International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 6 Issue 10, October - 2017

Finite Element Analysis of a Rocker Arm of a Diesel Engine Using Radioss Linear

Mr. Rahul Kirti
B. Tech-Mechanical,
M.tech-Machine Design Engineer
Training Department
Central Tool Room, Ministry of MSME, Govt. of India
A5, Phase5, Focal Point, Ludhiana-141010, Punjab, India

***Abstract*— 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 HyperMesh for static analysis. The von-Mises stress and displacement values of rocker arm made from material aluminum 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.

The stress analysis results help to improve component design at the early stage and also help in reducing time required to manufacture the rocker arm component and its cost. The reduction in weight is also helpful in better designing of the engine.

***Keywords*— *Rocker Arm, FE analysis, Hypermesh, von*-Mises stress**

I. INTRODUCTION

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. However, in Over Head Cam applications the Cam will ride directly on the Rocker Arm. In addition to just changing the direction of the motion from up on the rocker arm to down on the valve, the Rocker Arm changes the amount of motion transferred. Typically a Rocker Arm will "multiply" its motion by a Rocker Arm Ratio by a factor of 1.45 to 1.7, meaning that for each .100" of pushrod motion you would get .145" to .170" of valve motion.

Generally referred to within the internal combustion engine of automotive, marine, motorcycle and reciprocating aviation engines, the **rocker arm** is an oscillating lever that conveys radial movement from the cam lobe into linear movement at the poppet valve to open it. One end is raised and lowered by a rotating lobe of the camshaft (either directly or via a tappet (lifter) and pushrod) while the other end acts on the valve stem. When the camshaft lobe raises the outside of the arm, the inside presses down on the valve stem, opening the valve. When the outside of the arm is permitted to return due to the camshafts rotation, the inside rises, allowing the valve spring to close the valve.

Fig.1. A Rocker Arm

The drive cam is driven by the camshaft. This pushes the rocker arm up and down about the trunnion pin or rocker shaft. Friction may be reduced at the point of contact with the valve stem by a roller cam follower. A similar arrangement transfers the motion via another roller cam follower to a second rocker arm. This rotates about the rocker shaft, and transfers the motion via a tappet to the poppet valve. In this case this opens the intake valve to the cylinder head.

The effective leverage of the arm (and thus the force it can exert on the valve stem) is determined by the *rocker arm ratio*, the ratio of the distance from the rocker arm's center of rotation to the tip divided by the distance from the center of rotation to the point acted on by the camshaft or pushrod. Current automotive design favors rocker arm ratios of about 1.5:1 to 1.8:1. However, in the past smaller positive ratios (the valve lift is greater than the cam lift) and even negative ratios (valve lift smaller than the cam lift) have been used. Many pre-WW2 engines use 1:1 (neutral) ratios.

For car engines the rocker arms are generally steel stampings, providing a reasonable balance of strength, weight and economical cost. Because the rocker arms are, in part reciprocating weight, excessive mass especially at the lever ends limits the engine's ability to reach high operating speeds.

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 6 Issue 10, October - 2017

II. PROBLEM FORMULATION

Rocker arm is subjected to compressive load at the rocker pin on opening and closing of the valves, as a result of which stresses are developed in the rocker arm. This may lead to fracture in rocker arm or causes breakage of rocker arm. So this analysis is done to find out the safe limits of stresses and deflections for a rocker arm. Different materials have been tested and there stress values, deflections, total weight are also compared so as to find a suitable material and safe stress value.

The main objective of this work is:

1) To perform the Finite Element Analysis of rocker arm so as to determine the stress distribution, maximum deflections and its location in the rocker arm.

2) To compare materials used for making rocker arm so as to find a perfect material for rocker arm having low stress value and lesser weight.

The preprocessing is done in HyperMesh 11.0 and the solver used is RADIOSS Linear. The CAD model of ROCKER ARM has been generated in CATIA V5R20 and saved in IGES (Initial Graphics Exchange Specification) and then imported in HyperMesh.

The stress contours have been plotted and patterns are studied for different materials. The results are compared and verified with available experiments.

III. WORKING OF ROCKER ARM

The rocker arm is an oscillating lever that conveys radial movement from the cam lobe into linear movement at the poppet valve to open it. One end is raised and lowered by a rotating lobe of the camshaft (either directly or via a tappet (lifter) and pushrod) while the other end acts on the valve stem. When the camshaft lobe raises the outside of the arm, the inside presses down on the valve stem, opening the valve. When the outside of the arm is permitted to return due to the camshafts rotation, the inside rises, allowing the valve spring to close the valve.

