Analysis of Connecting Rod

[1]Dr. S. G. Jadhav, [2*]Sanket Dhabale
[1]Assistant Professor, [2]Student*
Department of Mechanical Engineering, VJTI Matunga Mumbai Maharashtra

[*]Corresponding Author
E-mail I.D:- *snktdhbl@gmail.com*

ABSTRACT

The internal combustion engine (ICE) consists connecting rod which is connected to piston and crankshaft to transfer power generated during working of engine from piston to crankshaft, which further is connected to gear box and simultaneously transferred to wheels as required for making the car move. This connecting rod is considered an important part of engine so as to transfer power which in case needs to be durable and efficient for a long time without effect the performance of engine in most cases. As the influence of stresses is more in connecting rod the connecting rod needs to be stable and perform efficiently under these stresses so to avoid any loss of power and obtain maximum performance.

The main objective of this paper is to carry FEA, static structural analysis on connecting rod using 2 different materials under these stresses and obtain knowledge of which material is more suitable for manufacturing connecting rod, through simulation as it is more efficient way of testing in depiction of real world scenarios then making prototype and performing physical tests.

Key notations*: Mpa, mm. Keywords: Connecting rod, stress.*

1: INTRODUCTION-

The connecting rod is internal part of engine which acts to transfer of produce during cyclic movement of piston to crankshaft. The connecting rod primarily goes under compressive and tensile stresses. The acts simultaneously with piston and crankshaft during the working of engine in cyclic revolutions. It consists of three major parts of which two are in connect with piston and crankshaft. Namely Crank end (bigger end), Piston end (smaller end) and shrank or body of connecting rod [1]. The connecting rod are mostly made of different materials according to the requirement of that particular engine, materials like steel, aluminium, titanium, etc. are used. But mostly connecting rod are made up of different grades of steel and aluminum as they are durable, long lasting and provide better efficiency accordingly.

In this paper we are going to test the connecting rod made of 2 different materials by FEA simulation using Ansys 18.1 under different compression and tensile stress to find out how the stresses are induced between these two materials and obtain a better material which can endure more stresses so as the reduce the resources, time and, man power to test physically. The basic requirement of a engine is to be light weight and provide high performance as required so has to increase fuel efficiency and achieve better product in automotive industrycompared to the competition in that segment and can be produced in large volumes [2]. Connecting rods are usually manufactured using forging technique, but before making prototype and testing it physically the e-simulation provides a better understanding of result of connecting rod and predict the

outcome based on these results.

For these necessary requirements the connecting rod needs to be rigid as the engine produces forces in revolution cycle by the piston and the piston end is induced by push and pull forces simultaneously and these forces according are induced to rotate crank which is connected through crank end to obtain rotational motion. As these movements induce friction so there is a small pin hole to squirt pressurized lubricating oil to lubricate piston and piston rings to reduce the friction and perform operation smoothly.

The loads applied on the engine depend upon various factors such as capacity of engine, type of engine. In this paper through literature review [3], we are going to use standard loads acting on connecting rod in Static Structural Analysis.

2: METHODOLOGY-

Through the data acquired during research and studies, by performing various test in different stages, the literature review helps use to understand the process and gather necessary data for this paper. To perform these tests the method needs to be followed in perfect manner to optimize the use of data gathered in literature. The stresses compressive and tensile induced during the piston cyclic movements are mostly due to reciprocating mass of the piston, maximum combustion pressure and forces due to inertia of connecting rod [1].

After the design of connecting rod is design using various CAD software in this paper it is design using Solidworks 2019 as shown in Figure 1. The 3-D model of the connecting rod is converted into .STEP file so as to import the 3-D model into the FEA software Ansys. After that the materials which are required to be tested are selected in Engineering data and properties of material are added accordingly or there might be predefined material in Ansys. After that fixing the ends is done.

Meshing – meshing is an important process so as to divide the element into fine elements andnodes to improve result accuracy for performing stress analysis. The materials used in this analysis are steel and Aluminium 7075. The Following steps need to be followed during the Simulation as mentioned below.

