# Using Shape Optimization Tool In Ansys Software For Weight Reducation Of Steel Connecting Rod

[1] *Mr. H.B.Ramani*, [2] *Mr. Neeraj Kumar*

[1]M.Tech.[Production Engineering] Students, Mechanical Engineering Department, Shri Balaji College of Engineering & Technology, Jaipur, Rajasthan, India
[2] Assistance Professor at Shri Balaji College of Engineering & Technology, Jaipur, Rajasthan, India, Mechanical Engineering Department

## Abstract:-

*The connecting rod is the intermediate member between the piston and the crankshaft. Its primary function is to transmit the push and pull from the piston pin to the crank pin, thus converting the reciprocating motion of the piston into rotary motion of the crank. This paper describes weight reduction of internal combustion engines connecting rod by ANSYS Workbench software. The objectives of this paper are to develop structural modelling, finite element analyse and the optimization of the connecting rod for robust design. The structure of connecting rod was modelled utilized Pro-E software and analysis was performed using ANSYS* Workbench *software. Static analysis was carried out for finding the stress/strain results. Shape optimization technique is used for performing optimization cause measurable reduction in weight of connecting rod. By the FEA analysis results, the crank end is suggested to be redesign based on the Shape optimization results. The optimized connecting rod is 15% lighter and predicted low maximum stress compare to initial design.*

***Keywords:*** connecting rod, finite element analysis, weight reduction, optimization, ANSYS Workbench ,Shape optimization

## 1. INTRODUCTION:-

Connecting rods are highly dynamically loaded components used for power transmission in combustion engines. The optimization of connecting rod had already started as early year 1983 by Webster and his team. However, each day consumers are looking for the best from the best. That's why the optimization is really important especially in automotive industry. Optimization of the component is to make the less time to produce the product that is stronger, lighter and less cost. The design and weight of the connecting rod influence on car performance. Hence, it is effect on the car manufacture credibility. Change in the structural design and also material will be significant increments in weight and performance of the engine. Mirehei et al.(2008) were performed the study regarding the fatigue of connecting rod on universal tractor (U650) by using ANSYS software application and the lifespan was estimated. The authors also investigated that the stresses and hotspots experienced by the connecting rod and the state of stress as well as stress concentration factors can be obtained and consequently used for life predictions. Rahman *et al.*(2008a, 2009a) discuss about FEA of the cylinder block of the free piston engine. The 4 nodes tetrahedral (TET4) element version of the cylinder block was used for the initial analysis. The comparison then are made between TET4 and 10 nodes tetrahedral (TET10) element mesh while using the same global mesh length for the highest loading conditions in the combustion chamber. A connecting rod is subjected to many millions of repetitive cyclic loadings. Therefore, durability of this component is of critical importance. It is necessary to investigate finite element modelling techniques, optimization techniques and new design to reduce the weight at the same time increase the strength of the connecting rod itself. Shenoy (2004) was explored the weight and cost reduction opportunities for a production forged steel connecting rod. The study has dealt with two parts which are dynamic load and quasi-dynamic stress analysis of the connecting rod, and second to optimize the weight and cost. Shenoy and Fatemi (2005) were explained about optimization study was performed on a steel forged connecting rod with a consideration for improvement in weight and production cost. Weight reduction was achieved by using an iterative procedure. In this study weight optimization is performed under a cyclic load comprising dynamic tensile load and static compressive load as the two extreme loads. Yang et al. (1992) describes a successful process for performing component shape optimization should be focused on design modelling issues. A modular software system is described and some of the modules are widely available commercial programs such as MSC/PATRAN and MSC NASTRAN and ANSYS. The upper end (pin end) of a connecting rod is optimized under a variety of initial assumptions to illustrate the use of the system. The objectives of the study are to develop structural modelling of connecting rod and perform finite element analysis of connecting rod. The main objective is to develop shape optimization model of connecting rod.

## 2. OPTIMIZATION APPROACH:-

The objective of optimization technique is to minimize the mass of the connecting rod and reduces the cost of production. The connecting rod subjected to tensile load at crank end, while using factor of safety 1.6 -1.7 maximum stress of the connecting rod monitored and make sure it is not over the allowable stress. The load of the connecting rod optimized is comprised of the tensile load of 86.4 kN at crank end. The optimization technique methodology flowchart is shown in Figure 1

