International Journal of Enhanced Research in Science, Technology & Engineering
ISSN: 2319-7463, Vol. 5 Issue 7, July-2016

Static Structural Analysis and Weight Reduction of Connecting Rod Using ANSYS

Ashish Kumar[1], Parveen Kamboj[2], Ketan Mittal[3]

[1]PG Scholar, Department of Mechanical Engineering, JCDM College of Engineering, Sirsa (Hry.), India
[2,3]Assistant Professor, Department of Mechanical Engineering, JCMD College of Engineering, Sirsa (Hry.), India

ABSTRACT

Connecting rod is the power transmission element which is used to transfer power form piston to the crank shaft in Internal Combustion Engine (IC Engine). Its main function is to convert reciprocating motion of piston into rotary motion of crank shaft. Every internal combustion engine requires at least one connecting rod and it must have the highest possible rigidity at the lowest weight for proper functioning. The major stress induced in the connecting rod is a combination of axial and bending stresses in operation. Therefore, the paper deals with the stress analysis of connecting rod by Finite Element Method using CATIA V5 and ANSYS Workbench software. The objective of present work is to optimize the material of connecting rod. The material used is aluminium fly ash silicon composite that is manufactured using industrial waste such as fly ash. In the work the model of connecting rod is designed in CATIA V5 software and saved in iges format. The model is then imported in Solid Works software and saved in Para solid (x_t) format file for no data loss. This Para solid file of connecting rod is then imported in ANSYS workbench. The deformation and stress contours have been plotted and the results obtained are compared with available results in literature survey. The design optimization of connecting rod is carried out with a view to reduce weight and cost.

Keywords: ANSYS, Connecting Rod, CATIA V5, FEA

I. INTRODUCTION

Connecting rod is the intermediate link between the piston and the crank. And is responsible to transmit the push and pull from the piston pin to crank pin, thus converting the reciprocating motion of the piston to rotary motion of the crank. Connecting rod, automotives should be lighter and lighter, should consume less fuel and at the same time they should provide comfort and safety to passengers, that unfortunately leads to increase in weight of the vehicle. This tendency in vehicle construction led the invention and implementation of quite new materials which are light and meet design requirements [1].

The earliest evidence for a connecting rod appears in the late 3rd century AD Roman Hierapolis sawmills. It also appears in two 6th century Eastern Roman saw mills excavated at Ephesus and Gerasa. The crank and connecting rod mechanism of these Roman watermills converted the rotary motion of the waterwheel into the linear movement of the saw blades. Sometime between 1174 and 1206, the Arab inventor and engineer Al-Jazari described a machine which incorporated the connecting rod with a crankshaft to pump water as part of a water-raising machine, but the device was unnecessarily complex indicating that he still did not fully understand the concept of power conversion.

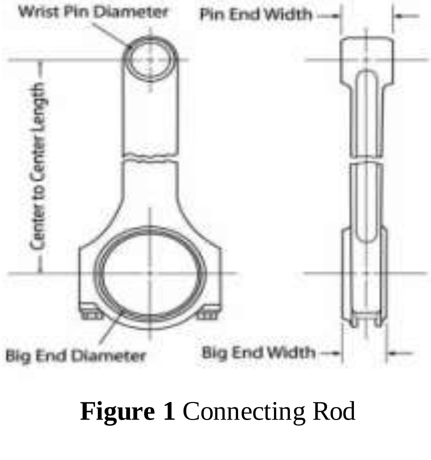

Figure 1 Connecting Rod

There are many types of connecting rod with different I section and H section. But there are basically two types of connecting rod.

- Connecting rod with nut and bolt: The connecting rod with cap at the larger end is joined by means of bolt. This type of connecting rod is most widely used in multicylinder engines. For example trucks, tractor etc
- Connecting rod without nut and bolt: This type of connecting rod consists of single parts itself. And mostly used in single cylinder engine. For example bikes, scooter etc.

