Optimization of Connecting Rod on the basis of Static & Fatigue Analysis

#1 V.A. Shedge, #2 C. Prof. K. H. Munde, #3 Prof. G.E. Kondhalkar

1 vikram.mechdesign@gmail.com
2 kashinathmunde@gmail.com
3 ganeshkondhalkar@gmail.com

#1 ABMSP’s Anantarao Pawar College of Engg. & Research, Parvati. Pune University.
#2 Dr. D. Y. Patil School of Engineering, Pune, India.
#3 ABMSP’s Anantarao Pawar College of Engg. & Research, Parvati. Pune University.

ABSTRACT

The 21st century competitive market demands light weight, low cost, fuel efficient engines, comfort drive with luxury at its best. Also with the on-going restrictive emission norms and increasing fuel prices it’s been crucial to cut down the weight of engines. Hence manufacturers are adapting light weight engines to survive the cut throat competitive market. The various components of an engine are piston, crankshaft, connecting rod, valves etc. The connecting rod being an essential component in an engine it becomes crucial to optimize connecting rod. The connecting rods function is to transmit energy from piston to the crankshaft. The paper deals with the static and fatigue analysis of existing design of automotive connecting rod which is having mass of 11.46 kg. Theoretical fatigue factor of safety of this existing design is 2.3. The most critical area is cross oil hole at shank. But this design is having Fatigue life nearly equal to 1.393E007 cycles which is very high and can be considered as infinite life. So by studying the scope for material removal, we can modify this existing robust design to get sufficient life i.e. in the range of E006 cycles. The mass of modified design is 9.47 kg. Here we have optimized the I-section of existing design. It is obvious that, due to removal of material there is increase in the stress level at the critical zone. But increase in the stress level is such that, fatigue factor of safety of modified design is 1.8 and having fatigue life of 1.5E006 which satisfying general criteria of E006 cycles. So, modified design is offering sufficient life with 1.99 kg material saving (17.37% weight reduction) with respect to existing design. Experimental validation is done on optimized Connecting Rod design by testing the same on servo hydraulic testing machine which shows same fatigue life as given by FEA analysis.

Keywords: Connecting rod, Mass Optimization, Static Analysis, Fatigue Analysis, Fatigue Factor of safety, Experimental Validation, Failure Analysis.

ARTICLE INFO

Article History
Received : 18th November 2015
Received in revised form : 19th November 2015
Accepted : 21st November , 2015
Published online : 22nd November 2015

I. INTRODUCTION

The automobile engine connecting rod is a high volume production, critical component. It connects reciprocating piston to rotating crankshaft, transmitting the thrust of the piston to the crankshaft. All internal combustion engines require number of connecting rod depending upon the number of cylinders in the engine. A 4-stroke IC engine undergoes 4 strokes. They are injection stroke, compression stroke, power stroke and exhaust stroke. The con rod is subjected to various forces in these four strokes. The major stresses induced in the connecting rod are a combination of axial and bending stresses in operation. The axial stresses are 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 are caused due to the centrifugal effects. It consists of a long shank, a small end and a big end. The cross-section of the shank may be rectangular, circular, tubular, I-section or H-section. Generally circular section is used for low speed engines while I-section is preferred for high speed engines. The most common type of manufacturing processes is casting, forging, and powdered metallurgy. Connecting rod is subjected to a complex state of loading. It undergoes high cyclic loads, which range from high compressive loads due to combustion, to high tensile loads due to inertia. Therefore, durability of this component is critical importance. Due to these factors, the connecting rod has been the topic of research for different aspects such as production technology, materials, performance, simulation, fatigue etc.
I. STATIC ANALYSIS OF EXISTING DESIGN

Existing Design geometrical details are as follows. Its mass is 11.46kg.

A. Calculation of Forces Acting on Connecting Rod

a) Input Parameters:
The input parameters of it are described below. The original mass of connecting rod is 11.47 kg. The calculations for original design are as follows.
- Maximum gas pressure, \( P_{\text{max}} = 2175.57 \text{ psi} \)
- Length of connecting rod, \( L = 345 \text{ mm} \)
- Reciprocating masses, \( M_r = 6000 \text{ gm} \)
- Bore diameter, \( D = 150 \text{ mm} \)
- Crank radius, \( R = 100 \text{ mm} \)
- Crank speed, \( N = 2500 \text{ RPM} \)
- Firing angle, \( \theta = 90 - 110 \)

b) Material Properties:
- Material density \( \delta_C = 7.85 \times 10^{-9} \text{ ton/mm}^3 \)
- Poisson’s ratio \( \mu = 0.3 \)
- Young’s Modulus \( E = 2.1 \times 10^5 \text{ MPa} \)
- Yield strength \( S_y = 600 \text{ MPa} \)
- Ultimate tensile strength, \( S_u = 900 \text{ MPa} \)

