Finite Element Analysis and Material Optimization for Equivalent Strength of Composite Connecting Rod

Mr. Vivek T. Fegade[1], Dr. Kiran S. Bhole[2]
[1] Research Scholar, [2] Head, Department of Mechanical Engineering,
Datta Meghe College of Engineering. Airoli, India,

Abstract

This work describes about weight reduction opportunities for forged steel connecting rod of internal combustion engine. The existing connecting rod is modified to make it composite connecting rod using two materials: forged steel and titanium without disturbing its existing geometry. The design and weight affects the performance of connecting rod. With the implementation of optimization approach connecting rod of stronger but equally lighter can be obtained with minimum cost. Therefore, this study has dealt with two subjects, first, static load with detailed stress analysis of the connecting rod and second optimization of rod to reduce the weight and production cost. A appropriate CAD model of connecting rod was designed in CATIA V5, after that FEA was carried out in ANSYS WORKBENCH to determine maximum vonmisses stresses acting on it. In response to an increasing demand for fuel-economy, weight reduction technique have been proposed to create an optimum connecting rod design. The research aims to maximize weight savings in a connecting rod, without sacrificing the structural performances such as strength and stiffness.

Keywords: *Connecting Rod, FEA, Optimization, Static*

I. INTRODUCTION

Automobile engine connecting rod is the critical component in internal combustion engine which connects reciprocating piston and connecting rod. Its function is to transmit thrust of 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. Connecting rods for automotive applications are manufactured by forging process from powder metal or wrought steel . They could also be cast. The forgings process produces blow-hole-free and better rods than rods obtained by casting process. Powder metal manufactured blanks have the advantage of being near net shape, reducing material waste but the cost of the blank is more 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. The main aspect was to optimize the weight and manufacturing cost of the steel forged connecting rod. Optimization of the connecting rod for its weight or volume will result in large-scale savings due to high volume production. It can also achieve the objective of reducing the weight of the engine component, thus reducing inertia loads, reducing engine weight and improving engine performance and fuel economy.

The important component factors such as material, cross section conditions etc. can be change. Weight reduction opportunity for connecting rod is taken here in this paper as a part of study by making it composite with some modifications without disturbing geometry of existing connecting rod.

A. Ulatowska [1] , has performed shape optimization using computer-aided optimization method of forged steel connecting rod to reduce large notch stresses. When there are lower stresses remove material in order to increase the stress in this point and when there is more stresses add material in order to decrease the stress in this point. P. Shenoy et al. [2], has done an optimization study on a steel forged connecting rod with a consideration for improvement in weight and production cost. Augugliaro G. et al.[3], has discussed practical application for the optimization of a two dimensional connecting rod to reduce weight of connecting rod. M. K. Lee et al.[4], studied buckling of connecting rods in design and concluded that when weight reduction of connection rod shank is attempted, buckling should be considered as an essential factor along with the other criteria such as yield and fatigue. T. G. Thomas et al. [5], has studied connecting rod of heavy duty application and observed stress concentration near the transition between small end and shank. Zhixue Wu [6], investigated the optimization problem of finding the optimal shape of a mechanical component with the aim of presenting a simple and efficient numerical approach for minimizing stress concentration factor.

X. Hou et al. [7], has concluded that in order to adapt connecting rod for new engines, it is necessary to optimize structure of connecting rod. Z. Bin et al. [8], has observed during evaluation of diesel engine connecting rod that the destructive position is lie in between connecting rod shank & transition location of small end for maximum compression condition. M.S. Shaari et al. [9], has used topology optimization technique to develop structural modelling, finite element analyse and the optimization of the connecting rod for robust design. Ridzuan [10], has revealed that results are more accurate when mesh size is smaller . Failure occurs if vonmises stress value is higher compared to yield strength of material. Ed. Platt. [11], presented the design of high speed diesel engine connecting rod in which the inertia of the moving part must be taken into account. D. Jia et al. [12] , introduced a method of the connecting rod structural finite element analysis and optimization and observed stress concentration between the connecting rod pin end and the rod body.

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 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 forged 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, optimization, 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 and boundary conditions.

A. *Design of Connecting Rod*

Design of various parts of connecting rod such as connecting rod shank, big end, small end, bolts for cap, cap of big end is done as per standard design procedure.

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 (α)	1500-2500rpm
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. 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

B. *Modeling*

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

C. *Meshing*

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

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.

