International Journal of Emerging Engineering Research and Technology
Volume 2, Issue 4, July 2014, PP 242-249
ISSN 2349-4395 (Print) & ISSN 2349-4409 (Online)

Numerical and Experimental Analysis of Connecting Rod

Fanil Desai

PG student, Mechanical Engineering
SITS, Pune, India
fanildesai2212@gmail.com

Kirankumar Jagtap

Department Mechanical Engineering
SITS, Pune, India
krjagtap_sits@sinhgad.edu

Abhijeet Deshpande

Department of Mechanical Engineering
VIIT, Pune, India
abhijit.deshpande32@gmail.com

Abstract: *The main function of connecting rod is to convert linear motion of piston to reciprocating motion of crankshaft. It is the main component of internal combustion (IC) engine. It is the most heavily stressed part if IC engine. During its operation various stresses are acting on connecting rod. The influence of compressive stress is more in connecting rod due to gas pressure and whipping stress. The objective of this paper is to investigate the compressive stress acting on connecting rod at different loading condition. Here two samples of connecting rods are taken for experimental analysis. Static structural analysis using ANSYS and experimental analysis was conducted on connecting rod made up of forged steel. The purpose of this study is to show the performance of connecting under different loading condition. Experimental results will be verified with the numerical results.*

Keywords: *Connecting rod, Internal combustion engine (IC), ANSYS, forged steel, static structural analysis, Universal Testing Machine (UTM), finite element analysis (FEA)*

1. Introduction

Connecting rod is the intermediate link between the piston and the crank. And is responsible to transmit the push and pull from the piston pin to crank pin, thus converting the reciprocating motion of the piston to rotary motion of the crank. For automotive it should be lighter should consume less fuel and at the same time they should provide comfort and safety to passengers, that unfortunately leads to increase in weight of the vehicle. This tendency in vehicle construction led the invention and implementation of quite new materials which are light and meet design requirements [9].

The major stresses induced during its operation are axial and bending stress. The compressive stress has more influence due to gas pressure. The gas pressure is the maximum force acting on connecting rod. Hence calculation of compressive stress is of more importance. So the compressive stress generated in connecting rod is calculated experimentally and will be verified with numerical results. From literatures it is observed that during its operation stress generated at the fillet section of connecting rod are much higher than the other region. P.K. Sharma et al. [1] carried out stress analysis of connecting rod by finite element method using pro-e wild fire 4.0 and ANSYS work bench 11.0 software. And concluded that the stress induced in the small end of the connecting rod are greater than the stresses induced at the bigger end, therefore the chances of failure of the connecting rod may be at the fillet section of both end. Vivek. C. Pathade et al. [2] investigate the stress analysis of connecting rod by finite element method using pro-e wild fire 4.0 and ANSYS work bench 11.0 software. And concluded that the stress induced in the small end of the connecting rod are greater than the stresses induced at the bigger end, therefore the chances of failure of the connecting rod may be at the fillet section of both end. H.B. Ramani et al. [3] performed detailed load analysis of connecting rod., In order to calculate stress in Different part of connecting rod, the total forces exerted connecting rod were calculated. It was observed that maximum stresses in different parts of connecting rod were determined by Analysis. The maximum pressure stress was between pin end and rod linkages and between bearing cup and connecting rod linkage.

2. Experimental Analysis

The experimental analysis was carried out on Universal Testing Machine (UTM). The equipment required to calculate the stress in connecting rod is UTM, Samples of Connecting rod, Design of fixture, Strain Gauge, Load bridge circuit and digital multimeter. Figure 1 and 2 presents the details of experimental setup.

Fig1. *Connecting Rod Sample 1 and 2*

Fig2. *Arrangement of fixture for holding connecting rod*

The strain gauge is an electrical transducer which translates changes in force (or weight) into change in voltage. The change in voltage can be calibrated directly in terms of the force (or load) applied to the cell [7]. The strain gauge is mounted at big and small end of connecting rod. The strain gauge is connected to load bridge circuit also called as Wheatstone bridge circuit. The amplifier is also attached to strain gauge because the deformation produce will give readings in millivolt. So amplifier circuit will convert signals from millivolt to volts. The readings are taken with the help of digital multimeter. The readings are converted to strain. With the known strain we will be able to calculate stress generated in connecting rod.

