International Research Journal of Engineering and Technology (IRJET) e-ISSN: 2395-0056
Volume: 04 Issue: 07 | July -2017 www.irjet.net p-ISSN: 2395-0072

An Overview of Design and Analysis of Rocker Arm

V. R. Magdum[1], S. H. Sawant[2]

[1] PG Student, Department of Mechanical Engineering, Dr. J. J. Magdum College of Engineering, Jaysingpur, India
[2] Professor, Department of Mechanical Engineering, Dr. J. J. Magdum College of Engineering, Jaysingpur, India

---

Abstract - *Rocker arm is a pivoted lever used in an internal combustion engine to transfer cam or pushrod motion to a valve stem. Increase strength and durability of rocker arm is still subject of research and investigation. There are many stresses acting on rocker arm of IC engine. Present work finds the various stresses in rocker arm. For this we are modeling the arm using design software like CATIA and the stressed regions are found out using ANSYS software. Using different materials for construction of rocker arms is one of the best methods to increase its durability and strength. This also helps to reduce stresses values in rocker arm. Here we are observing that how the stresses are varying by changing different materials of the rocker arm. After comparing values of stresses we are proposing best suitable material for the rocker arm.*

Key Words: ANSYS, CATIA, Design and Analysis, Rocker Arm, Stresses.

1. INTRODUCTION

A rocker arm is a valve train component in internal combustion engines. As a rocker arm is acted on by a camshaft lobe, it pushes open either an intake or exhaust valve. This allows fuel and air to be drawn into the combustion chamber during the intake stroke or exhaust gases to be expelled during the exhaust stroke.

Failure analysis is a broad discipline that includes metallurgy and mechanical engineering. There are numerous failure mechanisms that might occur, some appear more often than others, which includes various types of corrosion or wear itself, corrosion in combination with wear, and compression to name a few. Failure of engineering products and structures can occur by cyclic application of stresses (or strains), the magnitude of which would be insufficient to cause failure when applied singularly. Structural and mechanical components subjected to fluctuating service stress (or more appropriately, strain) are susceptible to failure by fatigue. Fatigue is considered as one of the most cause of structural and machinery component failures which are frequently found in engineering services. Fatigue failure is localized structural damage that occurs when a material is subjected to variable cyclic stresses. These stresses are much lower than the ultimate tensile stress limit when under the application of a single static stress.

Advancement in materials used in construction of rocker arm for reducing the noise, weight and higher strength for efficient operation is going on throughout the globe since long. The usual materials used for such purpose are Steel, Aluminum, and Forged steel to Stainless steel, alloys and composites. The success to investigate the possibility creating a light weight rocker arm that could provide a friction reducing fulcrum using needle bearings and a roller tip for reduced friction between the rocker and the valve stem but still be less expensive than steel lies in the development of composite rocker arms. Lighter mass at the valve is also allowed for increased speed while strength of the material caters to durability. The rocker arm usually operates at 40^0-500^0 C and the maximum pressure is exerted by the gas. Therefore in this investigation it has been thought proper to analysis a composite rocker arm of high density polyethylene (HDPE) reinforced with short S-glass fibers of 10% volume fraction. Finite element analysis may be carried out to determine the stresses and make a comparison between steel and composite to predict the failure modes.
The objective of this study is to measure stress, strain and stress concentration on rocker arm. Also increase strength and durability by finding best material for rocker arm.
By considering all above facts, this paper tries to cover literature which deals with Design and Analysis of Rocker Arm.

2. FINITE ELEMENT ANALYSIS OF A CAR ROCKER ARM

Tawanda Mushiri, et al. [1], worked on Finite Element Analysis of a Car Rocker Arm, they considered High Density Polyethylene (HDPE) composite rocker arm for analysis owing to its light weight, higher strength and good frictional characteristics. They carried out a 3-D finite element analysis to find out the maximum stresses developed in the rocker arms made of steel and composite. From their results it was noted that almost same stresses are developed for both the materials (steel and the composite). With this they concluded that the stresses developed in the composite are well within the limits without failure. Therefore they considered the proposed composite as an alternate material for steel to be used as rocker arm. Also they found that some rocker arms are made out of glass fiber reinforced high density polyethylene composite (HDPE).

3. ANALYSIS OF ROCKER ARM

Jafar Sharief, et al. [2], published paper on Analysis of a Rocker Arm, they found the various stresses under extreme load condition. For this they were modelled the arm using design software and the stressed regions are found out using

ANSYS software. Here in this thesis they were observed that by changing different materials how the stresses are varied in the rocker arm under extreme load condition. After comparing results they are proposed best suitable material for the rocker arm under extreme load conditions. The usual materials used for such purpose are Steel, Aluminum, and Forged steel to Stainless steel, alloys and composites. They investigated the possibility creating a light weight rocker arm that could provide a friction reducing fulcrum using needle bearings and a roller tip for reduced friction between the rocker and the valve stem but still be less expensive than steel lies in the development of composite rocker arms.