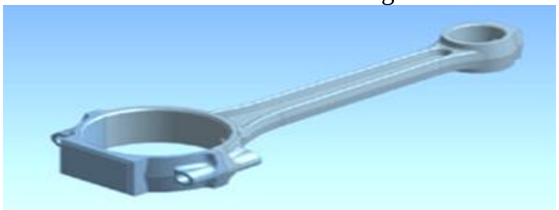

Fig. 1 CAD Model of Connecting Rod

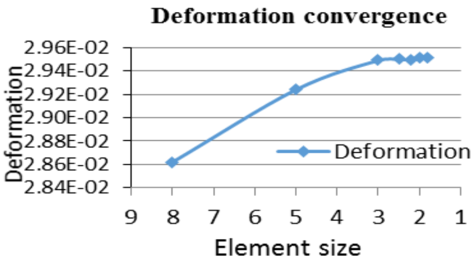

Fig. 2. Deformation Convergence Along Element Size

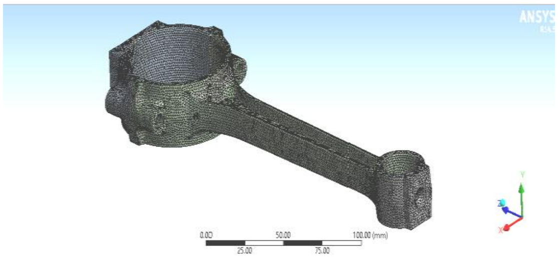

Fig. 3 Meshed Model of Connecting Rod

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:

$$Po = 15976/ (32 \times 34 \times \sqrt{3}) = 8.48 \text{ MPa}$$

Piston Pin End:

$$Po = 15976/ (12 \times 28 \times \sqrt{3}) = 27.45 \text{ MPa}$$

• Tensile Loading:

Crank End:

$$Po = 15976/ [32\times 34\times (\pi/2)] = 9.35 \text{ MPa}$$

Piston Pin End:

$$Po = 15976/ [12\times 28\times (\pi/2)] = 30.27 \text{ MPa}$$

Table 3 shows boundary conditions used for analysis of connecting rod for original model as well as optimized model.

E. 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.

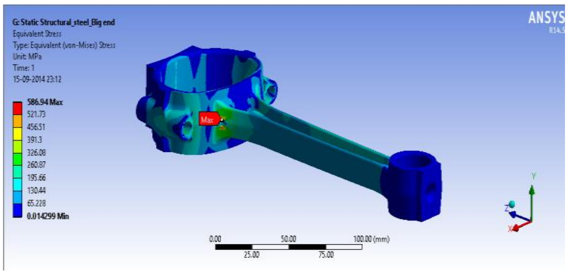

Fig.4 Vonmisses Steel Rod Crank End Compressive

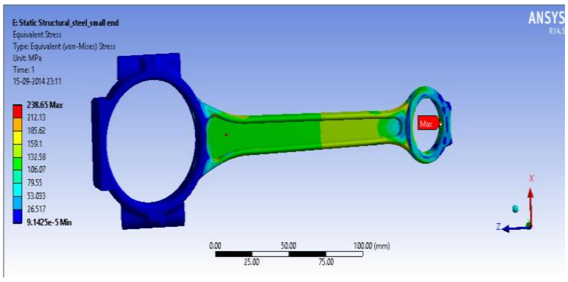

Fig.5 Vonmisses Steel Rod Pin End Compressive

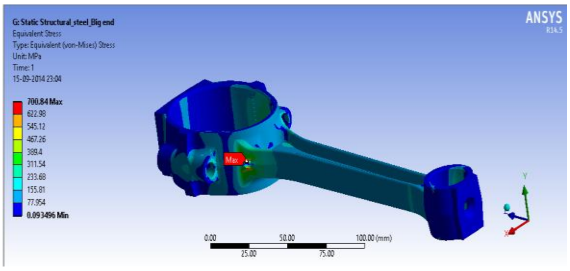

Fig.6 Vonmisses Steel Rod Crank End Tensile

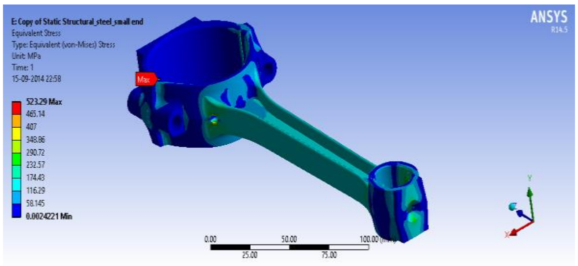

Fig. 7 Vonmisses Steel Rod Pin End Tensile

III. OPTIMIZATION

Simulation Methodology identifies the potential for weight reduction in the existing connecting rod. In actual operation, few regions of the connecting rod are stressed to much lower stress levels than under static load corresponding to the load at the crank end. The objective is to optimize the connecting rod for its weight and manufacturing cost, taking into account the recent developments.