II. LITERATURE

Ambrish Tiwari et al. (2014) presented the paper for connecting rod Finite Element Analysis for weight and cost reduction opportunities for a production of forged steel connecting rod. It was also performed a fatigue study based on Stress Life theory, considering the Modified Goodman diagram [1].

Kuldeep B et al. (2013) analysed and optimized the Connecting rod using alfasic Composites. In this work connecting rod is replaced by aluminium based composite material reinforced with silicon carbide and fly ash. FEA analysis was carried out by considering two materials. The parameters like von misses stress, von misses strain and displacement were obtained from ANSYS software. Compared to the former material the new material found to have less weight and better stiffness. It resulted in reduction of 43.48% of weight, with 75% reduction in displacement [2].

Leela Krishna Vegi et al. (2013) designed the Connecting Rod Using Forged steel and analysis is done using Ansys. A parametric model of Connecting rod is modeled using CATIA V5 R19 software and to that model, analysis is carried out by using ANSYS 13.0 Software. Finite element analysis of connecting rod is done by considering the material Forged steel. It was observed that Forged steel has more factor of safety, reduce the weight, increase the stiffness and reduce the stress and stiffer than other material like carbon steel [3].

Pushpendra kumar Sharma et al. (2012) performed the static FEA of the connecting rod using the software and said optimization was performed to reduce weight. Weight can be reduced by changing the material of the current forged steel connecting rod to crackable forged steel (C70). And the software gives a view of stress distribution in the whole connecting rod which gives the information that which parts are to be hardened or given attention during manufacturing stage [4].

Raviraj Yashwant Taware et al. (2015) carried out the analysis of Connecting Rod Used in Two Wheeler under Static Loading by FEA. In this work connecting rod is replaced by materials ASTM A216 GR WCB and Aluminum 360 for Hero Splendor motorbike. A 2D drawing is drafted from the calculations. A parametric model of connecting rod is modelled using CATIA V5R17 software. Analysis is carried out by using ANSYS Workbench 15.0 software. The best combination of parameters like Von mises stress and strain, deformation etc for two wheeler connecting rod was done in ANSYS Workbench15.0. Finally these results were compared with each other [5].

III. METHODOLOGY

Every internal combustion engine requires at least one connecting rod and it must have the highest possible rigidity at the lowest weight for proper functioning. The major stress induced in the connecting rod is a combination of axial and bending stresses in operation. Therefore, the paper deals with the stress analysis of connecting rod by Finite Element Method using CATIA V5 and ANSYS Workbench software. The objective of present work is to optimize the material of connecting rod to reduce the weight of connecting rod. The material used is aluminium fly ash silicon composite that is manufactured using industrial waste such as fly ash.

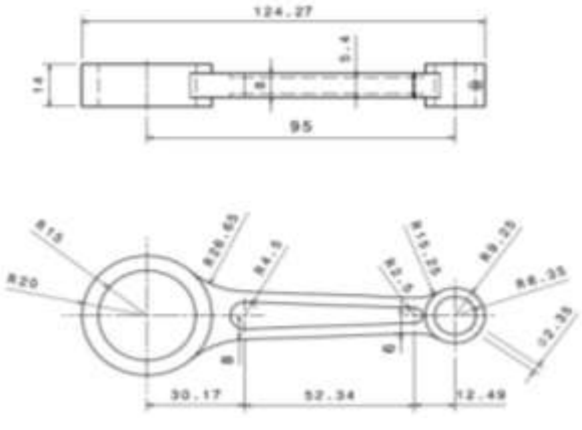

Figure 2 Drawing of Connecting Rod

The first step in Pre-Processing is to prepare a CAD model of connecting rod. The model of connecting rod is designed in CATIA V5 software and saved in iges format. The drawing of connecting rod is shown in Fig. 2[6].