The forces were applied by UTM and with the help of strain gauge the stresses are calculated both ends of connecting rod. The compressive stresses generated at the two ends of connecting rod are represented by Table 1 and 2.

Table1. *Experimental Results for Compressive Stress Acting at Two Ends of Connecting Rod sample 1*

Force (KN)	Piston End (MPa)	Crank End (MPa)
30	395.59	254.15
35	461.89	302.78
40	530.40	346.97
45	592.28	389.00
50	665.21	429.11

Table2. *Experimental Results for Compressive Stress Acting at Two Ends of Connecting Rod sample 2*

Force (KN)	Piston End (MPa)	Crank End (MPa)
30	461.89	291.72
35	529.19	344.76
40	616.59	400.00
45	685.10	446.42
50	771.29	512.72

3. MODELLING AND FEA ANALYSIS

The first step to start the analysis with the ANSYS programs is to select the type of analysis. The static structural analysis was selected for carrying out detail analysis of connecting rod. The geometry of connecting rod was created in ANSYS workbench by taking the parameter of rod. After creating geometry material properties were applied to connecting rod. Material properties for connecting rod are: Density-8000 kg/m^3, Young's modulus-221 MPa, Poisson's ratio-0.3 and Ultimate strength-827 MPa, Yield strength-625 MPa. Fig. 3 presents the 3-D model of connecting rod.

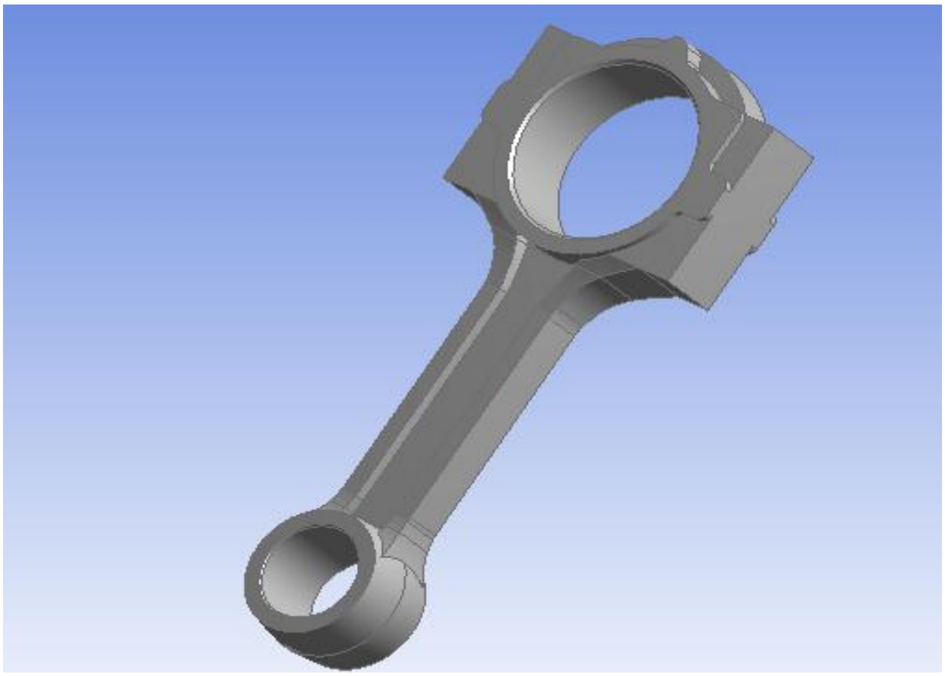

Fig3. *Model of Connecting Rod*

3.1. Meshing

After creating geometry and applying material properties next step in ANSYS is to generate a meshing. The tetrahedron meshing is used for this analysis. The computation time required for meshing is less and the results obtained with the help of tetrahedron meshing are fast and accurate. Total number of elements and nodes are 13425 and 24141 respectively. Fig. 4 shows the meshed model of connecting rod.

