RESEARCH ARTICLE OPEN ACCESS

Finite Element Analysis and Design Optimization of Connecting Rod

Ashwini Mane*, D. D. Date**, A. Z. Patel***
*PG Student (Department of Mechanical Engineering, TPCT's Collage of Engineering Osmanabad.
** Professor (Department of Mechanical Engineering, TPCT's Collage of Engineering Osmanabad.
*** Professor (Department of Mechanical Engineering, TPCT's Collage of Engineering Osmanabad.

ABSTRACT
The objective of this study is to improve the design of connecting rod of single cylinder four stroke Otto cycle engine by shape optimization. The main objective of this study is weight reduction of connecting rod and improving its performance without affecting its functionality. Finite element analysis is one of the most important tools of CAD/CAM CAE. For this study ANSYS analysis software is used for modeling, analysis and shape design optimization. Initially, according to design considerations maximum loads were calculated for various maximum operating loading conditions. Calculated loads used as a loading condition in various load steps of FEM analysis. Stresses generated across all the locations of connecting rod evaluated using ANSYS Workbench. For optimization ANSYS Shape optimization module is used and extracted the required shape of connecting rod. Final CAD model of optimized connecting rod is prepared in Design Modeler. Static structural analysis of modified design is performed and the results compared with baseline design. After result are validated with the help of Modified Goodman's Diagram. From the shape optimization we could able to achieve 14.73% weight reduction in existing connecting rod. Since the optimized design is having sufficient life, the design is much improved as compared to the existing design.
Keywords – optimization, Finite Element analysis, connecting rod, internal combustion engine, ANSYS, Weight reduction

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, as a result the reciprocating motion of the piston is translated into rotational motion of the crankshaft. Connecting rod consists of a pin end, a shank, and a crank end. Pin end and crank end holes are machined to permit accurate fitting of bearings. Pin end of the connecting rod is connected to the piston by the gudgeon pin. The crank end revolves with the crankshaft and is either split piece or one piece. Split piece crank end permit it to be clamped around the crankshaft split connecting rod are used where the crankshaft is manufactured in single piece. Single piece crankshafts are generally used in multi cylinder engine. The one piece connecting rod is used where the crankshaft is made up of press fitted crank pin in crankshaft. Every vehicle that uses an internal combustion engine requires at least one connecting rod depending upon the number of cylinders in the engine. Connecting rods are subjected to forces generated by mass of moving components and fuel combustion. These two forces results in axial and bending stresses. Therefore, a connecting rod must be capable of transmitting axial tension, axial compression, and bending stresses caused by the Thrust and pull on the piston and by centrifugal force [1]. Literature shows that there are number of studies carried out on the connecting rod analysis, but very little work was done on shape optimization of connecting rod. R. J. Yang et al. 1992 [2]: In this research work from Ford motor company 23400 Michigan Avenue USA they had perform shape optimization of upper end i.e. pin end of connecting rod using MSC/ NASTRAN. They had considered two different design cases, one with 5 design parameters and another with 10 design parameters. M. Sc. Anna Ulatowska 2008 [3]: In this research work they had performed shape design optimization of engine connecting rod made up of forged steel. They had optimized connecting rod for large notch stresses. Christina Schäfer et. al. 2007 [4]: In this research work they had performed shape design optimization of steel wheel of automobile. They had performed optimization for stress concentration reduction at various regions. Fanil Desai et. al. 2014 [1] worked on numerical and experimental analysis of connecting rod. This research explains about performance of connecting rod under different loading conditions and its validation with experimental results. In current study optimized design is also compared with the existing connecting rod for design reliability purpose.

II. NUMERICAL EVALUATION OF MAXIMUM LOADING CONDITION OF CONNECTING ROD

The There are total four loading conditions of each cycle of four stroke Otto cycle engine this are as follow- 1Tensile loading at suction stroke. 2. Compressive loading at compression stroke. 3. Compressive loading at power stroke. 4. Compressive loading at exhaust stroke.

The connecting rod is always designed at maximum operating speed at maximum loading conditions which occurs at 1st and 3rd loading conditions [5]

1. 1st Loading condition: Tensile loading during suction stroke

1.1 Tensile force at small end

Tensile force at small end is calculated by formula [5]

$$F_{t\ small} = -(m_p + m_{se}) \times \omega^2 \times R \times (\cos\theta + \lambda \times \cos 2\theta)$$

$$F_{t\ small} = -2370.2\ N \qquad (I)$$

1.2 Tensile force at big end

Tensile force at big end is calculated by [5]

$$F_{t\ big} = -\omega^2 \times R \times \left[(m_p + m_{crp}) \times (1+\lambda) + (m_{crc} - m_b)\right]$$

$$F_{t\ big} = -3777\ N. \qquad (II)$$

2. 3rd Loading condition: Compressive loading during power stroke

$$F_{Comp} = (P_{Za} - P_0) \times f_p - m_p \times R \times \omega^2 \times (\cos\theta + \lambda \times \cos 2\theta) \qquad (III)$$

