International Journal of Research Publication and Reviews

Journal homepage: www.ijrpr.com ISSN 2582-7421

Thermal Analysis of Piston, Connecting Rod and Crank Shaft Assembly using Catia and Ansys Software; A Review

Ravi Ranjan[1], Prof. D. S. Rawat[2] and Dr. Sumit Rai[2]

[1,2]Jabalpur Engineering College, Gokalpur, Jabalpur 482011, India

A B S T R A C T

This paper deals with the study of assembly of piston, connecting rod and crank shaft of a four wheeler petrol engine. The components of the assembly must be rigid and the assembly must move as a mechanism. Therefore, the analysis should include rigid-body analysis and flexible-body analysis. Therefore the forces exerted on the components when the engine is revving have to be calculated and these forces are used to calculate the dynamic stresses in the component of interest i.e. the connecting rod. It is proposed to change two new sets of materials for the components of the assembly and investigate the parameters by performing static, dynamic and thermal analyses, thermal analysis involves analyzing the thermal flow, temperature distribution by applying temperature on the surface of the piston. By looking at both analysis results, we can decide whether our designed piston is safe under the applied load conditions. Thus finally a comparative study is conducted by showing different results of analysis of pistons using different materials.

Keywords: Piston, Connecting rod, Crank shaft, Catia Ansys Software.

1. Main text

The piston is a component of the internal combustion engine. It forms a mechanism called crank mechanism with the help of connecting rod and crankshaft. The main function of the piston is to transform the pressure generated by the burning of the air-fuel mixture into force, acting on the crankshaft. The piston contributes to the dissipation of heat generated during combustion to the cylinder walls. It guarantees the sealing of the combustion chamber, preventing gas leaks and oil penetration into the combustion chamber and guides the movement of the connecting rod. The piston also ensures the continuous change of gases in the combustion chamber and thus generates the variable volume in the combustion chamber.

Zeng et al. configured a geometric piston model of a diesel engine in UG graphics. The piston temperature fields for burning diesel and DME separately are calculated using finite element analysis tool. The result shows that the temperature of the DME-powered diesel engine decreases along the piston axis from top to bottom. The piston temperature of the DME-fueled engine increases as a whole compared to burning diesel. However, the distribution of the temperature field does not show significant change, it decreases and then increases from the center of the combustion chamber to the edge, and decreases again towards the edge of the top of the piston. Saad et al., carried out numerical analyzes to analyze the stresses due to the thermal cycle with different aluminum piston alloys. The finite element method was used to evaluate the coupling field (thermal stress) in the piston. ANSYS5.4 Finite element code is used to perform the modeling process to determine coupling stress. Two models with three dimensions are created. The first is used to evaluate the temperature distribution across the piston volume, and the second is used to evaluate the thermal stress distribution due to the heat gradient and different materials. The result shows that the maximum temperature range is 4.3 °C and increases with the decrease in the thermal conductivity of the material. Thermal stress is concentrated at the edges of the piston and depends on the types of material. Reddy et al. In this work, the main emphasis is placed on the study of the thermal behavior of functionally graded coatings obtained through the use of a commercial code, ANSYS, on aluminum and zirconium-coated aluminum. Piston surfaces.

The analysis is performed to reduce the stress concentration at the upper end of the piston, i.e. (piston head/crown and piston skirt and sleeve). Using the computer-aided design software NX/Catia, the structural model of a piston will be developed. Additionally, finite element analysis is done using ANSYS computer-aided simulation software.

The research work deals with the study of assembly of piston, connecting rod and crank shaft of a four wheel petrol engine. The components of the assembly have to be rigid and the assembly has to move as a single mechanism. Therefore, the analysis should include rigid-body analysis and flexible-body analysis. Therefore the forces acting on the components when the engine is reciprocating have to be calculated and these forces are used to calculate the dynamic stresses in the component of interest i.e. the connecting rod.

