International Journal of Scientific Research and Management Studies (IJSRMS)
ISSN: 23493771 **Volume 1 Issue 12, pg: 465-468**

DESIGN EVALUATION OF FATIGUE LIFE OF CONNECTING ROD USED IN HERO HONDA MOTOR CYCLE

[1]Dipalee Bedse, [2]Mangesh Ahire, [3]Swapnil.S.Kulkarni
[1]M.E in Mechanical Design Engineering at Gokhale Education Society, R. H. Sapat College of Engineering ,Nashik, India
[2]Professor in Department of Mechanical Engineering at Gokhale Education Society R. H. Sapat College Of Engineering ,Nashik, India.
[3]Director, Ethika Engineering Solutions India Pvt. Ltd., Pune, India

ABSTRACT

The connecting rod is the most relevant part of an automotive engine. The connecting rod is subjected to an extremely complex state of loading. High compressive and tensile loads are due to the combustion of fuel and connecting rod's mass of inertia respectively. The objective of this dissertation is to investigate the failure analysis of the connecting rod of Hero Honda Motor Cycle. Detailed analysis will be done to determine the stress, strain and total deformation in the present design of the connecting rod for the given loading conditions using the suitable tools. Fatigue Analysis is compared with the experimental results and further experimental results will be validated by using suitable tools.

KEYWORDS: *FEA; Static; connecting rod load analysis; connecting rod stress analysis; etc*

I. INTRODUCTION

The connecting rod is a major link inside a combustion engine. It connects the piston to the crankshaft and is responsible for transferring power from the piston to the crankshaft and sending it to the transmission. There are different types of materials and production methods used in the creation of connecting rods. The most common types of Connecting rods are steel and aluminum. The most common types of manufacturing processes are casting, forging and powdered metallurgy. Connecting rods are widely used in variety of engines such as, in-line engines, V-engine, opposed cylinder engines, radial engines and Opposed-piston engines

Summary of Researchers Works on Connecting Rod / Literature Review

Table No.1: List Of Researchers Work On Connecting Rod

Sr. No.	Title	Year	Study
1	Fatigue Behavior and Life Predictions of Forged Steel and Powder Metal Connecting Rods	May-2004	This study investigates and compares fatigue behavior of forged steel and powder metal connecting rods.
2	Dynamic analysis of loads and stresses in connecting rods	6 Feb 2006.	Detailed load analysis under service loading conditions was performed for a typical connecting rod, followed by quasi-dynamic finite element analysis (FEA) to capture stress variations over a cycle of operation.
3	Fatigue analysis of connecting rod U650 tractor in the finite element code ANSYS	2008	FEM is evaluated as a useful approach to recognize the critical points and fatigue life time of the reciprocating components such as connecting rod.

4	Failure analysis of a fractured connecting rod	May-2010	Behavior of the connecting rod into the start point for crack initiation before the occurrence of catastrophic failure.
5	Stress Analysis of I.C.Engine Connecting Rodby FEM	March-2012	The stresses induced in the small end of the connecting rod and the stresses induced at the big end.
6	Nonlinear static finite element analysis and optimization of connecting rod	June-2012	Nonlinear static analysis is carried out on piston end and crank end ofconnecting rod then further study was conducted to explore weight reduction opportunities for a production of connecting rod.
7	Fatigue Numerical Analysis for Connecting Rod	Nov-Dec 2012	The complete understand of the mechanisms involved as well as the reliability of the numerical methodology to reduce project lead time and prototypes cost reduction.
8	Design And Finite Element Analysis Of Aluminium-6351 Connecting Rod	May - 2013	Static analysis is done to determine the von Misses stress, elastic strain, total deformation in the present design.
9	FEM analysis of the isothermal forging of a connecting rod from material previously deformed by ECAE	July -2013	Two set of dies are designed to perform a two stroke forging process and to obtain the desired nano structured part.
10	Stress and Fatigue of Connecting Rod in Light Vehicle Engine	Dec-2013	Stress distribution and fatigue life of CR in light vehicle engine were analyzed using the commercial 3D finite element software

Research Gap: From the literature review it is seen that most studies on IC Engines connecting rod, Heavy Vehicle connecting rod had focus on axial stress, critical surface of connecting rod ,weight of connecting rod, the connecting rod failure& structural analysis of connecting rod and all these results are carried only FEA software method there is no any experimentation and validation results are found . However very less research done on the stress distribution and fatigue life of CR in light vehicle engine are analyzed using the commercial 3D finite element software recommend the best alternative design for the connecting rod through experimentation and validation

Problem Definition: Past research and experiences had indicated that during the operation of the engine, the connecting rod undergoes tensile, compression, and buckling loading. In many cases, the major reason behind the engine failure is the occurrence of the connecting-rod failure and sometimes, such a failure can be attributed to the broken connecting rod. In particular, the new analysis will examine the calculations for the stresses and fatigue failure of the connecting rod of Hero Honda Motor Cycle.

