A CASE STUDY FOR IMPROVEMENT IN PERFORMANCE OF CONNECTING ROD USING FINITE ELEMENT ANALYSIS

[1]Akshay Nighot, [2] Kiran Patil, [3]Nilesh Chandgude, [4]Govind Badgujar.

[1,2,3,4]Assisant Professor

[1-4] Department of Mechanical Engineering,

[1-4] P.G. Moze College of Engineering, Wagholi, Pune, India

Abstract: The connecting rod is the mediating member between the piston and the Crankshaft. Its main function is to convert the reciprocating motion of the piston into rotary motion of the crank. The connecting rod is modeled using CATIA software for both existing solid and modified truss designs. Boundary conditions are applied to the models after finishing the pre – processing work in ANSYS 16.2 software. The best combination of parameters like Von misses stress and strain, Deformation, Factor of safety, fatigue and life cycle calculation, bi-ability indication for two wheeler piston were done in ANSYS 16.2 software. This paper also tends to optimize the design by calculating weight and stiffness for various materialistic designs by using the output values of mass and volume of the connecting rod which will also be obtained from the software. This paper will conclude whether the modified design is safe along above selected materials. And will be presenting the best design for future reference.

***Index Terms* - Connecting Rod, Design-Change, Truss Design, Optimization, Von Misses Stress**

I. INTRODUCTION

Connecting rod is one of the most critical components internal combustion. Connecting rod is connected in between the piston and crank shaft. While the crank shaft rotates piston moves from bottom dead centre to top dead centre vice versa. In this process connecting rod undergoes stress and deformation. Hence for the connecting rod when the load is applied, how the stresses and strain are induced in the component and deformation value, due to applied load is analyzed. Decreasing these stresses and increasing stability depends upon the materials applied. Thus in industrial purpose optimization of connecting rod had already started. Optimization is really important for automotive industry especially. Optimization of the component is to make the less time to produce the product that is stronger, lighter and less total cost productions. The design and weight of the connecting rod influence on car performance. Hence, it effects on the car manufacture credibility. Change in the design and material results a significant incremental in weight and performance of the engine. The structural factors considered for weight reduction during the optimization include the buckling load factor, stresses under the loads, bending stiffness, and axial stiffness. Thus, the component can give the higher strength, efficient design and lighter that would create a major success in the automotive and manufacturing industry. Among the main objectives are to improve the engine performance. Connecting rod failed due to insufficient strength to hold the load. Increasing the strength, automatically it will longer the life cycles of the connecting rod. In this study, the design of the connecting rod will be modeled and at the same time increase the strength. And different materials are applied for gaining more stability. The study will be focus on the finite element modeling and analysis. From the analysis results, the decision whether connecting rod needs to change in material, load, design etc factors which induces stress in the component.

II. LITERATUTE REVIEW

Kuldeep B et al. (2013) has performed analysis and optimization of connecting rod using ALFASiC composites. In their work connecting rod is replaced by aluminium based composite material reinforced with silicon carbide and fly ash. They carried out FEA analysis by considering two materials. The parameter like von misses stress, von misses strain and displacements 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. N.p.doshi et al. (2013) performed analysis of connecting rod using analytical and finite element method. For the project work they have selected connecting rod used in light commercial vehicle of TATA motors had recently been launched in the market. They found out the stresses developed in connecting rod under static loading with different loading conditions of compression and tension at crank end and pin end of connecting rod. Design of connecting rod which is designed by machine design approach is compared with actual production drawing of connecting rod. They found that there is possibility of further reduction in mass of connecting rod. Yoo et al. (2010) used vibrational equations of elasticity, material derivative idea of continuum mechanics and an ad joint variable technique to calculate shape design sensitivities of stress. The results were used in an iterative optimization algorithm, steepest descent algorithm, to numerically solve an optimal design problem. The focus was on shape design sensitivity analysis with application to the example of a connecting rod. The stress constraints were imposed on principal stresses of inertia and firing loads. But fatigue strength was not addressed. The other constraint was the one on thickness to bound it away from zero. They could obtain 20% weight reduction in the neck region of the connecting rod. Folgar et al. (1987) developed a fibre fp/metal matrix composite connecting rod with the aid of FEA, and loads obtained from kinematic analysis. Fatigue was not addressed at the design stage. However, prototypes were fatigue tested. The investigators identified design loads in terms of maximum engine speed, and loads at the crank and piston pin ends. They performed static tests in which the crank ends and the piston pin end failed at different loads. Clearly, the two ends were designed to withstand different loads.

