International Journal of Engineering Technology, Management and Applied Sciences

www.ijetmas.com February 2018, Volume 6, Issue 2, ISSN 2349-4476

Design, Analysis & Optimization of Connecting Rod – Literature Review

Aniruddha A. Bharamgunde

PG Student, M.E. Mechanical (CADME), MET's IOE BKC Nashik, Maharashtra, India

S.K. Dahake

Guide and Professor, M.E Mechanical, IOE BKC Nashik, Maharashtra, India

ABSTRACT

Each n everyday consumers are looking for the best from the best. Optimization of the component is to make the less time to produce the product that is stronger, lighter and less cost. The design and weight of the connecting rod influence on performance. Hence, it is effect on the manufacture credibility. That's why the optimization is really important in industry. This paper reviews the design, analysis & optimization methods used for the connecting rod by various researchers. These are stated in the following section. It is basically an assessment of the present state of art of the wide and complex field of design, analysis and optimization of connecting rod. The application of the various methods and software used by the researchers are described. Finally the results obtained for respective research are stated. Overall this literature review presents a complete overview of connecting rod optimization using various techniques and there outcomes.

***Keywords:* Connecting rod, design, analysis, optimization, FEA, anasys, photoelastic analysis.**

1. INTRODUCTION

The function of connecting rod in compressors is to transmit the power from crankshaft to the piston to compress the air and in engine to transmit the power from piston to the Crankshaft. The role of connecting rod is the conversion of rotary motion into reciprocating motion or vice-versa. The lighter connecting rod and the piston, greater resulting power and less the vibration because of the reciprocating weight is less. There are different types of materials and production methods used in the creation of connecting rods. The most common types of Connecting rods material are steel and aluminum. The most common types of manufacturing processes are casting, forging and powdered metallurgy. A connecting rod consists of a pin-end, a shank section, and a crank-end. Pin-end and crank-end pinholes at the upper and lower ends are machined to permit accurate fitting of bearings. The tensile and compressive stresses are produced due to pressure, and bending stresses are produced due to centrifugal effect & eccentricity. So the connecting rods are designed generally of I-section to provide maximum rigidity with minimum weight.

Figure: Connecting Rod

2. LITERATURE REVIEW

Review of Research Papers

Nagaraju K L (2016) [1] In his thesis, a connecting rod is demonstrated utilizing Catia v5, discretization utilizing HyperMesh and analysis utilizing Nastran. The outcome predicts the most extreme buckling load and basic locale on the interfacing pole. It is imperative to find the basic territory of concentrated stress for fitting adjustments. He discovered the stresses created in interfacing pole under static loading with various stacking states of compression and tension at crank end and pin end of connecting rod. The displacement plot shows a very small value which does not affect the performance of the connecting rod. The linear static analysis of the connecting rod shows that the stress generated in the model is within the acceptable limits or maximum allowable stress. The buckling mode analysis gives the buckling factor greater than 1 and hence it can be concluded that the connecting rod can withstand the load applied.

Akbar H Khan. (2017) [2] studied existing connecting rod is manufactured by using steel 16MnCr5. His paper describes Design, modeling and analysis of connecting rod. In his work connecting rod is replaced by steel alloy SAE 8620 and Aluminum alloy 360 for Discover 100cc motorbike. A 2D drawing is drafted from the calculations. A parametric model of connecting rod is modeled using Creo 2.0 software. Analysis is carried out by using Ansys 15.0 software. By comparing the von mises stresses in the materials of connecting rod he concluded that stresses occurs in the aluminium alloy 360 connecting rod are very less as compared to the steel 16mnCr5 and steel alloy SAE 8620. Instead of using the material Steel 16mnCr5 we can use the either aluminium alloy 360 or steel alloy SAE 8620 to reduce the weight and cost of the material and for better stiffness.

Mohammed Mohsin Ali Ha (2015) [3] modeled Connecting rod using CATIA software and FE analysis is carried out using ANSYS Software. Load distribution plays important role in fatigue life of the structure. Bush failure changes the loading direction and distribution. His study is concentrated around the fatigue life due to concentrated load and cosine type load distribution on the bigger end. The connecting rod analysis is carried out to check the fatigue life and alternating stress development due to service and assembly loads with variation in load distribution. The results are summarized as follows, Initially the connecting rod is built to the actual dimensions using Catia software. Axi-symmetric analysis is carried out to find interference effect on the stress behavior in the joint. 8 noded plane82 element with quadratic displacement variation is used for accurate results. The contact pair is created with Target69 and Contact72 elements. Interference is created through geometric built up. The result shows contact pressure development at the interface and higher compressive stress in the bush and tensile stress development in the small end. The results are plotted for radial, hoop and vonmises stresses. Also a three dimensional views are obtained through ansys axisymmetric options.

