NOVATEUR PUBLICATIONS
INTERNATIONAL JOURNAL OF INNOVATIONS IN ENGINEERING RESEARCH AND TECHNOLOGY [IJIERT]
ISSN: 2394-3696
VOLUME 3, ISSUE 2, FEB.-2016

OVERVIEW OF FATIGUE FAILURE OF CONNECTING ROD USED IN A LIGHT COMMERCIAL VEHICLE (LCV) THROUGH FINITE ELEMENT ANALYSIS

Mr.Gajanan Dinkarrao More,
M.E. Mech/ Design – 2nd yr Terna College of Engineering, Osmanabad

Prof.V.V.Mane
Mechanical Engg Dept., Terna College of Engineering, Osmanabad

Prof.M.S.Kadam
Bramadev Mane Institute of Technology Solapur

Prof.A.B.Ghalake
Mechanical Engg Dept., Terna College of Engineering, Osmanabad

ABSTRACT

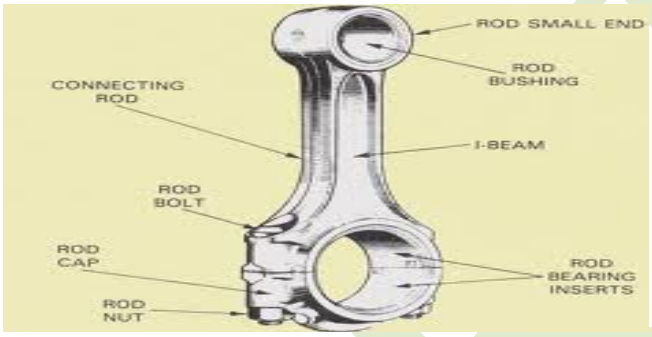
This study is carried out in view to verify and suggest design for reduction of mass & cost for connecting rod made by structural steel for 970 CC four cylinder four stroke engine. Every stroke in the engine is need to have to its nearby components to cyclic loading that push and pulls the crankshaft and connecting rod. The design of the connecting rod can be done in a specified manner, if study carried out to identify the effects of the operating loads on the component in the form of the type of stress induced with its higher value and the identification of areas where stresses over the component. While writing this paper, author has given main concentration mainly towards two major sections, finite element analysis and optimization for design and mass reduction.
In this paper, I-section is been analyzed using modeled using solid works. Also design is been verified using tetrahedron mesh using software hyper mesh. Ansys software is used Author has also tried to suggest few alternatives in the design of connecting rod by variation of different parameter.

INTRODUCTION

One of the important part of the combustion engine is the connecting rod, and the main purpose of the connecting rod is to transfer the energy from the pistons to crankshaft and convert the linear reciprocating motion of a piston into the rotary motion of a crankshaft, from the viewpoint of functionality; also it connects reciprocating piston to rotating crankshaft and some other design connected direct to the crosshead and then transmitting the thrust of the piston to the crankshaft of the combustion engine. The combustion engines connecting rod moves in two way rotating motion and connects both large and small end. The important property of the connecting

rods must have the highest value of possible rigidity keeping material weight as low as possible. In IC Engines used for automobile sector, connecting rod is a produced in very large quantity and it is always subjected to complex loading.

In the combustion engines bending stresses are due to eccentricities, crankshaft, rotational mass force and case wall deformation. Hence, a connecting rod must have capabilities of transmitting of axial tension at minimum axial compression. The connecting rod could be an low cost due to bending stresses are caused by the thrust and pull on the piston and by centrifugal force.



LITERATURE REVIEW

1. "Failure Analysis Of A Fractured Connecting Rod"
M.N. Mohammed[1] M.Z. Omar[2] Zainuddin Sajuri[3] A. Salah[4] M.A. Abdelgnei[5] M.S. Salleh[6]
In this paper author had focused on analysis of the connecting rod failure. The studies are been carried out by using finite element analysis and metallographic examination. Based on the experimentation, it is studies that, it is possible develop an casting defect based on cyclic loading behavior of connecting rod in combusting engines, it it's done successfully, then according to experimental results catastrophic failure can be avoided. The author based on findings in experiment, recommends that materials with good mechanical properties should be selected, and they should be free from manufacturing defects. Finally, lubrication engine system should be regularly checked, and all these are highly recommended to ensure long life of connecting rod in combustion engines.

2 "Stress Analysis of I.C.Engine Connecting Rod"
Vivek. C. Pathade, Bhumeshwar Patle, Ajay N. Ingale.
From the experimental results and analysis done using finite element method author concluded, that the stresses on the both end of connecting rods are different and stresses at the small end of the connecting rod are higher than the stresses on the larger end. Therefore risk of failure rod increases at fillet section of both end. Rigidity is the most important factor for the connecting rods and it must possess the highest possible rigidity at lowest possible weight. This paper is implemented using the ANSYS workbench 11.0 and PRO/E Wildfire 4.0 and this paper mainly deals with stress analysis of connecting rod in the combustion engines.

