IJREAT International Journal of Research in Engineering & Advanced Technology, Volume 2, Issue 2, Apr-May, 2014
ISSN: 2320 – 8791(Impact Factor: 1.479)
www.ijreat.org

Stress Analysis Of Connecting Rod Using Finite Element Method

Digvijay[1], Mohd.Ahmad[2], Ajay Mishra[3], Karunakar Mishra[4], Alok Panday[5], Shyam Bihary Lal[6]

[1,2,3,4] B. tech Students, Department of Mechanical Engineering, Buddha Institute of technology GIDA Gorakhpur Utter Pradesh (India)

[5,6] Asst. Professor Mechanical Engineering Department, Buddha Institute of technology GIDA Gorakhpur Utter Pradesh (India)

Abstract

In a reciprocating piston engine, the connecting rod connects the piston to the crank or crankshaft. The connecting rod is the intermediate member between the piston and crankshaft. Its primer function is to transmit the push and pull from the piston pin to the crankpin and thus convert the reciprocating motion of the piston into the rotary motion of the crank. There are different types of material and production method used in the creation of connecting rod. For the project work we have selected connecting rod used in light commercial vehicle of Tata motors. The main objective of this work is to explore weight reduction opportunities in the connecting rod of an I. C. engine by examining various materials such as steel alloy, cost Iron, Structural steel. This was entailed by performing a detailed load analysis. Therefore this study has deals with two subject first static load and stress analysis of connecting rod and second, Design optimization for suitable material to minimize the deflection. We used Solid works software for Design and ANSYS Workbench 14.0 for analysis.

***Keywords**: Connecting rod, FEM, Stress analysis, FEA, ANSYS Workbench 14.0.*

1. Introduction

One source of energy in automobile industry in internal combustion engine, I.C. engine converts chemical energy into mechanical energy in the form of reciprocating motion of piston. Crankshaft and connecting rod convert reciprocating motion into rotary motion [6]. The Automobile engine connecting rod is a high volume production, critical component. If connects reciprocating piston to rotating crankshaft, transmitting the thrust of the piston to crankshaft. Every vehicle that uses an internal combustion engine requires at least one connecting rod depending upon the number of cylinder in the engine. They are different types of materials and production method used in the creation of connecting rods. The major stresses induced in the connecting rod are a combination of axial bending stress 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 and compressive), where as bending stresses are caused due to the centrifugal effects [2]. It consists of a long shank, a small 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 engine while I-section is preferred for high speed engine [9]. This work investigated weight reduction and the suitable better material for minimizing deflection in connecting rod. First the connecting rod was digitized. Load analysis was performed which comprised of the connecting rod , small and big ends of connecting rod using analytical techniques and computer based mechanism simulation tools. FEA was then performed using the results from load analysis to gain insight on the structural behavior of the connecting rod and to determine the design loads for optimization and deflection.

2. Specification of the problem

The objective of the present work is to design and analyses of connecting rod made of structural steel material are used to design the connecting rod. In this project the material Steel alloy of connecting rod replaced with Structural steel connecting rod was created in solid works. Modal is import in ANSYS 14.0 for analysis .after analysis a given the FEA results.

3. Finite Element Method

The finite element method is numerical analysis technical of opening approximate solution to a wide verity of engineering problems. Because of its diversity and flexibility as an analysis tool , it is receiving much attention in engineering school and industries in more and more engineering situation today we find that it is necessary to obtain approximate solution to problems rather than exact close from solution it is not possible to obtain analytical mathematical solutions are many engineering's problems. An analytical solution is a

mathematical expression that gives value of the desire unknown quantity an any location in the body, as consequence it is valid for infinite number of location in the body.

For problem involving complex material properties and bounder condition, the engineer resource to numerical method that provide approximate that eatable solution.

Table.1 Design Calculation for Connecting Rod

Particulars	Value
Maximum engine speed	5700 RPM
Maximum gas pressure	37.3 bar
Crank shaft Radius	48.5mm
Connecting rod length	141mm

4. Modeling

Connecting rod of Tata Light Commercial Vehicle, market available is selected for the present investigation. The dimensions of selected connecting rod are measured using vernier calipers, screw gauge and are tabulated and present in a table. According to the model of the connecting rod is developed using solid works. The modeled connecting rod is shown in figure 1. In this analysis two materials are used.