- Create a new project file and import the model file which was created using Solidworks in Ansys by converting the Solidworks file .step file.
- Add the material properties in engineering data for here we have selected structural steel and aluminium as they are most widely used materials for forging connecting rod.
- By clicking on model, the workbench file opens in a new tab, select the components inGeometry and select the material in below.
- Meshing- In this analysis we have used 2mm element size meshing for better results. Solve and let the meshing be completed [3].
- In static structural drop box select fix support. In this we have to select the components where we want to keep fixed support so that we can apply pressure on other ends as required.
- In same drop box select pressure and by selecting the component where we want to givepressure. Give the pressure according to co-ordinate system in the place and input the pressure value respective to co-ordinate in negative or positive value to change the direction of pressure.
- Right click on Solution and select

solve. The software will work out on given data and the processed result will be displayed along with required images

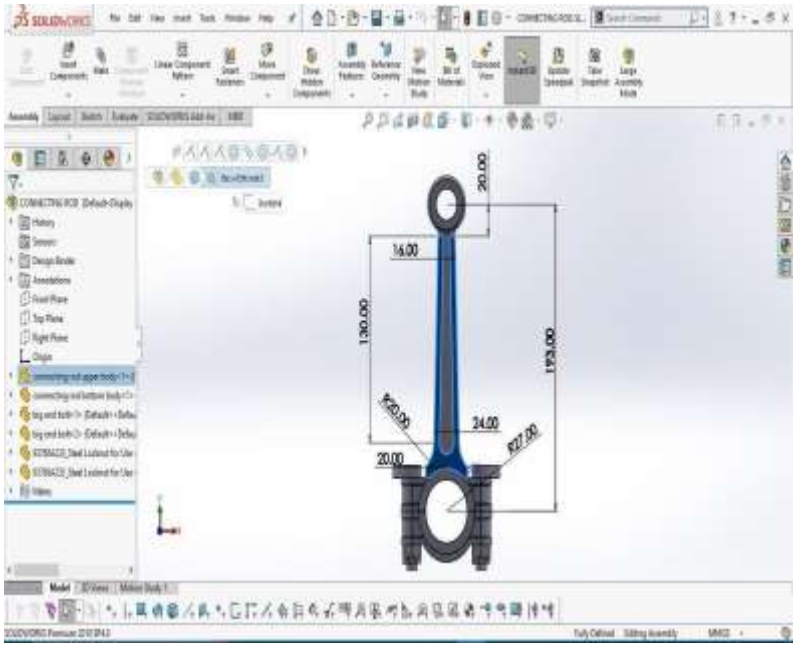


Fig. 1*. Designed model of connecting rod.*

3: Loads on connecting rod [Figure 2]-

Following load conditions are used to perform analysis.

a) Compression at crank end - 37.66 Mpa.

b) Compression at piston end – 69.98 Mpa.

c) Tension at crank end – 41.5 Mpa.

d)Tension at piston end –

77.17 Mpa.Example-

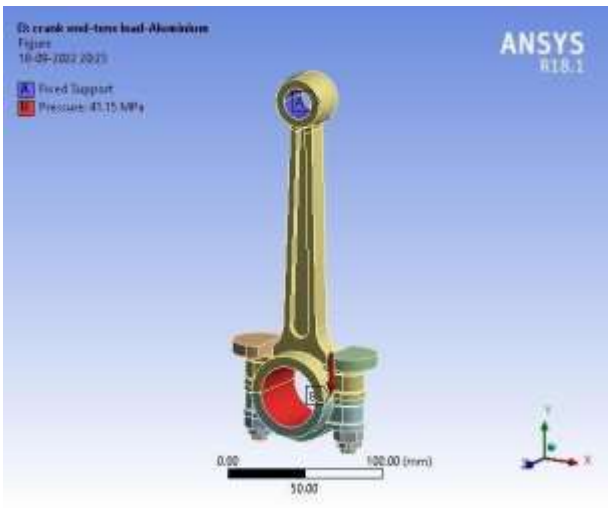


Fig. 2. *Tension load on Crank end.*

Here ‘A’ is kept as fixed support while applying tensile load on the Crank end.

