Design and Analysis of Motorbike Connecting Rod by FEM

Rajnish Saxena
Astt. Professor,
Mechanical Engineering Department,
FET, MRIU
rajnish.fet@mriu.edu.in

R.V. Singh
Professor & HOD,
Mechanical Engineering Department,
FET, MRIU
rvsingh.fet@mriu.edu.in

Abstract: *Connecting rod is a critical component used in internal combustion engine. It is subjected to different types of loading conditions like gas pressure, inertia loads, and frictional forces. It must be sufficiently strong, right and light in weight. In this paper, a connecting rod used in single cylinder four stroke petrol engines has been designed and developed using Pro-E wildfire 4.0. Forged steel material has been used which has high reliability and longer life due to high endurance limit. The design is based on load considering 5N/mm^2. Structural system of connecting rod has been analyzed using finite element techniques. Finite element analysis has been done to determine the* ***Von Mises*** *stresses in the model of connecting rod for given loading conditions. It was observed that Von Mises stress at big end shoulder is critical. That has been solved by changing the radius of curvature and considerable weight reduction has been achieved by changing the material.*

Keywords: Modeling, Connecting Rod, FEM analysis.

I. INTRODUCTION

Connecting rod is a critical component used in an automobile engine. It connects reciprocating part called piston to rotating part called crank shaft of internal combustion engine. It transmits the thrust of the piston to the crank shaft. The main function of connecting rod is to transmit the push and pull from the piston pin to the crank pin. In many cases, its secondary function is to convey the lubricating oil from the top end to the bottom to the crank shaft and i.e. from crank pin to piston pin and then for splash of jet cooling of piston crown. In connecting rod, an eye at the small end for the piston pin, bearing a long shank and a big end opening which is usually split to take the crank pin bearing shells. Adila Afzal et. al. [1] investigated and compared fatigue behavior of forged steel and powder metal connecting rods. A discussion of manufacturing cost comparison and recent developments in crack-able forged steel connecting rods are also included. Xiao Lei Xu et.al. [2] investigated on a diesel engine connecting rod failure used in a heavy duty truck. The failed connecting rod fractured at the small head end of the connecting rod and two fractured were formed. Pravardhan S. Shenoy et.al. [3] study was based on optimization of the weight and cost reduction. Reduction in machining operations achieved by using C-70 steel and utilization of the fracture splitting process reduces the production cost by about 25%. As compared with a powder metallurgy connecting rod, the cost saving is estimated to be about 15%. S B Jaju and P G Charkha (2009) studied on modeling and analysis of connecting rod of four stroke single cylinder engine for optimization of cost and material.

II. DESIGN OF CONNECTING ROD

Connecting rod is a critical component of an internal combustion engine. Connecting rod may be designed considering dimension of cross section of connecting rod; dimension of crankpin at the big end and the piston pin at the small end, size of bolts for securing the big end cap, and thickness of big end cap. Connecting rod takes

1. Force on piston due to gas pressure and inertia of the reciprocating parts,
2. Force due to inertia of the connecting rod or inertia bending forces,
3. Force due to friction of piston rings and of the piston, and

4. Force due to friction of piston pin bearing and the crank pin bearing.

Table 1: Basic design parameters of engine

Petrol Engine	
No. of Cylinders	1
Speed	1800 rpm
Maximum Pressure	$5 N/mm^2$
Compression Ratio	6.9:1
Length of Connecting Rod	95mm
Piston Diameter	56mm
Stroke	50mm
Factor of Safety	4

Table 2: Mechanical properties of AISI4140H

Specification	**AISI4140H**
Poisson's Ratio	0.293
Young's Modulus Yield Strength	2.1165 e5 Mpa 200 Mpa
Density	7.85 e-6 Kg/mm3

Calculation for Dimension of Rod

The cross section of connecting rod is assumed to be I-section. The dimensions have been calculated on the basis of following proportions;

Flange and web thickness of the section = t

Width of the section, B = 4t

Depth or height of the section, H = 5t
(I section dimension)

Dimensions of the I section are calculated using the empirical relations. Dimension of Connecting rod.

III. FEM APPROACH

Finite element method has been used for the load analysis and to check its failure under its working conditions. The approach used has been divided in following steps:

- Discretization
- Choosing the solution approx.Forming the element matrices and equation.

