Design and Analysis of Gudgeon Pin to Minimize Stress Concentration

Vijay Baban Kamble¹, Prof. K. S. Mangrulkar², Mithun D. Shah³, Hrushikesh B. Kulkarni³

Mechanical Engineering Department, NBNSCOE, Solapur-413001, MH, India

Abstract- This paper deals with the detailed 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. Present 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 minimizing it.

Keywords- Design and analysis, Gudgeon Pin, ANSYS, Stress concentration, FEM

I. INTRODUCTION

Gudgeon pin (Figure 1) 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 (Ramamurti, 2009). 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 (Figure 2). The load acts for the full width of the connecting rod (for nearly one third of the axial length of the Gudgeon pin) acting over a substantial arc of the outer cylindrical surface of the pin. A very simple method of analysis is to treat the pin as a simply supported beam subjected to lateral load of uniform intensity from the connecting rod.
This approach has two basic deficiencies. (i) The Gudgeon pin does not qualify to be treated as a beam since its length is much less than ten times its outer diameter. (ii) Besides the beam, when assumed as simply supported at the ends, ignores the fact that its support is along an arc of a circle. If, on the other hand, assumed fixed, it ignores the fact that there is a radial clearance between the piston bore and the cylindrical surface of the Gudgeon pin. However, when assumed simply supported, beam approach is very easy to analyze since the structure is statically determinate [1].

**II. FORCE ANALYSIS OF GUDGEON PIN**

The Gudgeon pin assembly is subjected to uniform intensity of pressure in the region of the connecting rod due to air compressed on the top flat face of the piston. Besides it is prevented from moving axially by the circlips. The deformation of the Gudgeon pin is to be within elastic limits under the action of the force along the width of the connecting rod for various angular positions. The bending deformation of the Gudgeon pin is to be assessed to address the adequacy of clearance between the pin and the small end of the connecting rod and also between the Gudgeon pin and piston bore.

The Gudgeon pin, in reality, is a cylinder of roughly length 5 times its diameter, supported by lubricating oil present in the clearance on the piston bore for roughly two thirds its length with the middle one third subjected to lubricating oil pressure on the annular space of the small end of connecting rod. The load that gets communicated to the Gudgeon pin from the connecting rod acts on the outer circumference along the middle one third of its length. The piston bore on both sides supports this pin through the lubricating oil. The pin oscillates through approximately 10° about its mean position. There are two aspects to be considered for the load distribution, one along the length of the pin and other along its circumference. Along the circumference, due to the lubricating oil pressure, it is periodic with the resultant along the line joining small end and the big end and along the width of the connecting rod, with uniform intensity. To meet these two requirements, semi analytical approach can be used (Ramamurti, 2009; Ramamurti and Gupta, 1978; Ramamurti and Narayanan, 1989; Quing et al., 2006; Zienkiewicz, 1991). In this connection, the following papers that have similar connected applications can be cited.

Ramamurti and Gupta (1978) have assumed the load of a kiln tyre supported by rollers to act over a small arc, whereas in another paper, Ramamurti and Narayanan (1989) have assumed that the load is transferred along several short arcs for a roller clutch. Quing et al. (2006) have determined the natural frequencies of a shell system by semi analytical approach in finite element method. In this paper, both beam and semi analytical approaches are attempted and results compared.
III. 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 micro hardness 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 factors 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].

Figure 3 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. 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 fatigue analysis tool. The aluminum is 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 on piston by using FEM is investigate and analyzed. 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 no optimized 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 no optimized 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]. In present work detailed design and analysis of gudgeon pin is done to minimize stress concentration

### IV. DETAILS OF ANALYSIS

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 maximum stress value and point of concentration of stress 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.

The study will be carried out on the following outline/methods:-

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

**Modeling:** Gudgeon pin is modeled in Design Modeler application of Ansys. The pin is modeled and the face is split at the location where it is held in piston for giving cylindrical support.

**Boundary conditions:** The gudgeon pin is held at both ends into the piston; hence two cylindrical supports are given. In this radial and axial movement are fixed whereas tangential movement is made free. One of the ends of pin (face) has given displacement constraints where displacement in x, y and z direction is made zero because pin is firmly held in piston and there is no relative movement between piston and pin. As there is small clearance in pin and piston, hence pin acts as a simply supported beam where it is supported by piston bore at two ends with line contact. Gudgeon pin is also held in small end of connecting rod. Hence contact area of connecting rod and pin is considered as loading zone. So pin is given load at middle area with pressure of 7 bar (Maximum pressure in the cylinder at the end of combustion).

**Meshing:** As the pin is uniform and is sweep able body, the sweep type meshing is carried out, so that all hexagonal brick mesh can be formed. Brick type of meshing gives accurate results, also brick mesh are densely arranged in the model. Brick mesh is suitable when volume to surface area ratio is high as like the gudgeon pin. Total 1824 number of elements and 10224 numbers of nodes are formed in the meshing. The global mesh size is 5 mm.

**Solution:** The above setup is solved in Ansys workbench; the total CPU time is 15 seconds. Here mathematical model is prepared and it is solved for given boundary conditions, constraints and loads.

**Results:** In the results we are interested in total deformation of the pin and equivalent von-Mises stress. The equivalent stress is evaluated and maximum stress found is 404.44 Mpa. The elements with maximum stress are shown with ‘Max’ tag. Also stress at any element or node can be read by inserting a probe. The total deformation found is 0.286 mm. Similar to stress maximum deformation is shown and deformation at any location read by probe.

V. ANSYS RESULTS

![Figure 4 meshing of gudgeon pin](image-url)
The figure 4 shows the meshing of pin Generated in Ansys. Here the type of Meshing is sweep type and all the Elements are of hexagonal brick type. Such arrangement gives accurate results. The total number of elements Generated are 1824 and total number of nodes generated are 10224. The size of each elements is 5mm, this element sizing is global sizing.

![Figure 5 Total deformation](image)

Figure 5 shows the total deformation obtained In Ansys. The maximum deformation observed is 0.028mm which is at the middle of the pin shown in Red zone. The deformation of pin at each point is Shown by legends of colours. The red colour shows Maximum deformation and blue shows minimum deformation. The deformation observed is in the direction of load applied. The overall deformation is found within limit because it is elastic deformation and the pin regains original shape and size after vanishing of load.

In this Case the pin is given cylindrical support on both ends and load is given in middle region of the pin where Connecting rod is in contact. The total load given is 7 bar pressure and thus the maximum deformation is 404.44 Mpa which is shown in red zone. The maximum stress is found at both ends where connecting rod is in contact and near to cylindrical support. Again the total stress is found within limit, because total stress is less than the yield strength of the pin material (which is above 1000 Mpa).

![Figure 6 shows the equivalent von-mises Stresses in the gudgeon pin after giving load.](image)
When gudgeon pin is considered as simply Supported beam (because of clearance fit in Pin and piston as well as pin and connecting rod, the pin acts as beam supported at two end points Of connecting rod) the stress distribution is shown in figure 7 that case in the image given. Here the total stress i.e. equivalent von-mises stress found is 65.13 Mpa which is very less as compared to the case when pin has cylindrical supports of piston. For a very small fraction of time the pin acts as simply supported (at the start of stroke) and most the time the pin has cylindrical supports (during entire stroke).

IV. CONCLUSION

In present work detailed design and analysis of existing Gudgeon pin is done to find stress concentration of the pin. The stress concentration is then minimised by making suitable changes in gudgeon pin design and mounting. Present 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 minimizing it.

REFERENCES

VI. 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
IX. 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,