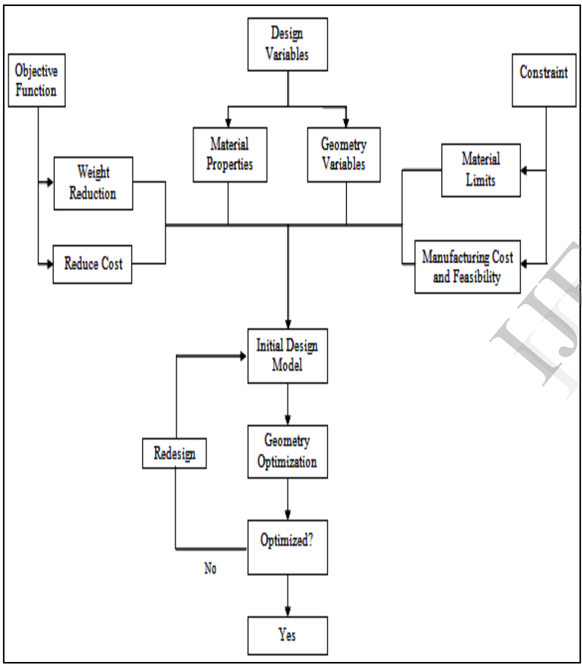


Fig.1 Optimization Process Chart

## 3. FINITE ELEMENT MODELLING AND ANALYSIS:

The connecting rod is one of the most important components in the internal combustion engine (Rasekh *et al.*, 2009). Therefore, the initial design is compared to other design before performing the optimization. A simple three-dimensional model of connecting rod was developed using Pro-E 5.0 software and finite element model was created using mesh size 1mm (Node 152873) as shown in Figure 3. Mesh study was performed on the FE model to ensure sufficiently fines sizes are employed for accuracy of the calculated result depends on the CPU time. During the analysis, the specific variable and the mesh convergence was monitored and evaluated. The mesh convergence is based on the geometry, model shape and analysis objectives. The uniformly distributed tensile load 120° on the inner surfaces of the crank end while the other part, pin end is restrain as in Figure 4. It is just same when load uniformly distributed on pin end surfaces, the crank end will restrain in all direction. This both cases also work exactly in compressive load. In Figure 5, shows the boundary condition of the connecting rod in three-dimensional FE model with load and constraints. In this study four finite element models were analysed. FEA for both tensile and compressive loads were conducted. Two cases were analysed for each case, Firstly, load applied at the crank end and restrained at the piston pin end, and secondly, load applied at the piston pin end and restrained at the crank end and the axial load was 86.4kN

Table.1 Properties of Connecting Rod Material

| | |
|---|---|
| Tensile Yield Strength | 550 MPa |
| Tensile Ultimate Strength | 900MPa |
| Compressive Yield Strength | 550MPa |
| Compressive Ultimate Strength | 600Mpa |
| Poisson Ratio | 0.3 |
| Density | 7850 Kg/m3 |
| Young's Modulus | 210000 MPa |



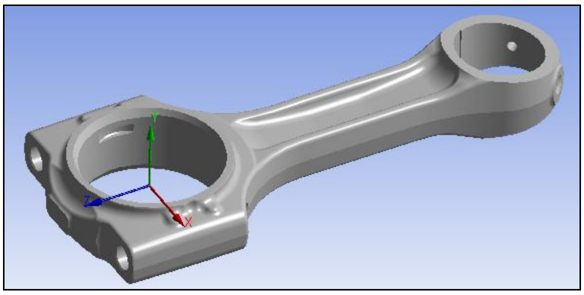

Fig. 2 3-D Model of Connecting rod

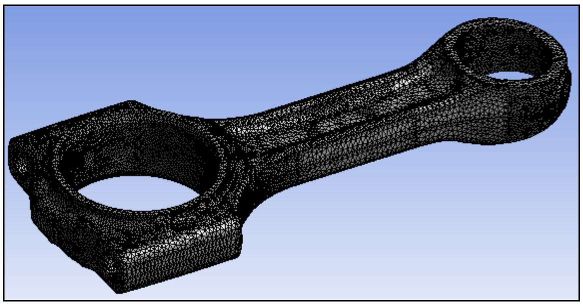

Fig.3 Mesh model of connecting rod (mesh size 1mm)

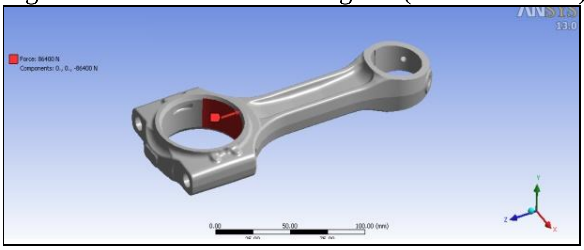

Fig.4 Load Apply on inner surface of connecting rod(120°)

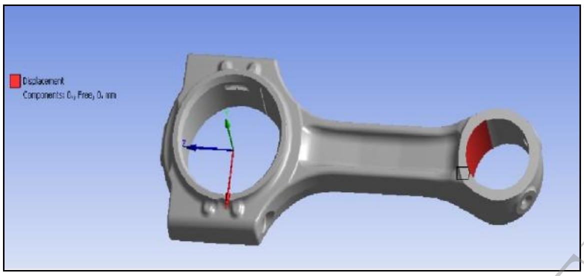

Fig.5 Constrain Apply on Pin End

.

