Comparative Study of Connecting Rod Using FEA and Experimental Analysis

Fanil Desai[1], Abhijeet Deshpande[2], Kirankumar Jagtap[3]

[1, 3]*Department of Mechanical Engineering, Sinhgad Institute of Technology and Science, Pune, India*
[2]*Department of Mechanical Engineering, VIIT, Pune, India*

Abstract— **The connecting rod is the main component of internal combustion (IC) engine. It is the most heavily stressed part in IC engine. During its operation various stresses are acting on connecting rod. The influence of compressive stress is more on 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. Static structural analysis in ANSYS and experimental analysis was conducted on connecting rod made up of forged steel. Experimental results are verified with numerical results.**

Keywords— Connecting rod, Finite Element Analysis (FEA), ANSYS, Static structural analysis, Universal testing machine (UTM)

I. INTRODUCTION

Connecting rod is the main component of the combustion engines which main purpose are transfer the energy from the pistons to crankshafts and convert the linear, reciprocating motion of a piston into the rotary motion of a crankshaft. Connecting rods must have the highest possible rigidity at the lowest weight. In Automobile internal combustion engine connecting rod is a high volume production component subjected to complex loading.

The major stress induced in connecting rod during its operation is combination of axial and bending stress. The bending stresses are produced due to centrifugal effects, while the axial stresses are produced because of cylinder gas pressure and they are compressive in nature. The inertia force arising due to reciprocating action of connecting rod causes both tensile as well as compressive forces. Therefore, durability of this component is of great importance. From literature survey it is observed that in majority of cases, the failure of connecting rod is due to high stresses generated at the big and small end of connecting rod.

Ram bansal et al [1] performed a dynamic analysis of connecting rod and it was observed that maximum stress is generated at the big and small end of connecting rod. A. Mireheri et al. [2] performed finite element analysis of MF-285 connecting rod. In this study, detailed load analysis was performed on connecting rod. The maximum stress was between pin end and rod linkages and between bearing cup and connecting rod Linkage. The maximum tensile stress was obtained in lower half of pin end and between Pin end and rod linkages. Webster et al. [7] performed three-dimensional finite element analysis (FEA) of a high-speed diesel engine connecting rod. Here they have used maximum compressive load which was measured experimentally, and maximum tensile load which is essentially the inertia load of piston assembly mass in their analysis. Load distributions on the piston pin end and crank end were also determined experimentally.

II. EXPERIMENTAL SETUP FOR ANALYSIS

The experimental analysis was carried out on Universal Testing Machine (UTM). The equipment required to calculate the stress in connecting rod is as under. Fig. 1 and 2 presents the details of experimental setup.

1. Universal Testing Machine – 100kN capacity
2. Connecting rod – 1 sample to be tested
3. Design of fixture – OHNS die steel
4. Strain gauge – 350 ohm
5. Amplifier – mV to V (V-volt)
6. Digital multimeter – for calculating voltage

Fig.1 Schematic Overview of Experimental Setup

Fig. 2 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 [4]. The strain gauge is mounted at big and small end of connecting rod as shown in figure 3.3. 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 excitation voltage is 0 to 10V. There is direct relation between volts and strain with which we are able to calculate the strain in connecting rod.

$$0V \text{ to } 10V - 0 \text{ to } 10{,}000\mu\varepsilon \quad (1)$$

The above equation 1 represents the relation to calculate strain. Now from strain we can calculate stress which is given by,

$$\sigma = \varepsilon E \quad (2)$$

Where, σ – Stress, ε – Strain, E – Modulus of elasticity

The forces were applied by UTM on connecting and following reading were calculated by using equation 1 and 2. The compressive stresses generated at the two ends of connecting rod are represented by Table I.

TABLE I
EXPERIMENTAL RESULTS FOR COMPRESSIVE STRESS ACTING AT TWO ENDS OF CONNECTING ROD

Force (KN)	Piston End (MPa)	Crank End (MPa)
5	61.6	39.6
10	127.73	81.54
15	192.042	123.31
20	260.78	165.75
25	324.87	207.74

III. MODELING AND FEA OF CONNECTING ROD

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.

