Nonlinear Static Finite Element Analysis and Material Optimization of Connecting Rod

Mr. Vivek T. Fegade
Research Scholar, Department of Mechanical Engineering,
Datta Meghe College of Engineering,
Airoli, India

Dr. Kiran S. Bhole
Head, Department of Mechanical Engineering,
Datta Meghe College of Engineering,
Airoli, India

Abstract— **This paper describes about nonlinear static analysis and optimization of forged steel connecting rod. With the implementation of optimization approach connecting rod of stronger but equally lighter can be obtained with minimum cost. This paper focus on finite element analysis and material optimization of Titanium alloys as an alternative material for connecting rod. For comparison Finite element analysis of connecting rod is completed by considering two materials viz. structural steel & Titanium alloys. A proper CAD model is developed using software CATIA V5 then FEA of connecting rod is carried out to determine the maximum von misses stresses for the given loading conditions using software ANSYS WORKBENCH. The design and weight of the connecting rod affect the engine performance. Specifications of connecting rod have been estimated to calculate the loads acting on it. Structural analysis is carried out on piston end and crank end of connecting rod. The component is to be optimized for material subject to constraint of allowable stress and factor of safety. The percentage weight reduction obtained was 1.5 % by optimization.**

Keywords: Connecting Rod, FEA, Nonlinear, Optimization, Static etc .

I. INTRODUCTION

As the use of automobiles is increase, there is huge demand of automobiles components. The increased demand is due to improved performance and reduced cost of these components. In order to minimize launch time for new product research and development and test engineers develop critical components in shortest time. It becomes necessary to understand and rapid incorporation of new technologies while developing the new products. Connecting rod is under severe dynamic load used for transmitting the power in internal combustion engines. It is considered as important component from the view of structural durability and efficiency. It acts as an intermediary link between the piston and crankshaft of internal combustion engine. Its function is to transmit thrust from piston to crankshaft. Connecting rods are subjected to forces generated by mass and fuel combustion which results in axial and bending stresses. Due to eccentricities, crankshaft, case wall deformation, and rotational mass force bending stresses are appear. That's why a connecting rod must be withstand for transmitting axial tension, axial compression, and bending stress occurred due to the thrust and pull of the piston and by centrifugal force. It is observed that for to produce light weight connecting rod, most efforts in weight reduction have focused on shape optimization and removal of material which is not always feasible. The requirement is that it should not alter the performance and safety of the component. Hence the material selection plays an important role in order to produce light weight connecting rod. This issue of material selection for connecting rod is taken here in this paper as a part of study to find out alternative materials for connecting rod by doing some modifications in design of existing connecting rod.

Finite element analysis has emerged as a powerful tool for supporting engineers in various fields of product development and research in the continuous improvement of computational capabilities [1]. Three dimensional FEA of high speed diesel engine connecting rod was first carried out by author [2].The connecting rod fouling with the camshaft in the engine of DI model of tractor of Mahindra & Mahindra was corrected by using the Computer aided modeling and analysis techniques [3]. Sensitivity analysis and optimization based on the combination of Pro/MECHANICA and ANSYS are applied to designing of the connecting rod of LJ276M electronic gasoline engine [4]. Author has suggested use of metamodel based shape optimization of connecting rod taking fatigue life in to consideration. The actual cost of computer simulations of connecting rod can be reduced by using metamodel based optimization as compare to classical design optimization [5]. Analysis and optimization of forged steel connecting rod for weight reduction has worked out by author [6]. The structural improvement in the structure of diesel engine connecting rod was evaluated and observed that the destructive position is lie in between connecting rod shank & transition location of small end for condition of maximum compression [7]. For to reduce weight & production cost of forged steel connecting rod, an optimization study is necessary [8]. The topology optimization is used to design internal combustion engine of connecting rod , to develop structural modeling, FEA & optimization [9]. Fully reverse loading is used to estimate longevity of a connecting rod and also find the critical points from where there is possibility of crack growth initiation in the universal tractor connecting rod (U650) [10]. They have introduced a new method of the connecting rod structural FEA and optimization. Stress distribution and strength of connecting rod are derived to optimize the design of the connecting rod [11]. Detail load analysis of connecting rod was studied for MF tractor followed by its FEA to capture stresses and fatigue cycles [12]. Investigated failure of diesel engine connecting rod by Visual and scanning electron microscopy (SEM) observations [13]. Connecting rod was redesigned by taking