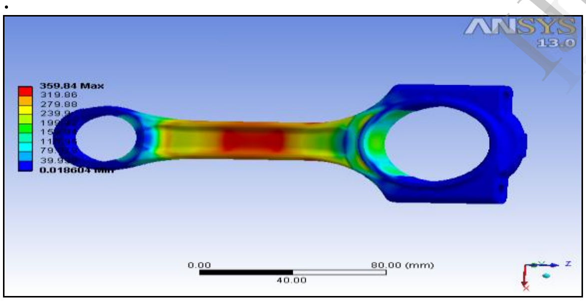


Fig 6 Von Mises Stress in Compressive Load of 86400 N

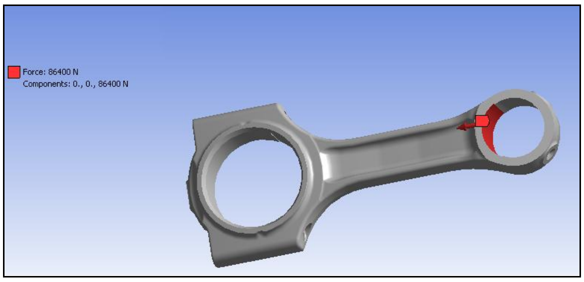

Fig.7 Load Apply on inner surface of connecting rod(180°)

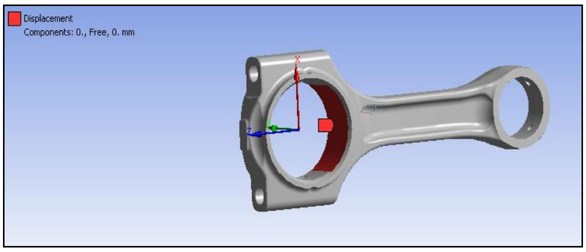

Fig.8 Constrain Apply on Big End.

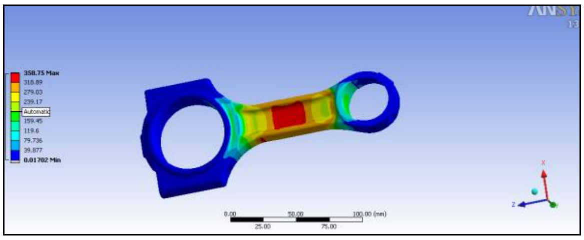

Fig.9 Von Mises Stress in Compressive Loading at Small End. (86400N)

Table 2. Comparison of Stress at Compressive load 864000 N and Tensile load 21600 N

| Sr. No | Load condition | Stress (MPa) |
|---|---|---|
| 1 | Crank Tensile | 429.02 |
| 2 | Pin Tensile | 469.88 |
| 3 | Crank Compressive | 359.84 |
| 4 | Pin Compressive | 358.75 |



## 4. OPTIMIZATION OF CONNECTING ROD

The optimization of the connecting rod carried out using shape optimization technique. The optimization focused on the uncritical sections which need to be reduced. From the shape optimization, it is suggest the unnecessary shape and design of the connecting rod. The results of shape optimization of the connecting rod are shown in Figure 8.The main objective is to minimize the weight of the connecting rod as well as the total production cost. It can be seen that the optimized model is reduce the weight from initial design until the value converges. The implementation of these optimizations is to find out the best design and shape of the connecting rod to improve the performance and the strength especially at the critical location. The possible modification section of the optimized connecting rod is indicated in the figure.8 the section with lower value than initial value considered as the suggestion to be optimized in the new design. Table 2 shows the comparison between initial and optimize designs on max principles stress and mass of the connecting rod. The optimize connecting rod was choose as the best optimize design due to the lowest occurred stress and mass. Even though the mass of the optimize connecting rod is not the lowest, but the decision was also based on the maximum stress which is 353 MPa. Figure 10 shows the new Design of the connecting rod and mass of the connecting rod is 0.706 kg compare to initial design 0.785 kg which is 10% lighter.

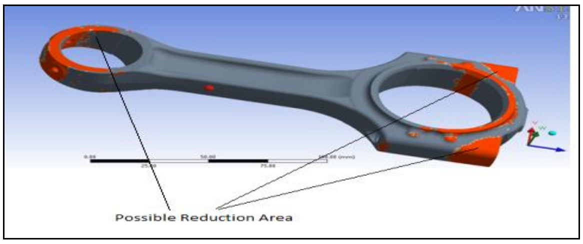


Fig.9 optimization Fig of Connecting rod (20%)

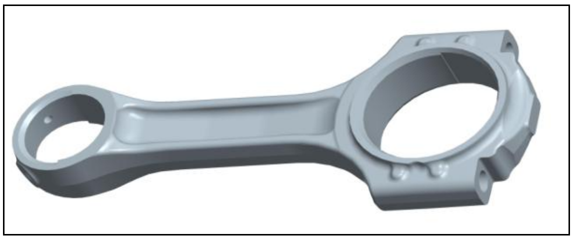

Figure 10 (a) Isometric 3D view of optimized design.