Fig. 3 3D model of connecting rod

A. 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.

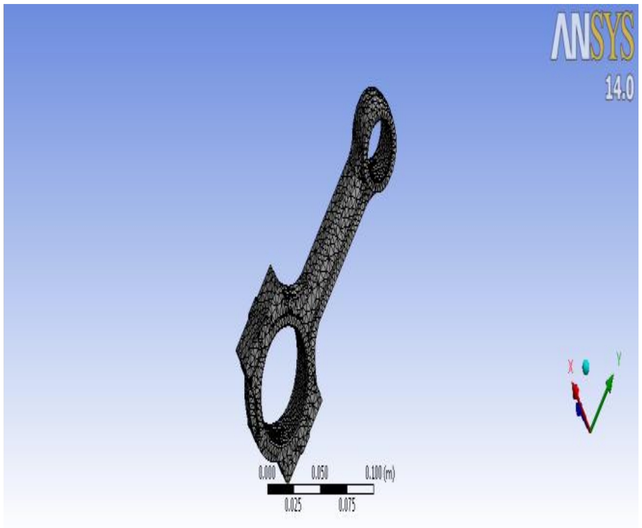

Fig. 4 Meshing of Connecting Rod

B. 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 equation.

C. Results of Finite Element Analysis

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 5KN to 25KN. The Von-mises (compressive) stress is calculated for each different force. The figures 5 to 14 presents the stress generated in connecting rod.

