NOVATEUR PUBLICATIONS
INTERNATIONAL JOURNAL OF INNOVATIONS IN ENGINEERING RESEARCH AND TECHNOLOGY [IJIERT]
ISSN: 2394-3696
VOLUME 3, ISSUE 3, March-2016

ANALYSIS OF FATIGUE FAILURE OF CONNECTING ROD USED IN A LIGHT COMMERCIAL VEHICLE (LCV) THROUGH FEA

Mr.Gajanan Dinkarrao More

M.E. Mech/ Design – 2nd yr Terna College of Engineering, Osmanabad

Prof.V.V.Mane

Mechanical Engg Dept., Terna College of Engineering, Osmanabad

Prof.M.S.Kadam

Bramadev Mane Institute of Technology Solapur

Prof.A.B.Ghalake

Mechanical Engg Dept., Terna College of Engineering, Osmanabad

ABSTRACT:

The main objective of this study was to investigate the design, mass and cost reduction of connecting rod made up of structural steel for 970 cc four cylinder four stroke engines. Every stroke of the engine subjected to its adjacent components to cyclic loading that pulls and pushes the connecting rod and crankshaft. The design of the Connecting rod can be done in a justifiable manner if an attempt is made to identify the effects of the operating loads on the component in the form of the type of stress induced with its peak value and the location of these stresses over the component. This study consists of two major sections, Finite Element Analysis and Optimization for design and mass reduction.

In this paper, existing connecting rod is been optimized with change of various design parameters for I- Section followed by 3-D CAD- modeling using Solid Works. Second order Tetrahedron mesh is been carried out in Hypermesh. 3-D finite element analysis is planned to be carried out by virtue of static stress analysis and followed by fatigue life prediction of Connecting rod using Ansys Workbench. Alternatives for Design would be suggested while attempting to modify the geometry of the Connecting rod by changing the different parameters.

KEYWORDS: Light commercial vehicle, connecting rod, ANSYS, FEM, Weight optimization.

INTRODUCTION:

The automobile engine connecting rod is a high volume production, critical component. It connects reciprocating piston to rotating Connecting rod, transmitting the thrust of the piston to the Connecting rod. Every vehicle that uses an internal combustion engine requires at least one connecting rod depending upon the number of cylinders in the engine. Connecting rods for automotive applications are typically manufactured by forging from either wrought steel or powdered metal. They could also be cast. However, castings could have blow-holes which are detrimental from durability and fatigue points of view. The fact that forgings produce blow-hole-free and better rods gives them an advantage over cast rods. Between the forging processes, powder forged or drop forged, each process has its own pros and cons. Powder metal manufactured blanks have the advantage of being near net shape, reducing material waste.
However, the cost of the blank is high due to the high material cost and sophisticated manufacturing techniques with steel forging, the material is inexpensive and the rough part manufacturing process is cost effective. Bringing the part to final dimensions under tight tolerance results in high expenditure for machining, as the blank usually contains more excess material so in order to reduce the material cost and thus for production cost it is better to optimize the weight or volume. And thus due to its large volume production, it is only logical that optimization of the connecting rod for its weight or volume will result in large-scale savings.

METHODOLOGY ADOPTED:

The methodology carried out in order to overcome the above problem would be studying & identifying the design of the existing Connecting rod, carrying out various geometric iterations of the I-Section. Building the solid model in Solid Work followed by carrying out meshing in Hyper mesh and Static Structure analysis along with fatigue life prediction in Ansys Work bench. Only the critical areas of connecting rod would be studied for the work and suitable recommendation can be find out while concluding the work. Practicality of the recommended solution pertaining to the cost and ease of deployment would be considered while suggesting the variants for design.