Objectives:

- Find out the problem areas by studying the existing system of linkages
- Identification of different parameters for redesign of the connecting rod
- Analyze the alternative geometry for connecting rod using CAE software
- Recommend the best alternative design for the connecting rod through experimentation and validation.

Scope:

The dissertation will focus on the stresses and fatigue failure of the component for evolving the best design of connecting rod. CAE software for analysis will be deployed for simulation and the results will be compared for improvement further.

Typical Steps involved in this project work are mentioned as below:

Following steps will be performed to execute this project. Following table gives details of steps and expected schedule.

Table No.2: Steps In Involved In Project Work

S. No.	Task	Remarks
1	Literature Review Understanding functional requirements of connecting rod	Compiled per the review of published papers on the research topic

2	To study existing functional requirements and list out advantages and limitations of existing design	Attempt to understand the formulation and application of empirical equations for the given case
3	CAD modeling for the existing application	3D model creation using CATIA/ UG/ or any other suitable 3D interface
4	Analysis using CAE	Design validation using Nastran or any other suitable CAE interface
5	Design modification for the application	By utilizing the inputs from CAE
6	Analysis for the modified configuration elements	Design validation using Nastran or any other suitable CAE interface
7	Testing and Validation	Experimentation over the test setup (if any)
8	Report Writing	Documenting results for proposed solution

II. METHODOLOGY

- **Finite Element Analysis**
  Finite element method is used to analyze structures by computer simulations and therefore it helps to reduce the time required for prototyping and to avoid numerous test series. The modeling and analysis will be done using Finite element Analysis software.
- **Steps for finite element analysis:**
  FEA is mainly divided into three following stages:
  - Preprocessing
    - Creating the model.
    - Defining the element type
    - Defining material properties
    - Meshing criteria
    - Applying loads
    - Applying boundary conditions
  - Solution 1: Assembly of equations and obtaining solution for structural analysis
    Solution 2: FE modeling for fatigue analysis using inputs from solution 1.
  - Post processing: Review of results

III. EXPERIMENTATION

Structural strength for the connecting rod would be verified over an UTM for tensile loading. The load applied at one end shall be determined through mathematical treatment for the problem. Static load would be applied to record the stresses over the component. Strain gauge shall be placed over the component as determined by FEA particularly for region of high stress. As for the fatigue life determined by the FE analysis the same shall be compared with historical data with the sponsoring company. The type and nature of the connecting rod resembling the previous application shall be referred to for comparison and validation. The physical experimentation for fatigue life is not feasible to conduct since the duration of testing exceeds 1000 hrs. of continuous exposure to cyclic loading. The validation for this work shall be exclusively effected by referring to static test for "Load vs. Stress"

IV. CONCLUSION

It has been concluded that Finite Element Analysis is best suitable method for predicting the Fatigue Life of any component. FEM has been evaluated as a useful approach to recognize the critical points and fatigue life time of the reciprocating components such as connecting rod.

REFERENCES

[1] Afzal, A., "Fatigue Behavior and Life prediction of Forged Steel and PM Connecting Rods," Master's Thesis, University of Toledo. 2004
[2] AhadRidzun Bin Ibrahim, "Analysis of Connecting Rod Fracture Using Finite Element Analysis". pp.110-112.
[3] Ali Fatemi and MehrdadZoroufi, "Fatigue Performance Evaluation of Forged Versus Competing Process Technologies: A Comparative Study", 24th Forging Industry Technical Conference, Cleveland, Ohio, 2002.

