Vol-07 Issue 11, November- 2023
ISSN: 2456-9348
Impact Factor: 6.736

International Journal of Engineering Technology Research & Management
www.ijetrm.com

FEA OF EXISTING ROCKER ARM OF COMPRESSION GARBAGE TRUCK

Amol Chandrakant Babar [1], Swapnil Dattatray Pawar [2]
[1]Head of Department, Department of Mechanical Engineering, Karmayogi Institute of Technology (Polytechnic), Pandharpur
[2] Lecturer, Department of Mechanical Engineering, Karmayogi Institute of Technology (Polytechnic), Pandharpur

ABSTRACT
Experimental analysis of existing rocker arm is performed to determine stress and deformation. Later topology optimization is to be carried out for best optimized model leads to saving of material, cost as well as sustaining existing boundary condition. Manufacturing of new optimized model is compared with existing and analyzed in ANSYS software.
This project focuses on design optimization of Rocker arm targeting weight reduction with required strength and stiffness. Optimized design of arm is mandatory with appropriate material selection as well as valid finite element analysis. Optimization was performed considering static analysis of stress, strain and total deformation along with the suitable material selection. The experimental testing will be performed on universal testing machine. After making the comparative analysis result and conclusion was drawn.

Keywords:
Rocker arm, Structural steel, FEA, Topology optimization of Rocker arm

INTRODUCTION

The flip barrel system is an important part of compression garbage truck. The main function is to grab the trash barrel and turn it so that the garbage can is dumped into the garbage truck. The flip barrel system mainly includes support, flip oil cylinder, rocker arm, pull rod, and dump rack. The flip oil cylinder will be outstretched driving the trash barrel to rotate counter clockwise around point A and dump the garbage into the compression garbage truck in working condition. The garbage barrel is returned to its original position when the flip oil cylinder is retracted. It is difficult to determine the working conditions of the rocker arm, which is the key structure and has complex forces, since the flip barrel system is a four-link mechanism. The analogy method is often used in the traditional design process. Topology optimization is to be carried on rocker arm of compression garbage component for material as well as cost. Optimized model is to be analyzed in ANSYS to compare the safety of existing design.

1. Literature Review

1. Finite Element Analysis of a Rocker Arm of a Linear by Mr. Rahul Kirti.
The rocker arm is an extremely important component in the operation of an internal combustion engine. It is responsible for the opening and closing the intake and exhaust valves. Rocker Arms are typically in between the pushrod and the intake and exhaust valves. They allow the pushrods to push up on the rocker arms and therefore push down on the valves. In the present work, a three dimensional solid model of Rocker Arm is designed with the help of CATIA, the model is saved in .igs format. The model is then transferred to ANSYS for static analysis. The von-Mises stress and displacement values of rocker arm made from material aluminium ALDC8 alloy is calculated and compared with the available values for validation. The FE analyses of rocker arm for two different materials are also compared. The two materials used are chrome moly steel, and a composite material E Glass/epoxy. The weight of Rocker arm is also compared for these materials.
2.Finite element analysis of a car rocker arm by Tawanda Mushiri.
High Density Polyethylene (HDPE) composite rocker arm has been considered for analysis owing to its light weight, higher strength and good frictional characteristics. A 3-D finite element analysis was carried out to find out the maximum stresses

iJETRM
International Journal of Engineering Technology Research & Management
www.ijetrm.com

developed in the rocker arms made of steel and composite. From the results it was noted that almost same stresses are developed for both the materials (steel and the composite). With this it may be concluded that the stresses developed in the composite is well within the limits without failure. Therefore, the proposed composite may be considered as an alternate material for steel to be used as rocker arm. Failure analysis is a broad discipline that includes sectors of engineering such as metallurgy and mechanical engineering. There are a number of failures that might occur, some appear more often than others, which include various types of corrosion or wear by itself, corrosion in combination with wear, and compression to name a few. Failure of engineered 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.

3. Design And Analysis of a Rocker Arm by Jafar Sharief, K. Durga Sushmitha.

Rocker arms are part of the valve-actuating mechanism. A rocker arm is designed to pivot on a pivot pin or shaft that is secured to a bracket. The bracket is mounted on the cylinder head. One end of a rocker arm is in contact with the top of the valve stem, and the other end is actuated by the camshaft. In installations where the camshaft is located below the cylinder head, the rocker arms are actuated by pushrods. The lifters have rollers which are forced by the valve springs to follow the profiles of the cams. Failure of rocker arm is a measure concern as it is one of the important components of push rod IC engines.