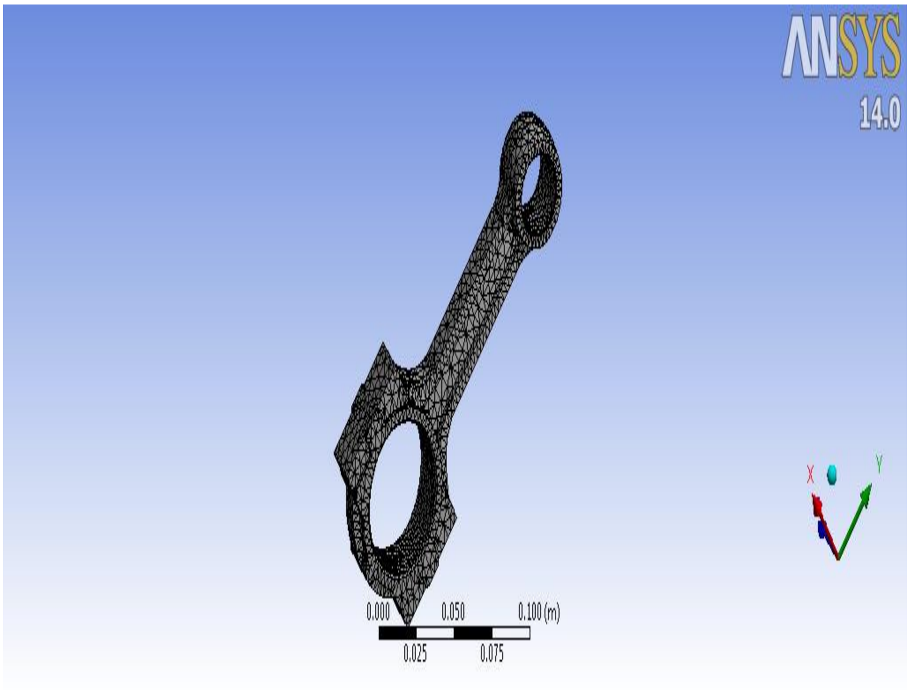

Fig4. *Meshing of Connecting rod*

3.2. Loading and Boundary Condition

After meshing the model, the boundary condition such as loads and constrains are imposed. One of the important factors to get accurate result is to apply correct the loads and the Boundary conditions. There are many ways to apply different loads and constraints to them model for example on nods, on edges, on surfaces or elements. Final stage of analysis is running a solver to get the desired results.

3.3. Results of FEA

In this study of finite element model the force is applied at piston end and the crank end is kept fixed. The force applied at the piston end is ranging from 25KN to 50KN. The Von-mises (compressive) stress is calculated for each different force. The figures 5 to 14 presents the stress generated in connecting rod.