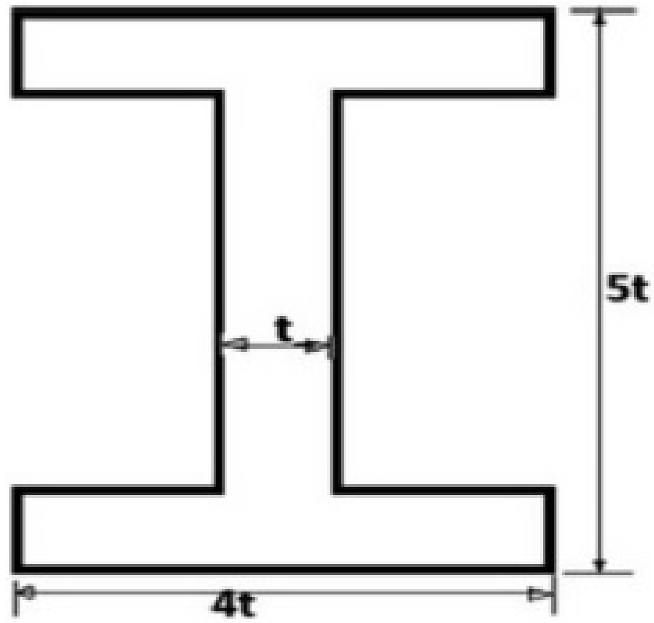

Fig. 1: Dimensions of I section

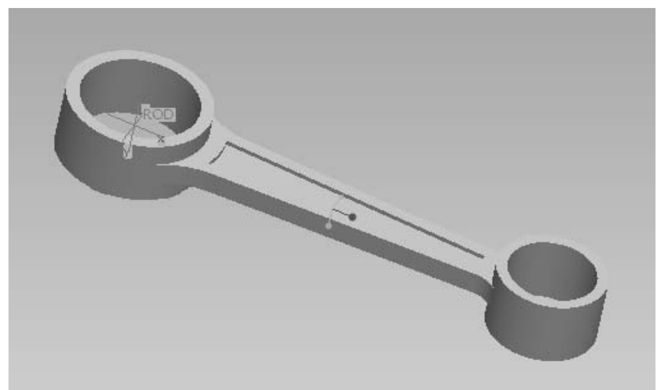

Fig. 2: Model of Connecting rod

- Assembling the matrices
- Finding the unknowns
- Interpreting the results

3.1 Simulation Process

Simulation is way to understand and improve process to increase the process demand. In this process problem is divided in to Phases as shown than solved the problem step by step.

3.2 Phase of Analysis

There are three main steps in a typical ANSYS analysis:

Pre Processor this phase consist of modeling of part, dividing the part in element and nodes ha material properties, Meshing and type of Analysis,

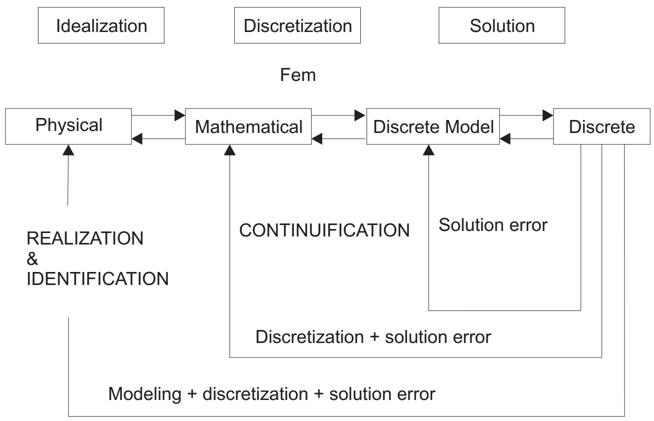

Fig. 3: for Simulation process

- Model generation
- Simplification, idealization
- Define element
- Define materials/Materials Properties
- Generate Mesh

Solution I this solver is used to solve the basic equation for Analysis type to compute the result

