RESEARCH ARTICLE OPEN ACCESS

FE-Analysis of Connecting Rod of I.C.Engine by Using Ansys for Material Optimization

Mr. J.D.Ramani*, Prof. Sunil Shukla**, Dr. Pushpendra Kumar Sharma***
*(M. Tech (Machine Design and Robotics) Student, Mechanical Engineering Department, NIIST Bhopal)
** (Assistant Professor, Department of Mechanical Engineering, NIIST, Bhopal)
*** (HOD, Department of Mechanical Engineering, NIIST, Bhopal)

ABSTRACT
In series of automobile engine components a connecting rod is highly critical and researchable one. The main idea of this study is to do analysis of connecting rod and get idea of stress producing during compressive and tensile loading. And then give idea about weight reduction opportunities for a production steel connecting rod. This has entailed performing a detailed load analysis. Therefore, this study has contain by two subjects, first, load and stress analysis of the connecting rod, and second, optimization for weight reducation. In the first part of the study, loads acting on the connecting rod and find out stress-time history at some critical point. The results were also used to determine the variation of Tensile and Compressive loading the component was optimized for weight reduction subject to space constraints and manufacturability.

Keywords – Ansys Workbench, Connecting Road, FEA, Optimization, Pro-E,

I. INTRODUCTION

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. A sizeable portion of the US market for connecting rods is currently consumed by the powder metal forging industry.

A connecting rod consists of a pin-end, a shank section, and a crank-end as shown in Fig. 1. Pin-end and crank-end pinholes at the upper and lower ends are machined to permit accurate fitting of bearings. These holes must be parallel. The upper end of the connecting rod is connected to the piston by the piston pin. If the piston pin is locked in the piston pin bosses or if it floats in the piston and the connecting rod, the upper hole of the Connecting rod will have a solid bearing (bushing) of bronze or a similar material. As the lower end of the connecting rod revolves with the crankshaft, the upper end is forced to turn back and forth on the piston pin. Although this movement is slight, the bushing is necessary because of the high pressure and temperatures.

The lower hole in the connecting rod is split to permit it to be clamped around the crankshaft. The bottom part, or cap, is made of the same material as the rod and is attached by two bolts. The surface that bears on the crankshaft is generally a bearing material in the form of a separate split shell. The two parts of the bearing are positioned in the rod and cap by dowel pins, projections, or short brass screws. Split bearings may be of the precision or semi precision type.

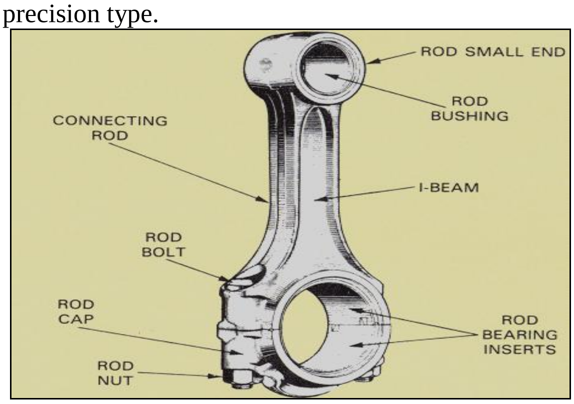

Fig -1: A**utomobile Connecting rod**

II. DESIGN PROCEDURE OF CONNECTING ROD

The connecting rod undergoes a complex motion, which is characterized by inertia loads that induce bending stresses. In view of the objective of this study, which is optimization of the connecting rod, it is essential to determine the magnitude of the loads acting on the connecting rod. In addition, significance of bending stresses caused by inertia loads needs to be determined, so that we know whether it should be taken into account or neglected during the optimization. Nevertheless, a proper picture of the stress variation during a loading cycle is essential from fatigue point of view and this will require FEA over the entire engine cycle.

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 crankpin because it is rigid, and thus convert the reciprocating motion of the piston into the rotary motion of the crank. The usual form of the connecting rod in internal combustion engines is, it consists of a long shank, a small end and a big end. The cross-section of the shank may be rectangular, circular, tubular, I-section or H-section. Generally circular section is used for low speed engines while I-section is preferred for high speed engines The length of the connecting rod (l) depends upon the ratio of l/r, where r is the radius of crank. It may be noted that the smaller length will decrease the ratio L/R.