4. Failure analysis of bolts on an end flange of a steam pipe by Hsing-Sung Chen, Pi-Tang Tseng, Shun-Fa Hwang.

This paper gives a case study about the failure of steam pipe fitting equipment in a chemical factory. The observation and analysis are done by way of SEM, EDS, tensile tests, and fracture mechanics. The reasons for failure, the fracture mechanisms, and the failure modes are revealed and discussed. A prediction of the fatigue life of a bolt of the fitting flange is made. In the initial stage of the normal operating period, the steam pressure on the end cover is 7.85–9.32 MPa,and the compound pressure of anti-leaking molds on the cover is 10.3 MPa. Hence, by combining the jointforce and the pressure load of vapor, the total maximum operating load carried by the 12 bolts is1296.265 kN. Assuming that there are no cracks in the bolts, the nominal stress is 188.192 MPa on eachbolt. If the stress concentration is considered, its factor Kt is about 2.5 evaluated by the dimensions ofthe threads of the bolt. Hence, the maximum local stress is 470.48 Mpa. It is still lower than the ultimatetensile strength ru (862.30 MPa) and the device is safe. But the safety factor of the combined device is 4.582and 1.833 for the nominal stress and the local concentration stress, respectively.

5. Safety evaluation of the rocker arm of a diesel engine by Chin-Sung Chung, Ho-Kyung Kim.

In order to evaluate the fatigue endurance for the rocker arm of a diesel engine, stress measurementswere performed using strain gages attached near the neck, which is one of the most critical regions in the rocker arm, while varying the engine speed. Fatigue life experiments were carried out on miniature specimens taken from rocker arms. To evaluate the fatigue endurance of the rocker arm, the S–N data were compared with the stress analysis results obtained through a Finite Element Modelling (FEM) analysis of the rocker arm. The von-Mises effective stress of the rocker arm neck region was determined to be22.4 MPa. The safety factors of this component are 2.6 and 3.8, based on the fatigue endurance limit and the modified fatigue endurance limit, respectively, suggesting that this safety factor is appropriate.

6. Experimental And Finite Element Analysis Of Rocker Arm For Bending Failure by Sagar Jadhav, Mr. P. J. Patil, Mr.P. V. Mulik.

A rocker arm, in the context of an internal combustion engine, is an oscillating lever that conveys rotating motion of cam lobe to linear motion of the inlet or exhaust valve of an engine. Rocker arms oscillate about rocker arm shaft because of action of push rod on one side and spring action on other side, which causes bending of rocker arm. As the result, the bending stresses are induced. Failure of rockers may take place due to these stresses. Theoretical analysis can been done using theoretical formulae but is necessary to carry out experimental analysis of rocker arm in working condition for better understanding of stresses. The paper deals with theoretical and finite element analysis of rocker arm. The results obtained from these two methods were compared with the results of experimental analysis.

7. Failure analysis of diesel engine rocker arms by Z.W. Yu, X.L. Xu.

This paper presents a failure analysis of two diesel engine rocker arms used in trucks, which failed in service. The fracture occurred at the hole of the rocker arm shaft in two cases. Beach marks and fatigue steps can be observed on the fracture surface. Multiple-origin fatigue is the dominant failure mechanism. A detailed metallurgical investigation was conducted on the failed rocker arms, and compared with a new one. The failed rocker arms present general metallurgical characteristics

ISSN: 2456-9348
Impact Factor: 6.736

iJETRM
International Journal of Engineering Technology Research & Management
www.ijetrm.com

that the periodization of cementite in pearlite appears in all the matrix structure, and a banded structure was observed in the crack origin region.

2. Methodology: FEA

3.1 CAD model and meshing

Computer-aided design (CAD) is the use of computers to aid in the creation, modification, analysis, or optimization of a design. CAD software is used to increase the productivity of the designer, improve the quality of design, improve communications through documentation, and to create a database for manufacturing. CAD output is often in the form of electronic files for print, machining, or other manufacturing operations. Analysis is done by selecting appropriate solver and carrying out the operations in various stages to obtain solution. Particularly analysis is carried out in three stages by performing various operations in software.

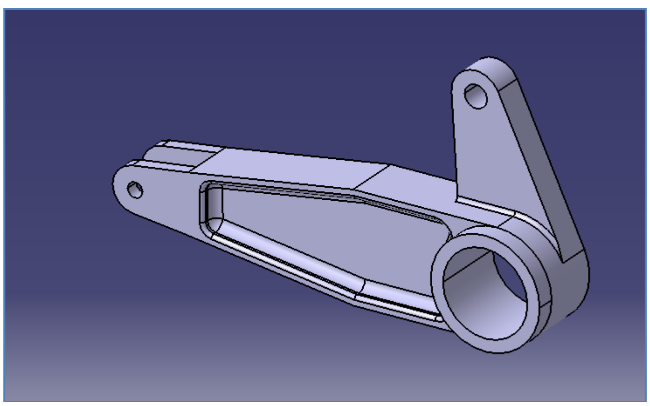

Fig.1 CAD model

Fig.2 Meshing

Table 1 Meshing details

Element Type	Tetrahedral Mesh
Number of Nodes	30912
Number of Elements	18848

3.2 Boundary conditions

A boundary condition for the model is that the setting of a well-known value for a displacement or an associated load. For a specific node you'll be able to set either the load or the displacement but not each. The main kinds of loading obtainable in FEA include force, pressure and temperature. These may be applied to points, surfaces, edges, nodes and components or remotely offset from a feature.