Table 3: Design of Connecting Rod

Name of Part	Numerical Value
Length of Connecting rod L	95
Diameter of Piston (D)	56 mm
Stroke of Piston	50 mm
Dimension of I section of connecting rod	
(a) buckling load	49232 N
(b) crank radius	25mm
(c) Thickness of flange and WEB of section (t)	3.5 mm
(d) width of section	14 mm
(e) Depth or height of section (H)	17.5mm
(f) Depth near the Big end(H_1)	21mm1
(g) Depth near the small end(H_2)	4mm
(h) Length of crank pin	40.3mm
(i) Dia. Of crank pin	20 mm
(j) width of big end cap	40mm
(k) thickness of big end cap	2.0 mm
Mass of Piston	0.28 Kg
Mass of Connecting rod	34g

- Apply Boundary Condition
- Apply Load
- Obtain the Solution

Post Processor this is the last phase for review the result or Analysis done

- Review result
- Deform & unreformed shapes
- Nodal Solution
- Element Solution
- Plot result
- check for validity.

3.3-D 10 Node Tetrahedral Structural **Solid (Solid 187)**

Solid 187 elements is a higher order 3-D, 10-node element. Solid 187 has a quadric displacement behavior and is well suited to modeling irregular meshes.

The element is defined by 10 nodes having three degree of freedom at each node, translations in the nodal x, y, z directions. The element has plasticity, hyper elasticity, creep, stress stiffening large deflection and large strain capabilities. It also has mixed formulation capability for simulating deformations of nearly incompressible elasto-plastic materials, and fully incompressible hyper elastic materials.

The geometry, node locations, and the coordinate system for these elements showed "SOLID 187 Geometry". In addition to the nodes, the element input data includes the orthotropic or anisotropic material properties. Orthotropic and anisotropic material direction corresponds to the element coordinate directions. The element coordinate system orientation as in Linear Material Properties. Element loads are described in Node and Element Loads. Pressure may be input as surface loads on the element faces as shown by the circled numbers on SOLID187 Geometry.

IV. IMPORTING THE DESIGNED MODEL

The CAD model of connecting rod is imported in the ANSYS Graphics user Interface. While importing the CAD model it is very important to sure that there should not be data loss while importing the model. Fig shows the imported CAD model in ANSYS.