The drive cam is driven by the camshaft. This pushes the rocker arm up and down about the turn-on pin or rocker shaft. Friction may be reduced at the point of contact with the valve stem by a roller cam follower. A similar arrangement transfers the motion via another roller cam follower to a second rocker arm. This rotates about the rocker shaft, and transfers the motion via a tappet to the poppet valve. In this case this opens the intake valve to the cylinder head.

IV. FINITE ELEMENT ANALYSIS

The finite element method is a numerical method for solving engineering and mathematical physics problems. The typical use of this method is to solve the problems in the field of stress analysis, heat transfer, fluid flow, and mass transfer and electromagnetic. This method can able to solve physical problems involving complicated geometrics, loadings and material properties which cannot be solved by analytical method. In this method, the domain in which the analysis to be carried out is divided into smaller bodies or unit called as finite elements.

The properties of each type of finite element is obtained and assembled together and solved as whole to get solution. Based on application, the problems are classified into structural and non-structural problems. Finite Element Analysis (or other numerical analysis), development of structures must be based on hand calculations only. For complex structures, the simplifying assumptions required to make any calculations possible can lead to a conservative and heavy design. A considerable factor of ignorance can remain as to whether the structure will be adequate for all design loads. In structural problems, displacement at each nodal point is obtained. Using these displacement solutions, stress and strain in each element are determined.

Similarly, the non-structural problems, a temperature or fluid property at each nodal point is/are obtained. Using these nodal values, properties like heat flux, fluid flow etc., for each element is determined. Since large computations are to be carried out, this method requires high-speed computation facility with large memory. Finite element method (FEM) and Finite element analysis (FEA) are both one and same term. But term FEA is more popular in industries while FEM is famous at universities.

V. PROCEDURE OF FEA

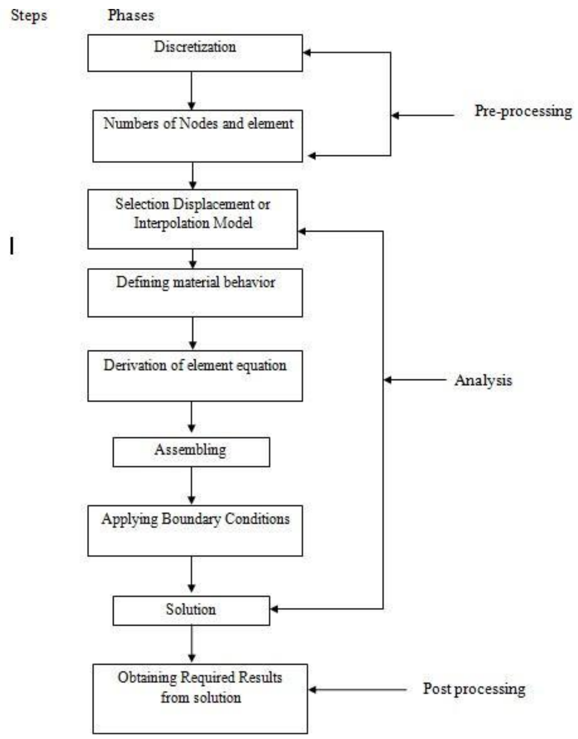

Fig.2. Procedure of FEA

In FEA software's, the general process of finite element method is divided into three main phases, pre-processing, solution, and post processing.

Following are the input data that needs to be given to the pre-processor:

1. Type of analysis (structural or thermal, static or dynamic, and linear or nonlinear)
2. Element type.
3. Real constraints.
4. Material properties.
5. Geometric model.
6. Meshed model.

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 6 Issue 10, October - 2017

7. Loadings and boundary conditions.
The input data will be pre-processed for the output data and pre-processor will generate the data files automatically with the help of users. These data files will be used by the subsequent phase.

Solution phase is completely automatic. The FEA software generates the element matrices, computes nodal values and derivatives, and stores the result data in files. These files are further used by the subsequent phase as for this purpose (postprocessor) to review and analyze the results through the graphic display and tabular listings.

Output from the solution phase (result data files) is in the numerical form and consists of nodal values of the field variable and its derivatives. For example, in structural analysis, the output is nodal displacement and stress in the elements. The postprocessor processes the result data and displays them in graphical form to check or analyze the result. The graphical output gives the detailed information about the required result data. The postprocessor phase is automatic and generates the graphical output in the form specified by the user. Result Viewer and Plot Result are used for Post-Processing in this problem.