III. MODEL FOR CASE STUDY

For this paper we have consider a case study of connecting rod having parameters as follows:

| Parameter Name	Dimension
Flange and web thickness (t)	7 mm
Width of the section (B)	28 mm
Depth of the section (H)	35 mm
I C engine speed	1800 r.p.m
Max Pressure	3.15 N/mm^2
Piston dia.	100 mm
Mass of the reciprocating parts per cylinder	2.25 kg
Length of connecting rod	380 mm
Stroke of piston	190 mm
Compression ration	6:1
Factor of safety	6
L/D for big end bearing	1.3
L/D for small end bearing	2
Big end Bearing Pressure	10 N/mm^2
Small end Bearing Pressure	15 N/mm^2
Density of connecting rod material	8000 kg/m^3
Allowable stresses in bolts	60 N/mm^2
Allowable stresses in cap	80 N/mm^2
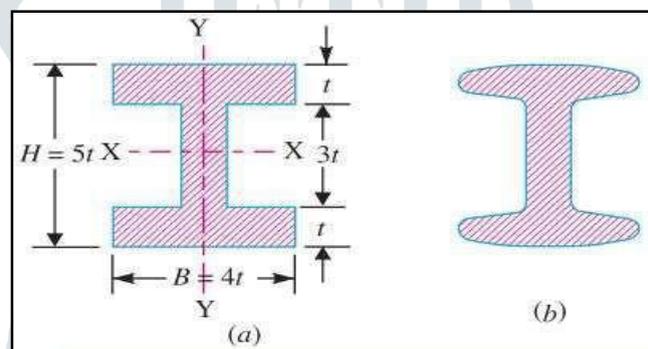
Fig 1. Cross section of connecting rod

Fig 2:CAD Model of connecting rod

The connecting rods are most usually made of steel for production engines, but can be made of Aluminium (for lightness and the ability to absorb high impact at the expense of durability) or titanium (for a combination of strength and lightness at the expense of affordability) for high performance engines, or of cast iron for applications such as motor scooters. The objective of the present study is to design and analysis of two wheeler connecting rod and to find the best alternative material of connecting rod. In the present study beryllium alloys and magnesium alloys have taken in place of currently using materials like Aluminium 360 for CAE analysis and a meaningful comparison made among AL360, Beryllium alloy and Magnesium alloy for choosing the alternative of existing material using for manufacture the connecting rod of single cylinder 4 stroke combustion engines. In this work, an analysis is done for Aluminium alloy, magnesium alloy and beryllium alloy. Beryllium alloys feature high fatigue strength and resistance to wear, corrosion, galling, and stress relaxation.

IV. PROPERTIES OF MATERIAL

Table 1- Properties of material

Properties	Aluminium Alloy	Titanium Alloy	Beryllium Alloy
Tensile strength (MPa)	317	862	1280
Yield strength (MPa)	170	970	965
Poisson's ratio	0.334	0.31	0.33
Modulus of rigidity (GPa)	69	40	50
Young's modulus (GPa)	120	110	125

V. ANALYSIS ON ANSYS WORKBENCH

Analysis software was used to check design feasibility of the design almost in all aspect. ANSYS software is made to be user-friendly and simplified as much as possible with lots of interface options to keep the user as much as possible from the hectic side of programming and debugging process.