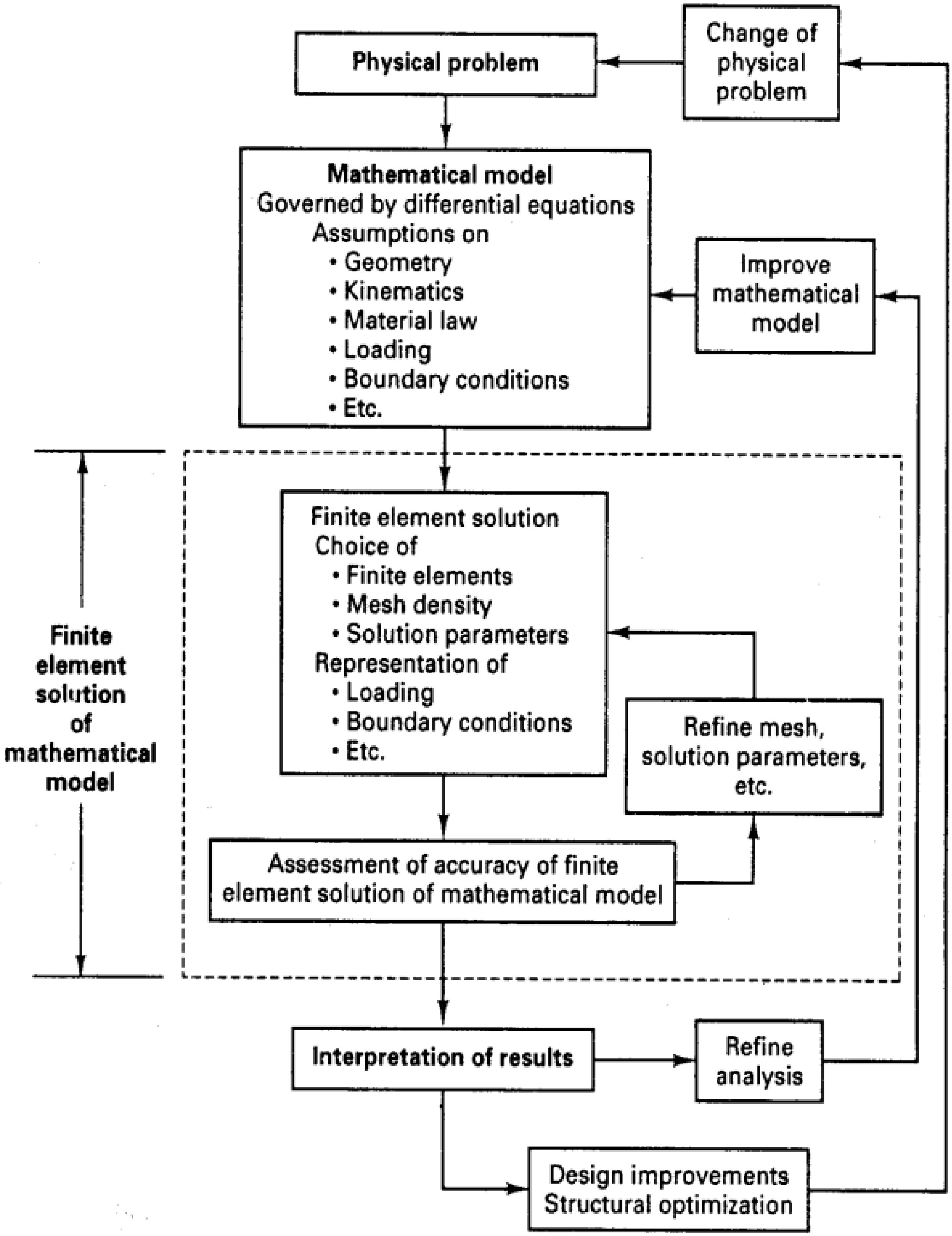

Fig. 4: Design and Optimization of CR

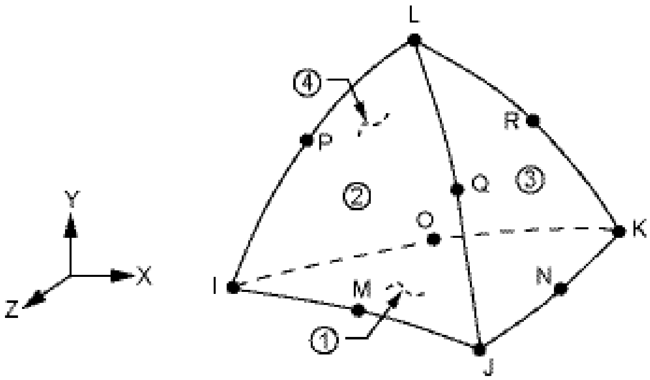

Fig. 5: Solid 187 type element

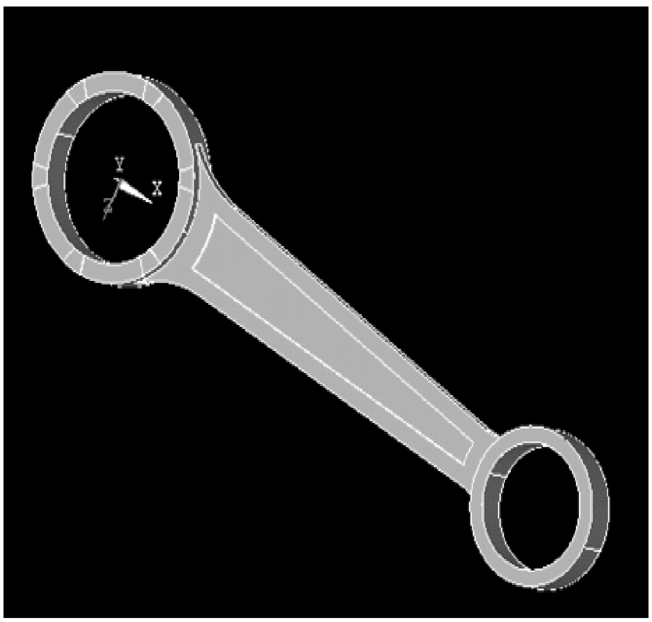

Fig. 6: CAD model in ANSYS

Meshing of Model. The most important step of any FEA is meshing. In this case I have given tetrahedral mesh. Since model consists of uneven geometry, have to check the nodal connectivity between segments under the condition of keeping the nodal connectivity we have opt for the target and contacts.