VI. MODELING METHODOLOGY

The first step in preprocessing is to prepare a CAD Model of rocker arm. CAD model of the complete rocker arm is generated using CATIA V5 R20 software. CATIA is having special tools in generative surface design to construct typical surfaces, which are later on converted into solid. CAD model of our problem consists of a single component. The model used in the present work is saved in IGES (Initial Graphics Exchange Specification) format which is compatible with all CAD software. After importing the CAD file into HyperMesh, it is saved in .hm format.

After importing the CAD data, the first step is geometry cleanup. Geometry cleanup tools are used to restore proper surface connectivity to part geometry. The geometry panel contains tools like quick edit, edge edit, point edit, auto cleanup etc. which help in preparing surface geometry for meshing. Meshing quality depends very much on the quality of geometry.

The element size used for 3D mesh is 4mm. Element size was decided after checking the convergence of von-Mises stress of Rocker arm. The following table 1 shows the value of maximum von-Mises stress and displacements for different element lengths.

Table No.1
Maximum Von-Mises Stress at Different Element Sizes

Element Size (mm)	Min. Element size(mm)	Maximum von-Mises Stress (Mpa)	Maximum Displacement (mm)
1	0.4	23.7	2.6
2	0.6	22.6	2.5
3	1	19.5	2.43
4	1	19.3	2.24
5	1	19.4	2.23

Element types used for 3D surface meshing were Quads and Tetra mesh type as these are more accurate for 3D parts. Tetra meshing and Volume tetra meshing is used for 3D mesh generation.

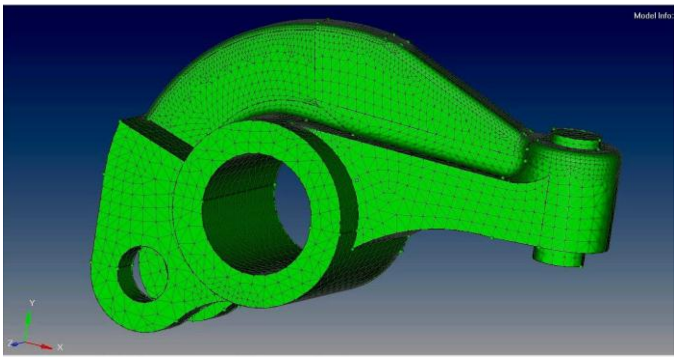

Fig.3. 3D meshed model of Rocker arm

Mesh details are as follows:
Element size 4mm
Minimum element size 1mm
Feature angle 30°
Number of Nodes generated: 4814
Number of Elements formed: 18687
Total no. of Degree of Freedoms: 14024
2D type elements- Quads
3D type elements- Tetras

After meshing various component collectors are created. The meshes of the different surfaces are put together in different component collectors. The next step after putting elements in various component collector is to create material collector an assigning the material to elements. After creating material collectors, property collectors are created. Element type, card image and material are assigned to various property collectors. For rocker arm model element type is 3D, card image used is PSOLID and material is cast aluminum ALDC8

VII. BOUNDARY CONDITIONS

Load collector is the collection of boundary conditions i.e. different forces, pressure, velocity, supports, constraints and any other condition required to complete analysis. Applying boundary condition is one of the most typical processes of analysis. A special care is required while assigning loads and constraints to the elements. Boundary condition is of two types:

1. Element based boundary condition
2. Node based boundary condition

In element based boundary condition a group of elements is selected on which various loads are to be applied. Whereas in node based boundary condition nodes of mesh are selected for applying load and forces. We have used the node based boundary condition in this analysis work for applying load at the rocker arm pin and constraints around the pin holes.
Chin-Sung Chung and Ho-kyung Kim [1] assessed the magnitude of an applied load on a rocker arm. The rocker arm used by them has been used in this analysis. To find the magnitude of applied load on the rocker arm, it is necessary to determine the compressive load on the valve spring during rotation of cam. The valve spring constant was determined to be 29.6 N/mm using a servo hydraulic tensile machine.

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 6 Issue 10, October - 2017

The maximum cam displacement was determined to be 6 mm by rotating the cam manually using a dial gauge. Thus the effective vertical stroke of the cam (L) is 4.48 according to the equation L=6 mm x cos36.3o. The compressive displacement of the valve spring (X) due to the effective vertical stroke of cam can be determined to be 8.87 mm by applying the level principle of 23.50 mm: 43.06 mm = 4.84 mm: X mm. The actual compressive displacement of the valve was determined to be 8.62 mm, taking into consideration the normal valve gap of 0.25 mm.