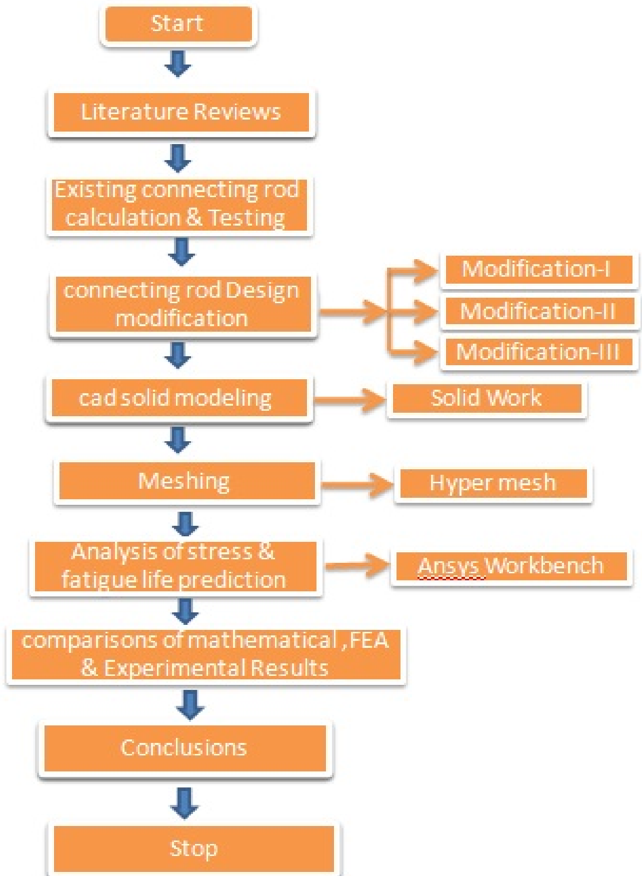


Figure 1: Methodology of Project

INPUT DATA:

ENGINE SPECIFICATIONS:

Specifications of Connecting rod:

Displacement : 970 CC

Max. Power : 103 kW@3750rpm

Gear Ratio : Max. 1.409 or Min. 2.268

MATERIAL:

The Structural Steel used for Connecting rods is high grade steel alloy which has the following composition and physical properties:

CHEMICAL COMPOSITION:

Carbon	0.15-0.30%
Chromium (Cr)	0.8-0.9%
Molybdenum (Mo)	0.08-0.25%
Silicon (Si)	0.40%
Manganese	0.50-1.70 %
Sulphur	0.035% max
Copper (Cu)	0.20 %
Nickel (Ni)	0.30-1.50 %

Table 1: Physical Properties of Original Connecting Rod

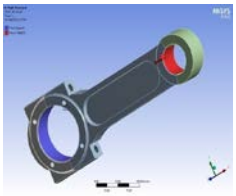
Density	7850kg m^-3
Tensile strength	460 MPa
Yield strength	250 MPa
Young's modulus	200000MPa

Table 2: Description of Original Connecting Rod

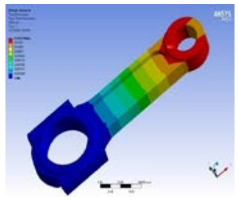
Sr. No.	Parameters	Default value (mm)
1	Connecting rod length	360
2	Crank end inner diameter	55
3	Crank end outer diameter	90
5	Piston Pin end inner diameter	30
6	Piston Pin end outer diameter	50
7	Shank Thickness	12

EXPERIMENTAL RESULTS:

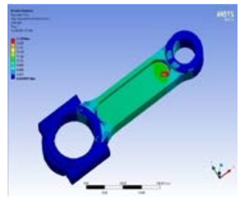
While implementing this project an analysis of four different connecting rods has been done including existing rod. Ansys and FEM analysis are presented below for four types of connecting rod.



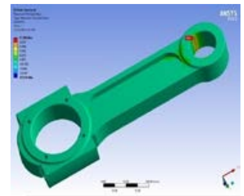
The Boundary Conditions for Compressive Loading



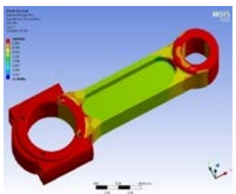
The Maximum Displacement of 0.014 mm under Compressive Loading



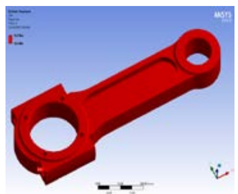
The Maximum Von Mises Stress Produced under Axial Compressive Load Comes Out to be 27.29 MPa which is at the Pin End.



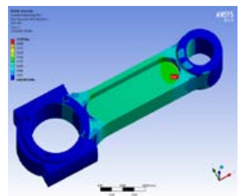
The Maximum Principle Stress Distribution in Connecting Rod under Compression which is 15.306MPa



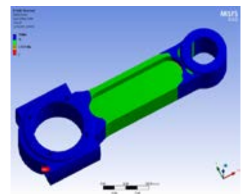
The Minimum Principle Stress Distribution in Connecting Rod under Compression which is 1MPa



The Minimum Life in the Simulation 1X10^6 (Within the Limit)



The Equivalent Alternating Stress for Fatigue Life Prediction which is 27.29MPa.