3.3 Topology optimization

Topology optimization is a mathematical approach that optimizes material layout within a given design space, for a given set of loads and boundary conditions such that the resulting layout meets a prescribed set of performance targets.

Results and discussion

Topology optimization excels at automated concept generation; shape optimization allows for efficient final fine tuning of designs. However, every fabrication technique has its shape/property accuracy limitations, particularly at the micro scale. Our activities on shape optimization concentrate on dealing with fabrication inaccuracy, and optimizing for robustness.

iJETRM
International Journal of Engineering Technology Research & Management
www.ijetrm.com

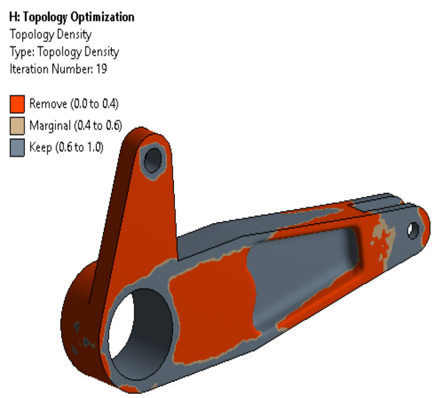

Fig.3- Topology optimization showing the area from where material can be removed.

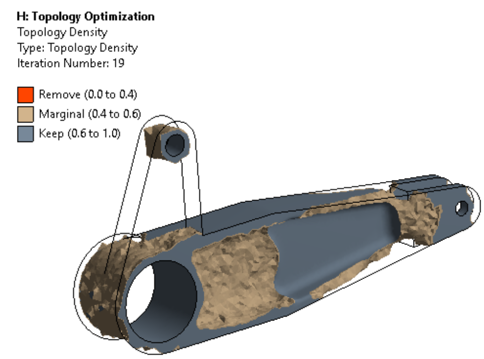

Fig.4 Topology optimization showing the area from where material cannot be removed.

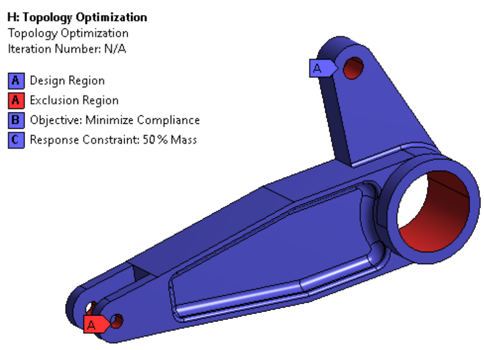

Fig.5 optimized design

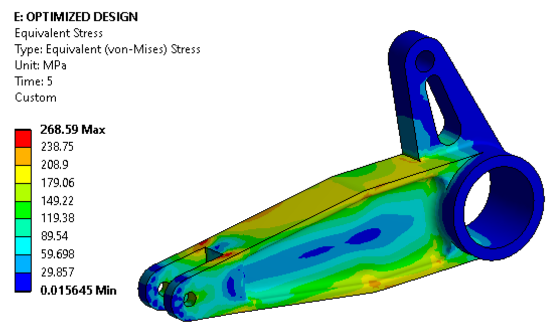

Fig. 6 Equivalent stresses

3. Conclusion

In this semester we have done static analysis of the rocker arm of compression garbage truck with the help of ANSYS Software. We have found the analytical results which shows the equivalent reaction force generated over the rocker arm body for one mm displacement application. After we performed the static analysis, the topology optimization operation over the existing rocker arm is done and theoretically find out the material which can be removed from Rocker arm body. Static structural analysis of Rocker arm is performed to determine deformation and equivalent stress. It is observed that around maximum deformation is 0.53 mm and equivalent stress is 262.37 MPa. An optimized model is obtained from topology optimization technique. Static structural analysis of optimized Rocker arm is performed to determine deformation and equivalent stress. It is observed that around maximum deformation is 0.7455 mm and equivalent stress is 268.59 MPa.

REFERENCES

1. Finite Element Analysis of a Rocker Arm ofa Diesel Engine Using Radioss Linearby Mr. Rahul Kirti.
2. Finite element analysis of a car rocker arm by Tawanda Mushiri.
3. Design And Analysis of a Rocker Arm by Jafar Sharief, K.Durga Sushmitha.
4. Failure analysis of bolts on an end flange of a steam pipe by Hsing-Sung Chen, Pi-Tang Tseng, Shun-Fa Hwang.
5. Safety evaluation of the rocker arm of a diesel engine by Chin-Sung Chung, Ho-Kyung Kim
6. EXPERIMENTAL AND FINITE ELEMENT ANALYSIS OF ROCKER ARM FOR BENDING FAILURE by Sagar Jadhav, Mr. P.J.Patil, Mr.P.V.Mulik.
7. Failure analysis of diesel engine rocker arms by Z.W. Yu, X.L. Xu.