2.1 Dimensions of Cross-Section of the Connecting Rod

Fig -2: Model of Connecting rod

Forces acting on the connecting rod, the various forces acting on the connecting rod are as follows:

- Force on the piston due to gas pressure and inertia of the reciprocating parts.
- Force due to inertia of the connecting rod or inertia bending forces.
- Force due to friction of the piston rings and of the piston.
- Force due to friction of the piston pin bearing and the crankpin bearing.

In designing a connecting rod, the following Dimensions are required to be determined.

- Dimensions of cross-section of the connecting rod.
- Dimensions of the crankpin at the big end and the piston pin at the small end.
- Size of bolts for securing the big end cap.
- Thickness of the big end cap.

The procedure adopted in determining the above mentioned dimensions is discussed as below.

III. FEA OF CONNECTING ROD

Finite element analysis (FEA) has become common place in recent years, and is now the basis of a multibillion dollar per year industry. Numerical solutions to even very complicated stress problems can now be obtained routinely using FEA, and the method is so important that even introductory treatments of mechanics of materials such as these modules should outline its principal features.

H.B.Ramani has do Analysis of Connecting Rod under Different Loading Condition Using ANSYS Software [1].In spite of the great power of FEA, the disadvantages of computer solutions must be kept in mind when using this and similar methods, they do not necessarily reveal how the stresses are influenced by important problem variables such as materials properties and geometrical features, and errors in input data can produce wildly incorrect results that may be overlooked by the analyst. Finite element codes are less complicated than many of the word processing and spreadsheet packages found on modern microcomputers. Nevertheless, they are complex enough that most users do not find it effective to program their own code. A number of prewritten commercial codes are available, representing a broad price range and compatible with machines from microcomputers to supercomputers. However, users with specialized needs should not necessarily shy away from code development, and may find the code sources available in such texts as that by O.C. Zienkiewicz to be a useful starting point. Most finite element software is written in FORTRAN, but some newer codes such as felt are in ‘C’ or other more modern programming languages.
During meshing 1mm node size is perfect preferable for analysis so for 1mm meshing size our connecting road have 179355 elements and 304977 nodes. And then on this element boundary condition will apply.

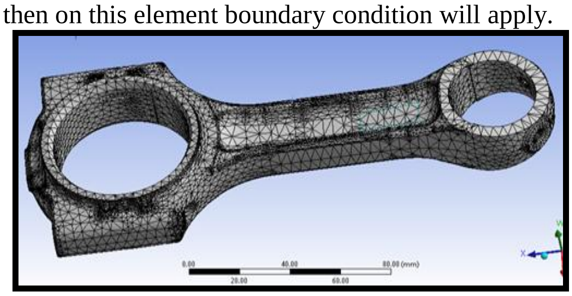

Fig -3 Boundary Conditions

Asadi, et al (2010) [2] developed detailed load analysis under service loading conditions was performed for connecting rod and The conclusions can be drawn from this study as follows the maximum pressure stress was obtained between pin end and rod linkage and the maximum tensile stress was obtained in lower half of pin end. The crank and piston pin ends are assumed to have a sinusoidal distributed loading over the contact surface area, under tensile loading, as shown in the Fig. 3. This is based on experimental results. [3]

In this study four finite element models were analyzed. FEA for both tensile and compressive loads were conducted. Four cases were analyzed for each case, one with load applied at the crank end and restrained at the piston pin end, and the other with load applied at the piston pin end and restrained at the crank end. Following four conditions were tested:

- Crank End Compressive Load
- Crank End Tensile load
- Pin End Compressive Load
- Pin End Tensile Load

Already mentioned, four FEA models were solved. Fig. 6 shows a FEA model in which tensile load is applied at the crank end and the piston pin end is restrained. Note that half of the piston pin inner surface is completely restrained 180^o contact surface area is totally restrained, i.e. X, Y, Z translations of all the node on this surface are set to zero if the connecting rod is in tension. Similarly, when the connecting rod is under axial compressive load, 120^o of contact surface area is totally restrained. Fig. 6 shows FEA model in which compressive load (86400 N) is applied at the crank end and pin end is restrained.