4: Properties of Material-Aluminium-

Compressive Yield Strength MPa	280
Tensile Yield Strength MPa	430
Tensile Ultimate Strength MPa	510

Steel-

Compressive Yield Strength MPa	250
Tensile Yield Strength MPa	250
Tensile Ultimate Strength MPa	460

Connecting rod is made of various materials according to the decades of research and advancement in creating new alloys and finding the more suitable alloy which can be manufactured on the large scale is considered best for making connecting rod as shown in Figure 3.

So has to find more suitable material various research are need to be conducted on different materials and there capacity needs to be analysed and upgrade the connecting rod through this research [4]. In this paper as the mentioned properties of Aluminium 7075 and steel are considered as they are of light weight, rigid and can be manufactured on the large scale.

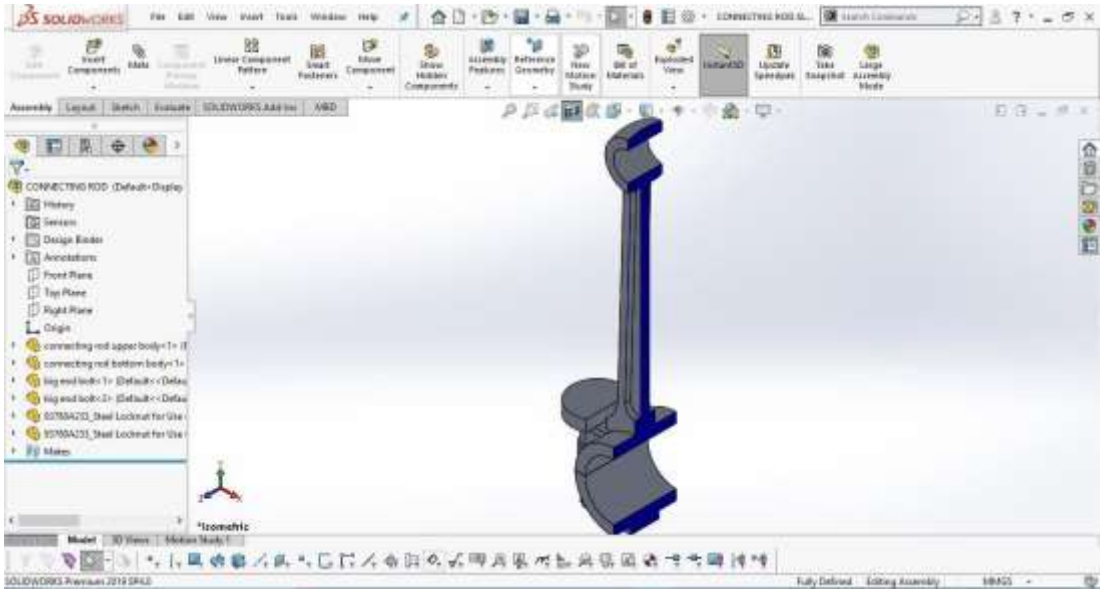
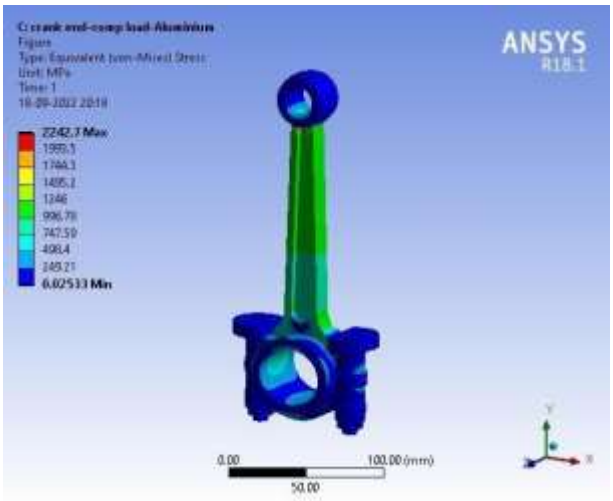