4. ANALYSIS OF A FAILED ROCKER ARM SHAFT OF A PASSENGER CAR ENGINE

G. A. Nassef, et al. [3], studied the failure of a rocker arm shaft of a passenger car. Analysis of a Failed Rocker Arm Shaft of a Passenger Car Engine, in this paper they found that the shaft was failed by brittle fracture across one of the four holes supported the shaft into the cylinder head. Microscopic observations of the failed shaft revealed that the four dark-etching areas are surface hardened zones of martensitic microstructure. Furthermore, they were scanned the microstructure along the failed shaft showed that the heat treatment was so mistakenly extended by excessive heating so that the structure of the shaft near the supporting holes contains considerable content of martensitic phase. Finally they confirmed the results of hardness measurements along the surface of the shaft.

5. DESIGN OF ROCKER ARM

Syed Mujahid Husain, et al. [4], worked on Design of Rocker Arm, they were discussed about Rocker arm of Tata Sumo victa that was designed and analysed to find the critical regions. They were created CAD models of Rocker Arm using Pro/E and ANSYS software was used for analysis of rocker arm. They found that the CAD model was inputted in ANSYS Workbench and Equivalent Stress and Maximum Shear Stress. The obtained results provided by ANSYS Workbench are compared to the results obtained by manual calculation. They found the shear stress at the pin of a rocker arm was evaluated by calculation and ANSYS software. The values of both shear stress and critical shear stress was nearly same. Thus they concluded that pin of rocker arm is under shear stress.

6. STRESS ANALYSIS OF GLASS/HDPE COMPOSITE ROCKER ARM BY FINITE ELEMENT METHOD

Antaryami Mishra, [5], published paper on Stress Analysis of Glass/HDPE Composite Rocker Arm by Finite Element Method. He used the light weight and reasonably high strength composite of Glass/HDPE as rocker arm. Even at rigorous loading conditions the composite rocker arm can withstand the load equivalent to that sustained by a steel rocker arm which is still in use. He found that maximum stress location that may be after a prolonged period of use the rocker will fail at sharp corners are existing. A die for compression molding of Glass/HDPE composite has already been developed in house so that a composite rocker arm can be fabricated in future.

7. STRUCTURAL ANALYSIS OF ROCKER ARM

Kim Ho-Kyung, et al. [6], studied on Structural Analysis of Rocker Arm. They done the modelling of the rocker arm by using Pro-e and analysis is performed by ANSYS. To find strength of the model in structural analysis they were taken 4 different materials and taken 3 load points on the model. They did analysis on model by applying loads at pin and end side by varying different 4 materials. By the results they were observed that the stress values of steel and alloy steel materials are nearer to each other and also for the total deformation the values of the steel and alloy steel got nearly same values. But only composite material got the better values in stress intensity and total deformation when compared to other materials. So by the investigation they were concluded that by using composite material the stress values are reduced by that the life time of the rocker arm increases.

8. CONCLUSIONS

From the literature survey it can be seen that the design of rocker arm has been a good research topic for many researchers, due to its important role in stress analysis in rocker arm. The researchers started from developing theories related to fatigue failure of rocker arm and further moving to finding out various parameters according to their application.

REFERENCES

[1] Tawanda Mushiri and Charles Mbohwa, “Finite Element Analysis of a Car Rocker Arm”- Case study review, 2015.

[2] Jafar Sharief and K. Durga Sushmitha, “Analysis of Rocker Arm”, November 2015.

[3] G.A. Nassef, Elkhatib and Mostafa Yakout, “Analysis of a Failed Rocker Arm Shaft of a Passenger Car Engine”- Case Study Review, July 2015.

[4] Syed Mujahid Husain and Siraj Sheikh, “Design of Rocker Arm”, International Journal of Mechanical Engineering, July 2013.

[5] Antaryami Mishra, “Stress Analysis of Glass/HDPE Composite Rocker Arm by Finite Element Method”- International Journal of Engineering Science and Innovative Technology, May 2014.

[6] Chung, C.-S. & Kim, H.-K. "Safety Evaluation of Rocker Arm of A diesel Engine", Journal of Materials & Design, 31(2), pp. 940-945, 2010.

[7] Dong Woo Lee, SeokSwoo Cho and Won SikJoo "An Estimation of Failure Stress Condition In Rocker Arm Shaft Through FEA and Microscopic Fractography", 2009.