The International Journal Of Engineering And Science (IJES)
|| Volume || 4 || Issue || 7 || Pages || PP.53-55 || July - 2015 ||
ISSN (e): 2319 – 1813 ISSN (p): 2319 – 1805

Finite Element Analysis and Optimization of Diesel Engine Connecting Rod

[1,] Mr. Ajinkya.A. Bhilare,[2,]Akash. R. Suryavanshi.
[1,] *P.G. Research Scholar in Mechanical Engineering, Dnyanganga College of Engineering & Research, Savitribai Phule Pune University, India*
[2,] *Associate professor in Mechanical Engineering, Dnyanganga College of Engineering &Research, Savitribai Phule Pune University, India*

ABSTRACT

The review report presents a procedure for weight optimization process through Ansys softwareusing finite element analysis.The automobile engine connecting rod is a high volume production component. It undergoes high cyclic loads of order which range 108 to 109 cycles from high compressive loads due to combustion, to high tensile loads due to inertia. It connects reciprocating piston to rotating crankshaft, transmitting the thrust of the piston to the crankshaft. Every vehicle that uses an internal combustion engine requires at least one connecting rod depending upon the number of cylinders. Connecting rods for automotive applications were typically manufactured by various manufacturing process like by forging from either wrought steel or powdered metal. They could also be cast, but castings could have blown-hole-free and better rods gives them an advantage over cast rods. But between the forging processes, powder forged or drop forged, each process has its own pros and cons. So in order to reduce the material cost and thus production cost it is better to optimize the weight or volume.This project entitled weight optimization process through Ansys software. In this with the help of generate code through Ansys API, we run the test to achieve the optimized weight condition of diesel engine connecting rod. It achieve saving in material, weight and cost for manufacturing diesel engine connecting rod.

KEYWORDS *-Ansys, Connecting Rod, Cost, Optimize, Weight.*

Date of Submission: 26-June-2015 Date of Accepted: 15-July-2015

I. INTRODUCTION

Connecting rod is the intermediate link between the piston and the crank. And is responsible to transmit the push and pull from the piston pin to crank pin, thus converting the reciprocating motion of the piston to rotary motion of the crank. The connecting rod is subjected to a complex state of loading. Therefore, durability of this component is of critical importance. Due to these factors, many research work is always going on the connecting rod and it is always area of research for different domains such as production technology, materials, performance simulation, fatigue and many more. Connecting rod of automotive applications should be lighter and lighter, should consume less fuel and at the same time they should provide comfort and safety to passengers, time they should provide comfort and safety to passengers that unfortunately leads to increase in weight of the vehicle. As we want the connecting rod with less weight, we done optimization work. In that, we modelled the connecting rod of diesel engine with ANSYS software by parametric modelling and thus we performed finite element analysis of connecting rod under static compressive load and static tensile load by giving standard boundary conditions. We get the structural behavior of connecting rod under the static load conditions. After the static finite element analysis of connecting rod we moved towards optimize the connecting rod process. On the basis of study of static finite element analysis of connecting rod we concluded that there is much scope of weight reduction in existing connecting rod.As weight is the main concern factor for any design issues. We try for optimized it and there is structural improvement. Every designer and manufacture always want the lighter and strong connecting rod. Connecting rod with less weight is always best option for further engine assembly. Therefore after static finite element analysis, optimization of connecting rod for weight is carried out under specified limits. We get the connecting rod with optimized weight. That is our requirement.

II. METHODOLOGY.

Here we use method for optimization of connecting rod. First step is the model creation. Themodel is created with command line interface with the use of ANSYS software. We have performed command line interface to model the connecting rod. Various dimensional parameters are fed into the program, thus this makes the parametric approach under command line interface, and thus programming is called PARAMETRIC MODELING. One of the main scope of parametric program modelling is that once the basic program for the modelling is generated we can model the connecting rod of various sizes by just changing the values of the parameters in program.Meshing, load and constraints are also applied via command line interface. Now main step of optimization is also carried with command line program.Program for optimization has been prepared to follow loops or iterations. ANSYS software carry out each loop and finds out the weight of connecting rod and accordingly it changes the dimensional parameters within the specified limits. ANSYS continues subsequent loop until the optimized weight is obtained and it takes care that dimensional parameters do not go beyond specified limits and maximum allowable stress.