Akbar H Khan (2017) [4] research work investigated Static structural and experimental stress analysis of two wheeler connecting rod using by theoretically, Finite element analysis and using Photoelasticity method. Connecting rod of two wheeler 100 cc petrol engine is taken for the analysis, Finite element analysis includes the Design and modeling of connecting rod using Creo 2.0 and Ansys 15.0 for the Static Structural analysis. Photoelasticity analysis method includes the casting of Photoelastic sheet using Araldite AY 103 and Hardener HY 991 and then connecting rod model is prepared by laser cutting machine. In his research paper static structural analysis is carried out to find the von mises stresses and Stress analysis is carried out to find maximum principle stress and reason behind the failure of connecting rod. Conclusion drawn from his study, It is been observed that the maximum stresses are induces at the fillet section of both ends of the connecting rod and chances of the failure of the connecting rod is found at the fillet sections of both ends of connecting rod. Therefore, to avoid that stresses and failure material need to be added at the fillet sections of connecting rod. By observing the different fringes developed in the connecting rod specimen and by calculating the maximum principle stress at that section we can say that the stresses induced in the small end of the connecting rod are greater than the stresses induced at the big end. Form the Photoelasticity analysis it is found that the stress concentration effect exist at both small end and big end and it is negligible in the middle portion of the connecting rod. Therefore, the chances of failure of the connecting rod may be at fillet section of both ends.

Mohd Nawajish, Mohd Niamuddin, Mayank (2015) [5] studied connecting rods are made up of carbon steel and nowadays aluminum alloys are also used for manufacturing the connecting rods. In this work a comparison study is made between the results of different materials for connecting rod. And the 3D modeling and FEA analysis was carried out by considering materials Al360, beryllium alloy25, Titanium alloy Ti-13v-11Cr-3Al. They developed solid 3D model of Connecting rod using SOLIDWORKS software and an analysis was also carried out by using SOLIDWORKS Software and useful factors like von mises stress, von mises strain and displacement were obtained. Results and comparison it can be noticed that Titanium alloy (Ti-13v-11Cr-3Al) may be used for connecting rod. It is a suitable material for manufacturing of connecting rod. In their work it is noteworthy that the economic consideration is not incorporated. From their study it may be concluded that (1) Maximum von mises stress is developed at small end and minimum von mises stress at big end of the connecting rod. (2) Maximum von mises stress is minimum in connecting rod of Titanium alloy. (3) Connecting rod of Titanium alloy is safer than AL360 and Alloy 25 based on the yield strength.

Suraj Pal (2012) [6] taken single cylinder four stroke petrol engines for the study of finite element analysis of connecting rod. Model is developed using cad software Pro/E Wildfire 4.0, static analysis is done to determine the von Misses stress, shear stress, elastic strain, total deformation in the connecting rod for the given loading conditions using finite element analysis software ANSYS v12. After that the work is carried out for safe design. Based on the observations of the static FEA and the load analysis results, the load for the optimization study was selected. The results were also used to determine of various stress and the fatigue model to be used for analyzing the fatigue strength. Outputs of the fatigue analysis of include fatigue life, damage, factor of safety, stress biaxiality indication. Then results in ANSYS are compared with the results of existing design in the reference paper. The results obtained are well in agreement with the similar available existing results. The model he presented, is well safe and under permissible limit of stresses. Conclusion is based on his work that the design parameter of connecting rod with modification gives sufficient improvement in the existing results. The weight of the connecting rod is also reduced by 0.477g. Thereby, reduces the inertia force. Fatigue strength is the most important driving factor for the design of connecting rod and it is found that the fatigue results are in good agreement with the existing result. The stress is found maximum at the piston end so the material is increased in the stressed portion to reduce stress.

