Analysis and Optimization of Connecting Rod by using FEM

Jaysukh N. Likhiya[1] Prof. Kevin M. Vyas[2]
[1]Student of M.E [2]Professor
[1,2]Department of Mechanical Engineering
[1,2]Nobel Group of Institution-Junagadh, Gujarat, India

Abstract— In the automobile industry connecting rod has high volume production. And connecting rod has a complex geometry also. Two major problems that arise in last many years are pollution and fuel crisis. This two problems can be reduce by reduce the weight of the moving part of IC engine. Among the various moving part of IC engine here choose connecting rod for research and reduce its by weight optimization technique. After the study of connecting rod conclude that mostly connecting rod fails at the piston end eyes. So we can modify the design of piston end by design optimization and by this we can reduce the failure rate of connecting rode. To improve the efficiency of IC engine we can reduce the weight of connecting rod by various optimization technique like topology optimization, material optimization etc. Material used for make the connecting rod in current is mostly forged steel. Because forged steel has high compressive strength. It can withstand against the high compressive stress that can produce by gas pressure in IC engine. But forged steel has high density. So weight of the forged steel also high. So if we find another material that has high compressive strength but less density than we can improve the efficiency of engine. Modeling of existing connecting rod is done by Pro E Creo 5.0 software. This model used for analysis of rod also. Static analysis is done by using ANSYS 14.5 software. Theoretical calculation of stress generated in connecting rod is done by using ranking formula. And after this comparisation of both result do. Both results are nearly same.

Key words: connecting rod, ANSYS, Pro E Creo

I. INTRODUCTION

Last two or three decade in the world two major problems arise. One is fuel crises and second is pollution. Fuel consumption and pollution are the most important drivers of the industry in the last twelve years and will continue to be in the next decade. The global warming problem, air pollution in large cities and the price of fuels are key points that leads us to development more efficient engines. To improve the efficiency of engine reduces the mass of the moving parts of internal combustion engines. By doing this, the engine consume less fuel (energy) to run and that means that the engine has efficiency and has less emission also.

IC engine has various reciprocating and rotating parts. Like piston, crank shaft and connecting rod etc. The piston and the crank are linked by connecting rod. It converts reciprocating motion of the piston into rotary motion of crankshaft. Each engine needs at least one connecting rod so volume of production of it is also high. It also subjected under high dynamic loading. So this component is a best selection for research work.

II. LITERATURE REVIEW

As the various part of IC engine the connecting rod is also subjected to a complex state of loading. Connecting rods are highly dynamically loaded components used for power transmission in combustion engines. It undergoes high cyclic loads of the order of 108 to 109 cycles, which range from high compressive load due to gas pressure, to high tensile loads due to inertia. Therefore, durability of connecting rod is of critical importance. Due to these factors, the connecting rod has been the topic of research for different aspects such as manufacturing processes, materials, performance simulation, fatigue, etc. For the current study, it was necessary to investigate finite element modeling techniques, optimization techniques, developments in production technology, new materials, fatigue modeling, and manufacturing cost analysis. This brief literature survey reviews some of these aspects.

K.Karthick and John Panner Selvam [2014] [1] present the design and analysis of connecting rod. The existing connecting rod is manufactured using carbon steel. The model of connecting rod is carried out using pro E software and analysis is carried out using ansys 14 software. Finite element analysis of connecting rod is done using forged steel. The parameters like von mises stress, strain, deformation and weight reduction were done in ansys software. Forged steel has increased stiffness, reduced weight and reduce stress and stiffer than other material.

They conclude that

1) ANSYS Equivalent stress for the both the materials are same.
2) The weight of the forged steel material is less than the existing carbon steel.
3) And also no. of cycles for forged steel is more than the existing connecting rod.
4) When compared to both materials, forged steel is cheaper than the existing connecting rod materials.

R A Savanoor, Abhishek Patil , Rakesh Patil and Amit Rodagi [2014] [2] evaluated connecting rods are manufactured using carbon steel and in recent days aluminum alloys are finding its application in connecting rod. In this work we are comparing the von mises stress and total deformation of 2 different aluminum alloys with the forged steel. FEA analysis was carried out by considering three materials. The parameters like von misses stress and displacement were obtained from ANSYS software. Then compared the aluminum alloys with the forged steel. Then Al5083 alloy found to have less weight. It resulted in reduction of 63.19% of weight. From the above analysis they can conclude that

1) Weight of the connecting rod can be reduced by repacing currently using forged steel in kirloskar engine by aluminium5083 alloy.
2) The section modulus of the connecting rod should be high enough to prevent high bending stresses due to inertia forces, eccentricities, as well as crankshaft and case wall deformations.