Optimization carried out here is not in the proper mathematical sense. This case is not like that, an optimum solution is the minimum or maximum possible value the objective function could achieve under the defined set of constraints. Here attempt was made to reduce weight of the component. Rather than using numerical optimization techniques for weight reduction, judgment has been used and FEA results for low stress zones. This optimization task was performed manually, considering manufacturing feasibility and cost. The following factors have been addressed during the optimization: the stresses under the loads, deformation, etc. All of these have been checked to be within permissible limits.

A. Optimization Statement

Objective of the optimization task was to minimize the mass of the connecting rod under the effect of a load range comprising the two loads tensile as well as compressive load such that maximum vonmises stress are within the limits of the allowable stresses and factor of safety less than 1.3 as discussed below. The connecting rod has to be interchangeable with the existing one in the current engine. This requires some of the dimensions in the existing connecting rod to be maintained. These dimensions are inner diameter of pin end, inner diameter of crank end and connecting rod length.

Mathematically stated, the optimization statement would appear as follows:

Objective Function:

Minimize Mass /Volume

Subject to Constraints:

[1] Maximum Vonmises stress < Allowable stress

[2] Factor of safety > 1.3

[3] Manufacturing Constraints.

B. Constraints

1) Applied Loads

The load range under which the connecting rod was optimized is comprised of the tensile load and the compressive load of 15976N . The compressive load of 15976N is independent of the geometry of the connecting rod. The tensile load is, however, dependent upon the specific geometry, as it is a function of the mass, moment of inertia, and location of C.G.

2) Allowable Stress

The ratio of tensile strength to von Misses stress in the existing geometry to that of the existing material, referred to as factor of safety and its value was found out to be 1.32.

The inserting material selected for the optimized connecting rod was titanium. As a result, the Factor of safety was defined with respect to the tensile strength. As the name implies, factor of safety will be an indication of the margin of safety for possibility. The factor of safety greater than one means the design is safe. So throughout the analysis factor of safety is kept greater than 1.3. FEA results predict small increase in factor of safety than the assumed. Since the new material has higher tensile strength, the connecting rod was modified to maintain the factor of safety. Then there is having scope for weight reduction.

3) Geometry Constraints

The optimized connecting rod is expected to be interchangeable with the existing one. Geometry constraint contains those parameters that will not be changed; this includes diameters of the crank end and piston end, overall thickness of the connecting rod, length on connecting rod and fillet radius at the crank end.

C. Optimized Model

After carrying out static structural analysis the stresses in each loading conditions were studied and identified that shank i.e. region between crank end and pin end offers greatest potential for weight reduction as the stress level lower i.e. 116 MPa and 155 MPa. Where factor of safety is higher than 1.3.So the web of shank was identified for material replacement without changing the existing geometry. Other dimensions were not changed to maintain component interchangeability with the existing connecting rod.

A metal strip of same dimension of different thickness was decided to insert on both side of shank of existing rod. 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 inserted 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. Optimized geometry which was modified in Design modeller of ANSYS Workbench is shown in fig 8.

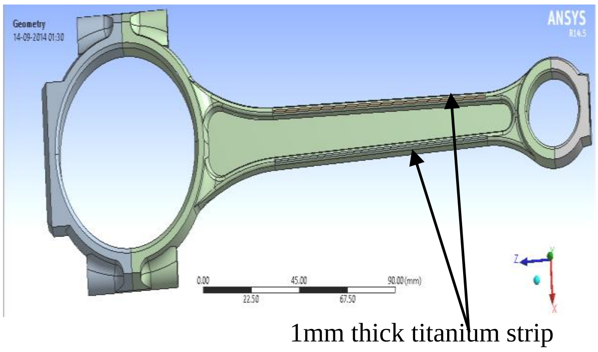

Fig. 8 Optimized Model of Connecting Rod

IV. RESULTS AND DISCUSSION

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.