Ram Bansal (2013) [7] conducted a Dynamic simulation was on a connecting rod made of Aluminium Alloy using finite element analysis. The connecting rod is one of the important part of an engine. Connecting rod of the single cylinder four stroke diesel engine is used. After measurements were taken, connecting rods were modeled using CATIA software and saved in 'IGES' format. Then, the model of connecting rod (IGES format) imported into ANSYS software. In his analysis of connecting rod was performed under Dynamic load for Stress analysis, and optimization. The pressure-volume diagram was used to calculate the load boundary condition in dynamic simulation model, and other simulation inputs were taken from the engine Specification chart. The data obtained at engine run were plotted on graph by Enginesoft Software. The maximum deformation, maximum stress point and dangerous areas are found by the stress analysis of connecting rod. This analysis uses a different mesh to get more precise results. The relationship between the stress and the nodal displacement is explained by the modal analysis of connecting rod. The results would provide a valuable theoretical foundation for the optimization and improvement of connecting rod. Dynamic load analysis was performed to determine the in service loading of the connecting rod and FEA was conducted to find stresses at critical locations. The maximum deformation appears at the center of big end & small end bearings inner fiber surface. The areas subjected to crushing due to crank shaft & gudgeon pin is shown through analysis after implementing boundary conditions. The connecting rod deformation was mainly bending due to buckling under the critical loading. And the maximum deformation was located due to crush & shear failure of the big & small end bearings. So these areas prone to appear the fatigue crack. Base on the results, we can forecast the possibility of mutual interference between the connecting rod and other parts. The results provide a theoretical basis to optimize the design and fatigue life calculation.

C.Juarez, F.Rumiche, A.Rozas, J.Cuisano, P.Lean (2016) [8] presents the results of a failure analysis investigation conducted on a connecting rod from a diesel engine used in the generation of electrical energy. The investigation included an extensive analysis of the con-rod material as well as the fracture zone. The investigation involved the following experimental procedures and testing techniques: visual inspection, fractography, magnetic particle inspection, chemical analysis, tensile and hardness testing, metallography, and microanalysis. The connecting rod was fabricated from an AISI/SAE 4140 low alloy steel; chemical composition, mechanical properties and microstructure were appropriate for the application. The connecting rod fractured at the body in a section close to the head; the origin of the fracture was located at the con-rod lubrication channel. The lubrication channel exhibited an area containing a tungsten based material, presumably from a machining tool, embedded in its surface as a result of a deficient manufacturing process. This area acted as nucleation site for cracks that propagate through the connecting rod section by a fatigue mechanism, reducing its section and finally producing its catastrophic failure. Failure occurred at the body of the con-rod, close to the head, and involved the propagation by a fatigue mechanism of cracks nucleated at the lubrication channel. A layer of a tungsten based material embedded in a portion of the lubrication channel served as the area for crack nucleation. The W layer was possibly generated during the connecting rod fabrication process, particularly, during the lubrication channel machining.

Kuldeep B (2013) [9] analyzed the connecting rod by replacing Al360 material by aluminium based composite material reinforced with silicon carbide and fly ash. He also described the modeling and analysis of connecting rod. FEA analysis was carried out by considering two materials. The parameters like von misses stress, von misses strain and displacements were obtained from ANSYS software. Compared to the former material the new material found to have less weight and better stiffness. It resulted in reduction of 43.48% of weight, with 75% reduction in displacement.

K. Sudershn Kumar (2012) [10] describes modeling and analysis of connecting rod. Existing connecting rod is manufactured by using Carbon steel. In his project connecting rod is replaced by Aluminum reinforced with Boron carbide for Suzuki GS150R motorbike. A 2D drawing is drafted from the calculations. A parametric model of connecting rod is modeled using PRO-E 4.0 software. Analysis is carried out by using ANSYS software. Finite element analysis of connecting rod is done by considering two materials, viz. Aluminum Reinforced with Boron Carbide and Aluminum 360. The best combination of parameters like Von misses stress and strain, Deformation, Factor of safety and weight reduction for two wheeler connecting rod were done in ANSYS software. Compared to carbon steel, aluminum boron carbide and aluminum 360, Aluminum boron carbide is found the working factor of safety is nearer to theoretical factor of safety in aluminum boron carbide. Percentage of reduction in weight is same in Aluminum 360 and aluminum boron carbide. Percentage of increase in stiffness in aluminum boron carbide is more. Percentage of reduction in stress aluminium boron carbide and aluminum 360 is same.

Sharma Manoj, Shashikant (2015) [11] taken connecting rod of a Mahindra Jeep CJ-340 and change its material from Al360 to PEEK. The modeling of the connecting rod is done on Pro-E wildfire 4.0 and analysis work is done on ANSYS 11.0. The parameters like Von misses stress, Von messes strain and displacement was obtained from ANSYS software which shows reduction in weight and improvement in strength.