All rights reserved by *www.ijsrd.com*

3) Comparison is also made between the three materials w.r.t. tensile stresses and al5083 alloy found least stresses.

Vivek C. Pathade and Dr. Dilip S. Ingole [2013] [3] performed the stress analysis of connecting rod by Finite Element Method using Pro/E Wildfire 4.0 and ANSYS WORKBENCH 11.0 software. The comparison and verification of the results obtained in FEA is 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 Hardner HY951, preparation of the model from Photo elastic sheet calibration of the sheet to determine material fringe value.

From the theoretical, Finite Element Analysis and Photo elastic Analysis they conclude that

1) The stresses induced in the small end of the connecting rod are greater than the stresses induced at the big end.
2) Therefore, the chances of failure of the connecting rod may be at fillet section of both ends.

M. Ravichandran [2013] [4] presents the design connecting rod of internal combustion engine using the topology optimization. The objectives his work are to develop structural modeling, finite element analyze and the optimization of the connecting rod for robust design. The structure of connecting rod was modeled utilized SOLIDWORKS software. Finite element modeling and analysis were performed using MSC/PATRAN and MSC/NASTRAN software. Linear static analysis was carried out to obtain the stress/strain state results. The mesh convergence analysis was performed to select the best mesh for the analysis. The topology optimization technique is used to achieve the objectives of optimization which is to reduce the weight of the connecting rod. From the FEA analysis results, TET10 predicted higher maximum stress than TET4 and maximum principal stress captured the maximum stress. The crank end is suggested to be redesign based on the topology optimization results. The optimized connecting rod is 11.7% lighter and predicted low maximum stress compare to initial design. For future research, the optimization should cover on material optimization to increase the strength of the connecting rod.

At the end of research he obtained In order to compare the equivalent stresses due to the challenge at traction of the connecting rod’s foot, obtained through the two methods. Using the classic method, the maximum values of the equivalent stresses in the outer and inner fibers were obtained, considering that the embedded section corresponds to the 1300 angle. It can be seen that the equivalent stresses obtained with the finite element analysis in the interest zones, have approached values to those calculated with the classic method. Concomitantly, it can be observed that in the interest zones, the stress values are maximum, fact which confirms the theory.

Anil kumar [5] worked to optimize weight and reduce inertia forces on the existing connecting rod, which is obtained by changing such design variables in the existing connecting rod design. The model was developed in Pro/E wildfire 5.0 and then imported as parasolid (XT) form in ANSYS workbench. In this work finite element analysis of the single cylinder four stroke petrol engine connecting rod is considered as case study. The Von Mises stress, strain and total deformation determined for the same loading conditions and compared with the existing results. Based on the observation of static FEA and the load analysis result, the load for the optimization study was selected same as on existing connecting rod. The current work consists of static structural analysis. The static analysis was carried out under axial and buckling load. The model is also selected for fatigue analysis to determine the fatigue strength.

By the experiment he observed that

1) The stress was found maximum at the piston end. This can be reduced by increasing the material near the piston end.
2) Fatigue strength plays the most significant role (design driving factor) in the optimization of this connecting rod.
3) Optimization was performed to reduce weight of the existing connecting rod. This optimization can also be achieved by changing the current forged steel connecting rod into some other materials such as C-70 steel etc.[5]

B.K.Roy [6] suggests various designs of connecting rod have been analyzed in this report and finally an optimal design has been selected for Finite Element Analysis. Using ANSYS-12.0 Workbench and CATIA V5R19, Various results are found out and compared with the existing results. It has been found out that the study presented here has came up with better results as well as safe design of connecting rod under permissible limits of various parameters and safe stresses.[6].

A.V.Akhare, Y. B. Dupare and A. S. Wagh [2013] [7] presents on case study of stress analysis of connecting rod is carried out. In this paper, finite element analysis of single cylinder four stroke petrol engines is taken as a case study. The work has to find out the stresses at various points on the connecting rod and the portion, which is more susceptible to failure and optimization of connecting rod. To evaluate the magnitude and location stresses in the existing connecting rod. This is of great interest to the auto manufactures that which is the portion of the connecting rod which mainly fails so that they can use various methods of hardening the specific area by using special hardening treatments. In response to an increasing demand for fuel-economy, more weight reduction techniques have been proposed to create an optimum connecting rod design. The research aims to maximize weight savings in a connecting rod, without sacrificing the structural performances such as bending strength, buckling strength, and torsion stiffness.