Analysis of Connecting Rod of Aluminium 360-
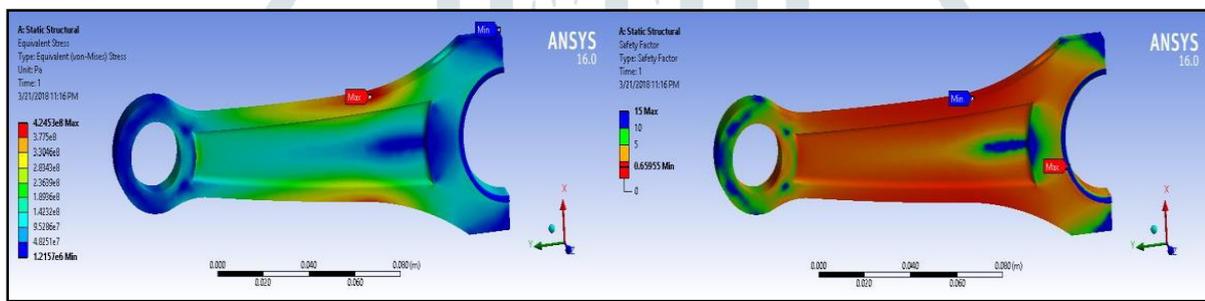
Fig 3 - Equivalent Stress Analysis   Fig 4 -Equivalent Safety factor Analysis
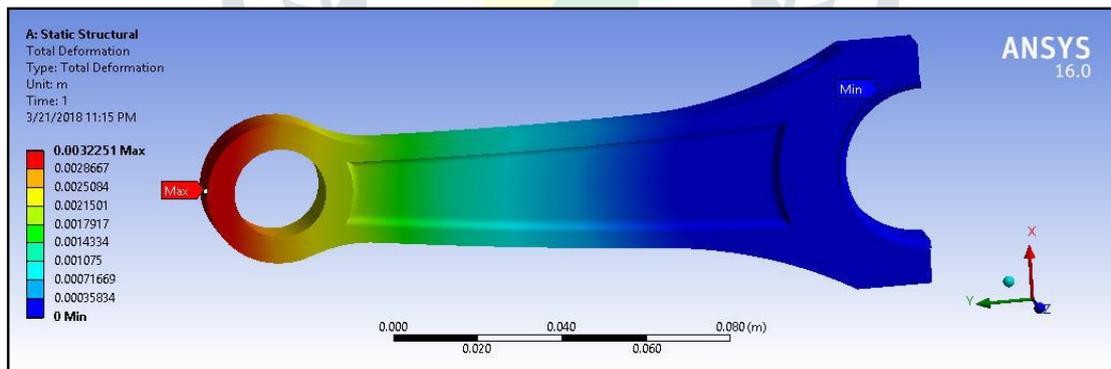
Fig 5 - Equivalent Deformation Analysis

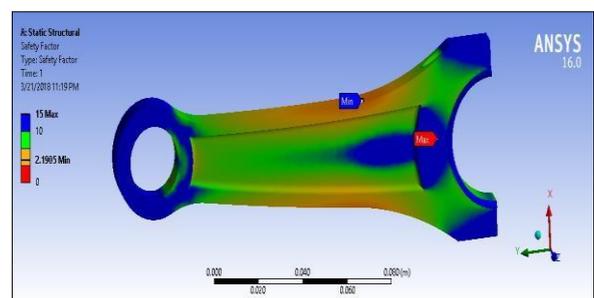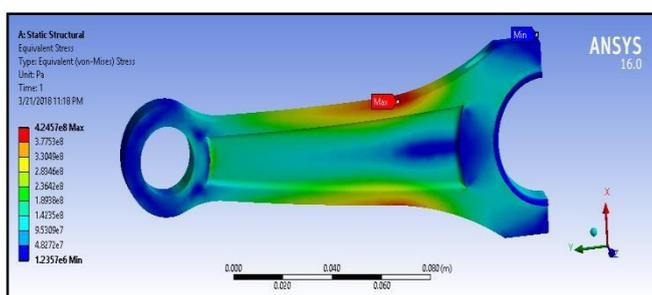**Analysis of Connecting Rod for Ti-13V-11Cr-3Al.**