III. LITERATURE REVIEW.

Prof. Vivek C. Pathade [1] proposed the major stress induced in the connecting rod was a combination of axial and bending stresses in operation. The axial stresses were produced due to cylinder gas pressure (compressive only) and the inertia force arising in account of reciprocating action (both tensile as well as compressive), whereas bending stresses were caused due to the centrifugal effects. The result of which was, the maximum stresses were developed at the fillet section of the big and the small end. Hence, the paper deals with the stress analysis of connecting rod by Finite Element Method using Pro/E Wildfire 4.0 and FEA WORKBENCH 11.0 software. The comparison and verification of the results obtained in FEA was done experimentally by the method of Photo elasticity (Optical Method). The method of Photo elasticity includes the casting of Photo elastic sheet using Resin AY103 and Hardener HY951, preparation of the model from Photo elastic sheet calibration of the sheet to determine material fringe value**Error! Reference source not found.**.

Kuldeep B et.alproposed the material of connecting rod was replaced by aluminum based composite material reinforced with silicon carbide and fly ash. And they also performed the modelling and analysis of connecting rod. FEA analysis was carried out by considering two materials. The parameters like von misses stress, von misses strain and displacement were obtained from FEA software. Compared to the former material the new material found to have less weight and better stiffness. It resulted in reduction of 43.48% of weight, with 75% reduction in displacement [2].

Zheng Bin Liu Yongqi et.al analyzed stress distribution, safety factor and fatigue life cycle of connecting rod by using 3D finite element method. The results show that the exposed destructive position was the transition location of small end and connecting rod shank at maximum compression condition. Maximum stress was 303MPa. Safety factor was 1.24. At maximum stretch condition, the exposed destructive position was I-shaped cross-section at big end. Maximum stress was 118MPa. Safety factor was 3.19. And structure of connecting rod was improved. Safety factor and fatigue life cycle of connecting rod increases. After structural improvement, maximum stress decreases and both safety factor and fatigue life cycle increases [3].

Mr. Pranav G Charkha et.al performed load analysis on connecting rod. Their study were deals with two subjects, first, static load stress analysis of the connecting rod, and second, optimization for weight. They performed finite element analysis on single cylinder four stroke petrol engine. Structural systems of connecting rod could be easily analyzed using Finite Element techniques. Optimization was performed to reduce weight. They found that weight can further more reduced by changing the material of the forged steel connecting rod to crack able forged steel (C-70). The optimized geometry was 20% lighter than the current connecting rod. Current connecting rods could be replaced by fracture split able steel forged connecting rods. These connecting rods were lighter in weight than existing connecting rod, with similar or better fatigue behavior [4].

Om Prakashet.al found the existing design performs by modelling and evaluates critical regions in the connecting rod under fatigue loading. The main objective of their work was to re-optimize the existing design of connecting rod of universal tractor (U650) by changing some of the design variables. Optimization of connecting rod was done under same boundary and loading conditions for variation in the few stress and fatigue parameters i.e. stresses, weight, life, damage and safety factor. The allowable numbers of cycles under fully reversed fatigue loading were increased and assumed up to a maximum limit. Stress concentration coefficient was varied to obtain the maximum cycles condition. The critical regions under both static and fatigue analysis were identified and improved. The connecting rod was then modelled and optimized for the reduction in weight [5].

Zhou Qinghui et.al they obtain the vibration characteristics and vibration frequency distributions, structural characteristics of the connecting rod mechanism using modal analysis. Then they prepare a physical model of connecting rod mechanism using CAD software. Then finite element analysis and simulation of the model is taken by Hyperworks and MSC. Nastran software's. Then its flexible multi-body dynamic model was established by ADAMS/View and the fatigue stress of connecting rod under the max combustion pressure and Inertia force condition was calculated using the durability Module. The stress was mainly produced on the joint of connecting rod shell and the bottom end or the top end. The simulation result showed that the stability of the mechanism was well. The simulation analysis was really an economical and efficient method to study [6].