The following conclusions can be drawnfrom this study:

1) Optimization was performed to reduce weight. Weight can be reduced by changing the material of the current forged steel connecting rod to crackable forged steel (C-70).
2) The parameter consideration for optimization are its 20 % reduction in weight of connecting rod, while reducing the weight, the static strength, fatigue strength, and the buckling load factor were taken into account.
3) The optimized geometry is 20% lighter than the current connecting rod. PM connecting rods can be replaced by fracture splitable steel forged

All rights reserved by *www.ijsrd.com*

connecting rods with an expected weight reduction of about higher than existing connecting rod, with similar or better fatigue behaviour.

4) By using other facture crackable materials such as micro-alloyed steels having higher yield strength and endurance limit, the weight at the piston pin end and the crank end can be further reduced. Weight reduction in the shank region is, however, limited by manufacturing constraints.[7]

Changyou Li, Changshuai Qiao, Yimin Zhang and Song Guo [2014] [8] worked the locomotive engine and derived that high reliability of a locomotive is important for the railway transportation. This high reliability was guaranteed by use of a large safety factor resulting in cost increase of railway transportation generally. To overcome this high cost without sacrificing reliability, this work focuses on an optimization design method for the design of the connecting rod used in locomotive traction equipment. The reliability model is formulated based on the stress–intensity distribution interference theory and the reliability of the original connecting rod was estimated using advanced first order and second moment method. Then, the reliability–sensitivity is analyzed. The results show that the reliability of the connecting rod used in China is almost equal to one and the reliability robustness is high. To minimize the quality of the connecting rod under the condition of ensuring high reliability and reliability robustness, the reliability optimization models are proposed for three cross sections. The optimization results show that the quality of the optimized connecting rod could be reduced to less than 40% of the original without sacrificing reliability and reliability robustness.

They derived that the reliability analysis, sensitivity and optimization of the connecting rod were presented in this work. The results show that the reliability and the reliability robustness of the connecting rods used in the Chinese locomotive are very well and these are consistent with the fact. Using the proposed reliability optimization method, the quality of the connecting rod under consideration was reduced to less than 40% of the original value under the condition that the reliability and reliability robustness were not decreased significantly.

Pravardhan S. Shenoy [2005] [9] performed an optimization study on a steel forged connecting rod with a consideration for improvement in weight and production cost. Since the weight of the connecting rod has little influence on its total production cost, the cost and the weight were dealt with separately. Reduction in machining operations, achieved by change in material, was a significant factor in manufacturing cost reduction. Weight reduction was achieved by using an iterative procedure. Literature survey suggests cyclic loads comprised of static tensile and compressive loads are often used for design and optimization of connecting rods. However, in this study weight optimization is performed 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. The fatigue strength was the most significant factor in the optimization of the connecting rod. An estimate of the cost savings is also made. The study results in an optimized connecting rod that is 10% lighter and 25% less expensive, as compared to the existing connecting rod.

At the end of research the following conclusions can be drawn from the results of this study:

1) Fatigue strength was the most significant factor (i.e. design driving factor) in the design and optimization of the connecting rod.
2) Stresses and displacements were observed to be significantly lower under conditions of assembly (with bearings, crankshaft and piston pin and bushing), when compared to stresses obtained from unassembled connecting rod subjected to cosine loading.
3) The optimized geometry is 10% lighter than the current connecting rod for the same fatigue strength, in spite of lower yield strength and endurance limit of C-70 steel compared to the existing forged steel.
4) Reduction in machining operations achieved by using C-70 steel and utilization of the fracture splitting process reduces the production cost by about 25%. As compared with a PM connecting rod, the cost saving is estimated to be about 15%.
5) By using other facture crackable materials such as micro-alloyed steels having higher yield strength and endurance limit, it may be possible to further reduce the weight at the piston pin end and the crank end. Weight reduction in the shank region is, however, limited by manufacturing constraints

Daniel Gaspari Cirne de Toledo [2009] [10]presents the design of a connecting rod for a lightweight spark-ignition four-stroke internal combustion engine by applying topology optimization. Topology optimization methodology combines FE analysis with a powerful optimization algorithm to find the optimum mass distribution inside the defined design volume concerning the loads and boundary conditions and considering a specified optimization objective function (e.g., minimum compliance, minimum mass, maximum first mode frequency) and constraints. This approach innovates the design process in the mechanical industry while changes the project information and decision flow, because the design is defined by the CAE engineer (supported by an optimization algorithm) and no longer by the CAD designer. The project includes the simulation of the combustion to predict the static load (gas pressure) that is applied to the connecting rod. With this it is possible to determine the loads that are applied to the connecting rod.

