Design and Analysis of Rocker ARM

V.Ponmuthu Selvan[1], Dr.P.Amuthakkannan[2]
[1, 2] Dept of Mechanical Engineering
[2]Associate Professor, Dept of Mechanical Engineering
[1, 2] PSR Engineering College

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 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. 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 ANSYS analysis. In this project, we are using three different materials preferred in industries which are, aluminium, carbon steel and HMCF (High modulus carbon fiber). Comparing ANSYS 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 and cost of material. The main purpose of this study is to determine the value of stresses in rocker arm at extreme conditions.*

Keywords- Design, Analysis, Rocker arm, CATIA, ANSYS.

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. 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. 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 ANSYS analysis method is the most popular approach and found commonly used for analyzing fracture mechanics problems.
Therefore it needs to carry out a detailed ANSYS 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

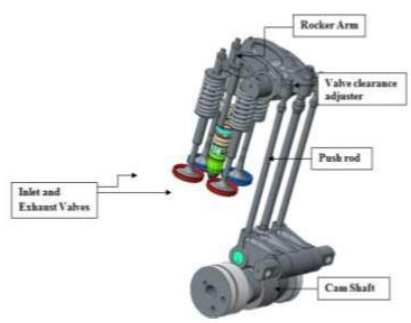

Figure 1-
Position of Rocker Arm in I.C. Engine

II. IDENTIFICATION OF PROBLEM

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.

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 a stress development factors using 3-D CAD models & analysis software in rocker arm by ansys engineering simulation program.

III. 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 ANSYS analysis on the designed model using OPTISTRUCT 13 static analysis solver. Thus we obtained various values of deformations and stresses by manual calculations and ANSYS results. This result was then compared to various above materials.

1) To design rocker arm of steel, aluminium, HMCF (High Modulus Carbon Fibre) using CAD software CATIA V5.
2) To carry out the ansys analysis on the designed model using OPTISTRUCT-RADIOSS 13.
3) ANSYS 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.

IV. 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	Aluminium	CarbonSteel	HMCF
Density (gm/cm^3)	2.7	7.85	1.6
Young's Modulus (GPa)	70	200	175
Ultimate strength (MPa)	310	585	1000
Yield Strength (MPa)	276	415	110
Poison's Ratio	0.35	0.285	0.3

V. METHODOLOGY

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

VI. ENGINE SPECIFICATIONS

Table 2

Type of vehicle	Diesel Engine
No. of Cylinders	6-cylinder V configuration
Capacity	2523cc
Max. Engine Power	46.3 kw @ 3200 rpm
Max. Torque	195NM @ 1440-2200 rpm

VII. DESIGN VARIABLES AND INPUTS

Table 3

Design Variables And Inputs		
Inner dia. of the fulcrum (d1)	22	mm
Bush thickness	2	mm
Mass of valve (Mv)	0.09	kg
sheet thickness (t)	3.5	mm
Diameter of the valve head (Dv)	40	mm
valve lift (h)	9.4	mm
rpm of engine	3200	Rpm
cylinder pressure (Pc)	0.4	N/mm^2
maximum suction pressure (Ps)	0.02	N/mm^2
diameter of boss (D1)	34	mm
length of valve side arm (Lv)	41	mm
Rocker arm ratio	1.64	
angle action of cam (Ø)	110	deg.
height of push rod contact to bush center (h2)	10	mm
height of valve contact to bush center (h1)	3	mm
Spring Rate (K)	23	N/mm

VIII. MODELLING 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

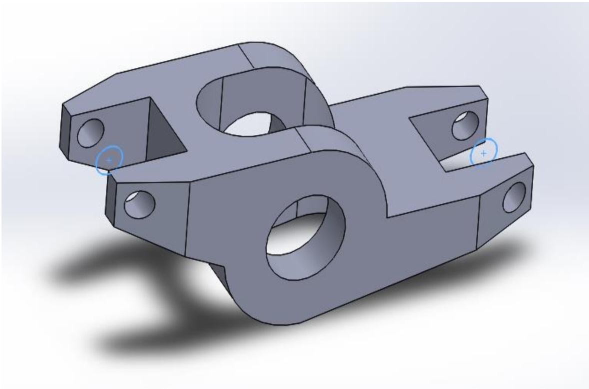

Figure 2 - 3D Model of Rocker Arm

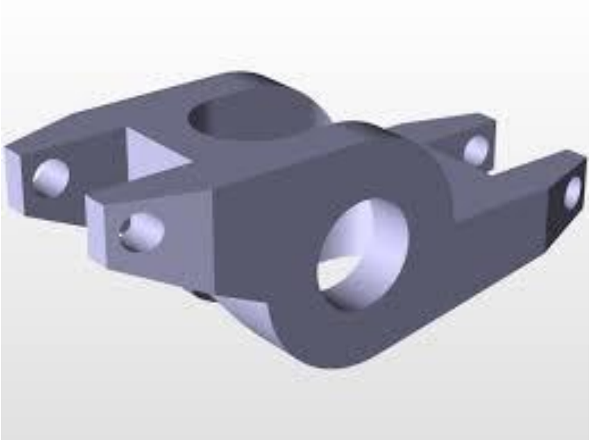

Figure 3 - Mesh Model

.

