ABSTRACT

The component rocker arm is one of the most important parts in valve actuating mechanism of an internal combustion engine. From many years research is going on in the automotive industries considering various factors of optimization like cost, weight, stresses, material compositions, etc. The failure of the rocker arm in valve actuating mechanism is measure concern. To study and analyze stresses generated, we are modelling rocker arm using CATIA CAD software and the various regions of stresses and deformations found out using OPTISTRUCT-RADIOSS structural analysis solver which gives solution for structural optimization using finite element analysis (FEA). In this project, we are using four different materials preferred in industries which are structural steel, aluminium, carbon steel and HMCF (High modulus carbon fibre). So by comparing analytical results and FEA results for all above materials we can study stress level and deformation in rocker arm at extreme load conditions and going to propose best suitable material among above materials on the basis of strength, durability, life and cost of material. The main purpose of this study is to determine the value of stresses in rocker arm at extreme conditions.

KEYWORDS: Rocker arm, Valve actuating mechanism, CATIA, OPTISTRUCT-RADIOSS, FEA.

I. INTRODUCTION

The rocker arm is an important part of the valve train mechanism in fuel injection systems of I.C. engines to satisfy functional requirement opening of inlet and exhaust valves [1]. It actuates the valves through a fulcrum using the lifter and the push rod. It also provides a means of multiplying the lift ratio. Recently there has been the advancement in the research of materials used in the construction of rocker arms. Researchers are looking for materials that reduce noise, weight and have higher strength for efficient operation. As of the material requirements of the rocker arm is low weight, a higher strength, low cost, good thermal stability, etc. The most popular materials used for construction of rocker arms are Steel, Aluminium, and forged steel to Stainless steel, alloys and composites [10]. If we discuss a functional requirement of 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 allowed to return due to the camshaft’s revolution, the inside rises, permitting 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. The friction may reduce at the point of direct contact with the valve stem by a roller cam follower due to roller. A similar arrangement transfers the motion via another roller cam follower to a second rocker arm. It rotates about rocker shaft and transfers the motion via tappet to the poppet valve. In this case, this opens the intake valve to the cylinder head of engine. The failure of rocker arm makes engine useless also requires costly replacement. Designers are facing a lot of problems especially, stress concentration and effect of loads and forces and other factors. The finite element analysis (FEA) method is the most popular approach and found commonly used for analyzing fracture mechanics problems [3].
Therefore it needs to carry out a detailed FEA analysis work to study and calculate deformations and stresses in rocker arm to understand the failure modes and to compare various materials on the basis of strength, durability, life, cost, etc.

II. OBJECTIVE
The objective is to design rocker arm of aluminium, carbon steel, structural steel, HMCF (High Modulus Carbon Fibre) using CATIA V5 and to carry out the finite element analysis (FEA) on the designed model using OPTISTRUCT 13 static analysis solver. Thus we obtained various values of deformations and stresses by manual calculations and FEA results. This result was then compared to various above materials.

2.1 Problem Description
Due to problems associated with rocker arms like strength, cost and weight, there is needed to consider other alternative materials for rocker arm. Analysis of the material properties is important before they can be implemented.

2.2 Necessity
To avoid rocker arm fatigue failure due to stresses created in rocker body. It is necessary to identify the main contributed stress development in rocker arm and how it can be reduced. This work presents design work analysis of stress development factors using 3-D CAD models & analysis software in rocker arm by finite element engineering simulation program.

2.3 Objective of Work
1) To design rocker arm of steel, aluminium, HMCF (High Modulus Carbon Fibre) using CAD software CATIA V5.
2) To carry out the finite element analysis (FEA) on the designed model using OPTISTRUCT-RADIOSS 13.
3) FEA analysis is performed according to ASME Code Section VIII, Division 2
4) To find the values of deformations and stresses in rocker arm for various materials.

III. MATERIALS AND PROPERTIES
The most common rocker arm materials are steel and aluminium. Clarifying on the material science of rocker arms, Comp Cams explains some interesting facts about steel rocker arms. Chrome-molybdenum steel, although heavier than other materials, offers some design advantages and has much thinner sections than aluminium and its alloy due to its superior strength density.