It is proposed to replace two new sets of materials for the components of the assembly and check the parameters by performing static, dynamic and thermal analysis. Piston is a component of internal combustion engine. It forms a mechanism with the help of connecting rod and crankshaft which is called as crank mechanism. The main function of the piston is to convert the pressure generated by the burning air-fuel mixture into force acting on the crankshaft. The piston contributes to the dissipation of heat generated during combustion on the cylinder walls. It ensures the sealing of the combustion chamber, prevents leakage of gas from it and ingress of oil into the combustion chamber and directs the movement of the connecting rod. The piston also ensures continuous change of gases in the combustion chamber and thus produces variable volume in the combustion chamber. Rotational motion is generated by the rotation of the crankshaft piston using a connecting rod. The motion of the engine's combustion gases and inertial components generate stresses that cause compressive and tensile stresses, respectively, in the connecting rod. Connecting rods fail due to overloading, bearing failure, irregular bolt adjustments, and faulty assembly or fatigue. It is important that the connecting rods are capable of withstanding the complex high tensile loads on which they operate. As a result, various design, material selection, working and fatigue test techniques for connecting rods have been studied and presented. The mechanical properties (such as hardness, tensile strength, toughness, and stress resistance) of the materials used in the manufacture of connecting rods on composites depend on the design of the connecting rod. The failure of the rod connection is due to the large force required to bear the load. This can be overcome by increasing the life cycle by increasing the strength. Rod finite element analysis has been conducted and presented by many researchers. In, the theory and FEA for the connecting rod of the motor IC was made. The obtained analysis result shows the causes of failures in both end slices due to induced stress. Continuous fatigue and deformation and weight optimization of the connecting rod are implemented and presented in FEA using ANSYS workbench. With the proposed design changes derived from the weight optimization result, the failure result is further updated to obtain better results. In the Bansal research paper, a dynamic stress analysis was performed on a single-cylinder four-stroke diesel engine with an aluminum connecting rod with FEA.

FEA is a commonly used computational tool for testing and modifying engineering structures within certain design limits. It involves diving into smaller units known as "elements" of static and dynamic analysis of a simple to complex model under various design constraints. Further research can be done to improve the design for optimal performance and lifespan with respect to design failures. Many authors have worked on weight correction. Gaikwad in his paper has modified the roller conveyor by performing the weight optimization process after performing static analysis on the roller conveyor. Further analysis of the structure optimization is also performed to identify a new optimized structure with new deformation and stress values, respectively. The analysis was performed using an ANSYS static structural mechanical analyzer, with a tensile force of 100 N applied at the large end.

2. Literature Review:

Zeng et al. A geometry model of a diesel engine piston was installed in UG Graphics. Piston temperature fields for burning diesel and DME separately are calculated using finite element analysis tools. The result shows that the temperature of a DME fueled diesel engine decreases from top to bottom along the piston axis. The piston temperature of an engine burning DME fuel increases compared to that burning diesel. **Saad et al.** have performed numerical analysis to analyze the stresses due to thermal cycling of pistons with different aluminum alloys. The finite element method was used to evaluate the coupling region (thermal-stress) on the piston. ANSYS finite element code is used to complete the modeling process to determine the coupling stress. Two models with three dimensions have been created. The first is used to evaluate the temperature distribution through the piston volume, and the second is used to evaluate thermal. Stress distribution due to temperature gradient and different materials. The result shows that the maximum limit of temperature is 4.3 °C and the thermal conductivity of the material increases with decreasing. Thermal stress is concentrated at the piston edges and depends on the type of material. **Reddy et al.** In that work, the main emphasis is on the study of the thermal behavior of functionally graded coatings obtained using the commercial code, ANSYS, on aluminum and zirconium coated aluminum piston surfaces. The analysis is done to minimize the stress concentration at the upper end of the piston i.e. (piston head/crown and piston skirt and sleeve). The structural model of the piston will be developed using Computer Aided Design NX/CATIA software. Furthermore, finite element analysis is performed using computer aided simulation software ANSYS.

G Gopal et al. The main parts of the assembly i.e. engine piston, connecting rod and crankshaft are modeled and assembled as per the given design. And finite element analysis is done in Ansys. Meshing is done in HyperMesh. Computer-aided design (CAD), also known as computer-aided design and drafting (CADD), is the use of computer technology for the design and design-documentation process. Ashish Kumar, Shubham Parmar, The main objective of this study is to analyze and optimize the connecting rod of Mahindra Piezo. This research shows that the performance of a connecting rod fundamentally depends on its shape optimization and material selection. The dimensions of the existing connecting rod are measured with the help of vernier calliper and micrometer. **Anil Kumar Vishwakarma** The present study was carried out using ANSYS software to analyze the connecting rod parameters. The main objective of this study is to investigate the induced stress in the connecting rod. This can be achieved by changing such design parameters in the existing design of single cylinder 4 stroke petrol engine by using FEA (Finite Element Analysis) for the study. During the analysis of the parameters of the connecting rod, it can be seen that there are several stresses acting during the loaded condition of the rod. **M Fratissa et al.** About Structural and Thermal Analysis of Diesel Engine Piston Using Ansys Software The main role of the piston is to vary the dimensions of the combustion chamber in an internal combustion engine. Therefore, steel pistons suffer from high inertial forces, and lightweight aluminum pistons cannot operate at high temperatures. The creative design of the bimetal piston has been chosen to combine the benefits of the two types of materials. **Mohammad Nawazish et al.** Structural analysis involves applying pressure to the piston and analyzing its stress, strain and deformation. Thermal analysis involves analyzing the thermal flux, the temperature distribution by applying temperature to the surface of the piston. By looking at both analysis results, we can decide whether our designed piston is safe under the applied load conditions.