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 1134N.

IX ANSYS

ANSYS is an engineering simulation software provider founded by software engineer John Swanson. It develops general-purpose finite element analysis and computational fluid dynamics software. While ANSYS has developed a range of computer-aided engineering (CAE) products, it is perhaps best known for its ANSYS Mechanical and ANSYS Multi-physics products.

ANSYS mechanical and ANSYS multi-physics software are non-exportable analysis tools incorporating pre-processing (geometry creation, meshing), solver and post-processing modules in a graphical user interface. These are general-purpose finite element modeling packages for numerically solving mechanical problems, including static/dynamic structural analysis (both linear and non-linear), heat transfer and fluid problems, as well as acoustic and electro-magnetic problems.

ANSYS mechanical technology incorporates both structural and material non-linear test. ANSYS multi-physics software includes solvers for thermal, structural, electromagnetic, and acoustics and can sometimes couple these separate physics together in order to address multidisciplinary applications. ANSYS software can also be used in civil engineering, electrical engineering, physics and chemistry.

In 2008, ANSYS acquired Ansoft Corporation, a leading developer of high-performance electronic design automation software, and added a suite of products designed to simulate high-performance electronics designs found in mobile communication and Internet devices, broadband networking components and systems, integrated circuits, printed circuit boards, and electromechanical systems. The acquisition allowed ANSYS to address the continuing convergence of the mechanical and electrical worlds across a whole range of industry sectors.

X. ANALYSIS AND RESULTS

Material 1: Aluminium

Equivalent (Von-Mises) Stress Plot

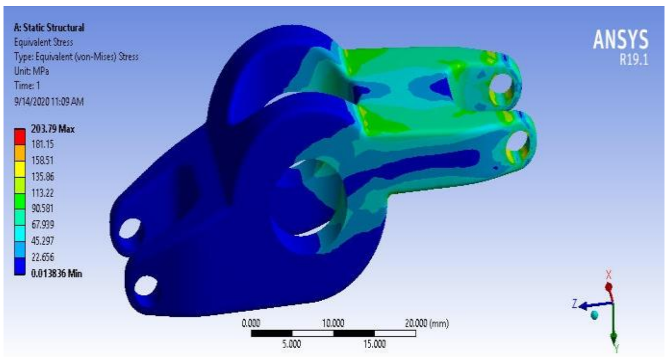

Figure 4

Equivalent Total Deformation Plot

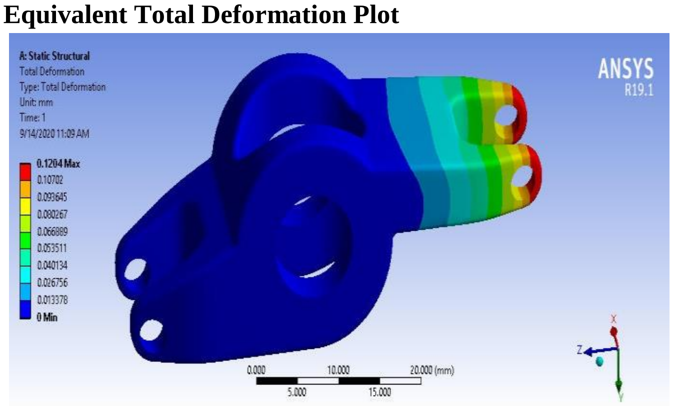

Figure 5

Equivalent Elastic Strain Plot

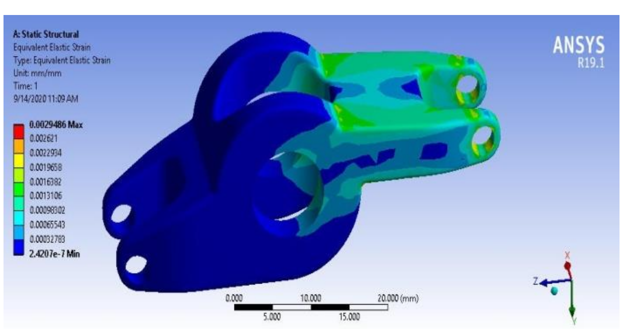

Figure 6

Result tabulation of Aluminium Rocker arm

Table 4

PARTICULARS	MAX	MIN
Equivalent stress (Mpa)	203.79	0.013836
Total deformation (mm)	0.1204	0
Equivalent elastic strain	0.00294	$2.4207e^{-7}$