c) Force due effect of gas pressure on piston

\[ F_g = (\text{Maximum Gas Pressure}) \times (C/s \text{ Area of Piston}) \]
\[ F_g = P_{\text{max}} \times \frac{\pi D^2}{4} \]
\[ F_g = (2175.57 \times 10^7 / 14.5037) \times \frac{\pi \times 0.15^2}{4} \]
\[ F_g = 265071.8801 \text{ N} \]

d) Force due to inertia of reciprocating masses

Inertial force due to reciprocating parts is given by,
\[ F_i = (\text{mass of Reciprocating Mass i.e. piston assembly}) \times (\text{Acceleration}) \]
\[ F_i = M_r \omega^2 (\cos \theta + \frac{\cos \beta}{L}) \]
\[ F_i = 6 \times \left( \frac{2 \times \pi \times 2500}{60} \right)^2 \times 0.1 (\cos 10 + \frac{\cos 10}{L}) \]
\[ F_i = 51699.555 \text{ N} \]

B. Load Cases and Boundary Conditions

The static analysis is carried out for tensile and compressive conditions under maximum gas load. There are four load cases in the analysis of con rod.

1) Loading at Piston End
- Tensile load i.e. load due to inertia 51699.555N is applied over 180° at piston end with crank end restrained over 180° of contact area.
- The compressive load i.e. load due to gas pressure 265071.881N is applied as a uniformly distributed load over 120° of contact surface at piston end with crank restrained over 120° of contact area.

2) Loading at Crank End
- Tensile load i.e. load due to inertia 51699.555N is applied over 180° at crank end with piston end restrained over 180° of contact area.
- The compressive load i.e. load due to gas pressure 265071.881N is applied as a uniformly distributed load over 120° of contact surface at piston end with piston end restrained over 120° of contact area.

C. FEA Results and Fatigue factor of safety

As we discussed, the inertia forces are responsible for tension loading and force due to gas pressure is responsible for compressive loading. For designing the component (fatigue life point of view) here we have considered 1st Principal stresses and 3rd Principal stresses. So, by applying these forces as per loading and boundary conditions discussed above, we have determined 1st Principal stresses for tension loading and 3rd Principal stresses for compression loading. Highest 1st Principal stress is 101MPa at cross oil hole at shank as shown in fig.2 (a). Also highest 3rd Principal stress is 350MPa at the same location as shown in fig. 2(b)

![Fig. 2 (a) - 1st Principal Stress](image-url)

![Fig. 2 (b) - 3rd Principal Stress](image-url)

Since, both 1st Principal and 3rd Principal stresses are highest at the same location, we can say that this cross oil hole area at shank is the most critical area from fatigue point of view. So now we will find out the Fatigue Factor of Safety by using Goodman theory,

\[ FOS = \frac{1}{2 \times F_s} \]
\[ FOS = 2.389 \]

III. FATIGUE ANALYSIS OF EXISTING DESIGN

Now on the basis of above FEA analysis, we have carried out fatigue analysis of existing design in N-Code dedicated fatigue analysis software.

A. N-code Work Environment
There are five important segmental windows in the N-Code work environment.

1. **FE Input Window:** This window is the starting point of fatigue analysis through which we take the FE input data. This data is nothing but the .rst file that we have prepared during above static analysis. This file contains two load cases i.e. tension loading and compression loading. This FE input data is used for fatigue life calculations.

2. **SN-Analysis:** This window is the main part of this software, where actual fatigue life calculations are carried out. Here, we go through different steps related to defining material and loading conditions and accordingly this will calculate fatigue life for each and every node.

3. **FE Display:** Visual display of fatigue analysis can be viewed through this window.

4. **Hot Spot Detection:** This window will show the most critical and most damaged areas which is having lower life along with corresponding node numbers.

5. **Data Value Information:** This window is used for data acquisition in which whole data related to the fatigue life is tabulated with corresponding node numbers.

### IV. TOPOLOGY OPTIMIZATION

Here topology optimization is carried out in order to optimize the original connecting rod. HYPERMESH Optistruct is used for optimization analysis. Here blue colour shows scope for material removal.

### Table I

<table>
<thead>
<tr>
<th><strong>TOPOLOGY OPTIMIZATION PARAMETERS</strong></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Objective</strong></td>
<td>Reduce volume of the component only in the design area.</td>
</tr>
<tr>
<td><strong>Stress constraint</strong></td>
<td>Yield stress is given as stress constraint parameter, 600MPa</td>
</tr>
<tr>
<td><strong>Optimization Response</strong></td>
<td>Mass or volume is the response.</td>
</tr>
<tr>
<td><strong>Boundary conditions.</strong></td>
<td>All the four load cases applied simultaneously.</td>
</tr>
</tbody>
</table>

### V. MODIFIED DESIGN- STATIC ANALYSIS

According to Optistruct results we have modified I-Section. Design geometrical details are as follows. Its mass is 9.47kg. Modified I-section is shown in fig.5 by magenta color and existing I-section by cyan color.