Table 4: Meshing description

Description	Quatity
Total Nodes	31323
Total Elements	22125

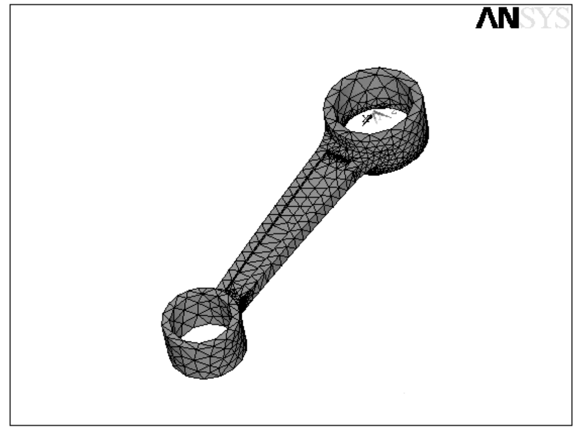

Fig. 7: Mesh Model of Connecting rod

deformed shape of the connecting rodwhenever load applied on the rod the shape of the rod will affected due to that.

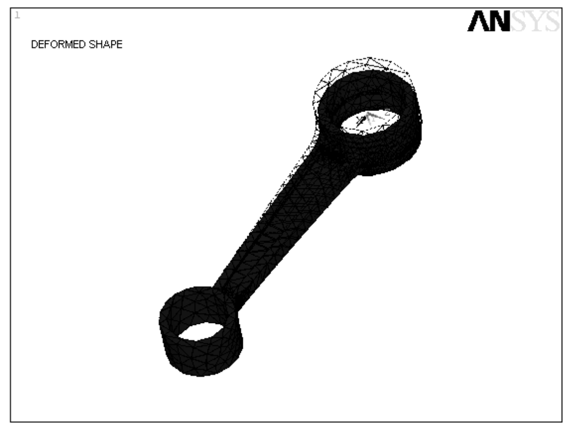

Fig 8: Defomatio uder load

Fig. 8 shows the elemental solution of rod as in FEA whole body is divided in to small segments called elements and nodes, so Fig. Shows effect on element when load applied on the rod.

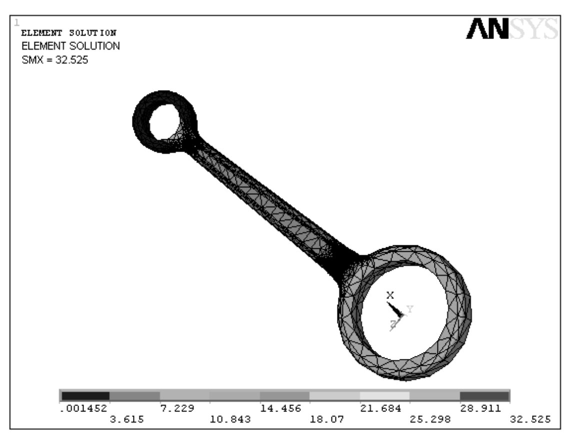

Fig. 9: Stress distribution on Elements and nodes

Fig. 9 shows the nodal solution of rod as in FEA whole body is divided in to small segments called nodes.

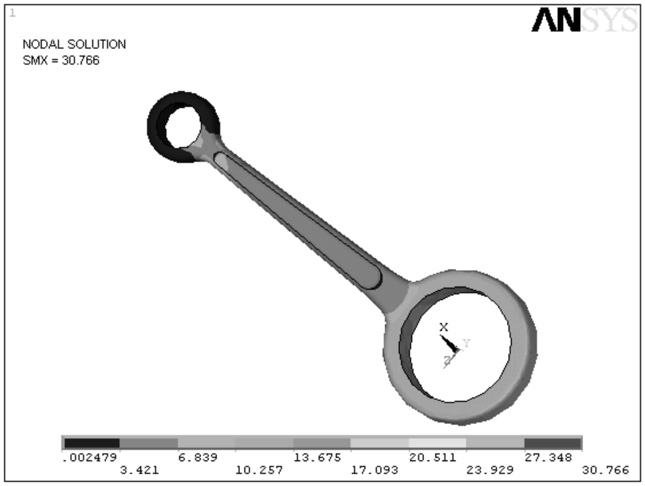

Fig. 10: Nodal solution

Fig. 10 shows the failture of connecting rod on big end due to compressive load act on it and failure occour.