ISSN: 2278-0181
ICNTE-2015 Conference Proceedings

manufacturing process effects in analysis to obtain good fatigue performance with weight reduction [14].

Connecting rod is subjected to forces generated by mass & fuel combustion which results in axial load and bending stresses. A connecting rod must be withstand for transmitting axial tension, axial compression, and bending stress occurred due to the thrust and pull of the piston and by centrifugal force. Finite element Model (FEM) is a new technique for fatigue analysis and estimation of the component. The important component factors such as material, cross section conditions etc can be change.

Steel and titanium are used as a material for connecting rod. The objective of the research is an attempt to design and analysis of composite connecting rod, made of strip of Titanium alloys inserted in structural steel connecting rod. CAD model is prepared in Catia V5. For analysis model is imported in Ansys workbench.

In this paper finally, the comparison is made between the structural steel and composite made connecting rod in terms of weight and stress.

The structure of paper is as follows:

First of all the introduction is presented with relevant literature review and objective of study, followed by simulation methodology, results and discussion, and finally the conclusions.

II. SIMULATION METHODOLOGY

In light of literature it was felt that simulation is important tool, which can be used for failure prediction or improvement in design with aim of reducing weight, cost etc. Simulation methodology contains design of connecting rod, modelling, meshing, boundary conditions, stress observation, stress prediction and optimization of connecting rod.

A. Design of connecting rod

Design of various parts of connecting rod viz: connecting rod shank, big end, small end, bolts for cap, cap of big end is done as per standard design procedure [24].

TABLE 1. INPUT PARAMETER FOR CONNECTING ROD

Parameters	Dimensions
Diameter of piston (d)	95 mm
Weight of reciprocating parts (m)	1.6 kg
Length of connecting rod (L)	200 mm
Stroke (r)	62.5 mm
Speed (ω)	1500-2500 rpm
Compression ratio	4:1
Maximum explosion pressure (p_{max})	2.5MPa

Outputs of the design:

Outputs obtained are the dimensions of connecting rod required for modeling it on CATIA software. Table 2 shows the dimension of connecting rod.

B. Modeling

Connecting rod was modeled using CATIA V5 software which is shown in Figure 1. It was then imported to Design Modeller of ANSYS Workbench

C. Meshing

Element used is 10 node Tetrahedron named Solid187. First convergence was checked by finding deformation against different element size .

Finite element mesh was generated using solid tetrahedral elements with various element lengths of 8 mm (5353 elements), 5 mm (16928 elements), 3 mm (118152 elements), and 2.5 mm (127410 elements), 2.2mm (141274 elements), 2mm (185932 elements), 1.8mm (240093 elements). The deformation was checked for convergence. Figure 2 below shown is deformation versus element size for checking convergence.

Thus element size was found out to be 2mm for working in convergence zone. Therefore, a finite element mesh was generated with a uniform global element length of 2 mm. This resulted in a mesh with 185932 elements and 285457 nodes. Figure 3 shown below is meshed model of connecting rod.