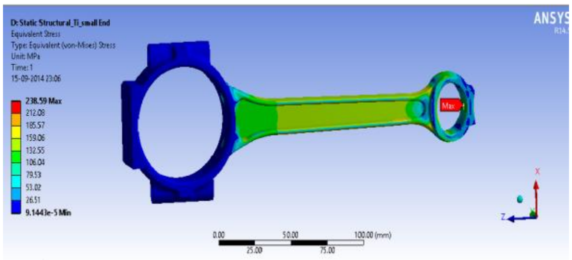

Fig. 9 Pin End Compressive

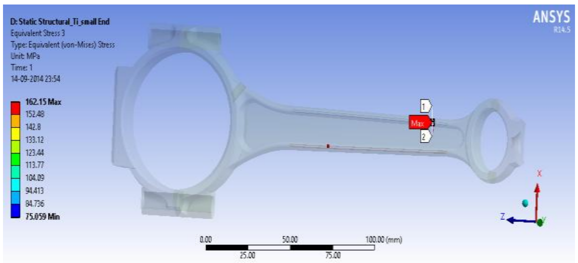

Fig.10 Pin End Compressive Along the Path

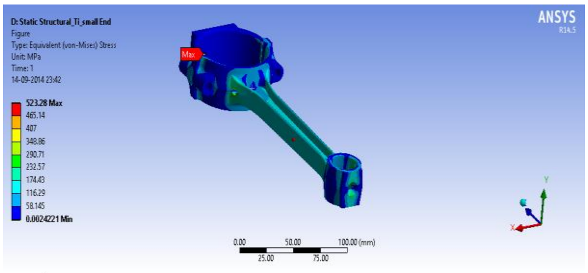

Fig. 11 Pin End Tensile

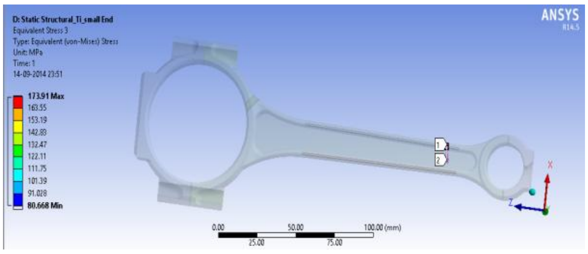

Fig. 12 Pin End Tensile Along the Path

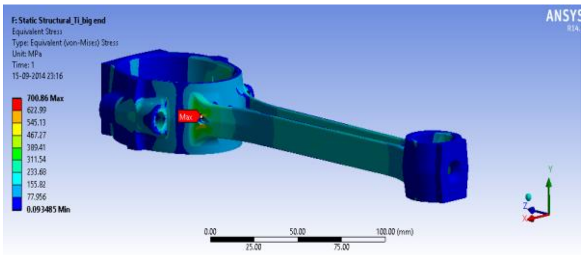

Fig. 13 Crank End Compressive

In ANSYS workbench static structural analysis of connecting rod was carried out to find out the maximum vonmises stress acting on it during tensile and compressive loading conditions. These stress values for original and optimized model were compared as shown in table **4**

Below shown is mass and factor of safety for both original as well as optimized model.

A. Original Model:

Mass =670.18 gm

Factor of Safety = 1.31

B. Optimized Model:

Mass= 659.91 gm

Factor of safety = 1.30

% of weight reduced = Total weight reduced / weight of steel rod x100

= (670.18-659.91) / 670.18 x100

= 0.01532 x 100

= 1.532 %

TABLE 4 Shows Maximum Vonmisses Stress Obtained at Different Loading Conditions for Original Model as Well as Optimized Model

Model		Maximum Vonmisses stress (MPa)		Maximum Vonmisses stress (Along path) (MPa)	
Pressure Applied	**Type of loading**	**Original**	**Optimized**	**Original**	**Optimized**
Small End	Tensile	150.44	173.91	700.84	700.86
	Compressive	140.32	162.15	523.29	523.28
Big End	Tensile	148.33	171.68	586.94	586.96
	Compressive	120.4	139.36	238.65	238.59

V. CONCLUSION

1. The variation of maximum vonmises stresses for each loading conditions in original model with comparison to optimized model was very less and it was found to be well below the limit of allowable stress.
2. For tensile loading with pressure applied at crank end and pin end restrained, maximum value of vonmises stress was observed in original as well as optimized which was critical loading condition used for optimization study.
3. 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.
4. 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.
5. Percentage weight reduction obtained by simulation in ANSYS Workbench based on stress observations was 1.5%.
6. Factor of safety was greater than 1.3 for all loading conditions for both original and optimized model obtained by weight reduction Methodology based on stress observations i.e. by judgment.
7. Insertion of titanium strip on both sides of shank in steel connecting rod increases the strength and stiffness of shank region.
8. Weight reduction will ultimately lead to cost savings in the material used for manufacturing of connecting rod with increased engine efficiency.