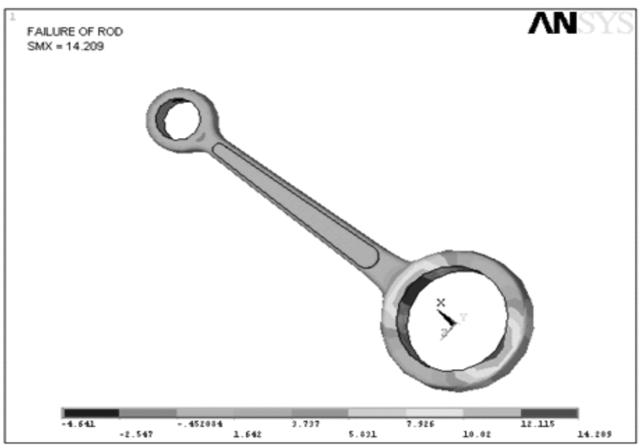

Fig 11: Faliure due to stress distriution on big end

V. DESIGN MODIFICATION

As shown in model of connecting rod the radius of curvature at the big end where failure occurs increase the curvature (as at corner and shoulder max. stress act and failure of part at shoulders so for smooth running increase the curvature). so analysis is done on rod by changing the radius of curvature. Here minimum stress produce -49.58N/mm^2 and max. 12.406N/mm^2 and rod part is within safe region.

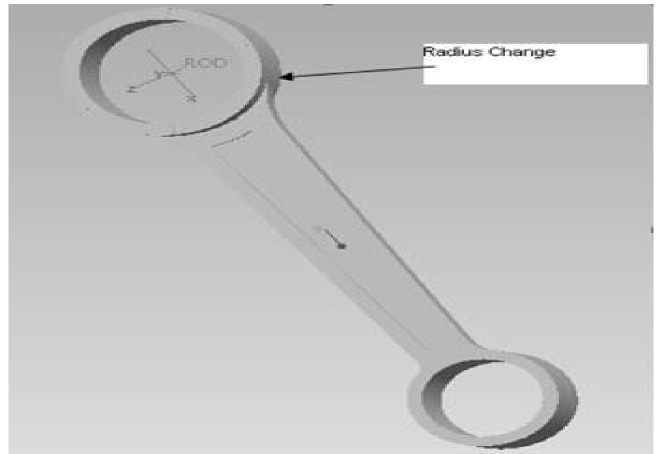

Fig. 12: Modified Curvature in designed model

After change of curvature again CR checks for failure but part is now safe on same working condition and plot Von Mises criteria for CR.

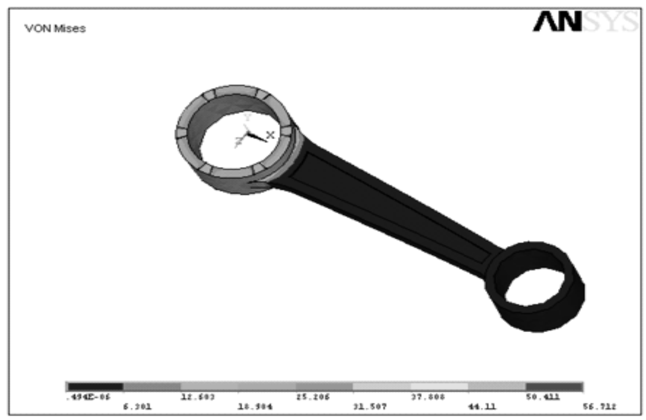

Fig. 7: Von Mises criteria

Now change the material from Forge steel to C-70 material and again calculation make for optimization of weight.