Table 3 Weight optimization Table.

| Sr.No. | Percentage (%) | Mass(kg) | Optimize weight(kg) |
|---|---|---|---|
| 1 | 10 | 0.785 | 0.706 |
| 2 | 15 | 0.785 | 0.667 |

Table 4 Result Comparison (20% optimized Weight)

| Sr. No. | Object | Load(N) | Stress(MPa) |
|---|---|---|---|
| 1 | Original | 86400 | 359.84 |
| 2 | Optimized | 86400 | 352.11 |

**5. CONCLUSION:**

By the finite element analysis method and the assistance of ANSYS Workbench software, it is able to analyse connecting rod for strees and strain. In this research we tried to simulate real condition by notice to all of effective forces on connecting rod. Shape optimization were analysed to the connecting rod and according to the results, it can be concluded that the weight of optimized design is up to 15% lighter and maximum stress also predicted lower than the initial design of connecting rod. The results clearly indicate that the new design much lighter and has more strength than initial design of connecting rod. Material optimization approach will be considered for future research.

**6. FUTURE SCOPE**

We can also reduce cost of connecting rod by change in manufacturing process and also by change in material.

**REFERENCES:-**

[1] Hexahedra, CAD-FEM GmbH, Germany.
Yang, R.J., Dewhirst , D.L., Allison, J.E. and Lee, A. 1992. Shape optimization of Connecting rod
[2] Rasekh, M., Asadi, M.R., Jafari, A. and Kheiralipour, K. 2009. Obtaining maximum stresses in different parts of tractor (MF-285) connecting rods using finite element method, Journal of Basic and Applied Sciences. 3(2): 1438-1449.
[3] Durability assessment of cylinder block for two stroke free piston linear engine using random loading. American Journal of Applied Sciences, 6(4): 726-735.
[4] Rahman, M.M., Ariffin, A.K., Abdullah, S., Noor, M.M., Bakar, R.A. and Maleque, M.A. 2008b. Finite element based fatigue life prediction of cylinder head for two-stroke linear engine using stress-life approach. Journal of Applied Sciences, 8(19):3316-3327.
[5] Rahman, M.M., Ariffin, A.K., Jamaludin, N. and Abdullah, S. 2007. Effect of nitriding treatment on fatigue life for free piston linear engine component using frequency response method: a finite element approach. Structural Durability and Health Monitoring, 3(4): 197-209.
[6] Mirehei, A.,Zadeh, H.M., Jafari, A. andOmid. M. 2008. Fatigue analysis of connecting rod of universal tractor through finite element method (ANSYS). Journal of Agricultural Technology. 4(2): 21-27.
Rahman, M.M., Ariffin, A.K., Abdullah, S., Noor, M.M.and Bakar, R.A. 2009b.
[7] Finite element based fatigue life prediction of a new free piston engine mounting. Journal of Applied Sciences, 8(9):1612-1621.
[8] Rahman, M.M., Ariffin, A.K., Rejab, M.R.M., Kadirgama, K., Noor, M.M. 2009c.
[9] Multiaxial fatigue behaviour of cylinder head for a free piston linear engine" Journal of Applied Sciences, 9(15): 2725 - 2734.
[10] Rahman, M.M., Kadirgama, K., Noor, M.M., Rejab, M.R.M. and Kesulai, S.A., 2009a, Fatigue life prediction of lower suspension arm using strain-life approach, European Journal of Scientific Research. 30(3): 437-450.
[11] Rasekh, M., Asadi, M.R., Jafari, A. and heiralipour, K. 2009. Obtaining maximum stresses in different parts of tractor (MF-285) connecting rods using finite element method, Journal of Basic and Applied Sciences. 3(2): 1438-1449.
[12] Shenoy, P.S and Fatemi, A. 2005. Connecting rod optimization for weight and cost reduction, SAE Technical Paper, Paper No. 2005- 01-0987.
[13] Shenoy, P.S. 2004. Dynamic load analysis and optimization of connecting rod. Master'sThesis. University of Toledo, USA.
Wang, E., Nelson, T. and Rauch, R. 2004, Back to Elements - Tetrahedra vs.
[14] connecting rod pin end using a generic model. Finite Elements in Analysis andDesign. 11: 257-264.
[15] Lee, M.K., Lee, H., Lee, T.S. and Jang

H. 2010, Buckling sensitivity of aConnecting rod to the shank sectional areaReduction, Materials and Design, 31: 2796–2803.