### Table 1. Material Properties

<table>
<thead>
<tr>
<th>Material</th>
<th>Aluminium</th>
<th>Carbon Steel</th>
<th>HMCF</th>
<th>Structural Steel</th>
</tr>
</thead>
<tbody>
<tr>
<td>Density (gm/cm$^3$)</td>
<td>2.7</td>
<td>7.85</td>
<td>1.6</td>
<td>7.85</td>
</tr>
<tr>
<td>Young’s Modulus (GPa)</td>
<td>70</td>
<td>200</td>
<td>175</td>
<td>200</td>
</tr>
<tr>
<td>Ultimate strength (MPa)</td>
<td>310</td>
<td>585</td>
<td>1000</td>
<td>460</td>
</tr>
<tr>
<td>Yield Strength (MPa)</td>
<td>276</td>
<td>415</td>
<td>110</td>
<td>250</td>
</tr>
<tr>
<td>Poison’s Ratio</td>
<td>0.35</td>
<td>0.285</td>
<td>0.3</td>
<td>03</td>
</tr>
</tbody>
</table>

**IV. METHODOLOGY**

1. Analytical Design calculations of the rocker arm.
2. Creating of 3-D Model by using any Modelling (CAD) Software.
3. Finite Element Analysis (FEA).
4. Comparison of Analytical results and FEA results.

**V. ENGINE SPECIFICATIONS**
VI. DESIGN AND CALCULATIONS

Table 3: Inputs to analytical calculations

<table>
<thead>
<tr>
<th>Design Variables And Inputs</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Inner dia. of the fulcrum (d1)</td>
<td>22 mm</td>
</tr>
<tr>
<td>Bush thickness</td>
<td>2 mm</td>
</tr>
<tr>
<td>Mass of valve (M_v)</td>
<td>0.09 Kg</td>
</tr>
<tr>
<td>Sheet thickness (t)</td>
<td>3.5 mm</td>
</tr>
<tr>
<td>Diameter of the valve head (D_v)</td>
<td>40 mm</td>
</tr>
<tr>
<td>Valve lift (h)</td>
<td>9.4 mm</td>
</tr>
<tr>
<td>Rpm of engine</td>
<td>3200 RPM</td>
</tr>
<tr>
<td>Cylinder pressure (P_c)</td>
<td>0.4 N/mm²</td>
</tr>
<tr>
<td>Maximum suction pressure (P_s)</td>
<td>0.02 N/mm²</td>
</tr>
<tr>
<td>Diameter of boss (D_b)</td>
<td>34 mm</td>
</tr>
<tr>
<td>Length of valve side arm (L_v)</td>
<td>41 mm</td>
</tr>
<tr>
<td>Rocker arm ratio</td>
<td>1.64</td>
</tr>
<tr>
<td>Angle action of cam (Θ)</td>
<td>110 deg</td>
</tr>
<tr>
<td>Height of push rod contact to bush center (h_2)</td>
<td>10 mm</td>
</tr>
<tr>
<td>Height of valve contact to bush center (h_1)</td>
<td>3 mm</td>
</tr>
<tr>
<td>Spring Rate (K)</td>
<td>23 N/mm</td>
</tr>
</tbody>
</table>
1. Total load on the valve
   \[ P_t = P_g + w \]
   \[ P_t = 503.53 \text{N} \]

2. Initial Spring Force, \( F_s \)
   \[ F_s = \frac{\pi}{4} \times d_v^2 \times P_s - w \]
   \[ F_s = 24.249 \text{ N} \]

3. Force due to the acceleration of the valve

   The speed of camshaft \( (Ns) = \frac{N}{2} = 1600 \text{ rpm} \)

   And angle turned by the camshaft per second
   \[ = \left( \frac{1600}{60} \right) \times 360 = 9600 \text{ deg/s} \]

   Therefore time is taken for the valve to open and close
   \[ t = \frac{\text{Angle of action of cam}}{\text{Angle turned by camshaft}} \]
We know that maximum acceleration of the valve
\[ a = \omega^2 \times r = \left(\frac{2\pi}{t}\right)^2 \times r \]
\[ = 1550.00 \text{ m/s}^2 \]
Therefore force due to valve acceleration, considering the weight of the valve,
\[ F_a = m \times a + w \]
\[ F_a = 140.38 \text{ N} \]
4. maximum load on the rocker arm for exhaust valve,
\[ F_e = P + F_s + F_a + (P_1 + K \times h) \]
\[ F_e = 1133.87 \text{ N} \]
Load on push rod side arm (\( F_c \))
\[ F_c = F_e \times \text{Rocker arm ratio} \]
\[ F_c = 1859.54 \text{ N} \]
6.1 Calculations for Bending Stress of Cross Section
The Rocker arm may be treated as a simple supported beam and loaded at the fulcrum point. Therefore, due to the load on the valve the rocker arm is subjected to bending moment.