4.1 Material Names: select material

Alloy Steel AISI4340 (present in market)

Structural steel (own paper)

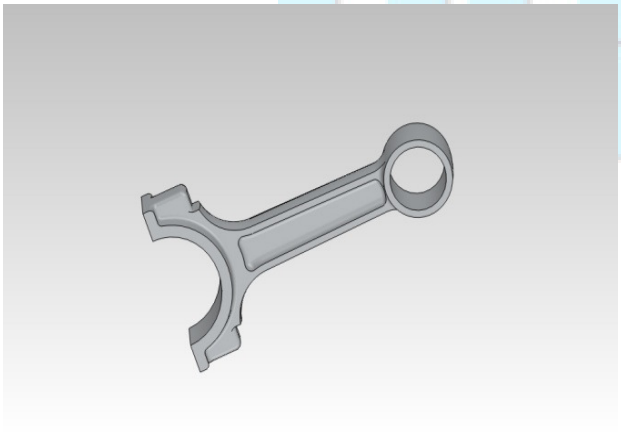

Fig. No.1:Modal of connecting rod

5. Methodology

5.1 Procedure of statics analysis

First of all, we have prepared assembly in Solid Works for connecting rod and save as this part as IGES for exporting into ANSYS workbench environment. Import IGES mode in ANSYS workbench simulation module.