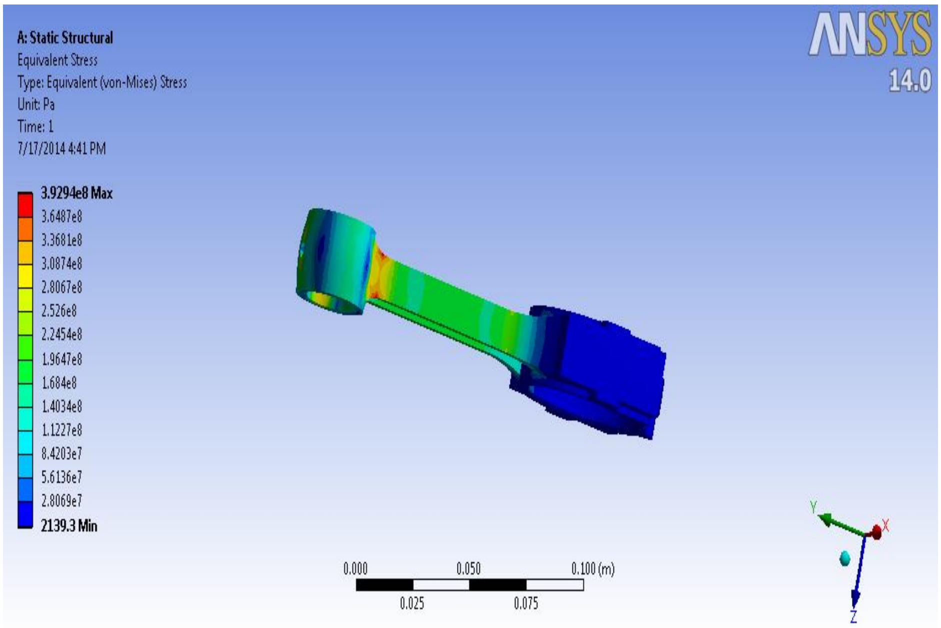

Fig5. *Equivalent (Von- Mises) Compressive Stress for Force Acting at 30KN*

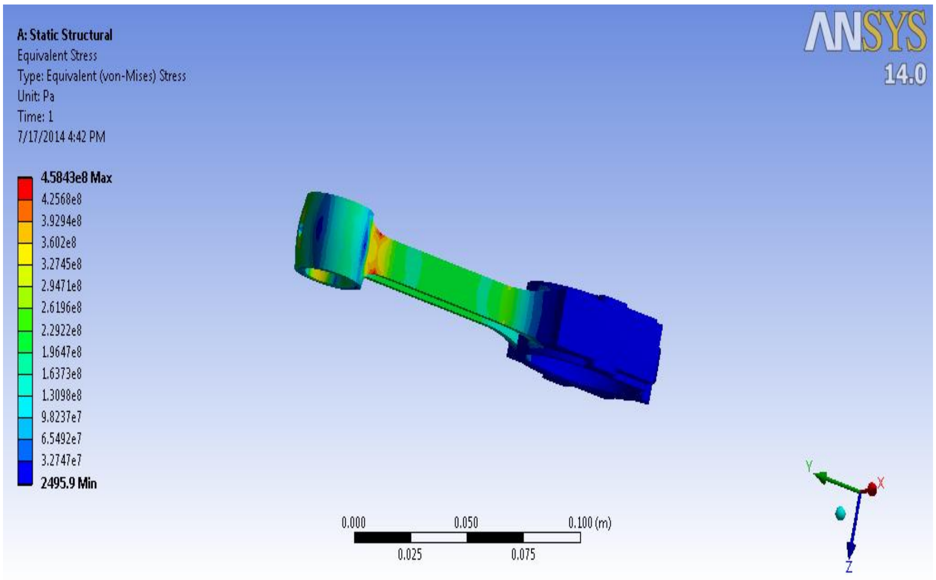

Fig6. *Equivalent (Von- Mises) Compressive Stress for Force Acting at 35KN*

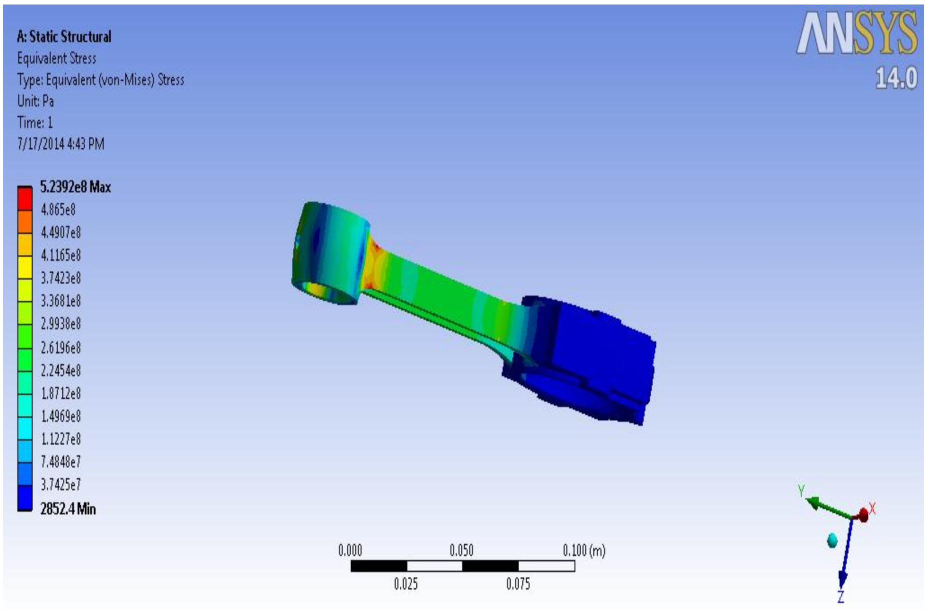

Fig7. *Equivalent (Von- Mises) Compressive Stress for Force Acting at 40KN*

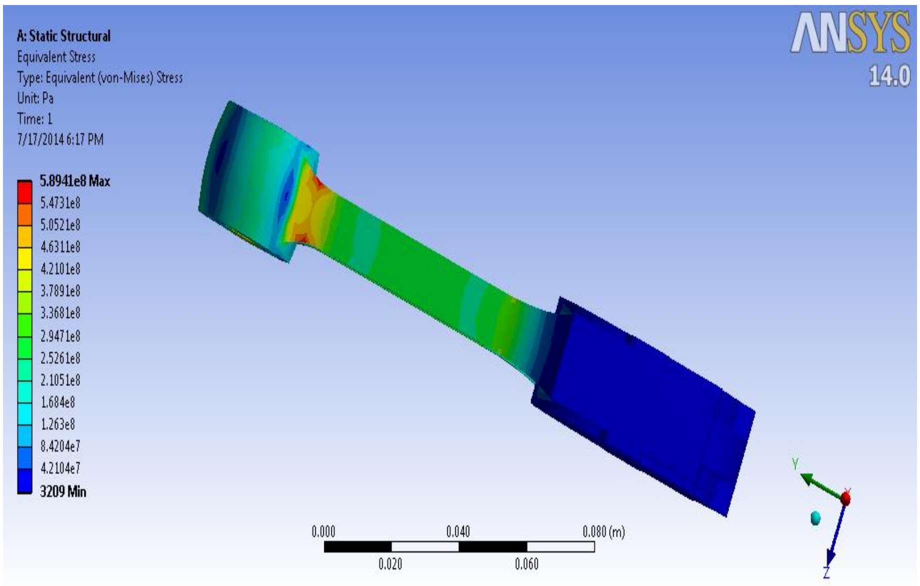