We know that maximum bending moment (\( M \)) of cross-section,
\[ M = F_e \times \left( l - \frac{d_1}{2} \right) \]
\[ M = 27212.89 \text{ N-mm} \]
The rocker arm is of C-section, Section modulus (\( Z \)),
\[ Z = \frac{l}{b-c_x} \]
\[ I = \frac{2ab^3 + (c-2a)a^3}{(3-AC_x)} \]
Where,
\[ A = 2ab + a(c-2a) \]
\[ = 203 \text{ mm}^2 \]
\[ C_x = \frac{2ab^2 + a^2(c-a)}{4ab + 2a(c-2a)} \]
\[ = 7.4396 \text{ mm} \]
Therefore,
\[ I = \frac{2 \times 3.5 \times 203^3 + (25 - 2 \times 3.5)3.5^3}{(3-203 \times 7.4396^2)} \]
\[ = 7688.176 \text{ mm}^4 \]
We know that, Section modulus is
\[ Z = \frac{l}{b-c_x} \]
\[ Z = 612.0992 \text{N-mm}^3 \]
Bending stress,

\[ \sigma_b = \frac{M}{Z} \]

\[ \sigma_b = 44.458 \text{ N/mm}^2 \]

VII. Modeling in CATIA

The rocker body was a forged body entity. The model was created in CATIA CAD software.

For analysis, a model was meshed using tetrahedron elements for rocker arm and hex elements for rocker shaft. The number of Nodes used in this meshing is 48261 and elements are 141408. Considering the practical application of rocker arm, we are constrained all degrees of freedom of rocker shaft. Force applied at push rod side is 1860 N and valve end side 1134 N.

Rocker Arm 1: Aluminium
Rocker Arm 2: Carbon Steel
Rocker Arm 3: Structural Steel
Rocker Arm 4: HMCF UD
VIII. RESULTS AND DISCUSSION
Numerically, by using mathematical formulae we calculated the stress level and shear stress at fulcrum pin in the rocker arm. The mathematical results are then compared or validated with FEA analysis results.

9.1 Numerical Results
Bending stress in all rocker arm is calculated analytically by using mathematical formulae results are as follows,

Maximum Bending stress ($\sigma_b$) = 44.458 MPa
Maximum Shear Stress ($\tau_{\text{max}}$) = 29.459 MPa

9.2 Finite Element Analysis Results:

<table>
<thead>
<tr>
<th>Material Used</th>
<th>Stress (MPa)</th>
<th>Shear Stress (MPa)</th>
<th>Deformation (mm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Aluminium</td>
<td>46.394</td>
<td>24.389</td>
<td>0.0842</td>
</tr>
<tr>
<td>Carbon Steel</td>
<td>48.358</td>
<td>25.113</td>
<td>0.0150</td>
</tr>
<tr>
<td>Structural Steel</td>
<td>47.95</td>
<td>24.964</td>
<td>0.0140</td>
</tr>
<tr>
<td>HMCF UD</td>
<td>47.95</td>
<td>24.964</td>
<td>0.0170</td>
</tr>
</tbody>
</table>

IX. CONCLUSION
Study and analysis of bending stresses, shear stresses and deformations level in rocker arm body are carried out as per the ASME section code and it can be concluded that Finite Element analysis is required to match the results of analytical calculations. This study is not exhaustive and conducted by Finite Element analysis of a Rocker Arm using 3D modelling and post-processing for analysis of rocker arm with rocker shaft according to ASME codes.

Stresses within a rocker arm body with rocker shaft on its periphery are investigated we can conclude that stress at the fulcrum pin and drain hole is maximum. Due to fatigue rocker may fail near fulcrum pin at push rod side.

2. Shear stress is maximum at the fulcrum pin, in the case of shear also there may chance of failure of the rocker body at fulcrum pin due cyclic load or fatigue. Thus we conclude that fulcrum of rocker arm is at maximum shear stress. Value of maximum deformation shows a valve end of rocker body that is negligible in value.
3. By analyzing obtained values of stresses, shear stresses and deformations weak areas are identified early in the product development process, allowing some design changes. Alternately, the design can be further optimized by reducing the amount of material used.

4. Stresses within the rocker body are calculated by using FEA and validated with the help of result obtained by using analytical mathematical formulae.

X. REFERENCES


CITE AN ARTICLE