International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 3 Issue 2, February - 2014

Stress Analysis of Connecting Rod: A Review

[1]Fanil Desai, [2]Kirankumar Jagtap, [3]Abhijeet Deshpande
[1]PG Student, Mechanical engineering, SITS, Pune.
[2]Department of mechanical engineering, SITS, Pune.
[3]Department of mechanical engineering, VIIT, Pune.

Abstract - The main objective of this review is to study the various stresses acting on connecting rod during its operation. The stresses acting on connecting rod are axial and bending stresses.it is one of the most heavily stressed components. The main function of connecting rod is to convert the linear motion of piston to rotary motion of crankshaft. Because of high speed, the stress generated in conrod increases, if this stress exceeds the designed parameter failure of connecting rod takes place. The review introduces different methods such as theoretical, FEA and experimentally with which stress can be calculated.

Keywords: connecting rod, FEA, stress analysis

I. INTRODUCTION

The connecting rod is connection between the crankshaft and the piston. The basic function of the connecting rod is to convert the linear motion of the piston to rotary motion of the crankshaft. It consists of big end at the crankshaft, small end at piston and a long shank. Another function of connecting rod is to transfer lubrication oil from crank pin to piston pin.

The connecting rod is subjected to a complex state of 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.

The length of connecting rod is of great importance. When the connecting rod is short compared to crank radius, it possesses greater angular swing which results in greater side thrust on piston. For high speed vehicles the length of connecting rod to crank radius is generally 4 or less. In low speed, the ratio varies from 4 to 5.

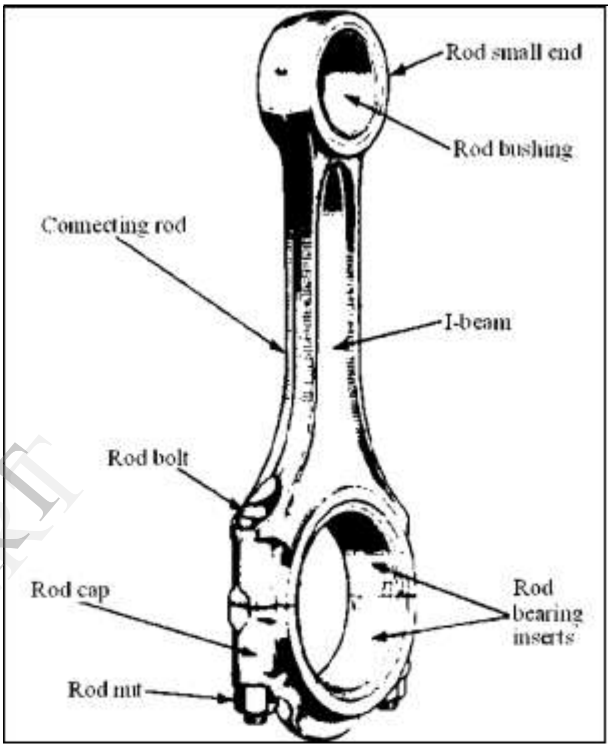

Figure 1 connecting rod [6]

Nowadays, engines require higher speed and power need of connecting rod with higher strength and stiffness, but must be lighter in weight and size. The lighter the connecting rod will results in greater power and less vibration because of the less reciprocating weight.

For automobile application connecting rod are manufactured by forging from either steel or powered metal. Every process has its own pros and cons. But for general automobile application metal formed by steel forging is preferred as the material is inexpensive and cost effective.

II. LITERATURE SURVEY

Many researches have been done on connecting rod but still a lot to be done. Different experiments were carried by researchers regarding reducing stress, weight, and selection of material.

2.1. Material selection

Material selection plays an important role in designing of connecting rod which will withstand the complex state of loading. Materials used for manufacturing of connecting rod is either carbon steels or alloy steels. Generally carbon steel are used for manufacturing of connecting rod and in recent days aluminium alloys are finding its application in connecting rod because they are lighter which causes overall increase in engine performance. Alloy steel includes nickel chromium or chromium molybdenum.

Lela Krishna vegi et al. [1] compared the material properties of connecting rod made up of forged steel and carbon steel. A parametric model of Connecting rod is modelled using CATIA V5 R19 software and to that model, analysis is carried out by using ANSYS 13.0 Software. Finite element analysis of connecting rod is done by considering the materials, forged steel. He used existing data of connecting rod made of carbon steel. On comparison following results were obtained.

1. Stress for both materials is same.
2. Factor of safety and stiffness for forged steel increased compared to carbon steel.
3. Also no. of cycles for forged steel (8500×103) is more than the existing connecting rod (6255× 103).
4. When compared to both of the materials, forged steel is cheaper than the existing connecting rod material.