Factor of Safety which is 15

Figure 2: Ansys analysis of existing rod

After finite element modeling we performed static analysis. The various individual loads acting on the connecting rod were used for finding the structural behavior of connecting rod and stress distribution was obtained. The loads included static compressive load and static tensile load. Analysis of connecting rod is performed under extreme loading Condition.

1. Small end Compressive loading and bigger end fixed.

For the current study we have analyzed connecting rod under static load conditions. The loads included static compressive load. For static analysis one of the ends of the connecting rod is fixed and other is given standard loads to determine stresses and deflection. The connecting rod has been separately analyzed for the compressive load due to gas pressure. The study also indicates that the buckling is important design factor that must be taken into account during the design process.

On the basis of the stress and strain measurements performed on the connecting rod, closed agreement has been found with applied static loads. Questions are naturally raised in light of such a complex structural behavior, such as: Does the peak load at the end of the connecting rod represents the worst case loading? Under the effect of axial load can one expect higher stresses than the experienced under axial load alone?

So this chapter discusses, such as how loads and constraints have been applied, weather connecting rod is safe against buckling, stress distribution along the length of the connecting rod and deflection in connecting rod.

CONCEPT OF WEIGHT OPTIMIZATION:

The process in which weight of the component reduced, maximum stresses at critical locations reduced and cost of the component reduced without minimizing the fatigue strength of the component is called as weight optimization. The optimization study performed on Connecting rod was not the mathematical optimization process. Design variables as a tool for Connecting rod optimization process. In the geometry optimization process the shape of the component remains same only size is modified. In the Connecting rod optimization mass reduction, cost reduction and improving fatigue performance using alternative materials and considering manufacturing aspects based on FEA simulation and dynamic load analysis. In the geometric optimization Connecting rod web thickness, web geometry, increasing inner hole diameter and their depth simulated to reduce the weight and final cost of the product.

CONSTRAINTS:

The following dimensions were not changed in the optimization process of Connecting rod.

1. Piston pin diameter remains constant.
2. Crank pin diameter remains constant.
3. Length between centers constant.
4. Material remains same.

DESIGN VARIABLES:

In the optimization process some parameters and dimensions should be changed are called as design variables. There is various design variables could be changed and considered in the Connecting rod optimization process they are as follows.

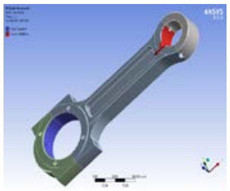
1. Thickness of the Connecting rod web.
2. Changes in pin fillet
3. Internal and external fillet radii of shank.

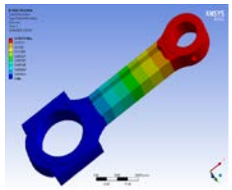
MODIFICATION -I OF CONNECTING ROD MODEL:

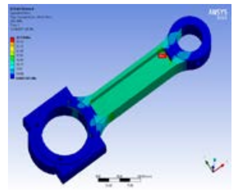
The modified design of Connecting rod is modeled using SOLIDWORKS software. Geometry provided had to clean and following modifications were done.

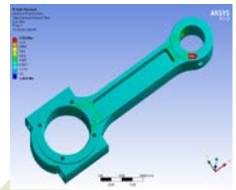
a) Model was cleaned up and was made symmetric.
b) Duplicate surfaces were removed.
c) Irregular surfaces were removed & replaced with proper surfaces.
d) Connectivity with neighboring surfaces was maintained.
e) Changing Connecting rod web thickness 12mm to 10mm.
f) We did small fillets as 1mm & big fillets as 8mm.
g) Lubricating hole as 3mm diameter.

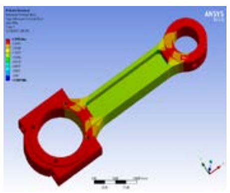
Because of this modification actual weight which was 10.12 kg was reduced to 9.47 kg.

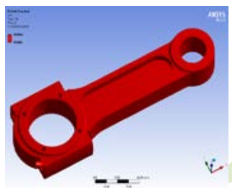

The Boundary Conditions for Modified Connecting Rod-I.

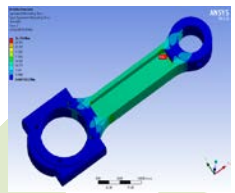

Maximum Displacement of 0.0165 mm under Compressive Loading


The Maximum Von-Mises stress Produced under Axial Compressive Load Comes out to be 32.31 MPa which is at the Pin End.


The Maximum Principle Stress Distribution in Connecting Rod under Compression which is 13.26 MPa.