REFERENCES

[1] M. SC. Anna Ulatowska, (2008) “Shape Optimization of the Connecting Rod”

[2] Pravardhan S. Shenoy and Ali Fatemi, (2005), “Connecting Rod Optimization for Weight and Cost Reduction”,The University of Toledo, SAE Interantional.

[3] Augugliaro G.a, Biancolini M.E.,(2003) “Optimization of fatigue performance of a Titanium Connecting Rod, SAE technical paper 980800, 1-3.

[4] Moon Kyu Lee, Hyungyil Lee,Tae Soo Lee, Hoon Jang, (2010) “Buckling Sensitivity of a Connecting Rod to the Shank sectional area Reduction” Journal of Materials and Design, 31, pp. 2796–2803.

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

[6] Zhixue Wu, (2005) “An efficient approach for Shape Optimization of components”, International Journal of Mechanical ciences 47 pp. 1595–1610.

[7] Xianjun Hou, Cuicui Tian, Dan Fang, Fuming Peng and Fuwu Yan,(2009) “Sensitivity Analysis and Optimization for Connecting Rod of LJ276M Electronic Gasoline Engine”, IEEE, 978-1-4244-4507-3

[8] Zheng Bin Liu Yongqi, Ji Lixia, (2010) “Finite Element Analysis and Structural Improvement of Diesel Engine Connecting Rod”, Second International Conference on Computer Modeling and Simulation.

[9] M.S. Shaari, M.M. Rahman, M.M. Noor, K. Kadirgama, A.K. Amirruddin, (2010) “Design of Connecting rod of Internal Combustion Engine: A Topology Approach” National Conference in Mechanical Engineering Research and Postgraduate Studies pp. 155-166.

[10] Ahamad Ridzuan Bin Ibrahim, (2010), “Analysis of Connecting Rod fracture using Finite Element Analysis”, Bachelor Thesis, University of Malaysia.

[11] Ed.Platt, (1922), “Design of connecting rods for High speed Engines”, Dailmer premium paper session.

[12] Dongkai Jia, Ke Wu, Shi Wu, Yuntong Jia, Chao Liang, (2011) “The Structural analysis and Optimization of Diesel Engine Connecting Rod” International Conference on Electronic & Mechanical Engineering and Information Technology, IEEE, pp. 3289-3292

[13] Pravardhan S. Shenoy, 2004, “Dynamic Load Analysis and Optimization of Connecting Rod”, Master’s Thesis, University of Toledo.

[14] Asadi, M.R., Rasekh M., Golmohammadi A., Jafari A., Kheiralipour, K., Borghei A.M., (2010) “Optimization of Connecting rod of MF-285 tractor” Journal of Agricultural Technology, Vol. 6(4), pp. 649-662.

[15] P. Brabec, P. Kefurt, C. Scholz, R. Vozenilek, “FEM Analysis of Connecting Rod for Stationary Engine” Josef Bozek Research Centre for Engine and Vehicle Technologies II

[16] B. Lawton, D. E. G. Crutcher, (1967) “Mechanical Stresses in Pistons, Gudegeon pins, and Connecting Rod”

Proceedings of Institution Mechanical Engineers, Volume 181

[17] Rabb R, (1996) "Fatigue failure of a connecting rod," Engineering Failure Analysis, Vol.3, No. 1, pp. 13-28.

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

[19] Yunliang Ding, (1986) "Shape Optimization of Structures: A Literature Survey" computer and Structures Vol. 24. No. 6. pp. 985-1004.

[20] Zhixue Wu, (2005) ,"An efficient approach for Shape Optimization of components", International Journal of Mechanical Sciences 47 pp. 1595–1610

[21] Xiao Lei Xu, Zhi Wei Yu, (2007) "Failure Analysis of a Diesel Engine Connecting Rod" Journal of Failure Analysis and Prevention", Vol. 7, pp. 316–320.

[22] Adila Afzal and Ali Fatemi, (2004) "A Comparative Study of Fatigue Behavior and Life Predictions of Forged Steel and PM Connecting Rods" The University of Toledo, SAE International.

[23] Mirehei, M. Hedayati Zadeh, A. Jafari, M. Omid, (2008), "Fatigue analysis of Connecting Rod of Universal Tractor through Finite Element Method" Journal of Agricultural Technology, Vol. 4, pp. 21-27.