



IJRASET

International Journal For Research in
Applied Science and Engineering Technology



INTERNATIONAL JOURNAL FOR RESEARCH

IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY

Volume: 6 Issue: III Month of publication: March 2018

DOI: <http://doi.org/10.22214/ijraset.2018.3326>

www.ijraset.com

Call:  08813907089

E-mail ID: ijraset@gmail.com

Design and Analysis of Gudgeon Pin to Minimize Stress Concentration: A Review

Vijay Baban Kamble¹, Prof. K. S. Mangrulkar², Mithun D. Shah³, Hrushikesh B. Kulkarni⁴

^{1, 2, 3, 4} Mechanical Engineering Department, NBNSCOE, Solapur-413001, MH, India

Abstract: This paper deals with the detailed review of design and analysis of existing Gudgeon pin to find stress concentration of the pin, and then minimize stress concentration by making suitable changes in gudgeon pin design and mounting. Proposed work consists of design of gudgeon pin and then the Finite Element Method is established using ANSYS software to analyse the stresses on gudgeon pin and minimization of it.

Keywords: Design and analysis, Gudgeon Pin, ANSYS, Stress concentration, FEM.

I. INTRODUCTION

While the engine developed quickly and engine performance improved greatly, the demand for engine parts is becoming higher and higher. As an important part in engine, piston endures the cyclic gas pressure and the inertial forces at work, and this working condition may cause the fatigue damage of piston, such as piston side wear, piston pin seat crack and so on. The investigations previous indicate that the greatest stress appears on the upper end of the piston pin seat and this stress concentration is one of the mainly reason for fatigue failure. Due to the complicated working environment for the piston, on the one hand, the FEA for the piston became more difficult; on the other hand, though there have many methods which are put forward to apply optimal design, the optimal parameters is not easy to determine.

Gudgeon pin connects the small end of the connecting rod with the piston providing a turning pair; Gudgeon pin can either turn relative to the connecting rod or relative to the piston bore or turn relative to both.

Prerequisite for this function is that the Gudgeon pin has a sliding fit with the other two. Since the Gudgeon pin transmits the load from the connecting rod to the piston, the deformation that it suffers during the operation must be such that it does not have surface contact with both simultaneously. This will lead to seizure. Besides, the Gudgeon pin should not be stressed beyond its endurance limit.

The Gudgeon pin is either a hollow or solid steel cylinder of length roughly five times its outer diameter. This is subjected to lateral load from the connecting rod. The piston pin is used to connect the piston and the connecting rods in the internal combustion engine, and the piston pin transfers gas pressure to the connecting rods. The rubbing pair is constituted by the piston pin and piston pin boss; the gas pressure will be delivered to the rubbing pair through the piston pin and piston pin boss, and then the piston pin may occur bending.

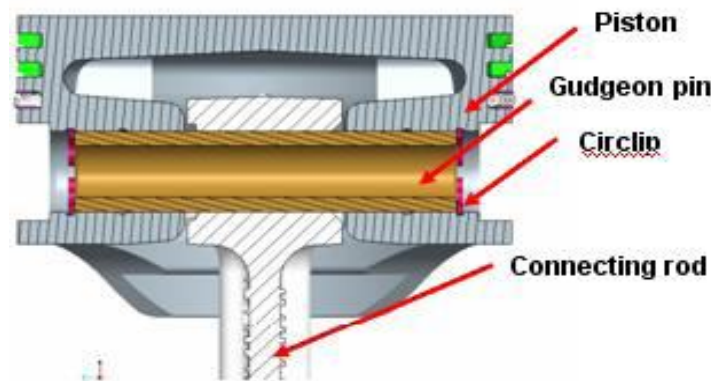


Fig. 1 Cross section of the piston assembly

So the piston and the piston pin should have enough stiffness which can endure the pressure and the friction between contacting surfaces. In addition, as an important part in engine, the working condition of piston is directly related to the reliability and durability of engine. So it is important for the piston pin and the piston pin boss to carry out structural and optimal analysis which can provide reference for design of piston.