Prof. Vivek C. Pathade (2013) [12] worked on the stress analysis of connecting rod by Finite Element Method using Pro/E Wildfire 4.0 and Ansys Workbench 11.0 software. The comparison and verification of the results obtained in FEA is done experimentally by the method of Photo elasticity (Optical Method). The method of Photoelasticity includes the casting of Photoelastic sheet using Resin AY103 and Hardener HY951, preparation of the model from Photoelastic sheet calibration of the sheet to determine material fringe value. From the theoretical, Finite Element Analysis and Photoelastic Analysis he found that the stresses induced in the small end of the connecting rod are greater than the stresses induced at the big end. Form the photoelastic analysis (from the fringe developed in the photoelastic model of connecting rod) it is found that the stress concentration effect exist at both small end and big end and it is negligible in the middle portion of the connecting rod. Therefore, the chances of failure of the connecting rod may be at fillet section of both ends.

Xiaolei Zhu, Jing Xu, Yang Liu, Bo Cen, (2017) [13] investigated in Connecting rod cap and connecting bolts of a reciprocating compressor are subjected to complex dynamic loads as they are of critical machine elements. The causes for the failure of connecting rod cap and connecting bolts after approximately 175200h in service have been investigated. To determine the failure mechanism of the connecting system and to figure out which of the connecting rod cap and connecting bolts was broken first, material characterization and numerical analysis are conducted on the connecting rod and connecting bolts. Scanning electron microscope (SEM) and optical microscope are applied to analysis the macro and micro structures. The chemical composition and metallographic structure investigation are carried out to study characteristics of the materials. Tensile tests, hardness and impact tests are performed to check the mechanical properties of the connecting rod cap and three connecting bolts. A finite element analysis of the connecting system is employed to evaluate the overall stress distribution with maximum stress criterion. Results of their investigation indicate that the reason of failure was high cycle fatigue and the initial crack location was consistent with high stress concentration at the curvature mutation position of connecting rod cap which was broken before the connecting bolts. The components of connecting rod cap and connecting bolts material conform to the requirements. The metallographic structure of connecting rod cap sample is ferrite and pearlite, and the connecting rod bolt sample's microstructure of substrate is tempered sorbite. Both of them correspond to the standards organization map. Connecting rod cap failure mode is high cycle fatigue fracture. Fracture position was located on the part of the curvature mutation position of the connecting rod cap, and this part will produce stress concentration under the normal operation condition and the initial micro crack would expand to fracture under dynamic load. The fracture of connecting rod cap resulted in releasing circumferential freedom of crankshaft to increase the connecting bolts load. The connecting bolts occurred plastic deformation at first and were broken finally. Strict control of condition change of the compressor in the operation process should be taken, and it should be stopped in time under abnormal phenomenon. Stress concentration on the terminal of the connecting rod cap shall be strictly controlled in the process of maintenance and inspection appraisal may be done on the part of the curvature change of the connecting rod cap when it is necessary.

Shriram A.Phad. (2013) [14] performed on a cast iron connecting rod of a compressor for the static and dynamic analysis. Literature survey suggests cyclic loads comprised of static tensile and compressive loads are often used for design and analysis of connecting rods. The S-N approach by Modified Goodman criterion to fatigue life prediction of the connecting rods is presented. The possibility failure of the connecting rod approach by failure index is the ratio of maximum principal stress to yield strength of material. The three-dimensional finite element model is constructed in PROE and used for Analysis. The model is meshed in ANSYS workbench and used for Analysis. The analysis shows that the tensile stresses are more than compressive stresses. The stress goes decreasing from pin end to crank end. Also the maximum stresses occurred at transition between shank and both ends, also at the fillet region of both pin end and crank end. There are lower stresses occurred at the shank region of connecting rod, hence there is scope for future development. The failure index of connecting rod is less than 1; the possibility of failure is less.

Borse Rajendra R. (2012) [15] investigated weight and cost reduction opportunities that steel forged connecting rods offer. His research is concentrated on the calculation of the stresses developed in the connecting rod and to find region more susceptible to failure. First the cad modeling of connecting rod with the help of cad software Pro/E Wildfire 3.0 and then Load analysis was performed with different cases consideration. The analysis was carried out with computer aided simulation. The tool used for analysis is ANSYS WORKBENCH 9.0.The following conclusions can be drawn from his study: 1) The optimization carried out in analysis gives deep insight by considering optimum parameter for suggestion of modification in the existing connecting rod. 2) Optimization was performed to reduce weight. Weight can be reduced by changing the material of the current forged steel connecting rod to crackable forged steel (C-70). 3) The parameter consideration for optimization are its 20 % reduction in weight of connecting rod, while reducing the weight, the static strength, fatigue strength, and the buckling load factor were taken into account. 4) The optimized geometry is 20% lighter than the current connecting rod. PM connecting rods can be replaced by fracture splitable steel forged connecting rods with an expected weight reduction of about higher than existing connecting rod, with similar or better fatigue behavior 5) By using other facture crackable materials such as micro-alloyed steels having higher yield strength and endurance limit, the weight at the piston pin end and the crank end can be further reduced. 6) Software gives a view of stress distribution in the whole connecting rod which gives the information that which parts are to be hardened or given attention during manufacturing stage.