The Minimum Principle Stress Distribution in Connecting Rod under Compression which is 1.77MPa


The Minimum Life in the Simulation 1X10^6 (within the limit).


The Equivalent Alternating Stress for Fatigue Life Prediction Which is 32.31MPa.

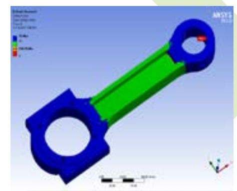
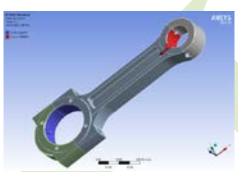
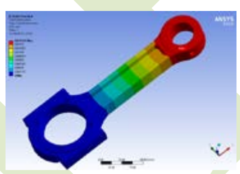
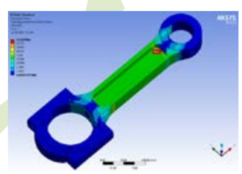

The Factor of Safety 15

Figure 3: Ansys analysis of modification – I

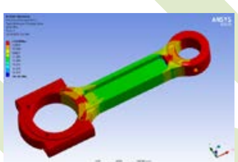
MODIFICATION – II


The Boundary Conditions for Modified Connecting Rod-II.

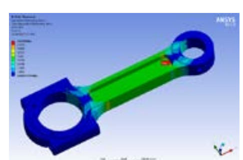

The Maximum Displacement of 0.0154 mm under Compressive Loading


The Maximum Von Mises Stress Produced under Axial Compressive Load Comes out to be 24.22 MPa which is at The Pin End.


The Maximum Principle Stress Distribution in Connecting Rod under Compression which is 13.00 MPa.


The Minimum Principle Stress Distribution in Connecting Rod under Compression which is 1.44MPa


The Minimum Life in the Simulation 1X10^6 (Within the Limit.)


The Equivalent Alternating Stress for Fatigue Life Prediction which is 24.22 MPa.

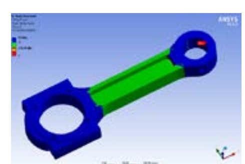

The shows the Factor of Safety 15

Figure 4: Ansys analysis of modification – II

Connecting rod was modeled using solid work software and all the specification was accordingly followed the relevant modification. The web thickness changes to 11 mm & small fillets as 2 mm

& big fillets as 9mm, where as the Connecting rod diameter and length is considered to be same as that of original Connecting rod design. The following images show the modified 3D modeling of Connecting rod.

A static structural analysis is carried out with the given loading condition in the ANSYS solver. Preprocessing of Connecting rod is done by using HYPERMESH software. Where the 3D tetrahedral mesh is done and the input deck is prepared for ANSYS solver. The result is shown below.

It is been observed that under compressive load, the critical regions are the crank end transition, pin end transition and the web at the crank end. Thus plot gives us the general idea of the stress variation on the connecting rod. The mashed model has number of nodes 3767855 and number of elements are 2520485. The static loads for which these stresses are plotted is a compressive load of 18 KN. From static analysis of Modified Connecting rod-II it is clear that when a force 18 KN is applied on the Connecting rod at piston pin location, we found that, the Maximum Deflection is 0.0154 mm and Maximum stress is 24.22 MPa. The Maximum principle stress distribution in connecting rod under compression which is 13.00MPa & The Minimum principle stress distribution in connecting rod under compression which is 1.44 MPa.

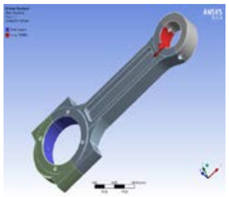
MODIFICATION- III

Connecting rod was modeled using solid work software and all the specification was accordingly followed the relevant modification. The web thickness changes to 13 mm & small fillets as 3mm & big fillets as 10mm, where as the Connecting rod diameter and length is considered to be same as that of original Connecting rod design. The following images show the modified 3D modeling of Connecting rod.

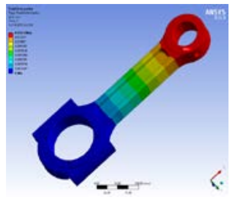
Connecting rod was modeled using solid work software and all the specification was accordingly followed the relevant modification.

A static structural analysis is carried out with the given loading condition in the ANSYS solver. Preprocessing of Connecting rod is done by using HYPERMESH software. Where the 3D tetrahedral mesh is done and the input deck is prepared for ANSYS solver.