II. LITERATURE REVIEW

V. Ramamurti, S. Sridhar, S. Mithun, B. Kumaravel, and S. Lavanya, The paper says that deformation and stress experienced by the Gudgeon pin of a reciprocating compressor used in air brake system is scientifically predicted when the pin is fully floating with lubricating oil surrounding it and when starved of oil. Both semi analytical approach in finite element method and simple bending theory of beams are used. Inadequacy of the beam approach is highlighted. The results obtained by both the approaches are compared. Role of clearance in piston bore and small end of connecting rod and effectiveness of lubrication are examined. Factor of safety associated with the design of Gudgeon pin is also looked into. From the foregoing analysis, it is clear that the maximum Von Mises stress experienced by the Gudgeon pin for a well lubricated assembly is unlikely to exceed 40 MPa. The material used for Gudgeon pin is Nickel-chromium alloy steel whose endurance limit is above 200 MPa. This provides a factor of safety around five. This implies that the Gudgeon pin is basically an over designed member which is not expected to fail during its operation [1] Zhiwei Yu, Xiaolei Xu, and Hongxin Ding, In this paper a diesel engine piston-pin used in a truck was smashed in four when servicing. The longitudinal and transverse cracking happened on the failed piston-pin. The cracks initiated from the internal hole surface and propagated toward the external circle. The occurrence of beach marks or fatigue striations on the fracture surfaces of all crack origin regions indicates that fatigue fracture is the dominant failure mechanism of the piston-pin. The internal hole and external circle surfaces are specified to be carburized. The microstructure and the microhardness profiles on the external circle and internal hole surface regions were examined to determine the depth of the carburized layer. However, not only no carburized layer found on the internal hole surface, but also the serious decarburization occurred on the surface region of the internal hole. Appearance of decarburization in the internal hole surface decreases intensely the fatigue strength of the internal surface so that the crack initiated from the internal surface and propagated toward the external circle, at last the fatigue fracture occurred. Improper carburizing technology is responsible for the appearance of the decarburization on the internal hole surface [2].

Yanxia Wang and Hui Gao[3], In this paper the fatigue failure and the fracture injury occurs under the alternative mechanical loads, the optimal design of the piston pin and the piston pin boss is presented depending on the FEA static analysis. The optimization is carried out using the Genetic Algorithm (GA), and the piston noncircular pin hole is used to further reduce the stress concentration on the upper end of the piston pin seat. The FEA is carried out for the initial piston model and the optimal one respectively. The results of analysis indicate that the maximum stress has changed from 171MPa to 145MPa, and the biggest deformation has been reduced from 0.359mm to 0.301mm [3]

K. Sandeep , Ajit Kumar , N. S. Mahesh, He says that Productivity improvement is a challenge to every industry and it means efficiency improvement at all stages i.e. man power, energy, machinery, process, money etc. Productivity is one of the major factor contributing to the profit of any company. The study focuses on improving the productivity of piston pin heat treatment process with continuous improvement and process improvement tools such as process mapping, FMEA's, control plans, and statistical techniques to increase productivity and create bottom-line savings. The study aims to achieve the optimum utilization of sealed quench furnace capacity by improving the fixture design, optimizing the lot size, improving the process parameters and thus improving the productivity of the heat-treatment process [4].

Fundamental tools for improving a process is done by identification of root causes with the help of a fish bone diagram and why-why analysis. Furthermore FMEA identifies potential product related process failure modes and prevents wastes, rework, reduce cycle time and cost. The proposal or solution for the material and design selection procedure was obtained with the help of literature review finding, prioritization of various solutions by using FEA analysis and CES. The ultimate results were improvement in productivity for selected five elements is 21.84%. The improvement of fixture design and wire holding system of parts helped to eliminate 14% of fixture weight to hold more parts. Effort has been made to improve the fixture design to hold more parts and FEA analysis is done to check the design behavior under static and thermal load. The productivity improvement resulted in a cost benefit of Rs. 10,00,000 per month. We conclude that the study has led to Significant improvements in process efficiency.

Andrew Homick, Aaron Turbeville, Mark Musick, Brent Clay. The objective of this project was to design the wrist pin bearing and crank pin bearing for a piston, connecting rod, and crankshaft system. The specifications for the system are a piston with a 2.375" diameter weighing 0.379 lbs. and a connecting rod of length 3.125" weighing 0.124 lbs. From the crank pin end the connecting rod has a mass eccentricity of 0.4" and the crank shaft has a length of 0.875". A constant angular velocity of 3000 RPM for the rotating crank was assumed. Using the provided engine data a rounded indicator diagram was created. In

order to design the wrist pin and crank pin bearings it was important to first calculate the dynamic forces as a function of the crank angle to find the maximum force caused by the reciprocating load. Calculations were based on SAE50W oil at a constant 200°F. The

necessary dimensions for the piston pin bearing were a length of 0.6875" and a diameter of 0.4375" and the crank pin bearings length and diameter were 1.125" and 1.25", respectively [5].