TABLE 2. DIMENSIONS OF CONNECTING ROD

Dimensions	Values (mm)
Web thickness t	3
Width of flange B	12
Height of I section H	15
Diameter of pin end	24
Length of pin end	32
Diameter of crank end	64
Length of crank end	34
No of bolts	2
Size of bolts	M14 X 1.5

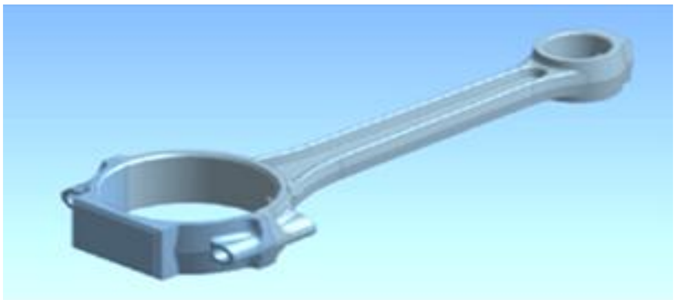

Fig. 1 Cad model of connecting rod

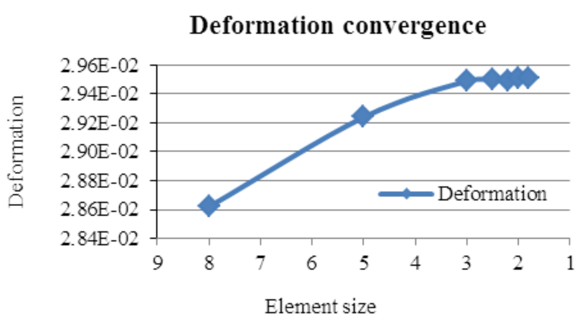

Fig. 2. Deformation convergence along element size

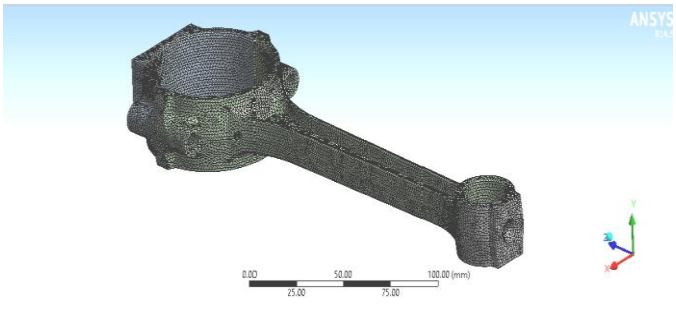

Fig 3. Meshed model of connecting rod

TABLE 3: BOUNDARY CONDITIONS USED FOR STATIC ANALYSIS OF CONNECTING ROD

Connecting Rod End Loading	Crank End	Piston Pin End
Compressive Loading	Pressure of 8.48MPa	Restrained
	Restrained	Pressure of 27.45MPa
Tensile Loading	Pressure of 9.35MPa	Restrained
	Restrained	Pressure of 30.27MPa

D. Boundary Conditions

By using the expressions from force analysis of connecting rod tensile and compressive loads acting on the connecting rod was obtained.

In the analysis carried out, the axial load was 15976 N in both tension and compression. For both tensile and compressive loads FEA was conducted. In this study four finite element models are analyzed. Finally the comparisons are done for optimization purpose. The pressure constants for 15976 N are as follows used for applying

• Compressive Loading:

Crank End: Po = 15976/ (32 x 34 x √3) = 8.48 MPa

Piston Pin End: Po = 15976/ (12 x 28 x √3) = 27.45 MPa

• Tensile Loading:

Crank End: Po=15976/ [32x 34x (π/2)] = 9.35 MPa

Piston Pin End: Po=15976/ [12x 28x (π/2)] = 30.27 MPa

Table3 shows boundary conditions used for analysis of connecting rod for original model as well as optimized model.

E. Nonlinearity of connecting Rod

Contact Nonlinearity was present between connecting rod and its cap. Bonded contact was defined between them, so contact elements and target elements were obtained between surfaces in contact i.e. between cap and connecting rod.

F. Stress Observation

Equivalent stress and deformation in connecting rod were obtained in both tensile as well as compressive loading condition using static structural analysis in ANSYS workbench. Factor of safety was calculated based on ratio of allowable stress to maximum stress. In case of compressive loading at crank end, due to stress concentration maximum stress occurred at oil hole as shown in fig 4 and at pin end maximum stress is occurred on the pin end as shown in Fig 5. In case of tensile loading at crank end maximum stress is occurred at oil hole as shown in fig 6 and at pin end stress distribution shown in fig 7.