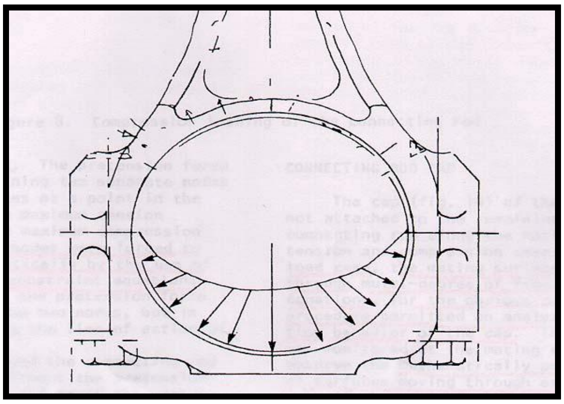

Fig -4 Tensile Loading of the Connecting Rod

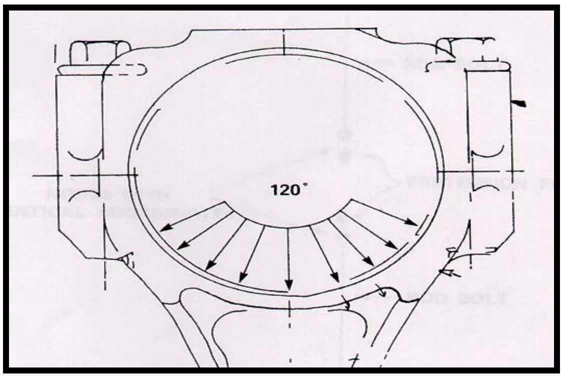

Fig -5 Compressive Loading of the Connecting Rod

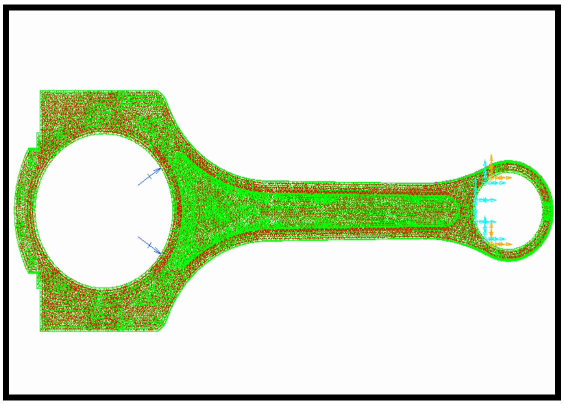

Fig. -6 FEA Model of the Connecting Rod with Axial Compressive Load at the Crank End Uniformly Distributed Over 120^0

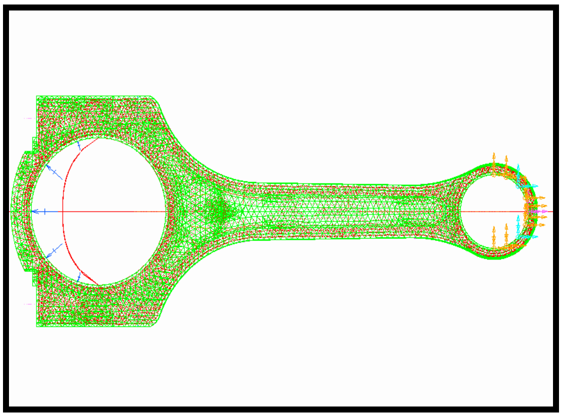

Fig. 7 FEA Model of the Connecting Rod with Axial Tensile Load at the Crank End with Cosine Distribution Over 180^0 and Piston Pin End Restrained Over 180^0

From the analysis we get different result related stress, strain and total deformation are as max. Stress 360.25 MPA Total strain 1.8 and total deformation is 156.7 micrometer. If F.O.S is find then it will between 1.6 to 1.7 and its totally safe.

Then Optimization tool is applied on current analysis.

IV. OPTIMIZATION

Pravardhan S.Shenoy and Ali Fatemi [4] performed an optimization study on a steel forged connecting rod with a consideration for improvement in weight and production cost. Since the weight of the connecting rod has little influence on its total production cost, the cost and the weight were dealt with separately. Constraints of fatigue strength, static strength, buckling resistance and manufacturability were also imposed. The fatigue strength was the most significant factor in the optimization of the connecting rod. An estimate of the cost savings is also made. The study results in an optimized connecting rod that is 10% lighter and 20% less expensive, as compared to the existing connecting rod. In this paper also used optimization tool trough the Ansys and can be useful to remove excess material from the existing connecting road. So automatically it can improve economic effect on manufacturing process. Objective function is the dependent variable that you are attempting to minimize. It should be a function of the DVs, i.e. changing the value of the DVs should change the value of the objective function. You can define only one objective function in an ANSYS design optimization problem. [5] The applied load distribution at the crank end and at the piston pin end was based on experimental results. They were also used in other studies in the literature by Folgar and Athavale and Sajan pawar. Since the details were not discussed by Webster, the applicability of the loading to this connecting rod could not be evaluated. [6]