Material 2: CarbonSteel

Equivalent (Von-Mises) Stress Plot

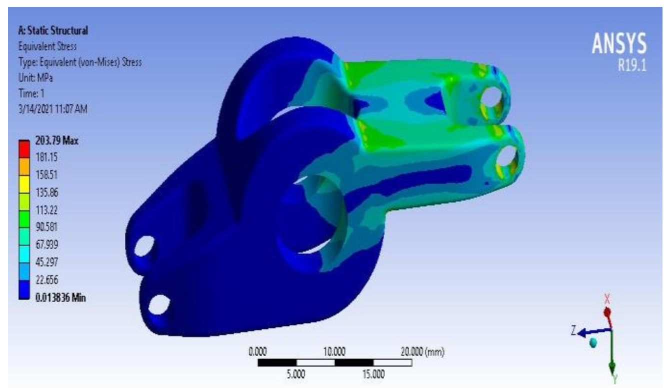

Figure 7

Equivalent Total Deformation Plot

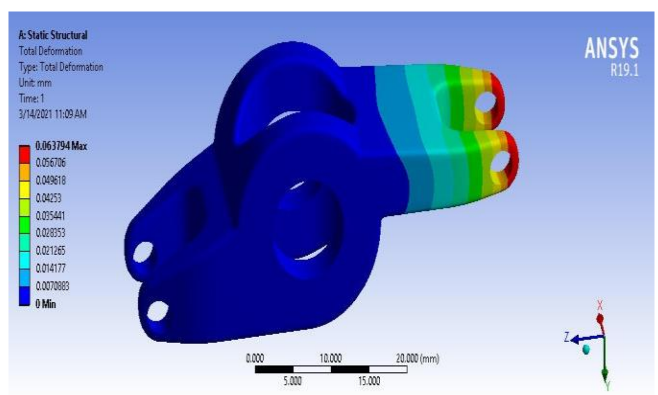

Figure 8

Equivalent Elastic Strain Plot

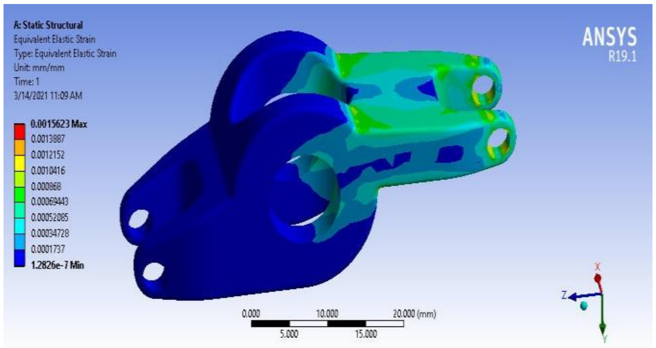

Figure 9

Result tabulation of CarbonSteel Rocker arm

Table 5

PARTICULARS	MAX	MIN
Equivalent stress (Mpa)	203.79	0.013836
Total deformation (mm)	0.063794	0
Equivalent elastic strain	0.0015623	1.2826-7