G. Stress Prediction

Stress concentration occurs at the regions of sudden changes in geometry, point of application of loads and point of application of forces.

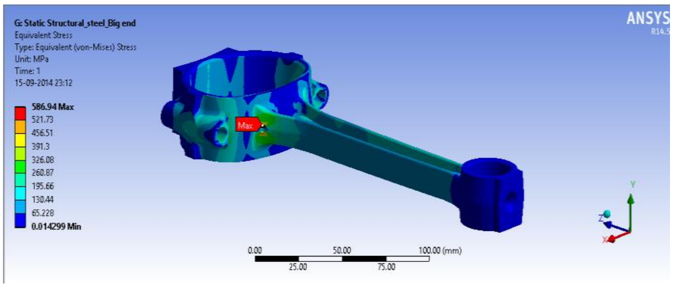

Fig. 4. Vonmisses steel rod crank end compressive

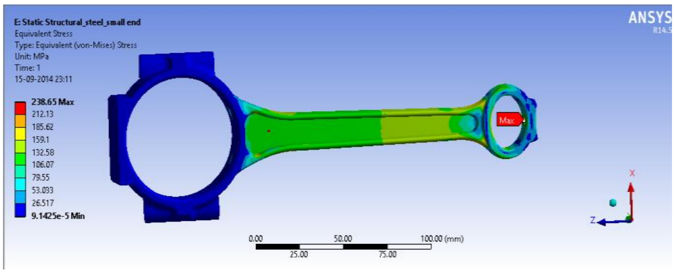

Fig. 5 Vonmisses steel rod pin end compressive

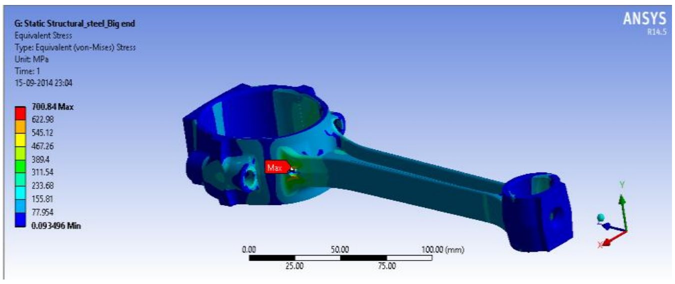

Fig. 6 Vonmisses steel rod crank end tensile

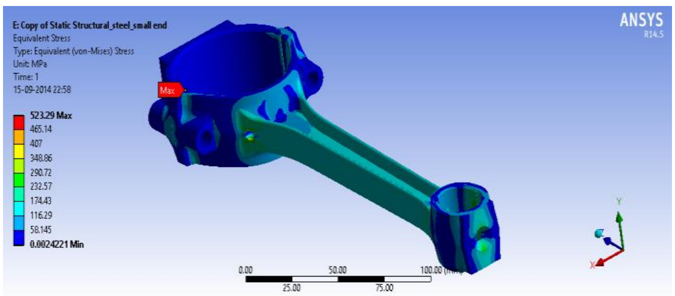

Fig. 7. Vonmisses steel rod pin end tensile

ISSN: 2278-0181
ICNTE-2015 Conference Proceedings

H. Optimization

Objective of the optimization task was to minimize the mass of the connecting rod under the effect of a load range comprising the two extreme loads, the peak compressive gas load to keep the maximum, minimum, and the equivalent stress amplitude within the limits of the allowable stresses.

a) Optimization Statement:

Objective Function: Minimize mass
Subject to Constraints
[1] Maximum Vonmises stress < Allowable stress
[2] Factor of safety > 1.3
[3] Manufacturing Constraints.

b) Optimized Model

The stresses in each loading conditions were studied after carrying out static structural analysis. As the thickness of web is 3mm , more than 1mm thickness plate will not fit there as surrounding steel will become too thin to hold it so a 1mm thickness titanium plate is added on both side of shank of connecting rod so that maximum vonmisses stress does not exceed allowable and factor of safety is kept above 1.3.