Kuldeep B et al. [2] performed a finite element analysis of connecting rod made up of composite named ALFASiC and compared with the existing connecting rod made up of Al360. The optimized connecting rod was 43.48% lighter than current connecting rod. He also found out that the new optimized connecting rod was 55.76% much stiffer than current connecting rod. Weight was also reduced by changing the material properties.

K. Sudershan kumar et al. [3] described modelling and analysis of Connecting rod. In his project carbon steel connecting rod is replaced by aluminium boron carbide connecting rod. Aluminium boron carbide is found to have working factory of safety is nearer to theoretical factory of safety, to increase the stiffness by 48.55% and to reduce stress by 10.35%.

Pushpendra kumar Sharma et al. [4] performed an experiment for weight reduction. He found out that weight can be reduced changing the material of the current forged steel connecting rod to crack able forged steel (C70), with the help of software detailed stress distribution in whole connecting rod was observed which gives information regarding which parts to be hardened or given attention during manufacturing.

2.2. Stress in connecting rod

It is one of the most heavily stressed components. Various stresses are acting on connecting rod (Axial and Bending force) which can be calculated with the help of different methods developed such theoretical, numerical and experimentally. For complex problem numerical analysis are preferred. For numerical analysis different software are preferred like ANSYS, ADAMS, NASTRAN, FORTRAN etc. the numerical analysis are cheaper and less time consuming than experimental analysis. Many researchers found out, the result obtained with this software is much reliable. Schematic representation on conrod failure is shown in figure 2

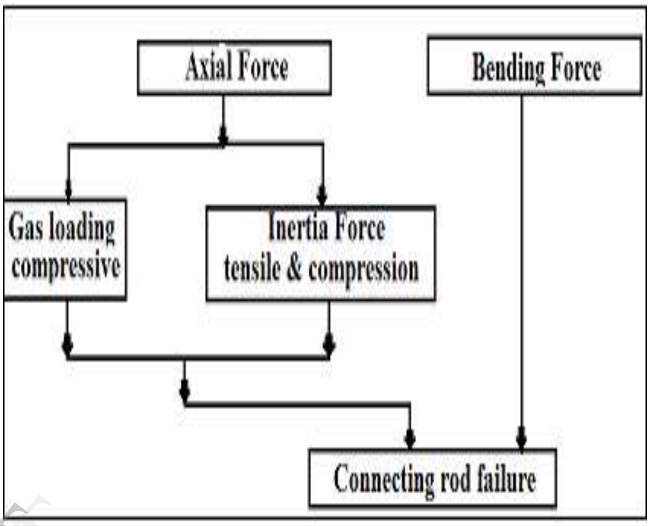

Figure 2 schematic diagram of connecting rod failure

Vivek C Pathade et al. [6] he dealt with 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.

B. Anusha et al. [12] in his paper a static analysis is conducted on a connecting rod of a single cylinder 4-stroke petrol engine. The model is developed using Solid modelling software i.e. PRO/E (creo-parametric). Further finite element analysis is done to determine the von-misses stresses shear stress and strains for the given loading conditions. He concluded that stress is maximum at piston end.

Mr. H. B. Ramani et al. [9] performed analysis of connecting rod in ANSYS. He found out that the maximum stress is between pin-end and rod-linkage, 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 linkage. He also concluded that results obtained by FEM method and experimental equations were similar (Maximum difference was only ±13%) this shows accuracy of our modelling, meshing and loading.

Ram bansal et al. [5] performed dynamic analysis of connecting rod made up of aluminium alloy using FEA. The modelled was drawn in CATIA software and imported to

ANSYS to perform dynamic simulation of in service connecting rod. From the result it was concluded that the maximum deformation appears at the centre of big end & small end bearings inner fibre surface. The areas subjected to crushing due to crank shaft & gudgeon pin is shown through analysis after implementing boundary conditions. Hence this areas are prone to failure. Further we can predict the mutual interference between the connecting rod and other parts.

M Rasekh et al. [7] performed finite element analysis of MF-285 connecting rod was carried out in ANSYS v9. In his study, detailed load analysis was performed on conrod. 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.

Anil Kumar et al. [8] presented the optimization of connecting rod parameters using CAE tools. The model was developed in pro/E Wildfire 5.0 and then imported in ANSYS Workbench. From the result he found out that maximum stress is at piston end, which can be reduced by increasing the material at that section.

2.3. Experimental Analysis

Stress induced in connecting rod can also be determined experimentally. Though experimental analysis are time consuming and costly but still more reliable. You are able to observe that how the object is going to behave on applying the loading condition. Basically software data can be validated with the help of experimental analysis. Hence many researchers performs experimental analysis were they are able to compare the numerical analysis with experimental analysis. Experimental analysis of connecting rod is performed in UTM (Universal Testing Machine).

Webster et al. [11] 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.