V. RESULT

After the analysis we get result is that after the optimization F.O.S is remains between 1.6 to 1.7 and still it indicates safe zone. By optimization our connecting rod of 0.785 Kg. can be optimized by 0.690 Kg.The FEM includes the axial compressive load applied to the pin at the piston pin end, the restraints applied to the crank pin, the interference simulated by applying pressure, and contact elements between the pins and the connecting rod. Static FEA results showed high stresses in the regions of the transitions to the shank at the crank end and piston pin end, the oil hole, and the cap. From these regions, representative locations were selected at which stresses could be traced. However, a node is created and clearly identified at a vertex.Fig.8 to 10 shows the force applied, stress, and strain in connecting rod.

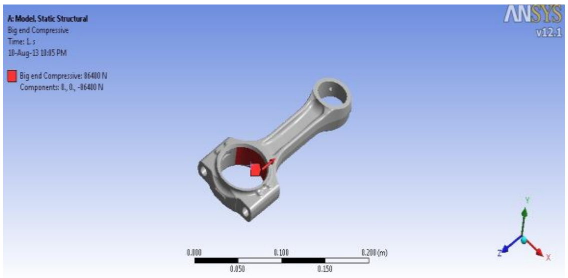

Fig. 8 Force Apply on Big End in Compressive loading

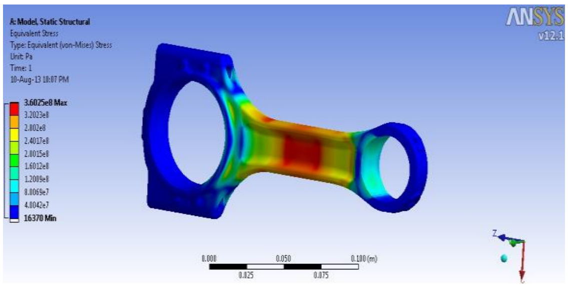

Fig. 9 Von Misses Stress in Compressive Load

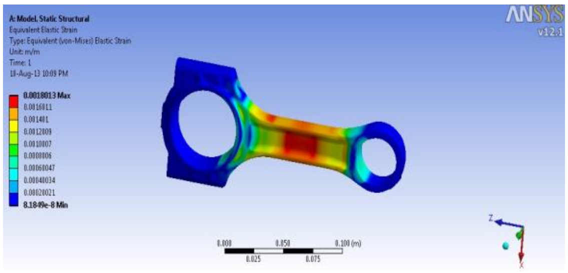

Fig. 10 Von Misses Stain in Compressive Load

VI. CONCLUSION

It is the conclusion of this study that the connecting rod can be designed and optimized under a load range comprising compressive load as one extreme load and tensile load. Furthermore, the existing connecting rod can be replaced by optimization with a new connecting rod made of lighter in weight (approx. 15%).

REFERENCES

[1] H.B.RAMANI, Mr. Neeraj Kumar, Mr. P. M. Kasundra"Analysis of Connecting Rod under Different Loading Condition Using ANSYS Software" International Journal of Engineering Research and Technology (IJERT) ISSN: 2278- 0181Volume 1 Issue 9 Published in November 2012

[2] Webster,W., D.Coffell R., and Alfaro D.,(1983),"A Three Dimensional Finite Element Analysis of a High Speed Diesel Engine Connecting Rod," SAE Technical Paper Series, Paper No. 831322

[3] Asadi, et al (2010), "paper of Optimization of connecting rod of MF-285 tractor"

[4] Pravardhan S. Shenoy, Ali Fatemi (2005) "Connecting Rod Optimization for Weight and Cost Reduction." SAE International Journal

[5] http://www1.ansys.com/customer/content/documentation/121/ans_tut.pdf

[6] Athavale S. and Sajanpawar, P. R., (1991), "Studies on Some Modeling Aspects in the Finite Element Analysis of Small Gasoline Engine Components," Small Engine Technology Conference Proceedings, Society of Automotive Engineers of Japan, Tokyo, pp. 379-389