Finally, the load due to the compression of the valve spring was found to be 255.2 N by multiplying the compressive displacement by the spring constant (8.62 mm X 29.6 N/mm = 255.2 N). The initial compressive load of the valve spring due to installation of the spring was determined to be 199.6 N by multiplying the valve spring constant (29.6 N/mm) by the maximum spring displacement of 6.75 mm.

Finally, the maximum static compressive force on the rocker arm was determined to be 454.8 N by adding a compressive load of 255.2 N, as induced by lifting the cam to the initial compressive spring load of 199.6 N, caused by installation of the valve spring as shown in fig. 4

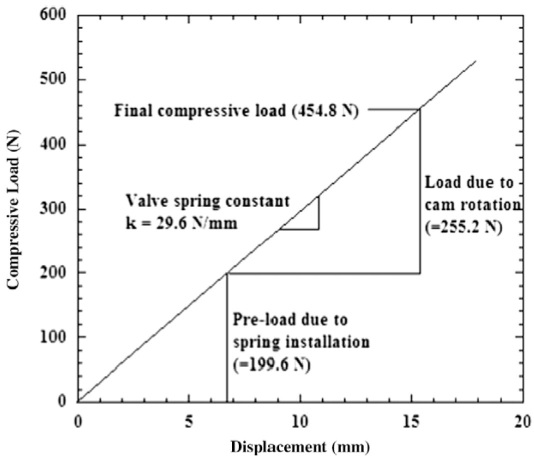

Fig.4. Determination of compressive load of the valve spring

The rocker arm was constrained by linking the nodes around the pin hole inner surface, representing only rotation along z-axis. The rigid element of the larger pin was linked by inner lower surface nodes, while that of the smaller pin hole was linked by inner upper surface nodes, this is shown in fig 5. Blue color triangles show SPC (single point constraint).

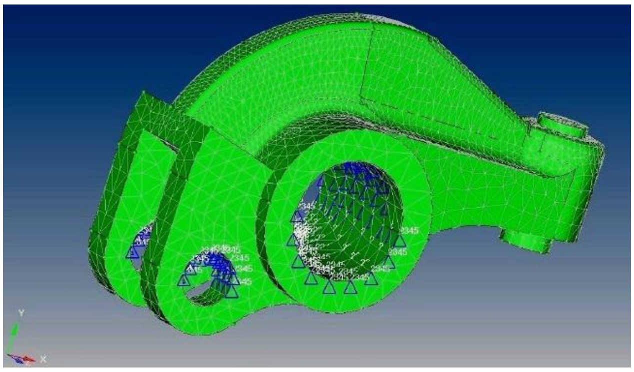

Fig.5. Constraints applied on Rocker arm

Figure 6 shows the distribution of force around the rocker pin, as the load is equally divided along the nodes on the rocker pin so:

No of nodes selected on rocker pin- 28

Total load on the rocker pin- 454.8N

Load on each node- 454.8/28 = 16.243N along y-axis.

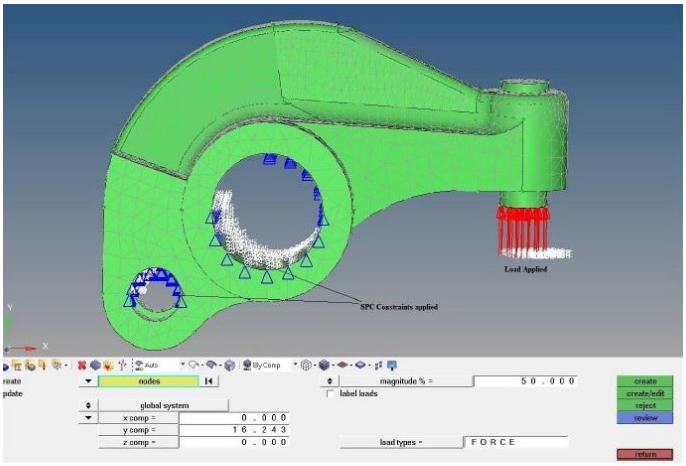

Fig.6. Distribution of load on the Rocker pin

VIII. RESULT AND DISCUSSION

Cast aluminum ALDC8 is a high strength, lightweight material for Rocker arm. Lightweight rocker arms are a plus for high rpm applications, but strength is also essential to prevent failure.