As shown in Fig.8 Following a 1mm thickness titanium plate is added on both side of shank. Optimized geometry was modified in Design modeler of ANSYS Workbench.

III. RESULTS AND DISCUSSION

The weight of original model was 670.18 gm & of optimized model is 659.91 gm. The total weight reduced is 10.27 gm.

Maximum vonmises stress and deformation was found out using static structural analysis in ANSYS workbench. Stress distribution for compressive loading at pin end, Maximum stress occurred at pin end as shown in Fig 9 and stress obtained by titanium plate along the path shown in fig 10. Stress distribution for Tensile loading at pin end is shown in Fig 11 and stress obtained by titanium plate along the path shown in fig 12. Stress distribution for compressive loading at crank end is shown in fig 13 in this case due to stress concentration max stress occurred at oil hole.

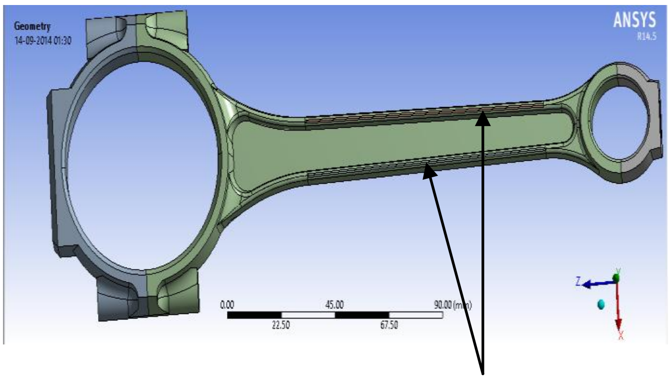

Fig 8. Optimized connecting rod

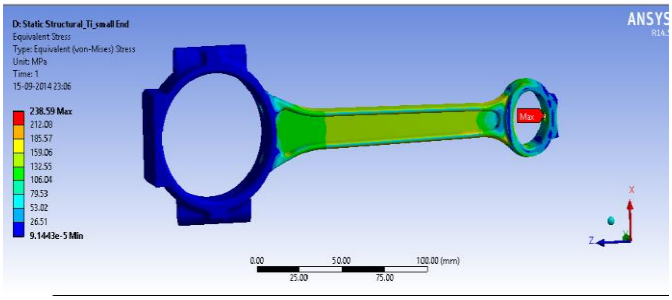

Fig. 9. Pin end compressive

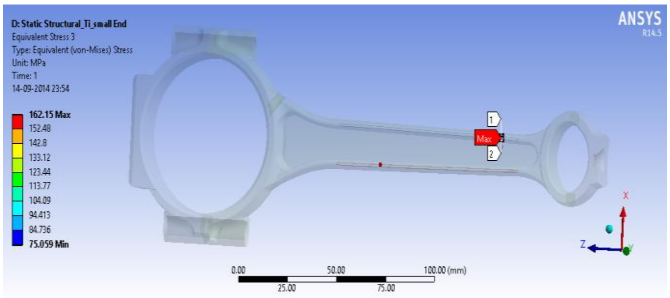

Fig.10. Pin end compressive along the path

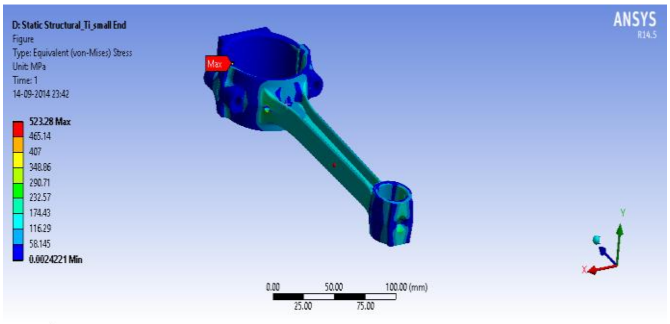

Fig 11. Pin end tensile

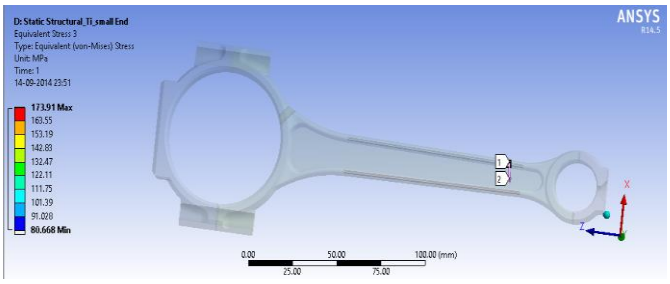

Fig. 12. Pin end tensile along the path

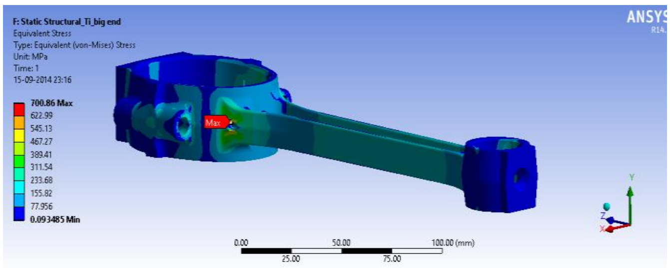

Fig. 13. Crank end compressive

TABLE 4. MAXIMUM VONMISES STRESSES ALONG THE PATH FOR ORIGINAL AND OPTIMIZED CONNECTING ROD

Model		Equivalent Vonmisses (Along path) (MPa)	
Pressure Applied	**Type of loading**	**Original**	**Optimized**
Pin End	Tensile	150.44	173.91
	Compressive	140.32	162.15
Crank End	Tensile	148.33	171.68
	Compressive	120.4	139.36

Table 4 shows Maximum vonmisses stress obtained along the path at different loading conditions original model as well as optimized model.

IV. CONCLUSION