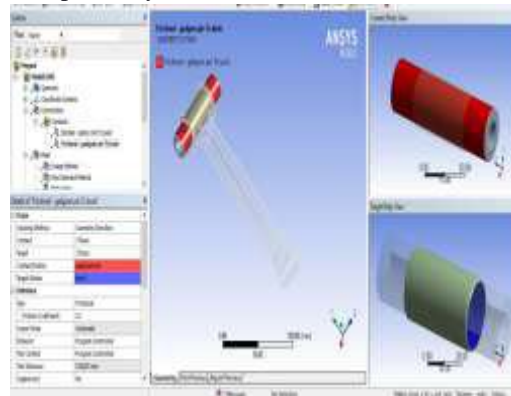


Fig2. Set the contact between parts of analysis and Mesh the CAD model with proper meshing techniques

Mr. Sajid Tamboli, Dr. N.K. Nath and Dr. S.B. Satpal reported that Gudgeon pin connects the piston and the small end of the connecting rod of engines. The wear of the Gudgeon pin and connecting rod concern for the company. In this Way frictional stress and Von-mises stresses are produced on Pin and they are determined by finite element analysis tool ANSYS .Fatigue life of pin is determined using fatiuge analysis tool. The aluminum having better strength as compared to steel [6].

Rahul D. Raut, Satish Mishra studied the performance of any automobile largely depends on its size and working in dynamic conditions. The piston is a "heart" of the engine and its working condition is the worst one of the key parts of the engine in the working environment. The good design of the piston optimization can lead to a mass reduction on the base of stress analysis satisfying the requirements of automobile specifications with cost and size effectiveness. Piston is the part of engine which converts heat and pressure energy liberated by fuel combustion into mechanical works. Engine piston is the most complex among automotive. This paper work describes stress optimization of Piston for I.C. Engine. The stress distribution of piston by using FEM to be investigate and analyze. The stresses due to combustion are considered to avoid the failure of the piston. Intensity of structural stresses should be reduced to have safe allowable limits. This paper introduces an analytical study of the structural effects on the piston head.CAD software Creo is used to model the piston and stress optimization is performed by using Creo-Simulate. The paper describes the piston optimization with using finite element analysis technique to predict the higher stress and critical region on the component. The optimization is carried out to reduce the stress concentration on the head of the piston. The optimization of piston is done and it is found that the mass of optimized piston is 0.2454 Kg. Hence percentage reduction in mass compared to nonoptimized piston (0.2574 Kg) is 4.66 %. The material is removed to reduce the weight of the piston so as to improve the efficiency. It is essential to obtain the optimized results for piston with reduced material. The factor of safety becomes greater than nonoptimized model i.e. design become more safe [7]

Shahanwaz Adam Havale, Prof. Santosh Wankhade, studied that the main heating part in the engine, piston works for a long time in high temperature and high load environment. The piston has the characteristics of large heating area and poor heat dissipation, so the thermal load is the most serious problem. This thesis presents a numerical method using thermos-mechanical decoupled FEM (Finite Element Method) to calculate the thermal stress only caused by the uneven temperature distribution. In this work, the main emphasis is placed on the study of thermal behavior of functionally graded materials obtained by means of using a commercial code ANSYS on aluminum alloy piston surfaces. The analysis is carried out to reduce the stress concentration on the upper end of the piston i.e. (piston head/crown and piston skirt and sleeve). With using computer-aided design, SolidWorks software the structural model of a piston will be developed. Furthermore, the finite element analysis is done using Computer Aided Simulation software ANSYS [8].

Aditya Chaudhary, Dr. Prashanta Kr Mahato elaborated that connecting rod is a critical element in an automotive power transmission system, being the kinematic link between the piston and the crankshaft. It is used to convert the reciprocating motion of the piston to the rotary motion of the crankshaft. In the present study, the stress and deformation analysis of the connecting rod has been carried out, and viable changes in three domains, namely the manufacturing processes, material and design of the connecting rod, have been proposed. Under the head of manufacturing processes, different processes including forging and sintering have been discussed. Effect of shot peening on fatigue strength and heat treatment on the overall performance of the connecting rod

have been taken into account. Newer and more efficient materials, namely C-70 steel, Micro-alloyed steels and aluminium based composites with particle and fiber reinforcements have been tested. Lastly, the design of the connecting rod has been modified to get the best combination of overall stress, stress concentration and deformation. In this process, areas of high stress concentrations are identified and attempts have been made to relieve stresses in these sections. Modifications in each of the above mentioned stages have led to stress and weight reduction and increase in the stiffness, thus enhancing the overall performance of the connecting rod. Static and fatigue analysis has been carried out in ANSYS 15.0 Workbench. Output parameters including Von Mises stress, total deformation, factor of safety, fatigue life and fatigue factor of safety have been used for comparative study of the modified models and their existing counterparts [9].