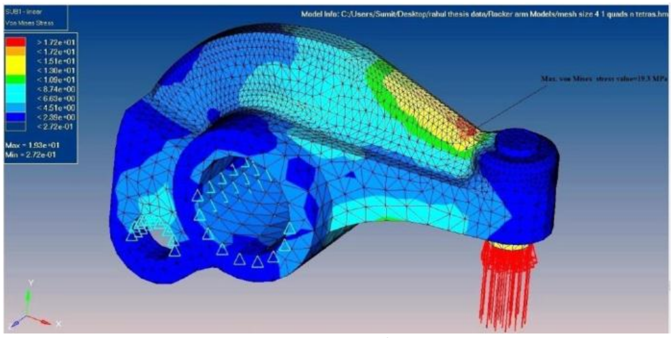

Fig.7. von-Mises stress contour of rocker arm/al ALDC8

Figure 7 shows Von-Mises stress developed in the rocker arm at normal loading conditions.

The maximum stress observed is 19.3MPa at the neck of rocker arm as shown in the figure. The value of stress is well below the yield stress which is 225MPa.

During analysis it was found that von-Mises stresses are almost same for three different material used (al ALDC8, Chrome moly steel alloy, E Glass/epoxy composite material).

The stress is found maximum at the neck of rocker arm. This can be reduced by increasing the material near the neck region. For validation of FEA results, the results have been compared with existing results [1] of the similar model under same loading conditions. For above loading conditions, the stresses developed are found to be well below the allowable yield stress (225MPa).The maximum von Mises stresses in the analysis is found to be 19.3MPa. The maximum displacement in the analysis is found to be 0.024mm which is in the acceptable range.

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 6 Issue 10, October - 2017

A. Weight comparison of the three materials:

By changing the material from chrome moly steel to aluminum alloy ALDC8 and to composite material E glass/ epoxy the weight of the rocker arm is optimized.

Table No.2

Comparison of weight of Rocker arm using different materials

Sr.No.	Name of material used for Rocker arm	Weight of Rocker arm (grams)
1.	Aluminum ALDC8	74.4
2.	Chrome Moly Steel	214.9
3.	E Glass/epoxy Composite material	52.35

Clearly from the table 2 by using composite materials we can reduce the weight of rocker arm, which in turn reduce the weight of engine and the automobile. By using E Glass/epoxy composite material the weight of Rocker arm becomes almost five times lighter than it was when using chrome moly steel as a material.

REFERENCES

[1] Chin-Sung Chung, Ho-Kyung Kim "Safety evaluation of the rocker arm of a diesel engine" Materials & Design, Volume 31, Issue 2, February 2010, Pages 940-945`

[2] Eric Palomaki and Kyle Hoffman "Stock Rocker vs. roller Rocker" Introduction to Finite Elements. Rensselaer Polytechnic Institute Troy, New York.

[3] Dong Woo Lee , Seok Swoo Cho and Won Sik Joo "An estimation of failure stress condition in rocker arm shaft through FEA and microscopic fractography", Journal of Mechanical Science and Technology 22 (2008) 2056~2061

[4] Z.W. Yu, X.L. Xu "Failure analysis of diesel engine rocker arms" Engineering Failure Analysis, Volume 13, Issue 4, June 2006, Pages 598- 605

[5] Xiao-lei Xu • Yu-zhou Gao • Zhi-wei Yu "Fatigue Fracture of a Truck Diesel Engine Rocker-Bracket" ASM International 2013.

[6] Bruce K. Geist & David Eovaldi "Analysis and Customization of Rocker Arm Joint Sliding Velocity", Journal of Engineering for Gas Turbines and Power November 2009, Vol. 131 / 062801-7.

[7] M.Merlin, C. Soffritti, R Vazquez, G.L. Garagnani "Tribological characterization of a Diesel Engine Hydraulic Tappets" Materials characterization in steel and metal industries, CETAS,Luxembourg,17-19 May 2011.

[8] N. Hendriksma, T. Kunz and C. Greene "Design and Development of a 2-Step Rocker Arm", SAE International, 2007 World Congress Detroit, Michigan April 16-19, 2007

[9] A.S More, S.P Deshmukh "Analysis of Valve Mechanism – A Review", IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE) ISSN (e): 2278-1684, ISSN (p) : 2320–334X, PP : 06-09.

[10] Mohd Hafiz Bin Ghazalli "Finite Element Analysis of Cam and its Follower Contact Stress Mechanism".

[11] Massimo Giorelli "Methodology for Correlating Experimental and Finite Element Modal Analysis On Valve Trains" Worcester Polytechnic Institute (2002).

[12] Jose Alejandro Escobar "Stress and Fatigue analysis of SVI- tested cam shaft lobes".

[13] Dragan D. Kreculj "Stress Analysis in an Unidirectional Carbon/Epoxy Composite Material" FME Transactions VOL. 36, No 3, 2008 -129.