1. The peak stresses mostly occurred in the transition area between pin end, crank end and shank region. The value of stress at the middle of shank region is below allowable limit. Forces at pin end are lesser in comparison to the forces in crank end. This decreases the strength of the pin end as compared to the strength of crank region.
2. Tensile loading is critical loading condition used for optimization study. In this loading pressure is applied at the crank end and pin end restrained maximum value of vonmisses stress was observed for both original as well as optimized model compared to other loading conditions.
3. Stress measured along the path showing that maximum stresses was obtained by titanium strip as compared to steel in tensile and compressive loading conditions when pressure applied at crank end and pin end.
4. Insertion of titanium strip on both sides of shank in steel connecting rod increases the strength of shank region, this reduces catastrophic and buckling failure.
5. In case of tensile as well as compressive loading , original as well as optimized model factor of safety was greater than 1.3.
6. Percentage weight reduction obtained is about 1.532 % which will save material with increase in engine efficiency and reduces the manufacturing cost.

REFERENCES

[1] S. Vijayaraja, S. Vijayaragavan,"Finite element analysis of critical components of gasoline engine", AVTEC Ltd, Pitampur, India,2008.

[2] W. D. Webster, R. Coffell and D. Alfaro,"A three dimensional finite element analysis of a high speed diesel engine connecting rod", SAE Technical Paper Series, Paper No. 831322, 1983.

[3] D. G. Gangwani, R. M. Metkar,"Computer aided modeling and finite element analysis of connecting rod fouling on camshaft of di engine of mahindra and mahindra tractor", International Conference on Emerging Trends in Engineering and Technology, IEEE, 2009.

[4] X. Hou, C. Tian, D. Fang, F. Peng and F. Yan, "Sensitivity analysis and optimization for connecting rod of lj276m electronic gasoline engine", Wuhan University of Technology, IEEE,2009.