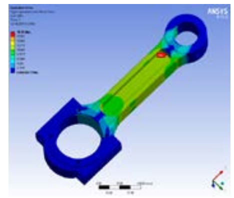
It is been observed that under compressive load, the critical regions are the crank end transition, pin end transition and the web at the crank end. Thus plot gives us the general idea of the stress variation on the connecting rod. The mashed model has number of nodes 3753355 and number of elements are 2516069. The static loads for which these stresses are plotted is a compressive load of 18 KN. From static analysis of Modified Connecting rod-III it is clear that when a force 18 KN is applied on the Connecting rod at piston pin location, we found that, the Maximum Deflection is 0.01374 mm and Maximum stress is 19.16 MPa. The Maximum principle stress distribution in connecting rod under compression which is 12.73MPa & The Minimum principle stress distribution in connecting rod under compression which is 1.45 MPa.



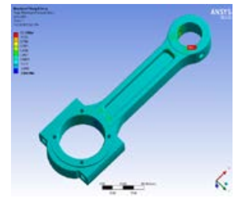
The Boundary Conditions for Modified Connecting Rod-III



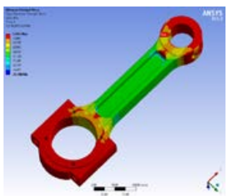
The Maximum Displacement of 0.01374 mm under Compressive Loading



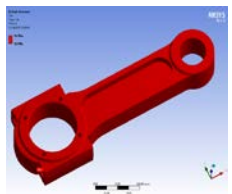
The Maximum Von Mises Stress Produced under Axial Compressive Load Comes out to be 19.16 MPa which is at the Pin End.



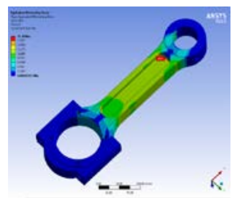
The Maximum Principle Stress Distribution in Connecting Rod under Compression which is 12.73 MPa.



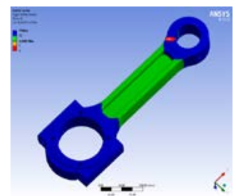
The Minimum Principle Stress Distribution in Connecting Rod under Compression which is 1.45MPa



The Minimum Life in the Simulation i.e 1X10^6 (Within the Limit)



The Equivalent Alternating Stress for Fatigue Life Prediction which is 19.16 MPa.



Factor of Safety 15

Figure 5: Ansys Analysis for modification –III

CALCULATION OF VARIOUS PARAMETERS FOR ALL FOUR CASES:

Table 3: Calculation of various parameters for all four cases

Parameter	Design Calculation of Existing Connecting Rod	Modification -I of Connecting Rod Model	Modification -II of Connecting Rod Model	Modification -III of Connecting Rod Model
Total volume of connecting rod model,	V_{tot} =1287898.089 m^3	V_{tot} =1206369.427 m^3	V_{tot} =1232356.688 m^3	V_{tot} =1256560.51 m^3
Mass density of connecting rod model,	ρ =7.85 x10^{-6} kg/m^3	ρ =7.85 x10^{-6} kg/m^3	ρ =7.85 x10^{-6} kg/m^3	ρ =7.85 x10^{-6} kg/m^3
Total weight of connecting rod model	10.12 kg	9.47 kg	9.67 kg	9.864 kg
Gas Force	F_p = 18 KN	F_p = 18 KN	F_p = 18 KN	F_p = 18 KN
Moment of Inertia (Ixx)=	724032mm^4	349162.69mm^4	510496.2283mm^4	9985252.937mm^4
Moment of Inertia (Iyy)	226368mm^4	109162.69mm^4	159112.2283mm^4	310388.9367mm^4