3. Methodology:

It is an effective method in modeling as well as analysis of structures. In this presentation, the weight and structural analysis of the connecting rod has been done through finite element method using ANSYS and presented together. The processes were sequentially executed on a connecting rod of structural steel material under a loading of 1000N static force. ANSYS is a general purpose finite element analysis (FEA) software package. Finite element analysis is a numerical method of dividing a complex system into very small pieces (of user-specified size) called elements. The software applies the equations that govern the behavior of these elements and solves them all. These results can then be presented in tabular or graphical forms. Static, dynamic and thermal analyzes are performed on piston, connecting rod and crankshaft assemblies. The aluminum alloys for the crankshaft are 6061 and titanium and EN308 and higher alloy steels.

Finite element analysis is a numerical approach described by partial differential equations to investigate problems and solve them to reach approximate exact solutions. Solving engineering problems involving complex structures is a good feature of Catia. ANSYS is a CATIA software package that generates equations that solve and control the behavior of elements. The geometry is first defined depending on the nature of the analysis to be performed.

A 3D model can be rendered in ANSYS by saving it as Initial Graphics Interchange Specification (IGES) format and then importing it into ANSYS Workbench, or by creating the entire structure in ANSYS Workbench. In this paper, the analysis is performed by importing the geometry from CATIA in IGES format into the program4. Online license transfer

All authors are required to complete the Procedure exclusive license transfer agreement before the article can be published, which they can do online. This transfer agreement enables Elsevier to protect the copyrighted material for the authors, but does not relinquish the authors' proprietary rights. The copyright transfer covers the exclusive rights to reproduce and distribute the article, including reprints, photographic reproductions, microfilm or any other reproductions of similar nature and translations. Authors are responsible for obtaining from the copyright holder, the permission to reproduce any figures for which copyright exists.

4. DESIGN OF PISTON

The piston in an IC engine must have the following characteristics:

1. Strength to resist gas pressure.
2. It must have minimum weight.
3. Must be able to reciprocate with minimum fuss.
4. It must have sufficient bearing area to avoid wear.
5. It must seal the gas on top and the oil on the bottom.
6. It must disperse the heat generated during combustion.
7. It must have good resistance to distortion under heavy forces and high temperatures.

From machine design and data hand books, following dimensions of the piston have been calculated according to the given design procedure. The dimensions are in terms of SI Units.

4.1 CONNECTING ROD

The connecting rod, also called the connecting rod, is the part of a piston engine that connects the piston to the crankshaft. Together with the crank, the connecting rod converts the reciprocating movement of the piston into rotation of the crankshaft. The connecting rod is needed to transmit compression and tension forces from the piston and rotate at both ends. The most common use of connecting rods is in internal combustion engines or steam engines. The connecting rod is the part that connects the piston and the crankshaft. It is the link between the two parties. The smaller end of the rod is connected to the piston with the help of a pin and the larger end of the rod is connected to the crankshaft. The purpose of the connecting rod is to provide fluid movement between the piston and crank.

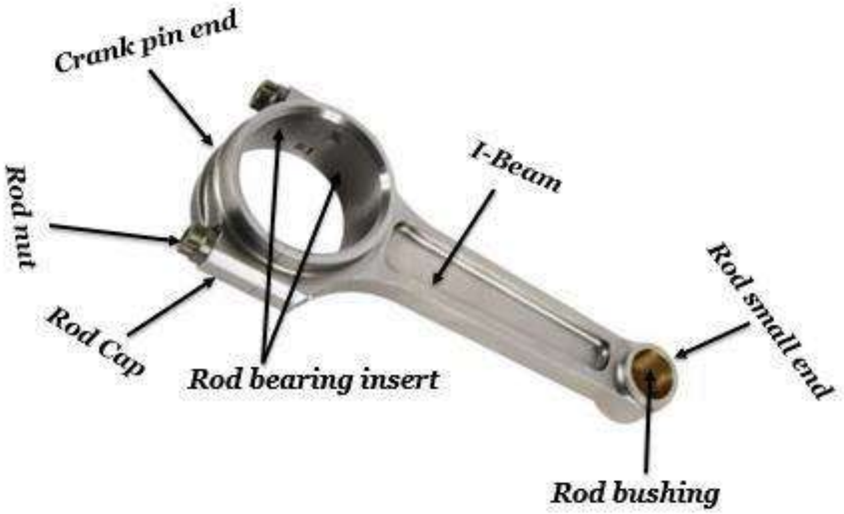

Figure 1. Connecting rod

4.2 PARTS

Pin End - (Small End) The short end of the connecting rod connects to the piston via the piston pin. The piston pin, or wrist pin, provides a pivot point between the piston and connecting rod. A spring clip, or piston pin lock, is used to hold the piston pin in place.