Fig8. *Equivalent (Von- Mises) Compressive Stress for Force Acing at 45KN*

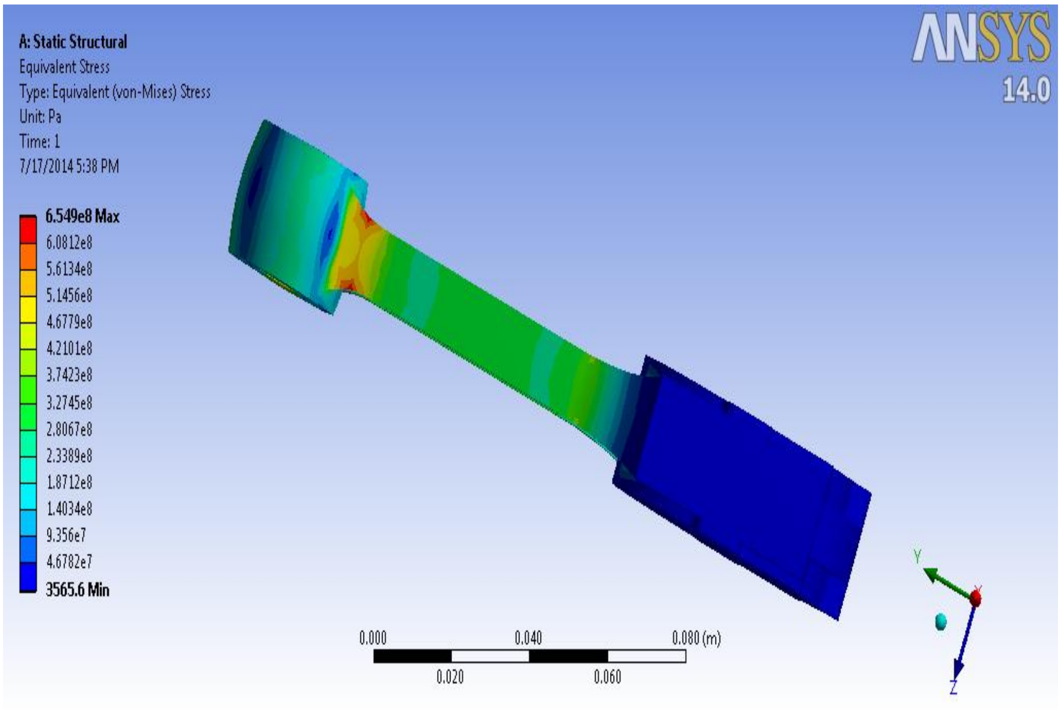

Fig9. *Equivalent (Von- Mises) Compressive Stress for Force Acting at 50KN*

Table3. *FEA result for compressive stress acting at two ends of connecting rod*

Force (KN)	Piston End (MPa)	Crank End (MPa)
30	329.00	252.41
35	458.98	294.48
40	523.43	336.53
45	589.67	378.04
50	654.32	421.16