www.ijetmas.com February 2018, Volume 6, Issue 2, ISSN 2349-4476

3. SUMMARY

These research papers are very useful in design, analysis and optimization of connecting rod. Researchers used many different methods and software's for design, analysis and optimization. Researchers used different 3D modeling software's like Pro-E, Catia, Solidworks etc., and for stress analysis they used Ansys, Abacus, and also experimental analysis using testing setups and photoelastic analysis. Optimization can be done by changing the material of the connecting rod and by changing the structural design.

REFERENCES

Papers from Journals

[1] Nagaraju K L and Chandan R , "Buckling analysis of connecting rod",IRJET ISSN 2395-0056,Vol 3, Issue 8, 2016, pg 1358-1361.

[2] Akbar H Khan and Dr. Dhananjay R Dolas, "Design modeling and static structural analysis of connecting rod",IJIR ISSN 2454-1362,Vol 3, Issue 1, 2017, pg 400-408.

[3] Mohammed Mohsin Ali H, Mohamed Haneef, "Analysis of fatigue stresses on connecting rod subjected to concentrated loads at the big end,4th International conference on material processing and characterization,2214-7853, 2015, pg 2094-2103.

[4] Akbar H Khan and Dr. Dhananjay R Dolas, "Static structural and experimental stress analysis of connecting rod using FEA and Photoelasticity",IJIRSET ISSN 2319-8753,Vol 6, Issue 1, 2017, pg 578-585.

[5] Mohd Nawajish, Mohd Naimuddin, Mayank, "A comparative study and analysis of connecting rod",IJETMAS ISSN 2349-4476,Vol 3, Issue Special, 2015, pg 332-338.

[6] Suraj Pal, Sunil Kumar, "Design evaluation and optimization of connecting rod using FEM",IJEMR ISSN 2250-0758,Vol 2, Issue 6, 2012, pg 21-25

[7] Ram Bansal, "Dynamic simulation of a connecting rod made of aluminium alloy using finite element analysis approach",IOSR-JMCE ISSN 2278-1684, Vol 5, Issue 2, 2013, pg 01-05.

[8] C. Juarez, F. Rumiche, A. Rozas, J.Cuisane, P. lean, "Failure analysis of a diesel generator connecting rod", Case studies in engineering analysis 7, 2016, pg 24-31.

[9] Kuldeep B, Arun L.R., Mohammed Faheem, "Analysis and optimization of connecting rod using ALFASiC composites",IJIRSET ISSN 2319-8753,Vol 2, Issue 6, 2013, pg 2480-2487.

[10] K. Sudershn Kumar, DR. Tirupathi Reddy, Syed Altaf Hussain, "Modeling and analysis of two wheeler connecting rod",IJMER ISSN 2249-6645,Vol 2, Issue 5, 2012, pg 3367-3371.

[11] Sharma Manoj, Shashikant, "Optimization of connecting rod with help of FEA",IJMET ISSN 0976-6359,Vol 6, Issue 7, 20115, pg 51-57.

[12] Prof. Vivek C. Pathade, Dr.Dilip S. Ingole, "Stress analysis of I.C. Engine connecting rod by FEM and Photoelasticity",IOSR-JMCE ISSN 2278-1684,Vol 6, Issue 1, 2013, pg 117-125.

[13] Xiaolei Zhu, Jing Xu, Yang Liu, Bo Cen, Xiaofeng Lu, Zhuo Zeng, "Failure analysis of a failed connecting rod cap and connecting bolts of a reciprocating compressor", Engineering failure analysis, PII S1350-6307(16)30748-8, Reference EFA3027, 2017.

[14] Shriram A. Phad and D.H. Burande, "Static and dynamic analysis of connecting rod of compressor", IJAnERD ISSN 2277-4785,Vol 3, Issue 3, 2013, pg 23-30.

[15] Prof. Pushpendra Kumar Sharma, Borse Rajendra R., "Fatigue analysis and optimization of connecting rod using finite element analysis", IJARSE ISSN 2319-8354,Vol 1, Issue 1, 2012.