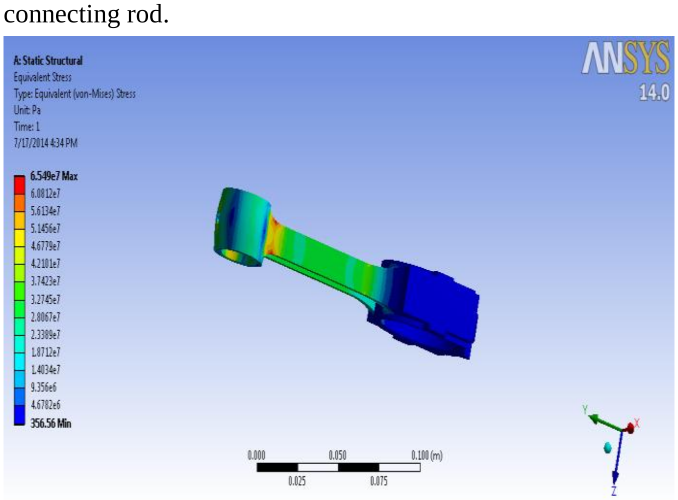

Fig. 5 Equivalent Von-mises (compressive) stress analyzed for 5KN force

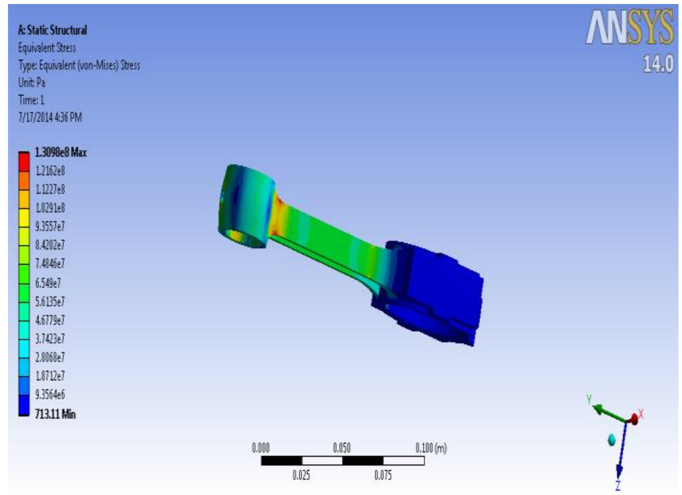

Fig. 6 Equivalent von-mises (compressive) stress analyzed for 10KN force

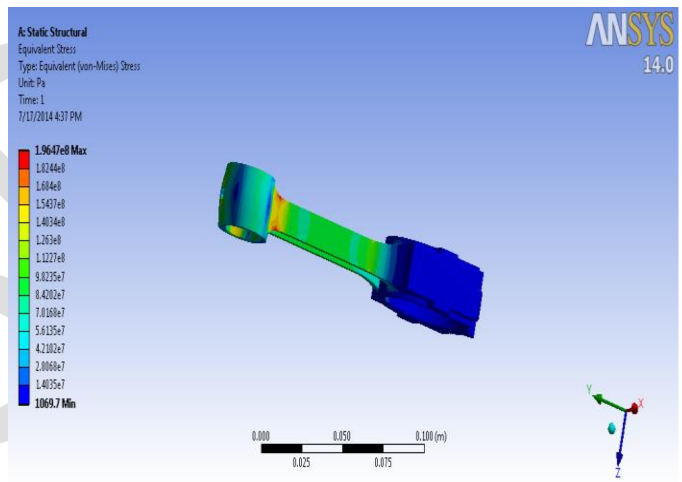

Fig. 7 Equivalent (Von- Mises) Compressive Stress analyzed for 15KN force

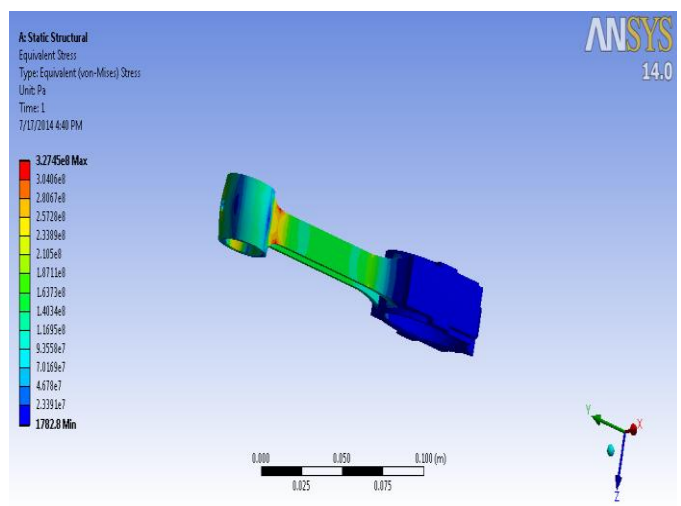

Fig. 8 Equivalent (Von- Mises) Compressive Stress analyzed for 20KN force

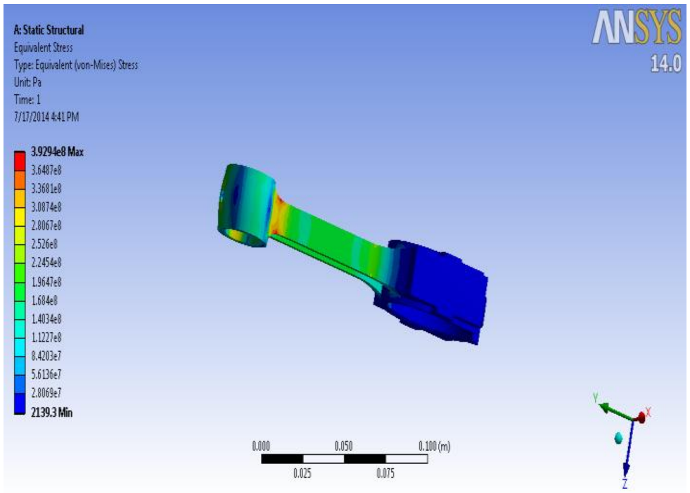

Fig. 9 Equivalent (Von- Mises) Compressive Stress analyzed for 25KN force

TABLE II
FEA RESULTS FOR COMPRESSIVE STRESS AT TWO ENDS OF CONNECTING ROD

Force (KN)	Piston End (MPa)	Crank End (MPa)
5	65.49	42.10
10	130.98	89.20
15	196.47	126.35
20	261.96	168.45
25	327.45	210.54

IV. RESULTS AND DISCUSSIONS

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 I and II.