By applying same loading and boundary conditions, we have carried out static stress analysis for modified design in the same manner as for existing design. Highest 1st Principal stress is 142Mpa at cross oil hole at shank as...
shown in fig.6 (a). Also highest 3rd Principal stress is 439MPa at the same location as shown in fig 6(b). Fatigue Factor of Safety for this modified design by using Goodman theory will be,

$$FOS=1.81$$

![Fig. 6 (a) - 1st Principal Stress](image1)

![Fig. 6 (b) - 3rd Principal Stress](image2)

VI. FATIGUE ANALYSIS OF MODIFIED DESIGN
As Fatigue Factor of safety is reduced from 2.38 to 1.81, it is obvious that this modified design will have less fatigue life than that of the existing design. Fatigue analysis of modified design is carried out in the same fashion as existing design and it shows that fatigue life of modified design is reduced to 1.581E006 cycles.

![Fig. 7 - Fatigue Life of existing design](image3)

Now this fatigue life is in the range of E006 cycles. So we can say that this design is having sufficient life with having 17.35% less mass than the existing design and modified design is one of the optimized design.

VII. EXPERIMENTAL VALIDATION
A vertical axis universal servo-hydraulic fatigue test machine of 1000 KN capacity is used for connecting rod fatigue tests. Custom made fixtures used in the previous project are modified to suit the test machine. Pin ground to big and small end journal dimensions are fitted to the rods and clamped in split blocks bolted together to aid assembly and dismantling of the test set up as shown in figure.

![Fig. 6 (a) – Servo Hydraulic Machine](image4)

![Fig. 6 (b) – Experimental Set Up](image5)

The rods are used to test under constant amplitude axial loading at a frequency of between 1 to 3.5Hz.

Load conditions used: Rated speed: 5270 kgf (i.e. equivalent to inertia force = 15699.555N.) to -27020 kgf (i.e. equivalent to Gas force = -265071.8801N.) Total 10 connecting rods are tested and their results are as follow.
A. Fatigue Life Results by Experimental Fatigue Test

![Fatigue Crack]

Fig. 7 - Failure Location (at oil way cross drill)

<table>
<thead>
<tr>
<th>Rod No.</th>
<th>No. of Cycles Passed</th>
<th>Failure location</th>
</tr>
</thead>
<tbody>
<tr>
<td>T1</td>
<td>1,482,326</td>
<td>Oil way cross drilling in shank</td>
</tr>
<tr>
<td>T2</td>
<td>1,43,598</td>
<td>Small end bush &amp; shank, Big end bearing cracked</td>
</tr>
<tr>
<td>T3</td>
<td>1,254,383</td>
<td>Oil way cross drilling in shank</td>
</tr>
<tr>
<td>T4</td>
<td>1,358,339</td>
<td>Oil way cross drilling in shank, small end bush cracked</td>
</tr>
<tr>
<td>T5</td>
<td>1,409,634</td>
<td>Oil way cross drilling in shank</td>
</tr>
<tr>
<td>T6</td>
<td>5,94,612</td>
<td>Big End oil way</td>
</tr>
<tr>
<td>T7</td>
<td>1,432,242</td>
<td>Oil way cross drilling in shank</td>
</tr>
<tr>
<td>T8</td>
<td>1,329,924</td>
<td>Oil way cross drilling in shank</td>
</tr>
<tr>
<td>T9</td>
<td>1,426,954</td>
<td>Oil way cross drilling in shank</td>
</tr>
<tr>
<td>T10</td>
<td>1,379,258</td>
<td>Oil way cross drilling in shank</td>
</tr>
</tbody>
</table>

B. Observations

From experimental results, it is clear that out of 10 tests, 8 connecting rod is failed at oil way cross drilling which is the same location shown by FEA Analysis and Life also matching with that of calculated by the N-Code software which in the range of 1E006 to 1.51E006 cycles.

VIII. CONCLUSIONS

- Existing design of connecting rod is having mass of 11.46kg and having fatigue factor of safety 2.38. Fatigue life of this design is 1.393E007 cycles, which somewhat more than the requirement criteria of E006 cycles.
- In Modified design, we have removed material from existing design so as to get 17.35% weight reduction. This design is having fatigue factor of safety 1.81 and having fatigue life of 1.581E006 cycles, which is satisfying the criteria of range of E006 cycles.
- So, removing the material from different sections, stress level increases and fatigue factor of safety along with fatigue life reduced. But if removal of material in such fashion that fatigue factor of safety is in between 1.5 to 2.0 then component will get sufficient fatigue life in the range of E006 cycles.
- Since, modified design is satisfying E006 cycle range, we can consider this design as one of the optimized design.

ACKNOWLEDGMENT

I take this opportunity with great pleasure to express my deep sense of gratitude towards my esteemed guide Prof. K. H. Munde for his valuable guidance and incessant encouragement and co-operation extended to me during this project work. I would also like to thank Prof. G.E. Kondhalkar (HOD Mechanical Engg. Department) and Prof. S. B. Thakare (Principal, Anantrao Pawar College of Engineering & Research, Parvati, Pune) for their unflinching help, support and cooperation during the project work.

I would like to thank Bharat Forge Ltd, Pune- Fatigue Testing & CAE Lab for their continuous support and proper guidance.

REFERENCES