Material 3: HMCF

Equivalent (Von-Mises) Stress Plot

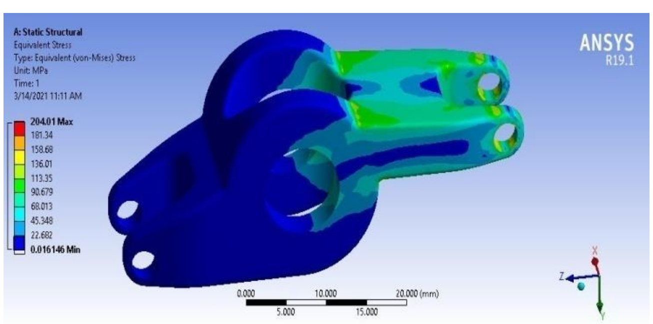

Figure 10

Equivalent Total Deformation Plot

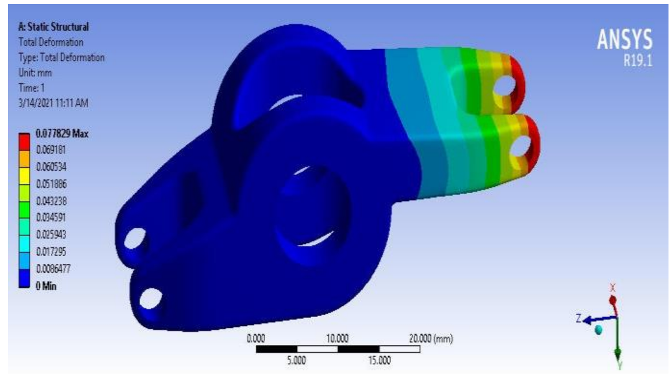

Figure 11

Equivalent Elastic Strain Plot

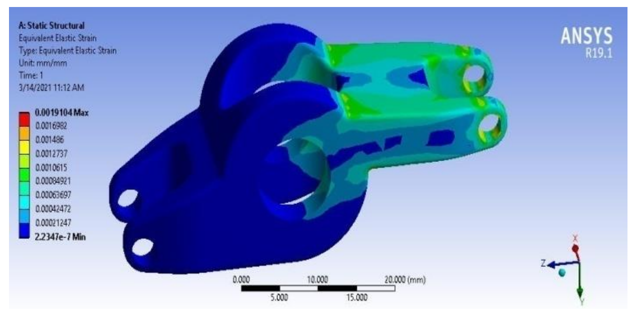

Figure 12

Result tabulation of HMCF Rocker arm

Table 6

PARTICULARS	MAX	MIN
Equivalent stress (Mpa)	204.01	0.016146
Total deformation (mm)	0.077829	0
Equivalent elastic strain	0.0019104	2.2347-7

XI. RESULTS AND DISCUSSION

Table 7 - Result tabulation

QUANTITY	ALUMINIUM	CARBON STEEL	HMCF
Equivalent stress (Mpa)	203.79	203.79	204.01
Total deformation (mm)	0.1204	0.063794	0.077829
Equivalent elastic strain	0.00294	0.0015623	0.0019104

The modeling of the rocker arm is done by using CATIA and the analysis is performed by ANSYS. The project consists of structural analysis of rocker arm which is done to find the strength of the model. To find the strength of the model in structural analysis we are taken 3 different materials and taken.

The equivalent stress in aluminium and carbon steel is same and high which is compared to HMCF.

The total deformation in aluminium is high compared to carbon steel and HMCF.

The equivalent strain in aluminium is high compared to carbon steel and HMCF.

So by the investigation we conclude that by using carbon steel the stress values are reduced by that the life time of the rocker arm increases.

XII. CONCLUSION

Study and analysis of Von-Mises Stress Plot , Elastic Strain and deformations level on rocker arm body are carried out as per the ASME section code and it can be concluded that ANSYS analysis. This study is not exhaustive and conducted by ANSYS 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.

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.

By analyzing obtained values of stresses, elastic strain 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.

So by the investigation we conclude that by using carbon steel the stress values are reduced by that the life time of the rocker arm increases.

REFERENCES

[1] Husain S.M., Prof. Siraj Sheikh, “Rocker Arm: A Review”. International Journal of Innovative Research in Science, Engineering and Technology Vol. 02, No.4, pp. 1120-1126, 2013

[2] JafarSharief, K.DurgaSushmitha, “Design and Analysis of a Rocker Arm” International Journal of Computational Engineering Research (IJCER) Vol. 05, No. 09, September – 2015.

[3] TawandaMushiri, Charles Mbohwa, “Finite Element Analysis of a Car Rocker Arm” International Conference on Operations Excellence and Service Engineering, Orlando, Florida, USA, September 10- 11, 2015, pp. 687-694, 2015.

[4] Anush Karki1, Anupam Raj Jha and RabindraNathBarman,“Design and Analysis of Rocker Arm by Finite Element Method using ANSYS”. International Journal of Advanced Research in Biology, Engineering, Science and Technology (IJARBEST) Vol. 02, No. 04, pp.1-6, 2016.

[5] Jim Miller, “Rocker Geometry” Engine Professionals Apr-Jun 2010, pp. 20-30, 2010.

[6] Mohd. Moesli Muhammad1, M. C. Isa, Syed Rosli S.B. “Failure Analysis of a Diesel Engine Rocker Arm”. Maritime Technology Division (BTM), Mechanical & Aerospace Technology Division (BTJA), Science & Technology Research Institute for Defence (STRIDE), Ministry of Defence, Malaysia.

[7] G.A. Nassef, A. Elkhatib, MostafaYakout, “Analysis of a failed rocker arm shaft of a passenger car Engine” Department of Production Engineering, Alexandria University, Egypt and Department of Mechanical Engineering, McMaster University, Hamilton, Canada. Case Study in Engineering Failure Analysis (ELSEVIER), Vol.05, No.06, pp. 10–14, 2016.