The model of connecting rod is then imported in Solid Works software and saved in Parasolid (x_t) format file for no data loss. This Parasolid file of connecting rod is then imported in ANSYS workbench. In the present work, material is optimized for reduction of weight. The material of connecting rod used for the analysis is aluminium fly ash silicon composite that is manufactured using industrial waste such as fly ash. The deformation and stress contours have find out using ANSYS workbench. The results obtained are compared with available results in literature survey.

Figure 3 CAD model of Connecting Rod in CATIA V5

The Figures 3 and Figure 4 show the CAD model of connecting rod in CATIA V5 Software and ANSYS workbench respectively.

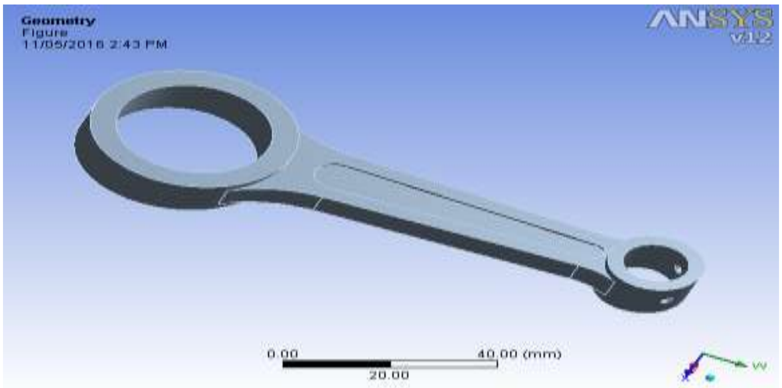

Figure 4 CAD model of Connecting Rod in ANSYS

Mesh generation Second step is to generate mesh using parabolic tetrahedral elements. An automatic method is used to generate the mesh in the present work. Fig 5 shows the meshed model of Connecting Rod in AYSYS workbench.

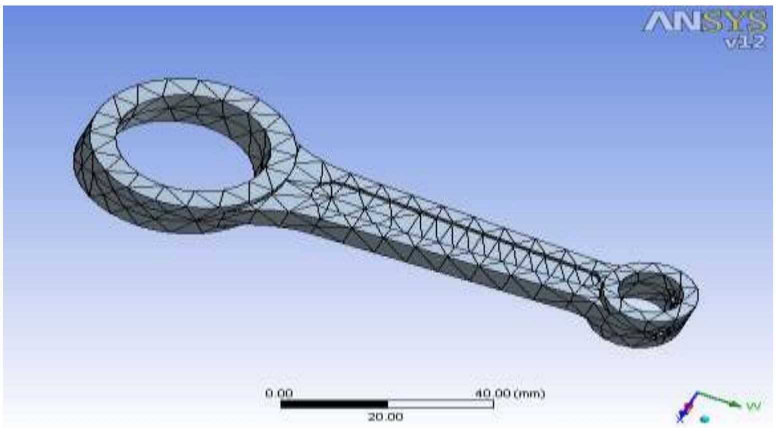

Figure 5 Meshed model of Connecting Rod in ANSYS workbench

Boundary Conditions:

After completion of the Finite Element Model, boundary condition and loads are applied. User can define constraints and loads in various ways. This helps the user to keep track of load cases. The boundary condition is the collection of different forces, supports, constraints and any other condition required for complete analysis. Applying boundary condition is one of the most typical processes of analysis. A special care is required while assigning loads and constraints to the elements. The load applied at the piston end and cylindrical support was given at the crank end. Two cases are 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. The analysis carried out under axial and buckling loads. Here the tensile or compressive load was equal to 4319N and buckling load is equal to 21598N. A static Load of 4319 N is applied on the connecting rod at the small end; and cylindrical support is given at the crank end as shown in Fig 6 [6].