Fig 6-Equivalent Stress Analysis　　　　　Fig 7-Equivalent safety factor Analysis
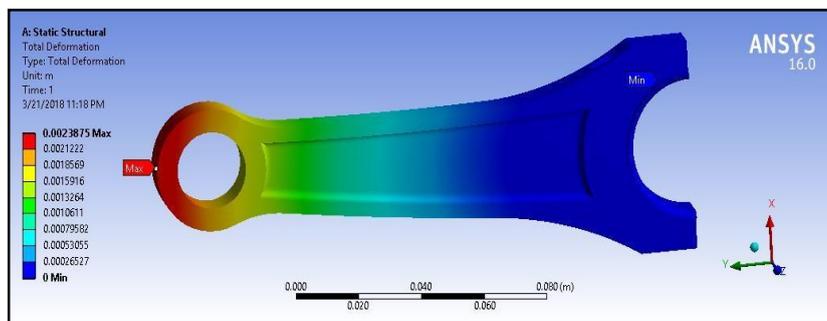

Fig 8-Equivalent Deformation Analysis
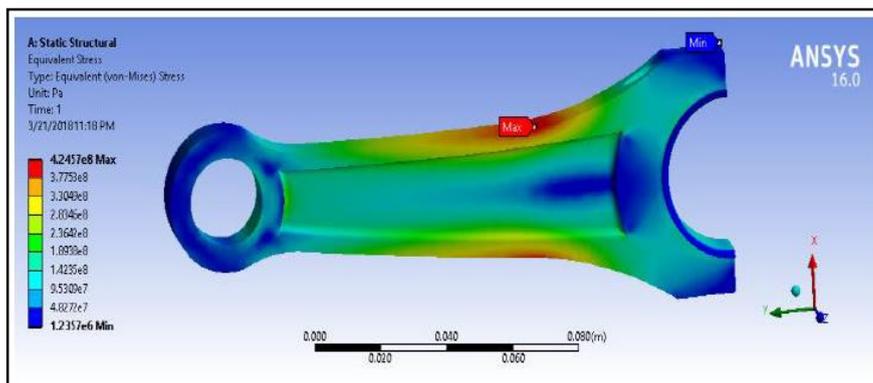
Analysis of Connecting Rod for Ti-13V-11Cr-3Al
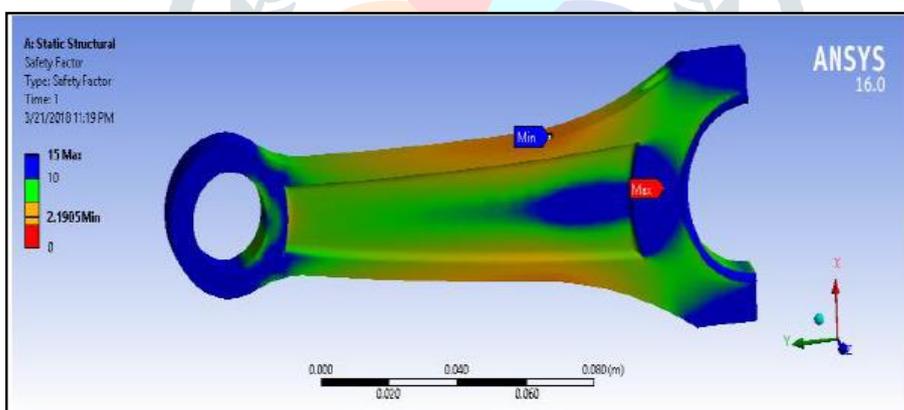

Fig 9 – Equivalent stress analysis

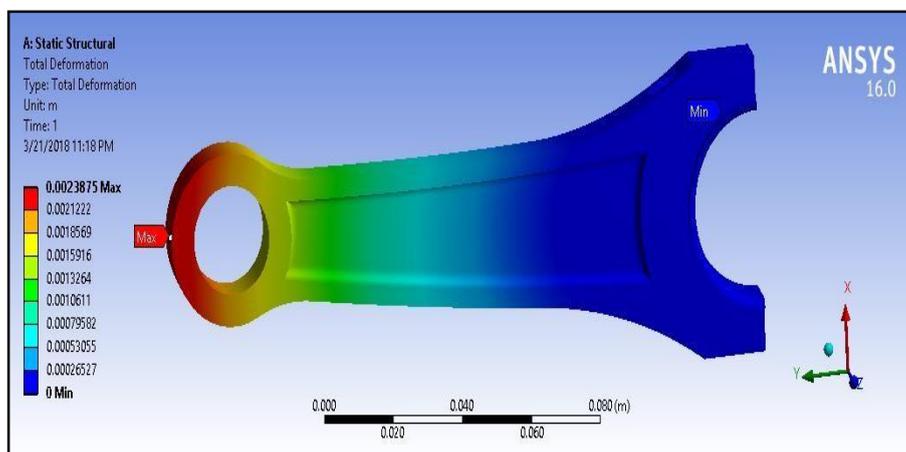Fig 10 - Equivalent safety factor Analysis