4. Results and Discussion

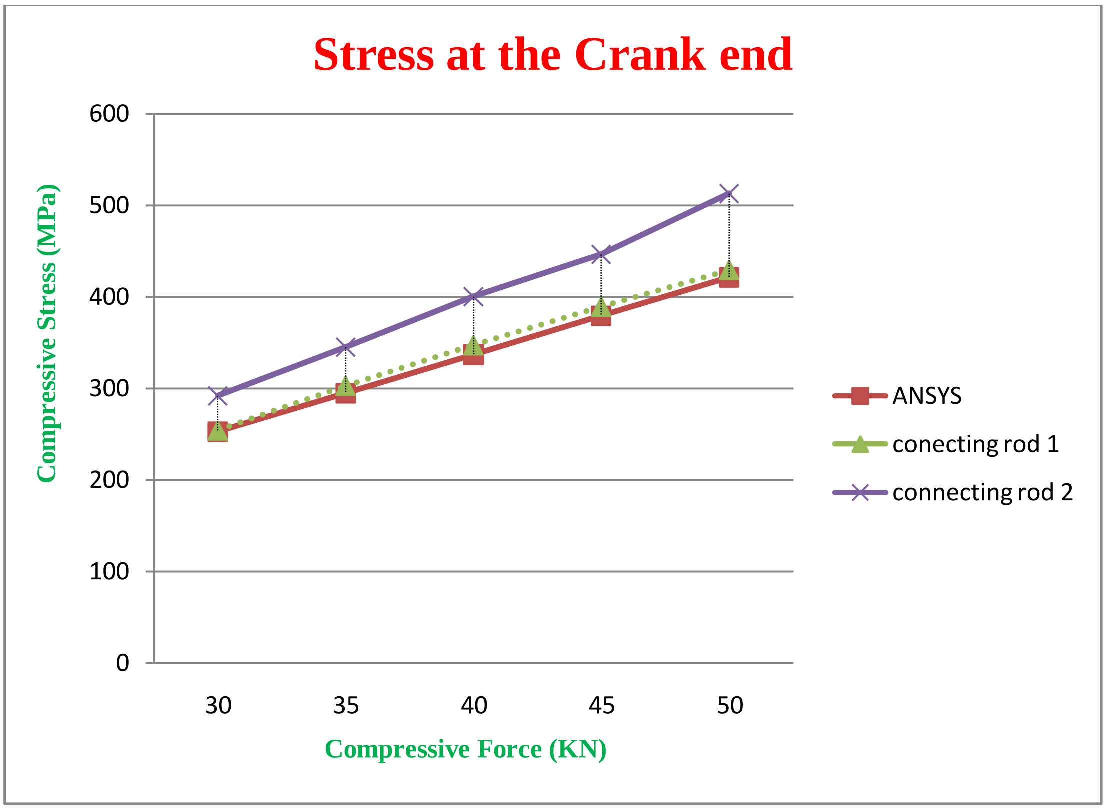

Fig10. *Stress Acting at Crank End of Connecting Rod*

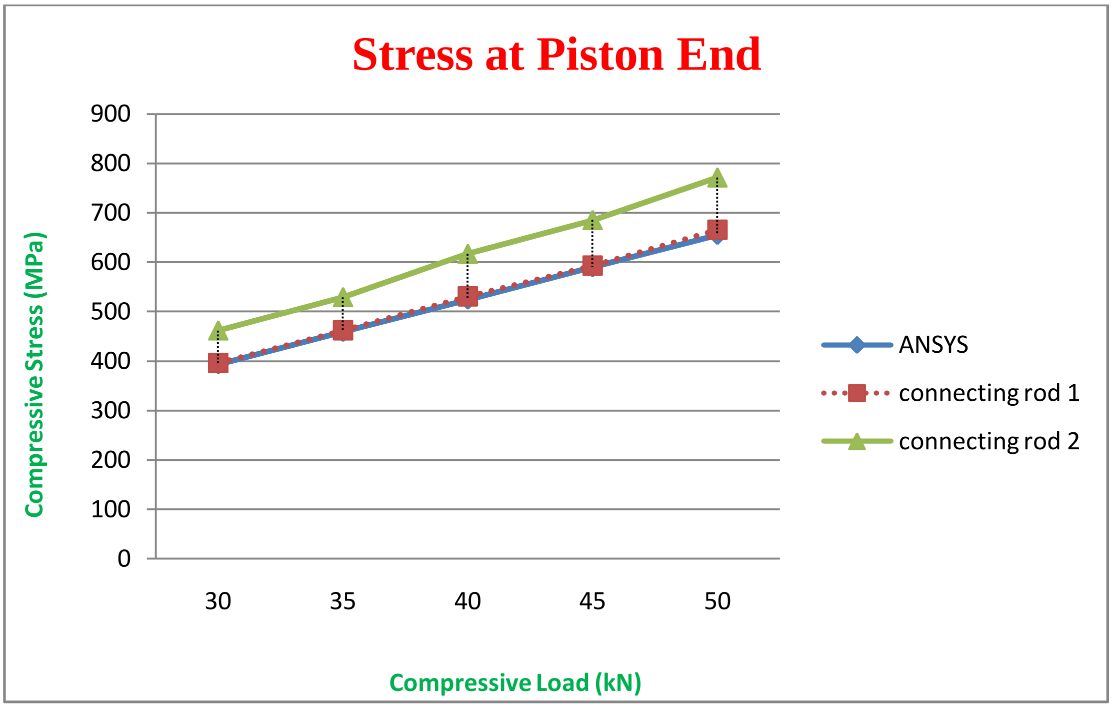

Fig11. *Stress at Piston End of Connecting Rod*

By performing numerical and experimental analysis, results are obtained for stress generated at piston end and the crank end of connecting rod. The result obtained from experimental analysis is compared with the results of numerical analysis and is plotted in the form of graph using table 1, 2 and 3.

The comparison of FEA results and experimental results on application of force from 30KN to 50KN is shown in fig. 10 and 11.

5. Conclusion

By performing numerical and experimental analysis of connecting rod, following conclusion is drawn:

1. For experimental analysis two connecting rods from different manufacturer are used. It is observed from figure 10 and 11 that, the stresses generated in connecting sample 2 is much higher than stresses generated in connecting rod sample 1. This is due to inaccurate chemical composition in connecting rod sample 2.
2. It is also observed from numerical analysis that maximum stress is obtained at big end, small end and shank region. The stress generated at shank is well below allowable limit. Based on experimental and numerical analysis it is also observed that maximum stress is generated at piston end of connecting rod. So the chances of failure are more at the piston end.
3. On comparison of experiment results with numerical results it was found that both the results are closer to each other.

References

[1] 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).