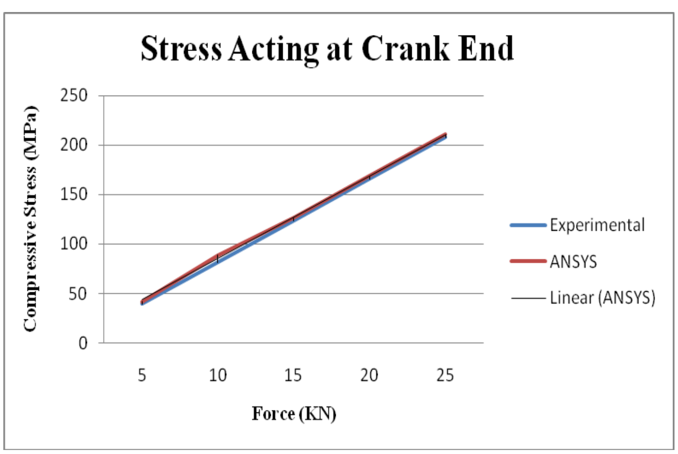

Fig. 10 Stress acting at the crank end of connecting rod

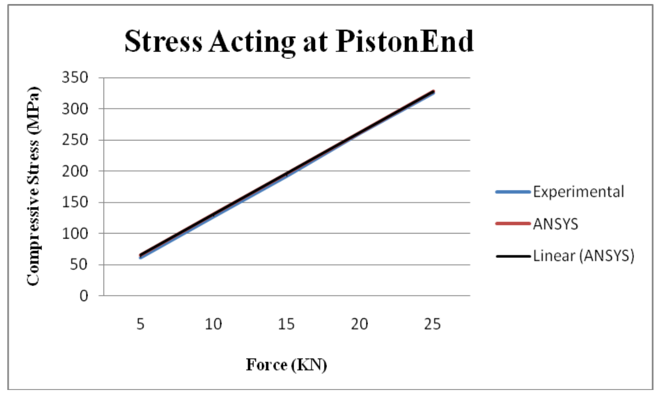

Fig. 11 Stress acting at the crank end of connecting rod

The comparison of FEA results and experimental results on application of force from 5KN to 25KN is shown in fig. 10 and 11.

CONCLUSION

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

1. It is 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.
2. On comparison of experiment results with numerical results it is found that both the results are closer to each other.

REFERENCES

[1] R. Bansal, Dynamic simulation of a connecting rod made of aluminium alloy using finite element analysis approach, ISOR Journal of mechanical and civil engineering 5(2013) 01-05.
[2] A. Mireheri, M. H. Zadeh, A. Jafari, M. Omid, Fatigue analysis of connecting rod of universal tractor through Finite element method (ANSYS), Journal of agricultural technology 1(2008) 21-27.
[3] 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).
[4] R. Djugum, specifications of strain gauges: fabrication processes and techniques.
[5] R. S. Khurmi, J. K. Gupta, A text book of Machine Design 14 edition. S. Chand, 1980.
[6] V. B. Bhandari, Design of machine element third edition, Tata McGraw Hill education private limited, New Delhi.
[7] Webster, W. D., Coffell R., and Alfaro D., 1983, “A Three Dimensional Finite Element Analysis of a High Speed Diesel Engine Connecting Rod,” SAE Technical Paper Series, Paper No. 831322.
[8] Rabb, R., 1996, “Fatigue Failure of a Connecting Rod”, Engineering Failure Analysis, Vol. 3, No. 1, PP. 13-28.

Biographies of Author

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

Abhijeet Deshpande is presently working as an Assistant Professor in VIIT, Pune. He is also pursuing Ph.D. from BVDU, Pune.