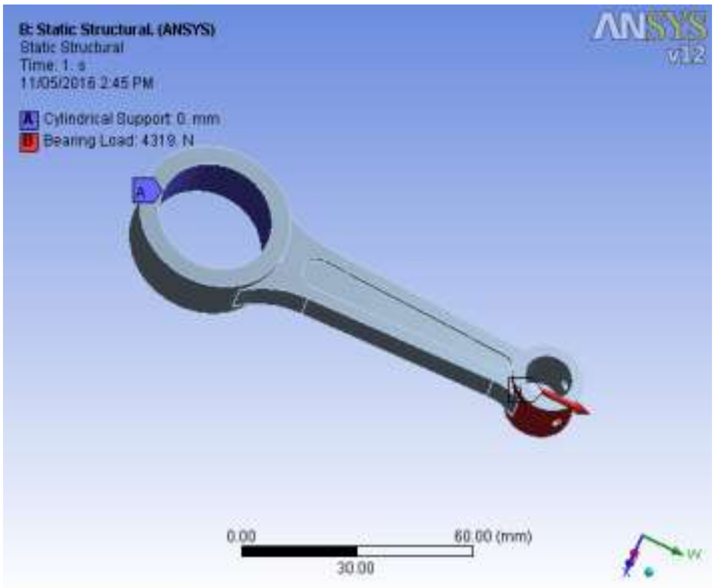

Figure 6 Applied constraints on Connecting Rod

IV. STATIC STRUCTURAL ANALYSIS

Static structural analysis is done in ANSYS workbench to find out the equivalent (von-mises) stress, shear stress, elastic strain and total deformation. Fig. 7 and Fig. 8, show equivalent (von-mises) stress and total deformation respectively in connecting rod.