Rod - (Middle) The rigid middle part that connects the small end and the large end.

Crank End - (Big End) The big end of the connecting rod connects to the crankpin journal to provide a pivot point on the crankshaft.

Rod Cap - Connecting rods are manufactured as one or two piece components. The connecting rod cap is the removable section of a two-piece connecting rod that provides a bearing surface for the journal journal. The connecting rod cover is attached to the connecting rod with two bolts for installing and removing the crankshaft.

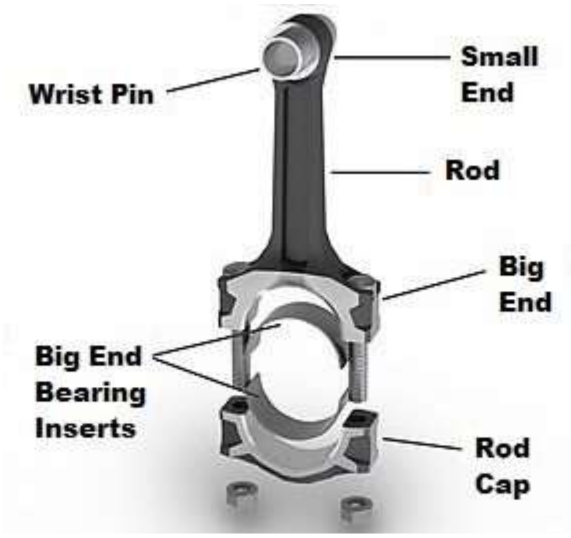

Figure 2. Parts of connecting rod

5. CONCLUSION:

The above study gives an idea about designing of connecting rods. It explains the different stresses to be considered while designing the connecting rod and the different materials used and compares the result for all the materials. Finite element analysis of the connecting rod is performed in ANSYS Workbench 18.0 considering all loading conditions. Maximum compressive stress was achieved between the pin end of the connecting rod and the rod.

Maximum shear stress is obtained at the pin end. Therefore the probability of failure of the connecting rod can occur at either end of the fitted section, but the probability of failure is higher at the piston end than at the crank end. Materials considered include aluminum alloy 6061 and aluminum alloy 2618 for the pistons, aluminum alloy 6061 and titanium for the connecting rods and EN308 and high alloy steel for the crankshaft. Having AL 2618 piston, titanium connecting rod and high alloy steel increases thrust and reduces displacement.

References

1. Balasubramaniam, B., Svoboda, M., and Bauer, W., 1991, "Structural optimization of I.C. engines subjected to mechanical and thermal loads," Computer Methods in Applied Mechanics and Engineering, Vol. 89.
2. Dr. B. Sudheer Prem Kumar, "Thermal Analysis and Optimization of I.C. Engine Piston Using Finite Element Method", International Journal of Innovative Research in Science, Engineering and Technology, Vol. 2, Issue 12, December 2013.
3. Ferguson, C. R., 1986, "Internal Combustion Engines, Applied Thermosciences," John Wiley and Sons, Inc.
4. Goenka, P. K. and Oh, K. P., 1986, "An Optimum Connecting Rod Design Study – A Lubrication Viewpoint," Journal of Tribology, Transactions of ASME, July 1986, Vol. 108.
5. Mbono Samba YC, Pascal M (2001) Dynamic analysis and numerical simulation of flexible multibody systems. Mech Struct Mach 29(3).
6. Montazersadgh, F. H. and Fatemi, A., 2007, "Dynamic Load and Stress Analysis of a Crankshaft," SAE Technical 2007, Society of Automotive Engineers, Warrendale, PA, USA.
7. Prem kumar, DESIGN & ANALYSIS OF CONNECTING ROD BY COMPOSITE MATERIAL, IJRDO, International-Journal-Of-Mechanical-And-Civil-Engineering, VOL 2 ISSUE 7 July 2015 Paper 1.
8. Taufeeque Hasan, Upendra Kumar, "FINITE ELEMENT ANALYSIS OF PISTON IN ANSYS" International Journal of Modern Trends in Engineering and Research (IJMTER) Volume 02, Issue 04, [April – 2015] ISSN (Online):2349–9745 ; ISSN (Print):2393-8161.
9. Madhura Chattopadhyay ,"Theoretical Analysis of Stress and Design of Piston Head using CATIA & ANSYS", International Journal of Engineering Science Invention, Volume 4 Issue 6, June 2015, PP.52-61.
10. Vivek C. Pathade, Dr. Dilip S. Ingole, "Stress Analysis of I.C.Engine Connecting Rod by FEM and Photoelasticity", IOSR Journal of Mechanical and Civil Engineering, Volume 6, Issue 1 (Mar. - Apr. 2013), PP 117-12