[5] T. H. Lee and J. J. Jung, "Metamodel-based shape optimization of connecting rod considering fatigue life", journal of engineering materials", Vol.306-308, pp. 211-216,2006.

[6] P. G. Charkha, S. B Jaju, "Analysis & optimization of connecting rod", International Conference on Emerging Trends in Engineering and Technology, IEEE, ,2009.

[7] Z. Yongqi, J. Lixia, "Finite element analysis and structural improvement of diesel engine connecting rod", International Conference on Computer Modeling and Simulation, IEEE,2010.

[8] P. S. Shenoy and A. Fatemi, "Connecting rod optimization for weight and cost reduction", University of Toledo, SAE Technical Paper Series, Paper No- 2005-01- 0987,2005.

[9] M. S. Shaari, M. M. Rahman, M. M. Noor, K. Kadirgama, A. K. Amirruddin, "Design of connecting rod of internal combustion engine: a topology approach" , National Conference in Mechanical Engineering Research and Postgraduate Studies,2010.

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

[11] D. Jia, K. Wu, S. Wu, Y. Jia, C. Liang, "The structural analysis and optimization of diesel engine connecting rod", International Conference on Electronic & Mechanical Engineering and Information Technology, IEEE,2011.

[12] M.R. Asadi, M. Rasekh, A. Golmohammadi, A. Jafari, K.Kheiralipour, A. M. Borghei, "Optimization of connecting rod of MF-285 tractor", Journal of Agricultural Technology, Vol. 6(4), pp. 649-664,2010.

[13] [13]Tony George Thomas, S. Srikari, M. L. J Suman, "Design of connecting rod for heavy duty applications produced by different processes for enhanced Fatigue Life" ,SASTECH Journal Vol. 10, Issue 1,2010.

[14] Xiao Lei Xu, Zhi Wei Yu, "Failure analysis of a diesel engine connecting rod", Journal of Failure Analysis and Prevention, Vol. 7, pp. 316–320,2007.

[15] Zheng Bin Liu Yongqi, Ji Lixia, "Finite element analysis and structural improvement of diesel engine connecting rod", International Conference on Computer Modelling and Simulation,2010

[16] Rabb R, "Fatigue failure of a connecting rod," Engineering Failure Analysis, Vol.3, No. 1, pp. 13-28,1966

[17] Moon Kyu Lee, Hyungyil Lee, Tae Soo Lee, Hoon Jang, "Buckling sensitivity of a connecting rod to the shank sectional area reduction", Journal of Materials and Design, 31, pp. 2796–2803,2010.

[18] Zhou Qinghui, Wang Yunying , Ji Wei , "The finite element analysis of connecting rod of diesel engine", International Conference on Measuring Technology and Mechatronics Automation, IEEE,2010.

[19] Ahamad Ridzuan Bin Ibrahim, "Analysis of Connecting Rod fracture using Finite Element Analysis", Bachelor Thesis, University of Malaysia,2010.

[20] Saharash Khare, O. P. Singh, K. Bapanna Dora, C. Sasun, "Spalling Investigation of Connecting Rod", TVS Motor Company Ltd., Research & Development, 2011.

[21] Dongkai Jia, Ke Wu, Shi Wu, Yuntong Jia, Chao Liang, "The Structural analysis and Optimization of Diesel Engine Connecting Rod", International Conference on Electronic & Mechanical Engineering and Information Technology, IEEE, 2011.

[22] M. SC. Anna Ulatowska, "Shape Optimization of the Connecting Rod",2008.

[23] Pravardhan S. Shenoy, "Dynamic Load Analysis and Optimization of Connecting Rod", Master's Thesis, University of Toledo,2004.

[24] C.S Sharma, K. Purohit, "Design of Machine Elements", Prentice Hall of India Publications, New Delhi 17, pp.631-653,2003.

[25] Dr. Rajendra Karwa,"Machine Design", Laxmi Publications, New Delhi, 20,pp.9-13,1999.