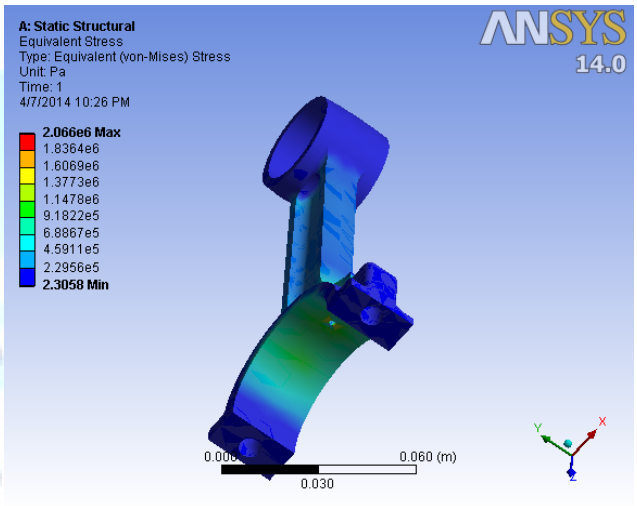

Fig. No.2: Equivalent (Von-Mises) Stress

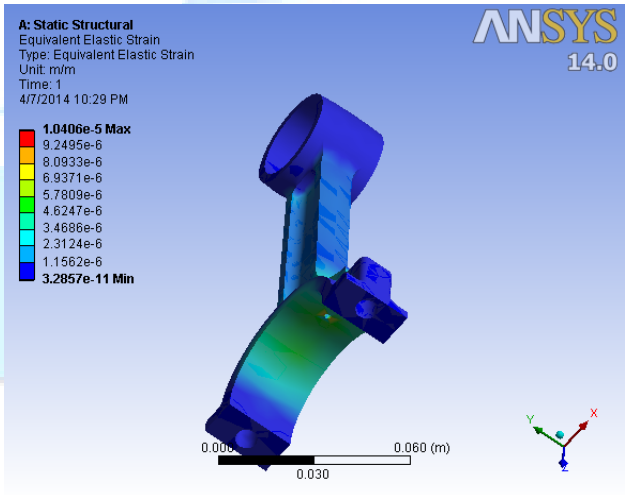

Fig. No.3: Equivalent Elastic Strain

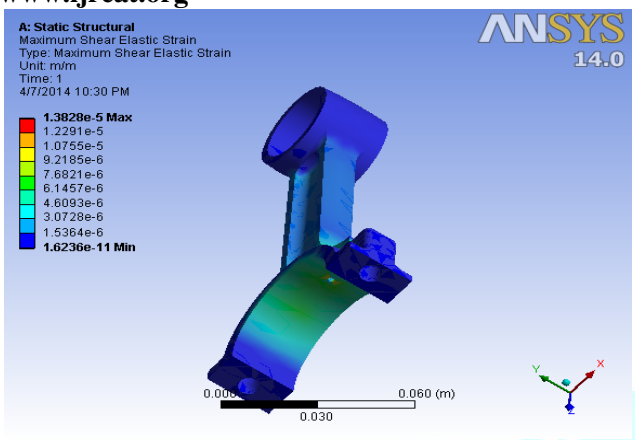

Fig No.4: Maximum Shear Elastic Strain

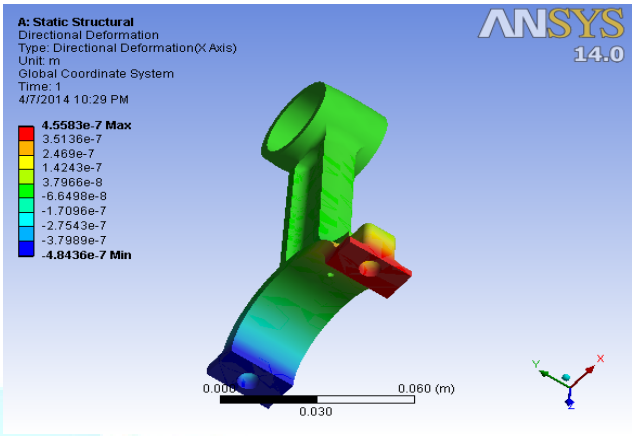

Fig No. 7: Directional Deformation

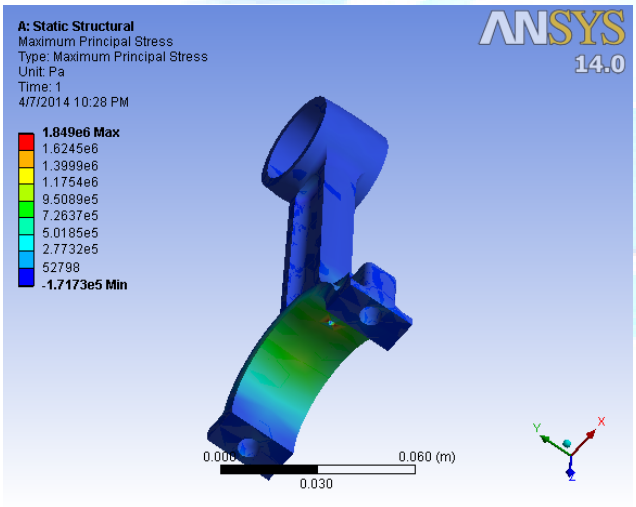

Fig No.5: Maximum Principal Stress

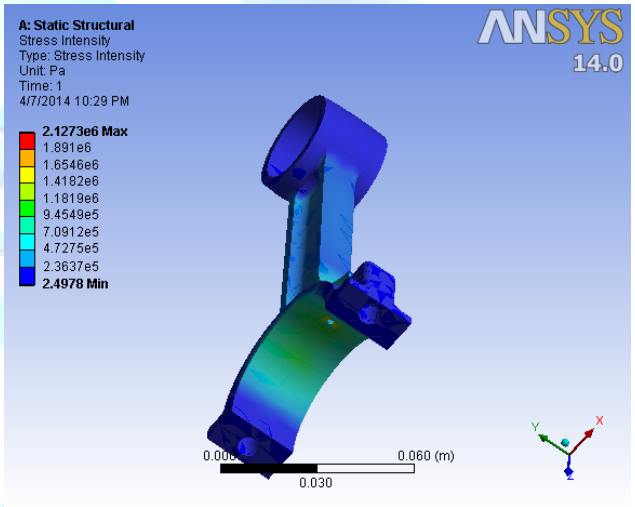

Fig No.8: Stress Intensity

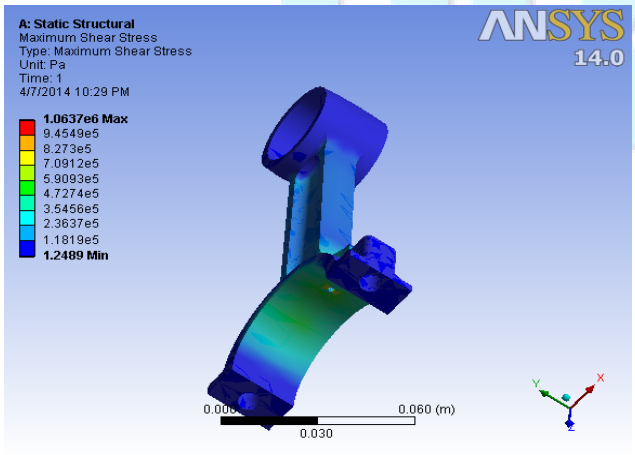

Fig No.6: Maximum Shear Stress

6. FEA Result

S.no	Type Of stress	FEA Analysis
1	Equivalent (Von misses stress)	2.3265e6 (Alloy Steel)
2	Equivalent (Von misses stress)	2.066e6 (Structural steel)
3	Equivalent Elastic Strain	1.046e-5
4	Maximum Principal Stress	1.849e6
5	Maximum shear stress	1.0637e6

7. Conclusions

In this paper , Finite Element analysis of the connecting rod of a Tata light commercial vehicle has been done using FEA tool ANSYS and are tabulated in a table. The following conclusion obtained from this study.

1- For the structural steel factor of safety and stiffness is increase compared to steel alloy.
2-FEA analysis shows the stress induced are lie within permissible limit. So our, this design of connecting rod safe buckling.
3- The material used for connecting rod is steel alloy which is replaced by structural steel .The weight of the connecting rod reduces, and reduces the deflection of connecting rod.
.

Acknowledgments

We gratitude Director Dr.Amar Singh for their valuable regards and highly obliged kindness to HOD ME (Mr. Satish Kumar Diwedi) BIT GIDA GORAKHPUR for his kind support and courage to write a paper.

References