III. STATIC ANALYSIS OF EXISTING CONNECTING ROD

Equivalent stresses are extracted from static structural analysis by using Ansys Mechanical module. Load cases are considered for the analyses are mentioned as (I), (II) and (III). Material properties for connecting rod are: Density-7990 kg/m3, Young's modulus-2.08E5 N/mm^2, Poisson's ratio-0.3 and Ultimate strength-700 MPa, Yield strength-400 MPa.

Fig 1 shows the result obtained because of tensile loading at small end of connecting rod. Equivalent stress observed in the model is 97 MPa. Tensile force at big end is 3777 N and is a second loading condition in the analysis. Stress observed in the model is 156 MPa. Whereas the high magnitude compressive load during compression stroke generates stress around 156 MPa.

IV. OPTIMIZATION FOR WEIGHT REDUCTION

Optimization is performed using the Shape optimization tool within ANSYS software. Prepared CAD model is used for optimization same as in structural analysis. Boundary conditions are also defined similarly. Preprocessing on geometry is done to define the design and non design space. Bothe ends (Big end and small end) are defined as non design space, where as the middle I section part is defined as design space. Non design space is determined according to the interfacing part constrains. Results are extracted from the optimization which will be a part with reduced weight.

Fig.5 shows result for optimized model for Load case 1. The results shows that the red region to remove the material. In load case 2 Fig.6 also have same results as in Load Case 1. The load case 1 and 2 are in tensile in nature, hence it is observed to have similar result. Only the change is in amount of material to remove from original design is more in case of Load Case 3 Fig 6. Compression stroke is simulated in Load case 3, which is higher in magnitude than all load cases, because of higher force exertion during compressive stroke [5].

It is important to take a decision on optimized design based on the available results of various load case from ANSYS shape optimization tool. It should be combined with all of the results which need to survive while various forces acting on connecting rod during its operating condition. Fig. 8 shows optimized design prepared in design modeler considering the results from shape finder in ANSYS shape optimization tool.

V. STATIC ANALYSIS OF MODIFIED CONNECTING ROD

While performing analysis on modified design all analysis setting needs to be same as used in existing design of connecting rod. It is necessary to take this precaution because the results are very sensitive with respect to the mesh size of the component. Results for different load cases are plotted in Fig 9, 10, & 11.

Load case 1 and Load case 2 does not make much difference if compared with existing design. Load case 3 is most critical load case because it has high magnitude of force in compressive nature. Equivalent stress in design in third load case if observed around 186 MPa. Whereas Load case 1 and 2 have equivalent stresses 101 Mpa and 159 Mpa respectively.

VI. RESULT & DISCUSSION

Existing design and optimized (modified) design is analyzed on similar platform. Similar platform means the loading conditions are exact same in both of the design. The additional parameters such as meshing methodology and mesh size is tried to keep with minimum deviation. Designs are meshed with all tetrahedral higher order element with same body size of 1 mm. Special care needs to take at filets, because these are critical stress location areas as per mentioned in [6] Mesh size across all the fillet is considered a 0.5 mm to get more accurate results.

Table 1 and 2 shows the comparison between existing and optimized design. It seems that the stresses in the modified design are increased as compare to the existing design. Stresses in Load case 1 and Load Case 2 are increased by almost 2%. It means that the stresses exerted by tensile load on connecting rod are not changing drastically. Whereas if we compare the result for Load case 3 it seems that the stresses are changed slightly. These stresses are occurred due to compressive force from big end of connecting rod. The difference 15% is observed between both the designs. Although there is slight increase is stresses because of compressive load will not confirm that the modified design is not better as compare to existing design. Increase in stresses is observed in the Modified design are compressive stresses. Various studies have been done on fatigue evaluation of steel, which explains that compressive stresses are cases of increasing the fatigue life of the component [7] and [8]. The stresses observed in the model are below Yield limit of 700MPa, hence design have good margin in static loading. So it is very important that the design needs to get validated with the fatigue life evaluation. The Fatigue life evaluation and comparison can only decide which design is better.

Fig. 12 shows Modified Goodman's diagram plotted for both design. Maximum Principle stresses are considered while evaluating maximum and minimum stresses acting on connecting rod. From maximum and minimum stresses mean and alternating stresses are calculated. For Existing design maximum and minimum stresses observed as 108MPa and -170Mpa respectively. In case of modified design these stresses are 144MPa and -198MPa. Extracting maximum and minimum stresses at critical location based on stress plot in the ANSYS. This location have maximum stress during tensile loading (Load Case 1 and 2 whichever in higher) and minimum value during compressive Loading (Load Case 3). Based on the material properties of connecting rod a Yield line and Goodman's line is plotted. A point under the area of Yield line and Goodman's line indicates infinite life (1E6 Cycles) of component.