Section Modulus:	11318.4mm^3	5458.1345mm^3	7955.6114mm^3	39834.1174mm^3
Slenderness Ratio	3.2	3.2	3.2	3.2
Von-Mises Stress Calculations: CASE-I	σ_x= 11.36 σ_y= 0	σ_x= 16.4684 σ_y= 0	σ_x= 14.6568 σ_y= 0	σ_x= 10.4272 σ_y= 0
Von-Mises Stress Calculations: CASE –II	σ_x= 11.30 σ_y= 17.091 σ_v= 15.05	σ_x= 16.42 σ_y= 35.4419 σ_v= 30.72	σ_x= 14.6155 σ_y= 24.2411 σ_v= 21.14	σ_x= 10.396 σ_y= 12.4256 σ_v= 11.5456
Buckling load	17402.147	260560.4662	223413.7781	420229.9896
Fatigue Life Calculation	σ_a = 7.525 MPa Mean Stress σ_m = 7.525MPa for forged steel S_e' = 230 MPa Endurance limit S_e = 184 MPa From modified goodman line n = 17.5438	σ_a = 15.3 MPa Mean Stress σ_m = 15.3MPa for forged steel S_e' = 230 MPa Endurance limit S_e = 184 MPa From modified goodman line n = 8.58	σ_a = 10.5 MPa Mean Stress σ_m = 10.5MPa for forged steel S_e' = 230 MPa Endurance limit S_e = 184 MPa From modified goodman line n = 12.51	σ_a = 5.77 MPa Mean Stress σ_m = 5.77MPa for forged steel S_e' = 230 MPa Endurance limit S_e = 184 MPa From modified goodman line n = 22.7842
Fatigue Strength	Modified Goodman line S_f = 7.650 MPa Finite Life N= 6.927226 X 10^{33} Cycles	Modified Goodman line S_f = 15.83 MPa Finite Life N= 2.069 X 10^{27} Cycles	Modified Goodman line S_f = 10.74 MPa Finite Life N= 5.982 X 10^{30} Cycles	Modified Goodman line S_f = 5.8432 MPa Finite Life N=1.7456 X 1036 Cycles

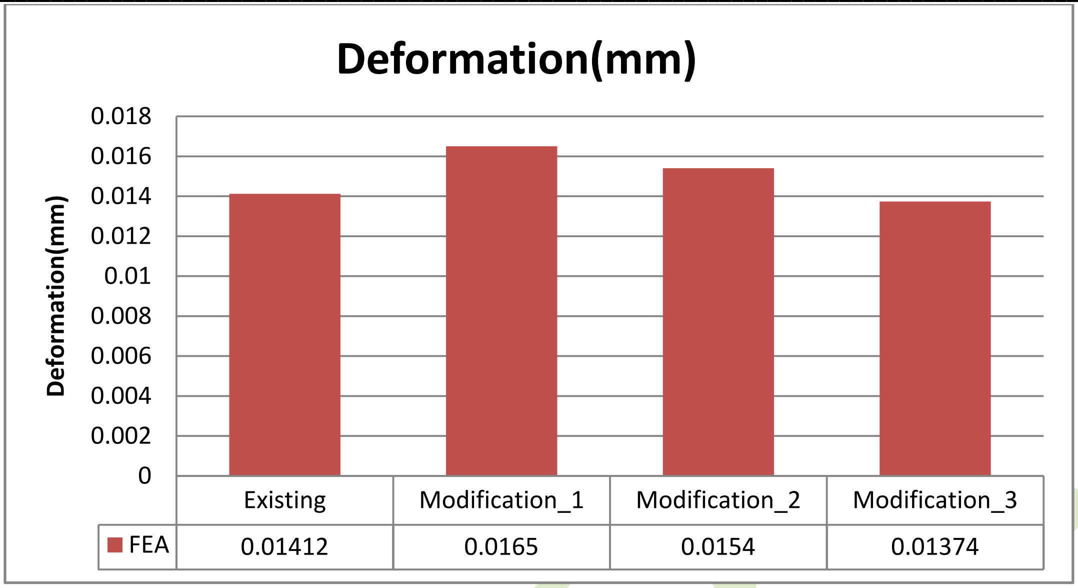


Figure 6: Comparison of deformation.