This paper concludes that the optimization problem defined as "case B2" lead to the best result. The mass reduction is 72%, the same of "case A". But, the von Mises stress is 9% lower and maximum total displacement is 10% lower. The comparison of the results of "case B1" and "case B2" shows how important is defining the right volume fraction constraint. Defining the volume fraction for "case B2" by the solution of "case A" optimization problem leads to a better result of the connecting rod design. This approach needs more computing and time, because two optimizations problem need to be solved. In the other hand it assures that the volume fraction definition is well defined for the connecting rod optimization considering the load steps, material and manufacturing constraints. Indirectly this

All rights reserved by *www.ijsrd.com*

approach considers both mechanical requirements of the connecting rod that are stress and stiffness.

III. Conclusion of Entire Literature Review

After studying above research paper and thesis following conclusions derived

1) The maximum stress occurs in the connecting rod near the piston end due to thrust of the piston.
2) At present material used for making connecting rod is mostly forged steel.
3) In the optimization of connecting rod we must reduce the weight of rod, reduce deflection of connecting rod and increase comparative and tensile strength of materials of connecting rod.
4) To optimize the connecting we can use various optimization techniques like topology optimization, design optimization and material optimization.
5) In topology optimization shape of connecting rod change and that way we can reduce the weight of connecting rod. In design optimization design of connecting rod change.
6) In material optimization material of connecting rod change. Properties of material which is used for making of connecting rod are less density, high stiffness for reduce deflation, high tensile and compressive strength for sustain high gas pressure and high thermal sustainable.
7) For material optimization aluminum alloy and C70 steel is best for replace exciting material forged steel.

References

[1] K.Karthick and John Panner Selvam, “Static Analysis of Connecting Rod Using Forged Steel,” IJESRT, ISSN: 2277-9655.

[2] R A Savanoor, Abhishek Patil, Rakesh Patil and Amit Rodagi, “FINITE ELEMENT ANALYSIS OF IC ENGINECONNECTING ROD BY ANSYS,” IJMERR, ISSN 2278 – 014,Vol. 3, No. 3, July 2014.

[3] Prof. Vivek C. Pathade , Dr. Dilip S. Ingole, “Stress Analysis of I.C.Engine Connecting Rod by FEM and Photoelasticity,” IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE) e-ISSN: 2278-1684 Volume 6, Issue 1 (Mar. - Apr. 2013), PP 117-125.

[4] M. Ravichandran, R. Anbazhagan, Binu. K. Soloman and C. Jagadeesh Vikram, “DESIGN AND ANALYSIS OF CONNECTING ROD BY USING CFRP MATERIAL,” International Journal of Automobile Engineering, Research and Development (IJAuERD),ISSN 2277-4785,Vol. 3, Issue 2, Jun 2013, 77-84.

[5] Anil kumar ,Kamaldeep Grover and Balvinder Budania, “Optimization of Connecting Rod Parameters using CAE Tools,” International Journal of Latest Trends in Engineering and Technology (IJLTET).

[6] DR.B.K.Roy, “Design Analysis and Optimization of Various Parameters of Connecting Rod using CAE Software’s,” International Journal of New Innovations in Engineering and Technology (IJNIET).

[7] A. V. AKHARE, Y. B. DUPARE, A. S. WAGH, “STRESS ANALYSIS OF CONNECTING ROD,” Akhare AV, IJPRET, 2013; Volume 1(8): 250-260.

[8] Changyou Li, Changshuai Qiao, Yimin Zhang and Song Guo, “Reliability optimization design of connecting rod of locomotive traction equipment,” Proceedings of the Institution of Mechanical Engineers, Part C: Journal of Mechanical Engineering Science published.

[9] Pravardhan S. Shenoy and Ali Fatemi, “Connecting Rod Optimization for Weight and Cost Reduction,” 2005 SAE International.

[10] Daniel Gaspari Cirne Ce Toledo, Emílio Carlos Nelli Silva, Rafael Augusto De Lima e Silva, Robson Ferreira Da Cruz and Luis Antonio Fonseca Galli, “ design of a connecting rod for internal combustion engine by applying topology optimization,” proceedings of cobem 2009.

All rights reserved by www.ijsrd.com