Fig 3. *Sectional view of connecting rod.*

5: Results -

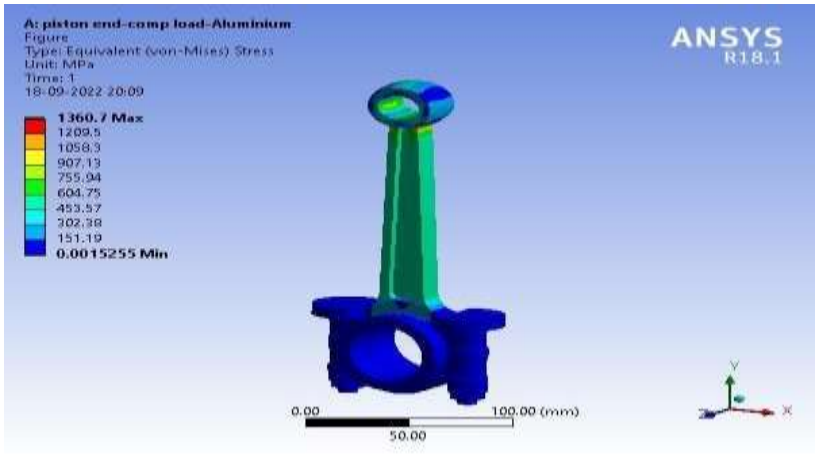
1: **For Aluminium**

a) Compression at crank end



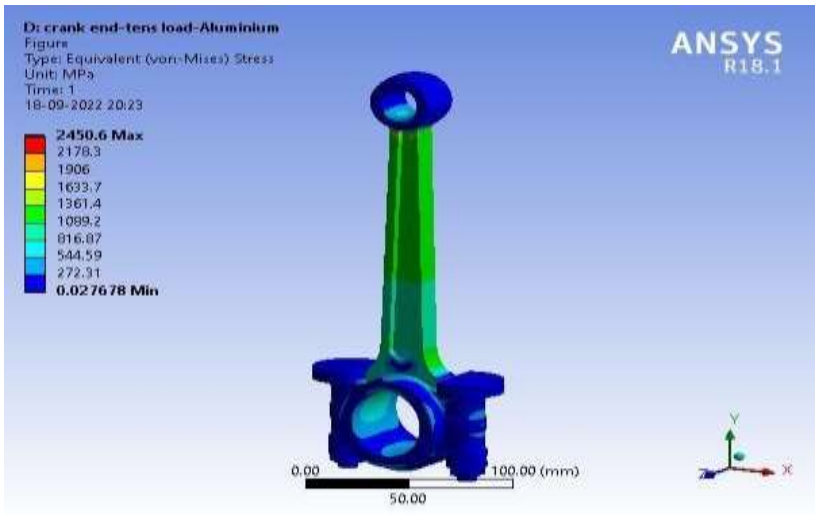
The maximum Stress induced is 2242.7 Mpa

b) Compression at piston end



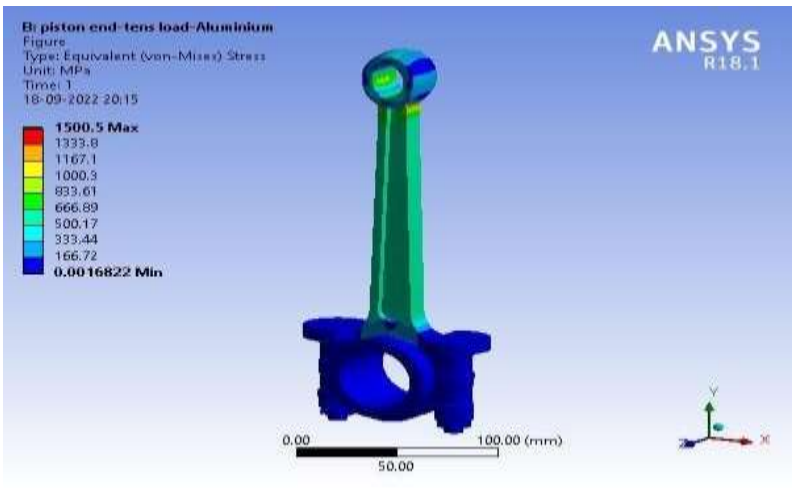
The maximum Stress induced is 1360.7 Mpa