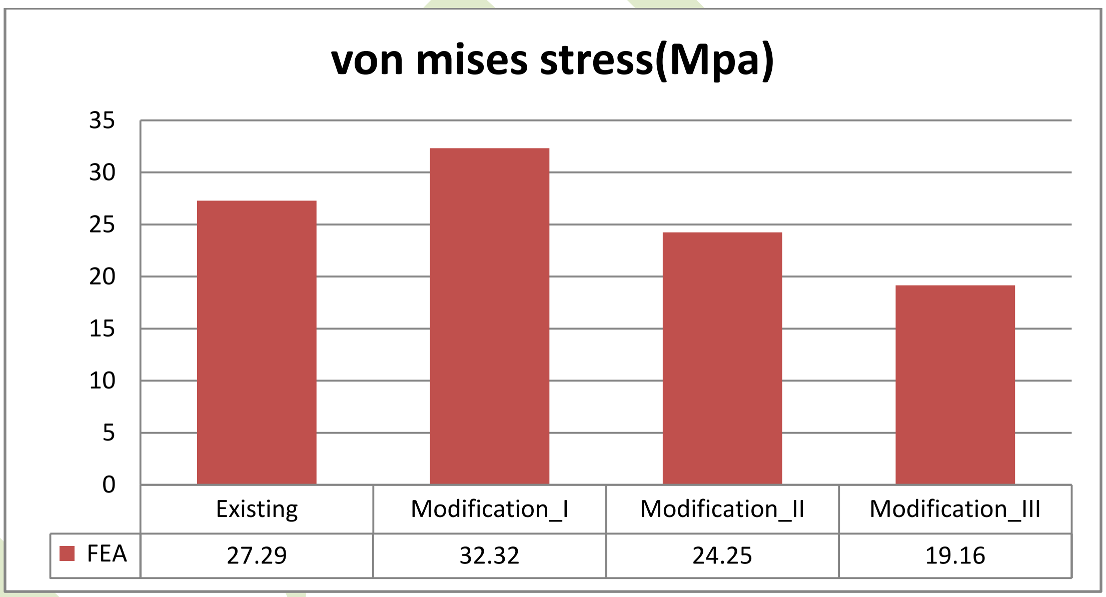


Figure7: Comparison of Von-Mises Stress.

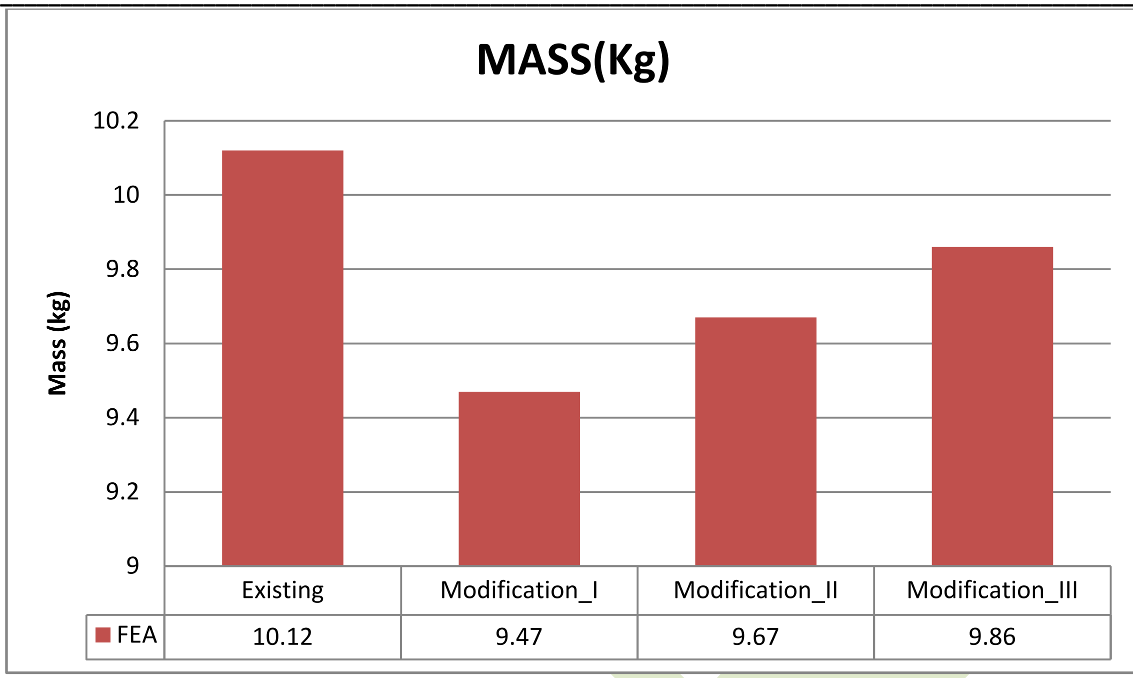


Figure 8: Comparison of Mass.

CONCLUSIONS:

1. Original Connecting rod shows the maximum deflection of 0.01419 mm.
2. Original Connecting rod shows the Maximum Von-Mises stress is observed to be 27.29 MPa near the piston pin end.
3. The minimum life predicted by simulation is about 6.9E33 cycle for the original Connecting rod design which shows the infinite life of connecting rod.
4. The mass of original Connecting rod is 10.12K g
5. Selecting Modification I as a best Optimized Connecting rod because the maximum Von-Mises stress is to be 32.31 MPa which is less than the material yield stress.
6. Modified I Optimized Connecting rod shows the maximum deflection of 0.01657 mm which is well within the permissible limit.
7. The minimum life predicted by simulation is about 2.069E27 cycle for the Modified-I Connecting rod design.
8. The mass of Modified-I Connecting rod is 9.47 Kg.
9. Material and cost saving is 0.65Kg and 250/- per piece respectively. Hence we have saved 6.5% of each.