Now if we compare the points plotted for both designs on Modified Goodman's diagram, both points lies within the safe zone of infinite life of component. Hence increase in the compressive stresses is not making any significant changes in the life of design. The connecting rod will be having much enough fatigue life.

VII. CONCLUSION

Results summary explains all about the comparison between existing and optimized design of the connecting rod. The main achievement from this study is to reduce the weight of connecting rod by 16 gms from existing design. Modified Goodman's plot shows design margin for both the designs. Hence from all above discussion it can be concluded couple of points listed below:

- Optimization technique used for this study is successful to achieve weight reduction by almost 15%.
- Static Structural analysis results shows sufficient design margin without failing in yielding criteria, It concludes that design is passing the acceptance criteria.
- Fatigue validation is also an important parameter because connecting rod has to withstand for infinite life (1E6 cycles). From the Goodman's plot it can be concluded that design meets the fatigue life criteria.
- From limitation point of view if we think, the design complexity is increased as compared to the earlier design, though the manufacturing processes too.

VIII. FIGURES AND TABLES

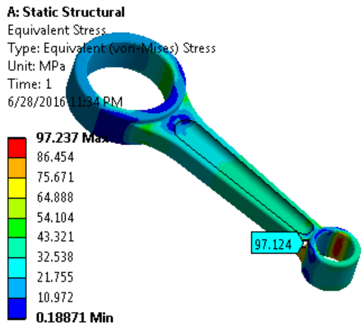

Figure 1, Existing Design Equivalent Stress Load Case 1

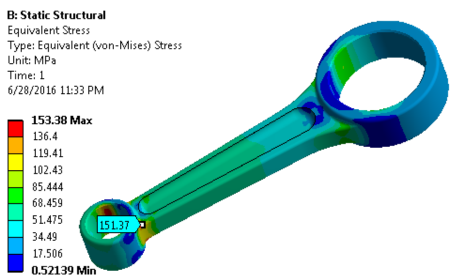

Figure 2, Existing Design Equivalent Stress Load Case 2

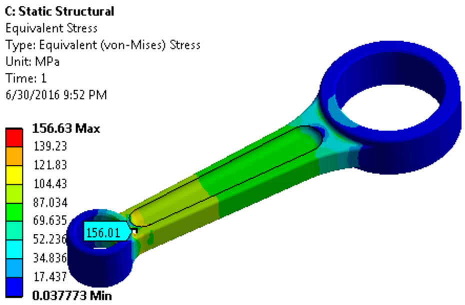

Figure 3, Existing Design Equivalent Stress Load Case 3

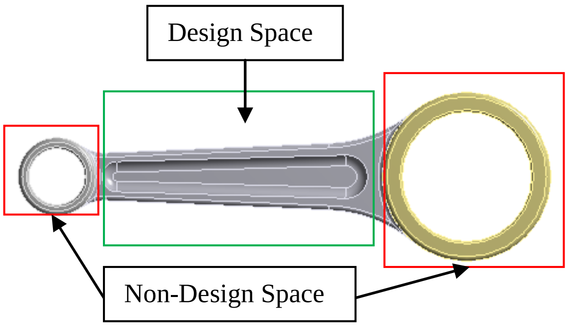

Figure 4, Design and non design space

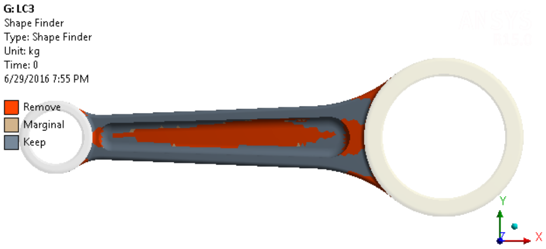

Figure 5, Shape Finder Load Case 1

Figure 6, Shape Finder Load Case 2

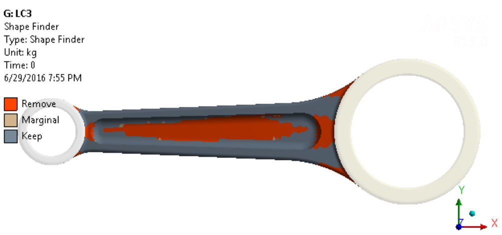

Figure 7, Shape Finder Load Case 3

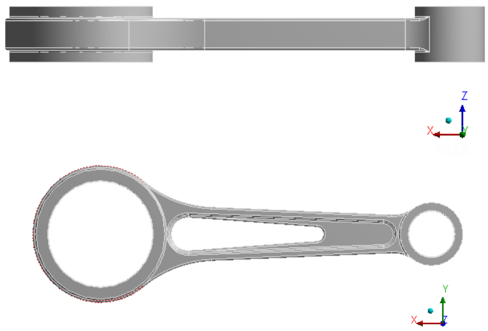

Figure 8, Optimized Design

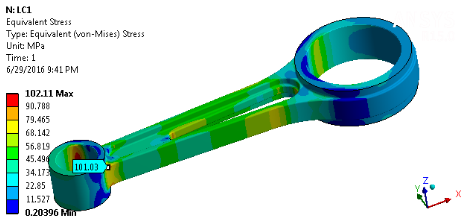

Figure 9, Optimized Design Equivalent Stress Load Case 1

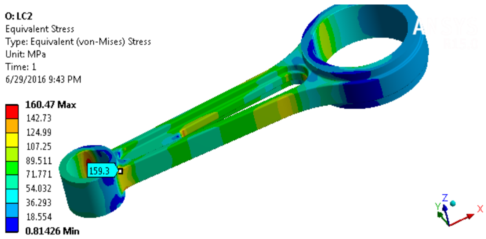

Figure 10, Optimized Design Equivalent Stress Load Case 2

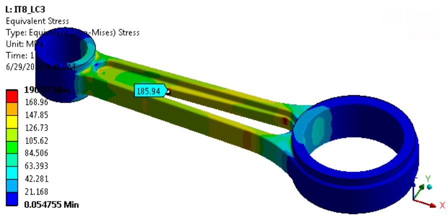

Figure 11, Optimized Design Equivalent Stress Load Case 3