Shenoy, P et.al performed the optimization study on a steel forged connecting rod. They have the aim to improve weight and production cost related parameters. As we know weight of the connecting rod has little influence on its total production cost. They also change the material of connecting rod and got the result in terms of reduction in machining operations.They performed the weight optimization under a cyclic load comprising dynamic tensile load and static compressive load as the two extreme loads.Constraints of fatigue strength, static strength, buckling resistance and manufacturability were also imposed. Then they found that the fatigue strength was the most significant factor in the optimization of the connecting rodthe study results in an optimized connecting rod that is 10% lighter and 25% less expensive, as compared to the existing connecting rod.[6].

Dr. K. B. Roy analyzed various designs of connecting rodand finally he select an optimal design for Finite Element Analysis.He used ANSYS-12.0 Workbench and CATIA V5R19 and got various results and then he compared with the existing results. In existing design parameters, he done slight and careful variation. That result in good design which can be made feasible by a number of analysis using CAE tools and Software's. He also studied and analyzed the Stress, Strain, Deformation, Life, and Damage, Biaxiality Indication etc. parametersto get the good design parameters with taking into account the safe permissible stresses and factors which would have affect the design if not taken into account [6].

IV. CONCLUSION

The Optimization carried out in analysis gives a deep insight by considering optimum parameter for suggestion of modification in the existing connecting rod. Change in the design and material results in significant effect on weight. As it achieves the objective of reducing the weight of the engine component, thus reducing inertia loads, reducing engine weight and improving engine performance and fuel economy. Fatigue strength was the most significant factor in the design and optimization of the connecting rod. The shank region of the connecting rod offered the greatest potential for weight reduction.

REFERENCES

[1]. Prof. Vivek C. Pathade ,Dr. Dilip S. Ingole (2013) 'Stress Analysis of I.C. Engine Connecting Rod by FEM and Photo elasticity' IOSR Journal of Mechanical and Civil Engineering, PP 117-125.

[2]. Kuldeep B, Arun L.R, Mohammed Faheem (2013) 'Analysis andOptimization of Using ALFASic Composites'. International Journal of Research in Science, Engineering and Technology.

[3]. Zheng Bin Liu Yongqi, JiLixia(2010) 'Finite Element Analysis and Structural Improvement of Diesel Engine Connecting Rod'. Second International Conference on Computer Modeling and Simulation.

[4]. Mr. Pranav G Charkha, Dr Santosh B Jaju (2009) 'Analysis & Optimization of Connecting Rod' Second International Conference on Emerging Trends in Engineering and Technology.

[5]. Om Prakash et al.(2013) 'Optimizing the Design of Connecting Rod under Static and Fatigue Loading. International Journal of Research in Management, Science and Technology.

[6]. Zhou Q, Wang Y, J. Wei (2010), 'The Finite Element Analysis of Connecting Rod of Diesel Engine', International Conference on Measuring Technology and Mechatronics Automation, IEEE.PP 870-873.

[7]. P S Shenoy and A Fatemi (2005) 'Connecting rod optimization for weight and cost reduction', SAE Publications, Paper No 2005-01-987.

[8]. DR.B.K.Roy (2012) 'Design Analysis and Optimization of Various Parameters of Connecting Rod using CAE Software'. International Journal of New Innovations in Engineering and Technology.

[9]. Deng Banglin et al (2011), 'Fatigue and Bearing Analysis of An Engine Connecting Rod Based on Multi-body Dynamics' Third International Conference on Measuring Technology and Mechatronics Automation.

[10]. E MusangoMunyao et al (2014) 'Simulation of Thermal-Mechanical Strength for Marine Engine Piston Using FEA' Int. Journal of Engineering Research and Applications. PP319-323

[11].