[2] V. C. Pathade, B. Patle, A. N. Ingale, Stress analysis of I.C. Engine connecting rod by FEA, International Journal of engineering and innovative technology 1(2012).

[3] H. B.Ramani, P. M. Kasundra, Analysis of connecting rod under different loading condition using ANSYS software, International journal of engineering research and technology1(2012).

[4] R. S. Khurmi, J. K. Gupta, A text book of Machine Design 14 edition. S. Chand, 1980.

[5] P. S. Shenoy, A. Fatemi, Connecting Rod Optimization for Weight and Cost Reduction, SAE International by Automotive Research Association of India (2014) 1-8.

[6] D. R. Merritt, O. Mian, D. Wang, Connecting Rod Bearing EHD Analysis Including Inertia Effects due to the Distributed Rod Mass, SAE Technical paper (2007)1-7.

[7] S. Singh, Experimental Stress Analysis 3rd edition, Pearson publication, New Delhi, 2009.

[8] B. Kuldeep, L. R. Arun, M. Faheem, Analysis and optimization of connecting rod using ALFASiC composite, International journal of innovative research in science, engineering and technology 6(2013).

[9] V. B. Bhandari, Design of machine element third edition, Tata McGraw Hill education private limited, New Delhi, 2010.

AUTHOR'S BIOGRAPHY

Fanil Desai is currently pursuing M.E in design engineering at SITS, Narhe, Pune affiliated to Pune University. He has completed B.E in Aeronautical engineering from Gujarat Technological University in 2012.

Kirankumar Jagtap is currently working as an Assistant Professor in the Department of Mechanical Engineering, Sinhgad Institute of Technology and Science, Narhe, Pune.

Abhijeet Deshpande is currently working as an Assistant Professor in Vishwakarma Institute of Information Technology, Pune.