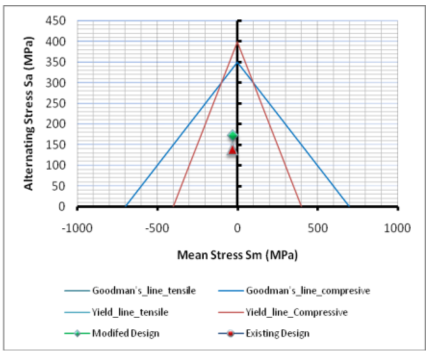

Figure 12, Modified Goodman's Diagram

Table 1: Results for existing design

Load Case No.	Load Case Description	Force (N)	Max Principle Stress (MPa)	Min Principal Stress (MPa)	Von Mises Stress (MPa)
1	Tensile loading during suction stroke at small end	2370	99	1.5	97
2	Tensile loading during suction stroke at big end	3777	159	2	151
3	Compressive loading during power stroke at big end	5847	-6	-168	156

Table 2: Results for modified design

Load Case No.	Load Case Description	Force (N)	Max Principle Stress (MPa)	Min Principal Stress (Mpa)	Von Mises Stress (MPa)
1	Tensile loading during suction stroke at small end	2370	101	3	102
2	Tensile loading during suction stroke at big end	3777	167	5	160
3	Compressive loading during power stroke at big end	5847	-3	-197	190

REFERENCES

[1] F. Desai, K. Jagtap and A Deshpande, Numerical and Experimental Analysis of Connecting Rod, *International Journal of Emerging Engineering Research and Technology, 4(2),* 2014, PP 242-249

[2] R. Yang, D. Deversit and J. Alliset, Shape optimization of connecting rod pin end using genetic model. *Ford Motor Company*, 23400 Michigon Avenue, vp1100,dearbon,mi48124, USA.1992

[3] M. Sc. Anna Ulatowska, *Shape optimization of the connecting rod* (Politechnika Krakowska, 2008)

[4] Christina Schäfer, Eckhard Finke, Shape optimization by design of experiments and finite element methods-an application of steel wheels, *Structural and Multidisciplinary Optimization, 36,* 2008, PP 477-491 (Wolfsburg, Germany, 2007)

[5] A. Kolchin, V. Demidov, *Design of Automotive Engines (* MIR Publishers, Moscow, 1985)

[6] P. K. Sharma, B. Rajendra, Fatigue Analysis and Optimization of connecting rod using Finite Element Analysis. *International journal of advance research in science and engineering 1,* 2012.

[7] Baskaran Bhuvaraghana, Sivakumar M.Srinivasanb, Bob Maffeoc, Optimization of the fatigue strength of materials due to shot peening : A Survey, *International Journal Of Structural Changes In Solids – Mechanics and Applications, 2 (2),* 2010, PP. 33-63

[8] Yoshiaki Akiniwa, Hidehiko Kimura And Takeo Sasaki, Effect Of Residual Stresses On Fatigue Strength Of Severely Surface Deformed Steels By Shot Peening, *International Centre for Diffraction Data,* 2009