VI. MASS OPTIMIZATION

Mass of Connecting Rod (Forged steel)

mass = density × vol.

m_{cr} = 7850 × 0.095 × 0.014 × 0.0175 = 0.182 Kg

m_r = mass of reciprocating parts

m_r = mass of Piston + $\frac{1}{3}$ mass of Connecting rod

mr = 0.28 + $\frac{1}{3}$ × 0.182 = .34 kg

Mass of Connecting Rod (C-70 steel)

mass = density × vol.

$m_{cr} = 1160 \times 0.095 \times 0.014 \times 0.0175 = 0.027$ Kg

m_r = mass of reciprocating parts

m_r = mass of Piston + $\frac{1}{3}$ mass of Connecting rod

$m_r = 0.28 + \frac{1}{3} \times 0.0.27 = .28$ Kg

So from these calculations it is noted that mass of connecting rod reduce by 6%.

Results:

Table 5: Max. & Min. stress at

Analysis for	Minimum Stress	Maximum Stress	Result
Forged steel	-4.6 N/mm^2	14.209 N/mm^2	Failure

Table 6: Max. & Min. stress at changing the material into C-70 steel

Analysis for	Minimum Stress	Maximum Stress	Result
Forged steel	-14.35 N/mm^2	46.213 N/mm^2	Ok

VII. CONCLUSION

In this work, connecting rod was designed using traditional calculation and modeled in Pro-E wildfire 4.0.

Static Analysis of connecting rod was performed on ANSYS 9.0 software and the deformation and stress were compared. The Von Mises stress in existing connecting rod using traditional method is 9.0N/mm^2 and using finite element analysis is 54.11N/mm^2 under same boundary conditions.

New design of connecting rod by changing the curvature can reduce the failure (as on corner max. stress act) problem under the given loading condition.

Since the entire possible load are analyzed by CAE. This saves the product development time and testing material. This analysis gives a clear picture about the stresses and deformations occurred during working condition of connecting rod.

Analysis was made from condition that pressure applied on the big end and due to load failure occurs due to that pressure. Figure shows that maximum stress act on big end so we can say that failure may occur at that point and rod will fail due to stress concentration at big end. When radius of curvature increase than the stress concontration on that reduce and value of produce stress are within certain limits and failure not occours rod part is safe. After changing the radius of curvature failure in the rod part not occurs and part is safe. The results are validated by comparing the results obtained by ANSYS 9.0 software with the results of traditional method. Thus results are validated.

REFERENCES

[1]. Afzal, A., 2004, "Fatigue Behavior and Life prediction of Forged Steel and PM Connecting Rods," Master's Thesis, University of Toledo.

[2]. Xiao Lei Xu Zhi Wei Yu, 2007, "Failure Analysis of a Diesel Engine Connecting rod". pp. 316-320.

[3]. Pravardhan S Shenoy and Ali Fatemi, 2005, "Connecting Rod Optimization for Weight and Cost Reduction". pp. 1-8.

[4]. S B Jaju, P G Charkha , 2009, "Modeling & Analysis of Connecting Rod of Four Stroke Single Cylinder Engine for Optimization of Cost & Material". pp. 1277-1288.

[5]. Mirehei, M. Hedayati Zadeh, "A. Jafari, M. Omid, 2008, "Fatigue analysis of connecting rod of universal tractor through finite element method (ANSYS), Journal of Agricultural Technology 2008, V.4(2): 21-27.

[6]. Zhou Qinghui , Wang Yunying , Ji Wei, 2010, "The Finite Element Analysis of Connecting Rod of Diesel Engine" 2010 International Conference on Measuring Technology and Mechatronics Automation pp. 870-873.

[7]. R.Vozenilek, C. Scholz, P. Brabec,2004 "FEM Analysis of connecting rod" Medunarodni naucni simpozijum Motorna Vozila I Motori International Scientific Meeting Motor Vehicles & Engines Kragujevac, 04.

[8]. Dilip G. Gangwani , Prof. R.M. Metkar, "Computer Aided Modeling and Finite Element Analysis of Connecting Rod Fouling on Camshaft of DI Engine of Mahindra and Mahindra Tractor. Second International Conference on Emerging Trends in Engineering and Technology, ICETET-09 pp. 104-109.

[9]. P. Brabec, P. Kefurt, C. Scholz, R. Vo•enílek, "Fem analysis of Connecting Rod for Stationary Engine".

❒