Y.X. Wanga , Y.Q. Liub , H.Y. Shic, explained that piston is one of the most stressed components of an engine. In this paper, a 1/2 3-D solid model of a new designed piston was built by using ANSYS software. The stable stress distribution and the deformation under the thermo-mechanical coupling condition were firstly calculated. Calculating results indicates that the maximum stress concentration is at the upper end of piston pin boss inner hole, and is mainly caused by the peak pressure of the fuel gas. Then the finite element dynamic analysis was conducted based on the mechanical fatigue testing method, and the mechanical fatigue life-span was calculated. All these work indicate that the design of the piston is reasonable [10].

III. PROPOSED WORK

The objective of this work is to Design and Analysis of existing Gudgeon pin to find stress concentration of the pin, and then minimize stress concentration by making suitable changes in gudgeon pin design and mounting. This work consists of design of gudgeon pin and then the Finite Element Method is established using ANSYS software to analysis of gudgeon pin. Then I will solve the model in Ansys software. Then solve it and find the maximum stress value and point of concentration. As I got the maximum stress value and point of concentration of stress. I will work on that for minimizing stress value or Stress concentration. Then make suitable Geometrical changes in Gudgeon pin design to minimize the stress value or stress concentration.

A. *The Study Will Be Carried Out On The Following Outline/Methods:-*

- 1) To study the gudgeon pin design and Analysis of pin.
- 2) Design the gudgeon pin as CAD Modeling of pin using CATIA/Pro-E.
- 3) Analyze the Gudgeon pin by establishing finite Element Method using Ansys Software.
- 4) Find the maximum stress value and point of concentration.
- 5) Work on minimizing the stress value or concentration of that point.
- 6) Comparison of the results for the designed pin.

REFERENCES

- [1] V. Ramamurti, S. Sridhar, S. Mithun, B. Kumaravel, and S. Lavanya, "Design Considerations of Gudgeon Pin in Reciprocating Air Compressor by Semi Analytical Approach", Key semi analytic method, JMERE vol. 4(3), pp. 75-88, March 2012.
- [2] Zhiwei Yu, Xiaolei Xu, and Hongxin Ding, "Failure Analysis of a Diesel Engine Piston Pin", Wlectromechanical and Material Engineering College, Dalian Maritime University, Dalian 116026, P R China, Dec 2005.
- [3] Yanxia Wang and Hui Gao, "Reseach on Optimization for the Piston Pin and Piston Pin Bodd", School of Traffic and Vehicle Engineering, Shandong University of Technology, Zibo 255049, Shandong, China, Open mechanical engineering journal, vol 5, 2011.
- [4] K. Sandeep , Ajit Kumar , N. S. Mahesh, "Improving Productivity in a Heat Treatment Shop for Piston Pins", M. S. Ramaiah School of Advanced Studies, Bangalore 560 058, SASTECH Journal, Vol 11, Issue 2, Sept 2012.
- [5] Andrew Homick, Aaron Turbeville, Mark Musick, Brent Clay, "Wrist Pin & Crank PinJournal Bearing Design", MAE 415-003 Analysis for Mechanical Engineering Design, NC State University, October 24th, 2011.
- [6] Mr. Sajid Tamboli, Dr. N.K. Nath, Dr. S.B. Satpal, 'Design And Analysis of Gudgeon pin', IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE) e-ISSN: 2278-1684, p-ISSN: 2320-334X, PP. 38-4, NCRIME-2018
- [7] Rahul D. Raut1, Satish Mishra2, Stress Optimization of S.I. Engine Piston, International Journal of Science and Research (IJSR) ISSN (Online): 2319-7064, Volume 4 Issue 9, September 2015.
- [8] Shahanwaz Adam Havale1, Prof. Santosh Wankhade2 , DESIGN, THERMAL ANALYSIS AND OPTIMIZATION OF A PISTON USING ANSYS, International Research Journal of Engineering and Technology (IRJET), Volume: 04 Issue: 12 | Dec-2017, e-ISSN: 2395-0056
- [9] Aditya Chaudhary1, Dr. Prashanta Kr Mahato 2, Fabrication, Material and Design Modification based Analysis of Connecting Rod Using FEA, International Journal of Research in Advent Technology, Vol.4, No.11, November 2016, E-ISSN: 2321-9637,
- [10] Y.X. Wanga, Y.Q. Liub, H.Y. Shic, Finite Element Static and Dynamic Analysis for a Piston, Advanced Materials Research, ISSN: 1662-8985, Vols. 97-101, pp 3323-3326, doi:10.4028/www.scientific.net/AMR.97-101.3323



10.22214/IJRASET



45.98



IMPACT FACTOR:
7.129



IMPACT FACTOR:
7.429



INTERNATIONAL JOURNAL FOR RESEARCH

IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY

Call : 08813907089  (24*7 Support on Whatsapp)