3 "Fatigue Numerical Analysis for Connecting Rod"

S B Chikalthankar, V M Nandedkar, Surendra Prasad Baratam

The paper has been written to demonstrate the analysis of connecting rod used in combustion engines using finite element analysis (FEA). The modified Goodman diagram is used to carry out fatigue study based on stress life (SxN) theory. Based on results obtained, the very high amount of stresses were develop in the small end region and calculation of fatigue factors calculated for almost all critical parts in connecting rod for combustion engines. The methodology presented in this paper, develops an important tool should be applied during the research and development phase of connecting rod.

4 "Stress and Fatigue of Connecting Rod in Light Vehicle Engine"

Bin Zheng, Yongqi Liu and Ruixiang Liu.

Author has written this paper after carrying out extensive study for stress distribution and fatigue life of connecting rod in combustion engines in light vehicle engine were analyzed using the commercial 3D finite element analysis software and ANSYSTM. The results shown in this paper are for medial surface of small end will be the critical surface, where chances damage will ignite maximum stretch condition. It has seen that maximum stretch will occur at medial surface of smaller end. The stresses are high at the shrank of the connecting rod, but because of development and high rigidity material, the stresses are relatively uniform over the medial surface of the connecting rods.

5 " Design And Finite Element Analysis Of Aluminium-6351 Connecting Rod"

Priyank D. Toliya, Ravi C. Trivedi, Prof. Nikhil J. Chotai.

This sole objective of this technical paper, is to investigate the failure analysis of the connecting rod of the automotive combustion engine. The author has used failure mode analysis is been carried out by using failure mode analysis. The analysis focused on the frequency of the failure of the connecting rod due to fatigue. Crack in the connecting rod due to excessive fatigue, causes uneven distribution of the stress over growth mechanism, there are various reasons for that. But mainly it occurs due to manufacturing defect. The crack is very dangerous in connecting rod, as it may cause catastrophic failure. In this paper author strongly recommends to lubricate the engine on regular basis.

The results of the experiments obtained by analyzing the finite element in the present study, it can be concluded that the occurrence of the connecting rod failure was due to the fatigue crack growth mechanism which came as a result of higher stress being combined with the porosity (manufacturing defect) in initiation and growth of a fatigue crack followed by catastrophic failure. The authors highly recommended that lubrication engine system should be regularly checked to ensure long life connecting rod.

PROBLEM STATEMENT

Past research has 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 catastrophic 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 component.

AIM

To calculate the stresses and fatigue failure in the connecting rod due to cyclic loads.

OBJECTIVES

- Identify the problem areas by studying the existing system of linkages
- Document the challenges to be addressed for uniformity in load distribution for connecting rod
- Consider feasibility 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 OF THE WORK

The paper work would focus on the parameters above for evolving the best design of connecting rod. Different options would be studied while arriving at the most suitable design for the connecting rod. CAE software for analysis (Patran/Nastran/ANSYS or any suitable FEA solver identified by the Sponsoring Company) would be deployed for simulation and the results would be compared for improvement further.

STEPS FOR FATIGUE ANALYSIS

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

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 Solid work/ or any other suitable 3D interface
4	Analysis using CAE	Design validation using Ansys 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 Ansys 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

METHODOLOGY

- **Finite Element Analysis**

  FEM(Finite element analysis) is adopted for analysis of structures, and various different software's is been used for doing an simulation. The advantage of using an computer based approach is helps in reduction of time drastically for analysis. For implementation of presented method the modeling and analysis is been carried out by 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
    - Applying loads
    - Applying boundary conditions
  - Solution: Assembly of equations and obtaining solution
  - Post processing: Review of results

CONCLUSION

Experiment is to be carried out over Fatigue Testing SPM. Cyclic loads are to be applied and the life to be recorded before failure. The failure could be in the form of excessive permanent strain or noticeable cracks or complete breakage of the component. The numbers of cycles are to be determined with respect to the application for the Automotive Vehicle under consideration. Typically, the component is exposed to around 30 to 75 million cycles in its lifetime. The test could be planned for limited cycles for extrapolating the results. Alternatively, an accelerated life testing can be engaged for securing real-time results.

REFERENCES

1] "*Failure Analysis Of A Fractured Connecting Rod"*M.N. Mohammed1 M.Z. Omar2 Zainuddin Sajuri3 A. Salah4 M.A. Abdelgnei5 M.S. Salleh6

2] "*Stress Analysis of I.C.Engine Connecting Rod" FEM* Vivek. C. Pathade, Bhumeshwar Patle, Ajay N. Ingale

3] "*Fatigue Numerical Analysis for Connecting Rod*" S B Chikalthankar*, V M Nandedkar**, Surendra Prasad Baratam* (*Department of Mechanical Engineering, Government College of Engineering, and Aurangabad) (SGGS Institute of Engineering and Technology, Nanded, India)*

4] "*Stress and Fatigue of Connecting Rod in Light Vehicle Engine*" Bin Zheng, Yongqi Liu* and Ruixiang Liu

5] *" Design And Finite Element Analysis Of Aluminium-6351 Connecting Rod*"Priyank D. Toliya, Ravi C. Trivedi, Prof. Nikhil J. Chotai *Department of Mechanical Engineering Marwadi education foundation's group of institutions, Rajkot, 360 003, Gujarat, India*