[1]. Tukaram S. Sarkate, Sachin P. Washimark, "Optimization of steel connecting rod using Finite Element Analysis"

[2].Marthanapalli haripriya, K.Manohar Reddy "Materialized optimization of connecting Rod for Four Stroke Single Cylinder Engine" International Journal of Computational Engineering Research||Vol, 03||Issue, 10||

[3].Pravardhan S. Sheony and Ali Fatemi "Connecting Rod Optimization for Weight and Cost Reduction" (The University of Toledo)

[4].G. Naga MalleshwareRao "Design Optimization and Analysis of the Connecting Rod using ANAYS" International Journal of Science and Research (IJSR), India Online ISSN: 2319-7064.

[5].Prof. N. P. Doshi, Prof. N. K. Ingole "Analysis of connecting rod using Analytical and Finite Element Method" International Journal of Modern Engineering Research (IJMER) Vol.3, Issu.1, Jan-Feb. 2013 pp-65-68.

[6].Bin Zheng, Yongqi Liu, and Ruixiang Liu "Stress and Fatigue of Connecting Rod in Light Vehicle Engine" School of Transportation and Vehicle Engineering, Shandong University of Technology, Zibo 255049, Chine The Open Mechanical Engineering Journal, 2013, 7, 14-17.

[7].Vivek C. Pathade, BhumeshwarPatle, Ajay N. Ingal "Stress Analysis of I.C. Engine Connecting Rod by FEM" International Journal of Engineering and Innovation Technology (IJEIT) Volume 1, Issue 3, March 2012.

Books:

[8]Abdulla sheriff, Hand Book of Properties of Engineering Material and Design Data for Machine Element (Tenth Edition, Delhi, DhanpatRai& Co, 2011)

[9].R.S. Khurmi& J.K. Gupta, A Textbook of Machine Design (Multi ColourReviced& Update Edition, New Delhi, S. Chand Publishing, 2011)

[10]. Er. R.K. Rajput, Strength of Material