Fig 11 - Equivalent Deformation Analysis

VI. RESULTS AND COMPARISON

Following table shows results obtained for different materials.

Table 2 Results of stress, strain and deformation of connecting rod for different materials

Sr. No.	Material Name	Stress		Strain		Deformation	
		Min.	Max.	Min.	Max.	Min.	Max.
1	Aluminium 360	1.33×10^6	2.96×10^8	1.9632×10^{-5}	0.0044616	0.0	0.00033186
2	Beryllium	1.334×10^6	2.95578×10^8	1.1151×10^{-5}	0.0025342	0.0	0.00018849
3	Ti-13V-11Cr-3Al	1.4385×10^6	2.9774×10^8	1.2413×10^{-5}	0.002651	0.0	0.00019652

VII.　CONCLUSION

From results it is found that stresses developed in Beryllium material is less as compared to Aluminium 360 and Ti-13V-11Cr-3Al also strain developed in Beryllium is less as compared to Aluminium 360 and Ti-13V-11Cr-3Al. Further it is found that deformation of beryllium material is less as compared to Aluminium 360 and Ti-13V-11Cr-3Al. From results it concludes that beryllium is suitable material for connection rod.

REFERENCES

[1] Kuldeep B, ArunL. R, Mohammed Faheem "Analysis and Optimization Of Connecting Rodusing Alfasic Composites", ISSN: 2319-875, International Journal of Innovative Research in Science, Engineering and Technology,Vol.2,Issue6,June2013

[2] Prof.N.P. Doshi, 1 Prof. N.K. Ingole "Analysis of Connecting Rod Using analytical And finite Element Method " International Journal Of Modern Engineering Research" (IJMER) www.ijmer.comVol.3,Issue.1,Jan-Feb.2013,ISSN:2249-6645.

[3]FOLGAR Analysis of Connecting Rod Using Analytical and Finite Element "International Journal of Modern Engineering Research(IJMER),JanFeb.2013,Vol.3,Issue.1,pp-65-68

[4]RABB"FINITE ELEMENT ANALYSIS OF CONNECTIN GROD USING ANSYS", Proceedings of Third IRF International Conferenceon 8thFebruary2015,Cochin, India,ISBN:978-93-84209-88-9

[5]PAI, "Dynamic analysis of laods and stresses in connecting", Proc. I Mech EVol.220 Part C: J.Mechanical Engineering

[6] SONSINO and ESPER, Stress Analysis of I.C.Engine Connecting Rodby FEM, International Journal of Engineering and Innovative Technology(IJEIT) Volume1,Issue3,March2012

[7] BALASUBRAMANIAM. Vijaya Bhaskar Reddy, Modelingand Analysisof Two Wheeler Connecting Rodby Using Ansys, IOSR Journal of MechanicalandCivilEngineering(IOSR-JMCE)Volume6,Issue5(May.-Jun.2013),PP8387.

[8] SARIHAN and SONG, Design Evaluation and Optimization of Connecting Rod Parameters Using FEM International Journal of Engineering and Management Research,Vol.-2,Issue-6,December2012.

[9] ISHIDA "Structural Analysis of Bush Bearing For Small End Connecting Grod Using "Promechanica" ISSN0975–668X|NOV12TOOCT13|VOLUME–02.2344-02 [10]Webster.w.d,coffellR,and AlfaroD. "Modelling and analysis of two wheeler connecting rod whileusing Ansys software " IOSR Journal mechanical and civil engineering.