REFERENCES:

1) Kuldeep B, Arun L.R, Mohammed Faheem. ***"****ANALYSIS AND OPTIMIZATION OF CONNECTING ROD USING ALFA Sic COMPOSITES."* 'International Journal of Innovative Research in Science', Engineering and Technology. ISSN: 2319-8753, Vol. 2, Issue 6, June 2013.

2) Mohammad Reza Asadi Asad Abad, Mohammad Ranjbarkohan and Behnam Nilforooshan Dardashti. *"Dynamic Load Analysis and Optimization of Connecting Rod of Samand Engine."* Australian Journal of Basic and Applied Sciences. ISSN 1991-8178, 5(12): 1830-1838, 2011

3) Yogesh Kumar Bharti, Vikrant Singh, Afsar Husain, Dipanshu Singh, Shyam Bihari Lal, Satish Kumar Dwivedi. *"Stress analysis and optimization of connecting rod using finite element analysis"*. 'International Journal of Scientific & Engineering Research', ISSN 2229-5518, Volume 4, Issue 6, June-2013

4) Prof. Pushpendra Kumar Sharma, Borse Rajendra R. *"Fatigue Analysis & optimization of connecting rod use fea"*. Head Department Mechanical Engineering NRI-IST, RGPV University, Bhopal, Research Scholar M.tech. NRI-IST, RGPV University, Bhopal.

5) FEM Vivek. C. Pathade, Bhumeshwar Patle, Ajay N. Ingle, *"Stress Analysis of I.C.Engine Connecting Rod"*

6) S B Chikalthankar, V M Nandedkar, Surendra Prasad Baratam, *"Fatigue Numerical Analysis for Connecting Rod"*, 'Department of Mechanical Engineering, Government College of Engineering, and Aurangabad) (SGGS Institute of Engineering and Technology, Nanded, India)'

7) Bin Zheng, Yongqi Liu and Ruixiang Liu, *"Stress and Fatigue of Connecting Rod in Light Vehicle Engine"*

8) Priyank D. Toliya, Ravi C. Trivedi, Prof. Nikhil J. Chotai, *"Design And Finite Element Analysis Of Aluminium-6351 Connecting Rod"*, Department of Mechanical Engineering Marwadi education foundation's group of institutions, Rajkot, 360 003, Gujarat, India

9) Prof. Pushpendra Kumar Sharma, Borse Rajendra R, *"FATIGUE ANALYSIS AND OPTIMIATION OF CONNECTING ROD USING FINITE ELEMENT ANALYSIS"* Head

Department Mechanical Engineering NRI-IST, RGPV University, Bhopal 2Research Scholar M.tech. NRI-IST, RGPV University, Bhopal.

10) Adila Afzal and Ali Fatemi, *"A Comparative Study of Fatigue Behavior and Life Predictions of Forged Steel and PM Connecting Rods"*, the University of Toledo,2004-01-1529.

11) Uraj Pal, Sunil kumar, *"Design Evaluation and Optimization of Connecting Rod Parameters Using FEM"*, 'International Journal of Engineering and Management Research', Issue-6 December 2012, Vol.-2, ISSN No.: 2250-0758, Pages: 21-25.

12) Mr. H.B.Ramani, Mr. Neeraj Kumar, *"Using Shape Optimization Tool in Ansys Software for Weight Reduction of Steel Connecting Rod"*, 'International Journal of Engineering Research & Technology (IJERT), Vol. 2 Issue 2, February- 2013.

13) CH.Venkata.Rrajam, P.V.Murthy, M.V.S.MuraliKrishna, G. M.Prasada rao. *"Design Analysis & Optimization of Piston Using SOLIDWORKS & Ansys"*, 'International Journal of Innovative Research In Engineering & science'.ISSN 2319-5665, ISSUE 2 Volume 1, Jan 2013.

14) Rankle Garg, Sunil Baghla, *"Finite Element Analysis and Optimization of Crankshaft Design"*, 'International Journal of Engineering and Management Research', Issue-6, December 2012, ISSN No.: 2250-0758, Pages: 26-31

15) Dr. K.H. Jatkar, Mr. Sunil S. Dhanwe , *" Dynamic Analysis of Single Cylinder Petrol Engine"* , 'International Journal of Engineering Research and Applications (IJERA)', Vol. 3, Issue 3, May-Jun 2013, pp.1177-1183.