Priyank D. Toliya et al. [10] performed a FE analysis and experimental analysis of FM-70 connecting rod. Different parameter like von misses stress, total deformation, and elastic strains were calculated. From the results it was concluded that maximum stress is at pin end, crank end and shank region. Based on experiments it was observed that failure of connecting rod was due to fatigue crack growth mechanism which came as a result of higher stress combined with the porosity in initiation and growth of fatigue crack followed by catastrophic failure. Experimental results were nearer to analysis result. He also compared the results of experimental and Ansys analysis. The results obtained for fatigue life of three specimen are 3.124 x 10^5, 2.984 x 10^5 and 3.096 x 10^5 respectively, where Ansys software result was 3.0817 x 10^5. So here Ansys analysis is nearly equal to experimental analysis

III. DISCUSSION

The connecting rod is a basic element of IC engine, which performs the basic function of converting linear motion of piston into rotary motion of crankshaft. The main objective of this review is to study various stresses acting on connecting during its operation. From the discussion it is seen that maximum stress is generated at the big and small end of connecting rod. It is also seen from the experiments that material can be removed from minimum stressed parts and it can be added where the stress is maximum. The paper also suggests that optimization of connecting rod can reduced the weight as well as cost of the connecting rod. It also suggests that FEA is one of the best tools for calculating the stress in connecting rod. Different materials are tested for maximum stiffness, maximum stress and less weight using finite element analysis.

REFERENCES

[1] Leela Krishna vegi, venu gopal vegi, “Design and Analysis of connecting rod using Forged Steel”, International Journal of Scientific & Engineering Research, Volume 4, Issue 6, June-2013, Vol. 32, 2002.

[2] Kuldeep B, Arun L.R, Mohammed Faheem, “Analysis and Optimization of Connecting rod using ALFASiC Composites”, International Journal of Innovative Research in Science, Engineering and Technology, Vol. 2, Issue 6, June 2013.

[3] K. Sudershan Kumar, Dr. K. Tirupathi Reddy, Syed Altaf Hussan, “Modelling and analysis of two Wheeler connecting rod”, International Journal of Modern Engineering Research, Vol -2, Issue- 5, pp-3367-3371, Sep-Oct-2012.

[4] Pushpendra Kumar Sharma1, Borse Rajendra R, “Fatigue analysis and Optimisation of Connecting rod using finite element analysis”, International Journal Of advance research in Science and Engineering, Vol. No.1, Issue No. I, pp-3367-3371, September 2012.

[5] Ram Bansal, “Dynamic simulation of connecting rod made of aluminium alloy using finite element analysis approach”, IOSR Journal of Mechanical and Civil Engineering, Volume 5, Issue 2, PP 01-05, Jan. - Feb. 2013.

[6] Vivek.c.pathade, Bhumeshwar Patle, Ajay N. Ingale, “Stress Analysis of I.C. Engine Connecting Rod by FEM”, International Journal of Engineering and Innovative Technology, Vol-1, Issue-3, pp-12-15, March2012.

[7] M. Rasekh, M. R. Asadi, A. Jafari, K. Kheiralipour, “Obtaining Maximum Stresses in Different Parts of Tractor (Mf-285) Connecting Rods Using Finite Element Method”, Australian Journal of Basic and Applied Sciences, Vol.3, pp 1438-1449.

[8] [8] Anil Kumar, Kamaldeep Grover, Balvinder budina, “Optimization of Connecting rod parameters using CAE tools”, International Journal of Latest Trends in Engineering and Technology, Vol- 1, Issue- 3 , Sep2012.

[9] Mr. H. B. Ramani, Mr. Neeraj Kumar, Mr. P. M. Kasundra,“ Analysis of Connecting Rod under Different Loading Condition Using Ansys Software”, International Journal of Engineering Research & Technology (IJERT) Vol. 1 Issue 9, November- 2012.

[10] Priyank D. Toliya, Ravi C. Trivedi, Prof. Nikhil J. Chotai, “Design and Finite Element Analysis of Aluminium-6351

Connecting Rod", International Journal of Engineering Research & Technology (IJERT), Vol. 2 Issue 5, May – 2013.

[11] Webster, W. D., Coffell R., and Alfaro D, "Design and Finite Element Analysis Of Aluminium-6351 Connecting Rod", A Three Dimensional Finite Element Analysis of a High Speed Diesel Engine Connecting Rod, SAE Technical Paper Series, Paper No. 831322.

[12] B. Anusha, C.Vijaya Bhaskar Reddy, "Modelling and Analysis of Two Wheeler Connecting Rod by Using Ansys", IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE), Volume 6, Issue 5 (May. - Jun. 2013).

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.

Professor Kirankumar Jagtap is currently working as the head of department of Mechanical engineering in SITS, Narhe, Pune. He is also pursuing Ph.D. from SVNIT, Surat, Gujarat.

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