c) Tension at Crank end



The Maximum stress induced is 2450.6 Mpa.

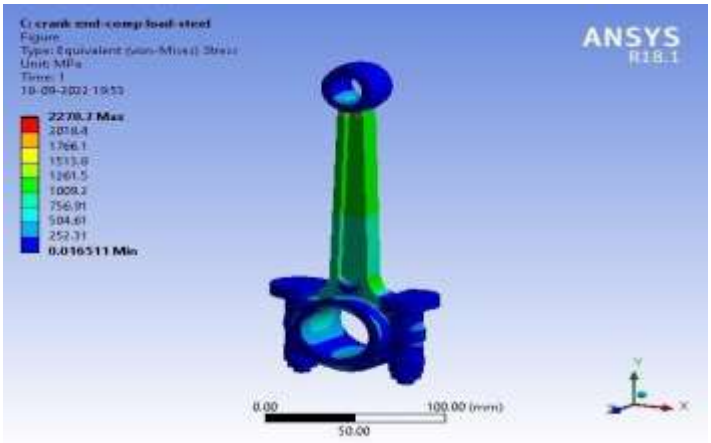
d) Tension at piston end



The Maximum Stress induced is 1500.5 Mpa.

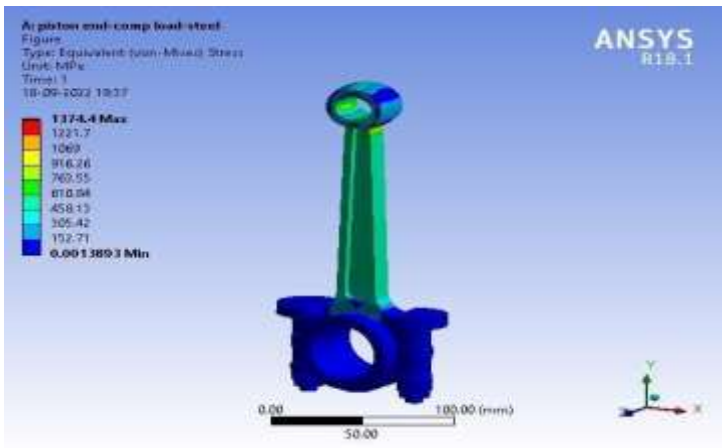
2: For Steel

a) Compression at crank end.

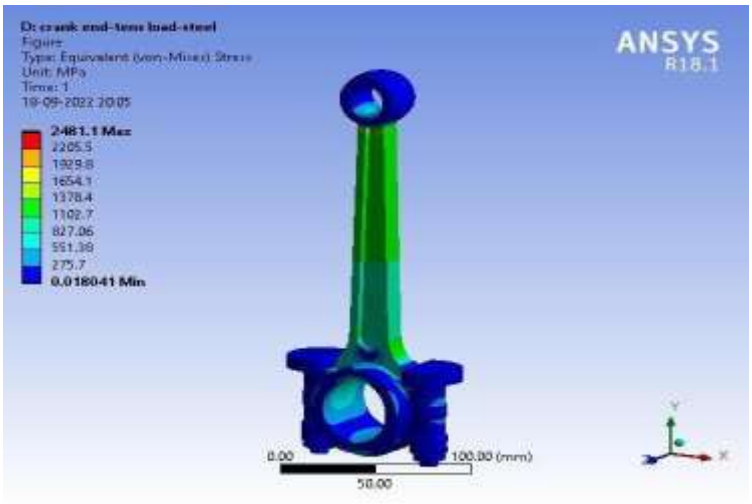


The Maximum Stress induced is 2270.7 Mpa.

b) Compression at piston end.

