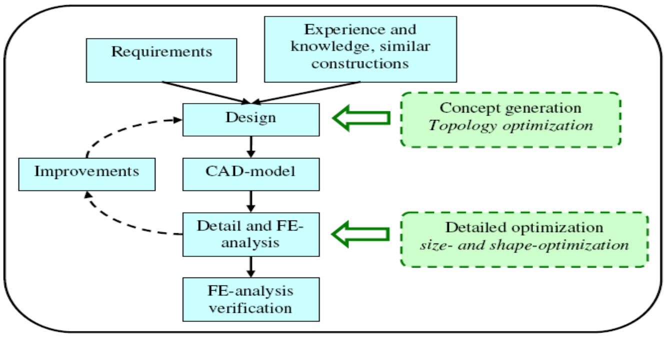

Figure.6. Overview of the design process

The factor of safety observed for given loading condition is more than 1.5 (specified limit for rotating components) thus the structure is safe and existing design of crankshaft is possible to optimize.

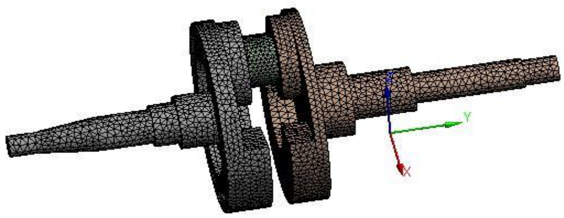

Figure.7. FEM Model

The Optimized CAD model was subjected to same loading conditions, boundary conditions and material properties as

mentioned in linear static analysis and dynamic modal analysis of existing model.
Frictional contact is used to define contact between all components.

A. *FEA Results*

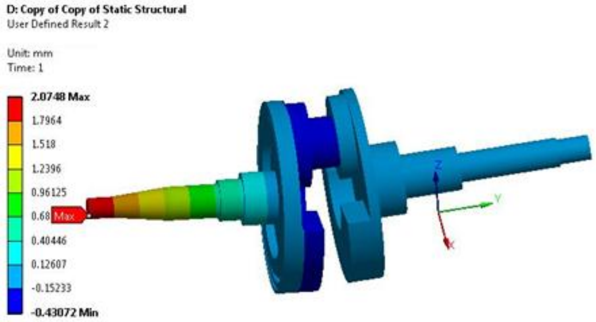

Figure.8. Displacement Contour plot

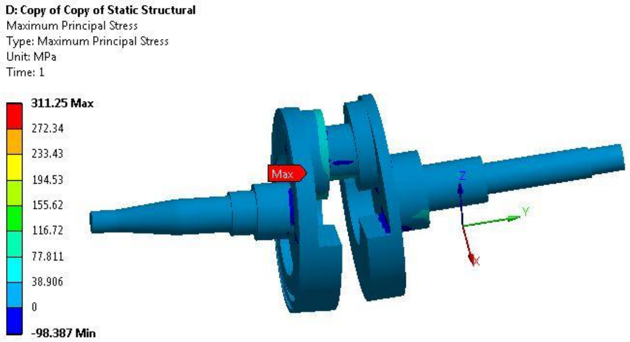

Figure.9. Contour Plot for Max Principal Stress

Results:-
1. Maximum Deflection = 2.07 mm.
2. Max Principal Stress Stress observed = 311.25MPa.
From Fig. 9 the encircled region shows the high stress region. Here the maximum stress is observed which is less than the yield strength of the material. So design is safe against load.

VI. CONCLUSIONS

From the analytical and experimental investigation on crankshaft. The manufacturing of crankshaft using AISI1045 steel material was successful. However it is strong enough to fulfill the loading conditions. The investigation has shown that 20% of weight reduction is achieved. If we compare the experimental test value which is displacement 2.3mm and our analytical displacement is 2.07mm. Hence we can conclude that optimum amount of material is used to manufacture the crankshaft no any excess material is used. The factor of safety observed for given loading condition is more than 1.5 thus the structure is safe.

VII. REFERENCES

[1].Jaimin Brahmbhatt, Prof. Abhishek choubey "Design And Analysis Of Crankshaft For Single cylinder 4-Stroke Deisel Engine." International Journal of Advanced Engineering Research and Studies.

[2].Amit Solanki, Jaydeepsinh Dodiya. "Design and Stress Analysis of Crankshaft for Single

[3].Sujata Satish Shenkar, Nagraj Biradar. "Design and Static Structural Analysis of Crank Shaft". International Journal of scientific Engineering and Technology research.

[4].P.N.Rohini, S.Nagu. "International Journal Of Engineering Sciences & Research Technology."

[5].G. Thirunavukkarasu, Anantharaman Sriraman. "Design Optimization and Analysis of Crankshaft for Light Commercial Vehicle." Indian Journal of Science and Technology.

[6].Amit Solanki, Ketan Tamboli, M.J.Zinjuwadia. "Crankshaft Design and Optimization." National Conference on Recent Trends in Engineering & Technology.

[7].Nitin S Gokhale. "Practical Finite Element Analysis." (2008)