[4] AtishGawale, A. A.Shaikh1 and VinayPatil."Nonlinear static finite element analysis and optimization of connecting rod. World Journal of Science and Technology 2012, 2(4):01-04. ISSN: 2231 – 2587.
[5] Biancolini M.E., Brutti C., Pennestrì E. and ValentiniP.P"Dynamic, Mechanical Efficiency and Fatigue Analysis of the Double CardanHomokinetic Joint" Int. J. of Vehicle Design. Vol. 33, No.1, pp. 47-65.
[6] D. G. Gangwani, R. M. Metkar, 2009."Computer Aided Modelling and Finite Element Analysis of Connecting Rod Fouling on Camshaft of DI Engine of Mahindra and Mahindra Tractor", International Conference on Emerging Trends in Engineering and Technology, IEEE, pp.104-109.
[7] D. Jia, K. Wu, S. Wu, Y. Jia, C. Liang, "The Structural analysis and Optimization of Diesel Engine Connecting Rod" International Conference on Electronic & Mechanical Engineering and Information Technology, IEEE, pp. 3289- 3292.
[8] James R. Dale, "Connecting Rod Evaluation", January 2005.
[9] J. Sugita, T. Itoh, T. Abe, Honda R. & D. "Engine Component Design System Using Boundary Element Method", 905206.
[10] K. HO-LE, "Finite Element Mesh Generation Methids", Computer Aided Design, , January 1998,vol. 20, pp. 27-38
[11] Mr. H. B. Ramani, Mr. Neeraj Kumar, Mr. P. M. Kasundra, "Analysis of Connecting Rod under Different Loading Condition Using Ansys Software". International Journal of Engineering Research & Technology (IJERT) Vol. 1 Issue 9, November- 2012 ISSN: 2278-0181
[12] P. G. Charkha, S. B Jaju, "Analysis & Optimization of Connecting Rod", Second International Conference on Emerging Trends in Engineering and Technology,2009,IEEE, pp. 86-91.
[13] Pravardhan S. Shenoy, "Dynamic load analysis and optimization of connecting rod", 2004,pp. 30-42.
[14] P. S. Shenoy and A. Fatemi, "Connecting RodOptimization for Weight and Cost Reduction", University ofToledo, SAE Technical Paper Series, Paper No- 2005-01-0987.
[15] Roman Celin, Boris Arzenek, DimitrijKmeti "A Metallographic Examination of a Fractured Connecting Rod "Professional article/Strokovni ~lanek MTAEC9, Vol. 42, No.2, pp.93.
[16] R. Roger, "Fatigue failure of a connecting rod", Engineering Failure Analysis, ,January 1996,vol. 3, pp. 13-28.
[17] S. Khare, O.P. Singh, K. Bapanna Dora, and C. Sasun, "Spallinginvestigation of connecting rod", Eng. Failure Analysis, January 2012,vol. 19, pp. 77-86.
[18] Suraj Pal, Sunil kumar, "Design Evaluation and Optimization of Connecting Rod Parameters Using FEM". International Journal of Engineering and Management Research, December 2012,Vol.-2, Issue-6 ISSN No.: 2250-0758
[19] W. D. Webster, R. Coffell and D. Alfaro,. "A Three Dimensional Finite Element Analysis of a High Speed Diesel Engine Connecting Rod," SAE Technical Paper Series, ,1983,PaperNo. 831322..
[20] X. Hou, C. Tian, D. Fang, F. Peng and F. Yan, "Sensitivity Analysis and Optimization for Connecting Rod of LJ276M Electronic Gasoline Engine", Wuhan University ofbTechnology,2009, IEEE

AUTHORS' BIOGRAPHY

DIPALEE BEDSE Miss. Dipalee Bedse is currently pursuing her M.E in Mechanical Design Engineering at Gokhale Education Society R. H. Sapat College of Engineering ,Nashik, India. She did her B.E in Mechanical Engineering from Sinhgad Academy of Engineering Pune (Maharashtra) . She has 2 year of teaching experience , Now she working in Javahar Institute Technology ,Nashik, Maharashtra. Her area of Interest are Design of Machine Elements , Strength of Material .

MANGESH AHIRE Prof. Mangesh Ahire is currently working as an Associate Professor in Department of Mechanical Engineering at Gokhale Education Society R. H. Sapat College Of Engineering ,Nashik, India. His area of Interest are Design of Machine Elements , Strength of Material .

S.S.Kulkarni Director, Ethika Engineering Solutions (I) Pvt. Ltd., Pune. The Company offers Engineering Services Automotive OEM's and Tier I and Tier II Companies. He is a Graduate in Industrial Engineering with PG in Operations Management. With around 20 years of working experience in the domain of R&D, Product Design and Tool Engineering, he has executed projects in the Automotive, Medical and Lighting Industry. His area of interest is Research and Development in the Engineering Industry as well as the emerging sector of Renewable Energy.