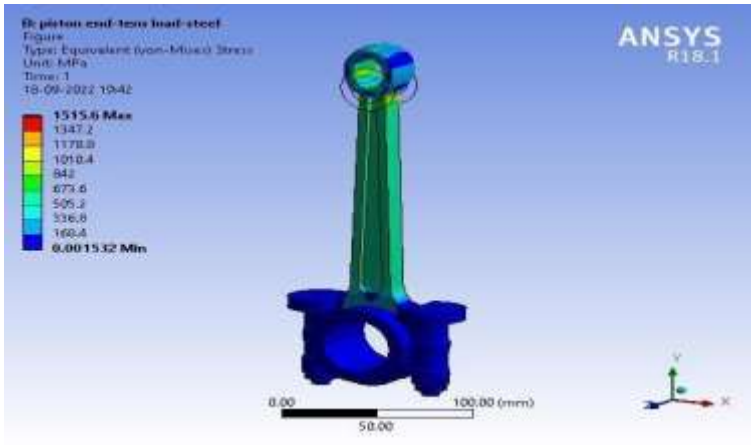


c) Tension at crank end.



The Maximum Stress induced is 2481.1 Mpa.

d) Tension at piston end.



Loading Conditions	Aluminium (Mpa)	Steel (Mpa)
Compression at Crank end	**2242.7**	**2270.7**
Compression at Piston end	**1360.7**	**1374.4**
Tension at Crank end	**2450.6**	**2481.1**
Tension at piston end	**1500.5**	**1515.6**

6: CONCLUSION

As the simulation is done through proper method and by using available data according to the methodology mentioned. There are differences in stresses induced on the connecting rod varying from material to material. In this paper it is noticed that there is higher intensity of stresses induced in Steel as compared to Aluminium [5]. Therefore Aluminium 7075 provides better result and can be used for manufacturing into prototype compared to steel in this paper. As Aluminium is light weight, and is more

durable it improves the efficiency of the engine and helps in reducing the weight of engine further more [6]. Furthermore various alloys of Aluminium can be researched and tested for more better performance and can be analysed, tested according to the given method in this paper andupgraded to the needs of engine.

7: REFERENCES

1. Design and Structural Analysis of Connecting Rod Adnan Ali Haider1, Akash Kumar2, Ajinkya Chowdhury3, Moin Khan4, P. Suresh5 1,2,3,4,5 School of Mechanical Engineering. Galgotias University, Greater Noida
2. Bhandari, V. B., 1994, "Design of Machine Elements," Tata McGraw-Hill. Clark, J. P., Field III, F. R., and Nallicheri, N. V., 1989, "Engine state-of-the-art a competitive assessment of steel, cost estimates and performance analysis," Research Report BR 89-1, Automotive Applications Committee, American Iron and Steel Institute.
3. G. Naga Malleswara rao Design Optimization and Analysis of a Connecting Rod using ANSYS International Journal of Science and Research (IJSR),july-2013
4. Design and Analysis of a Connecting Rod. Sebastian Antony, Arjun A., Shinos T. K B.Tech. Mechanical Dept. Muthoot Institute of Technology and Science Ernakulam, India. International Journal of Engineering Research & Technology (IJERT) http://www.ijert.org ISSN: 2278-0181 IJERTV5IS100142 Vol. 5 Issue 10, October-2016.
5. B. Anusha, C. Vijaya Bhaskar Reddy, "Modelling and Analysis of Two Wheeler Connecting Rod by Using Ansys", IOSR-JMCE, Volume 6, Issue 5, e-ISSN: 2278- 1684, May 2013.
6. Vikas Gupta, Om Prakash, Vinod Mittal "Optimizing the Design of Connecting rod under Static and Fatigue Loading" International journal of Research in Management, Science & Technology (IJRMST) Vol. 1, issue. 1, June 2013.