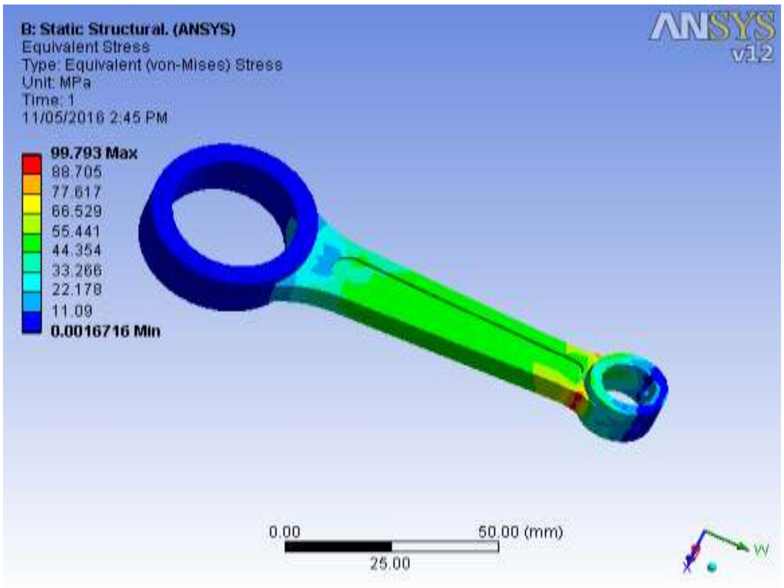

Figure 7 Equivalent (von-mises) stress

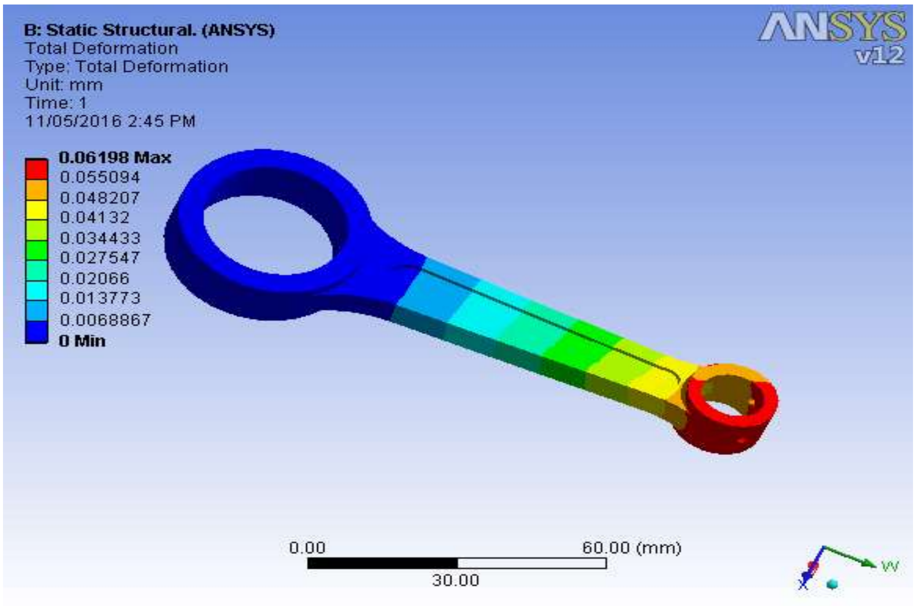

Figure 8 Total Deformation

The analysis results of connecting rod after optimization of the existing model are shown in table 1.

Table 1 FEA results of Connecting Rod

Sr. No.	Parameters	Existing Results	FEA Results
1.	Equivalent (von-mises) stress	71.20 MPa	99.79 MPa
2.	Shear stress	13.04 MPa	38.92 MPa
3.	Equivalent elastic strain	3.56 e-4	1.42e-3
4.	Total deformation	-	0.061mm
5.	Mass	0.126kg	0.0579

From the above comparison Table it has been observed that for the same static load and boundary conditions, the deflection in the connecting rod 0.061 mm. Also the value of von-mises stress is 99.79 MPa which is less than the ultimate tensile strength that shows our design is safe. At the same time a mass also reduces from the 0.126 kg to 0.0579 kg.

CONCLUSION

Finite Element Analysis of the connecting rod has been done using ANSYS Workbench. From the results obtained from FE Analysis, many discussions have been made. The results obtained are well in agreement with the available existing results. The model presented here, is well safe and under permissible limit of stresses.

1. On the basis of the current work, it is concluded that the proposed material give sufficient improvement in the existing results.
2. The weight of the connecting rod is reduced by 54 %, thereby reducing the cost.
3. The stress is found maximum near the hole and sharp edges.

REFERENCES

[1]. Ambrish Tiwari, Jeetendra Kumar Tiwari, Sharad Kumar Chandrakar "Fatigue Analysis of Connecting Rod Using Finite Element Analysis to Explore Weight and Cost Reduction Opportunities for a Production of Forged Steel Connecting Rod", International Journal of Advanced Mechanical Engineering, Volume 4, Nov 2014.

[2]. Kuldeep B, Arun L.R, Mohammed Faheem "Analysis and optimization of connecting rod using alfasic composites" International Journal of Innovative Research in Science, Engineering and Technology , Vol. 2, Issue 6, June 2013
[3]. Leela Krishna Vegi, Venu Gopal Vegi "Design And Analysis of Connecting Rod Using Forged steel", International Journal of Scientific & Engineering Research, Volume 4, Issue 6, June-2013
[4]. Pushpendra Kumar Sharma1, Borse Rajendra R, "Fatigue analysis and optimisation of connecting rod using finite element analysis", International Journal Of advance research in Science and Engineering, Vol. No.1, Issue No. I, pp-3367-3371, September 2012.
[5]. Raviraj Yashwant Taware, Abhay Arun Utpat, "Analysis of Connecting Rod Used in Two Wheeler under Static Loading by FEA", International Journal of Engineering Trends and Technology (IJETT), Volume 20, Feb 2015.
[6]. Suraj Pal and Sunil Kumar, "Design Evaluation and Optimization of Connecting Rod Parameters Using FEM", International Journal of Engineering and Management Research, Vol.-2, Issue-6, December 2012.