INTERNATIONAL JOURNAL OF ADVANCE RESEARCH, IDEAS AND INNOVATIONS IN TECHNOLOGY

OPTIMIZATION OF STEEL CONNECTING ROD BY ALUMINUM CONNECTING ROD USING FINITE ELEMENT ANALYSIS

*Corresponding Author: Sahil Guleria , ✉ sguleria37@gmail.com
Department of mechanical Engineering,
Rayat and Bhara Institute of Engineering and Technology ,
Mohali.

ABSTRACT

The main aim of the project is to analyze various stresses and fatigue parameters acting on connecting rod, Optimize shape and weight. Connecting rod is an intermediate & important engine component which connects piston & crankshaft. It is subjected to multiple compressive & tensile forces. Major consideration in this case is gas force. The high magnitude gas force is responsible for many kinds of failure. These failures need to be prevented & for this purpose analysis was needed to be done. In this project, connecting rod of Tractor is chosen as a model for study. This project considers two cases, first, static load stress analysis of the connecting rod, and second, optimization for weight. In this project analysis is done on four stroke single cylinder petrol engine connecting rod. The model was developed in PRO-E software, saved in IGES format and then imported to ANSYS workbench. Using ANSYS workbench 14 model was analyzed for various stresses by applying suitable boundary conditions & using different modules of ANSYS workbench 14. The Von Misses stresses, shear stresses, elastic strain, total deformation and various fatigue parameters like life, damage, safety factor etc. are analyzed. Here two materials were studied for their performance, viz., Structural Steel & Aluminum (LM6) Alloy. Shape and weight optimization was done for both the materials. Aluminum being light in weight and having more yield strength became the suitable material. The results obtained from the stress analysis were used to modify the design of existing connecting rod, so that better performance i.e. reduced inertia, fatigue life and manufacturability can be obtained under varying load conditions.

KEYWORDS: ANSYS Workbench, Connecting Rod, FEA Optimization, Von mises stress, Stress Analysis

1. INTRODUCTION

In this project FEM software has been used to study the strength and distortion characteristics of connecting rod and perform various stress analysis on it. The automobile engine connecting rod is a high volume production critical component. It connects reciprocating piston to rotating crankshaft and transmits the thrust of piston to the crankshaft. And thus, it converts the linear, reciprocating motion of a piston into the rotary motion of a crankshaft [1]. Every vehicle that uses an internal combustion engine requires at least one connecting rod depending upon the number of cylinders in the engine. Because of limitation of strength of material there are chances of permanent deformation & hence failure in case of high loads. Combustion in I.C [2]. Engine produces very high load which transmits to crankshaft via connecting rod. So connecting rod is susceptible to many stresses including equivalent, shear, etc. also fatigue failure is possible because of frequent alternate loading & change of direction [5].

Forces acting on the connecting rod:

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

2. MODELLING AND ANALYSIS

In this project, the first two forces have been considered. The connecting rod is designed of I-section to provide the highest possible rigidity at the lowest weight. So, firstly a proper Finite Element Model is developed using PRO-E software [6]. Then using Finite Element Analysis software ANSYS, analysis is done to determine the von misses stresses in the existing connecting rod for the given loading conditions [4]. And then, from the results obtained the load for the optimization study was selected. Boundary conditions for connecting rod are chosen such that critical case can be observed and studied. Two cases are analyzed for each case, one with load applied at the crank end and restrained at the piston end, and the other with load applied at the piston end and restrained at the crank end [3]. A pressure of 64.28Mpa applied on the connecting rod at the small end, and cylindrical support is given at the crank end. A pressure of 34.29Mpa is applied on crank end.

2.1 MESHING OF CONNECTING ROD

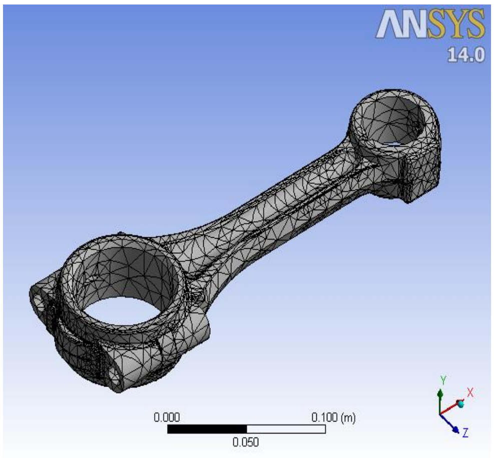

Figure 2.1: Meshed Model of Connecting Rod

2.2 PROPERTIES

Table 2.1: Properties of Structured steel

Material	Structured steel
Density[Kg/m3]	7850
Young's Modulus (Pa)	2.e+011
Tensile Yield Strength (MPa)	250
Compressive Yield Strength MPa)	250
Poisson’s ratio	0.3

Table 2.2: Properties of Aluminum- Lm6

Material	Aluminum- Lm6
Density[Kg/m3]	2650
Young's Modulus (Pa)	7.e+010
Tensile Yield Strength (MPa)	240
Compressive Yield Strength MPa)	230
Poisson's ratio	0.28

3. RESULTS AND DISCUSSIONS

3.1 Figure shows results of structured steel and Aluminium-Lm6 after analysis

- EQUIVALENT STRESS

  Equivalent Stresses are minimum at both the ends and moderate at shank

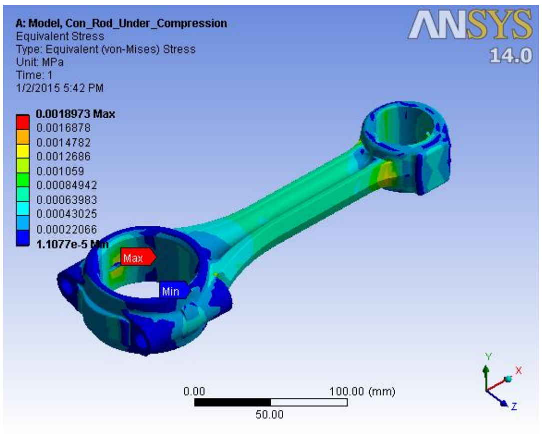

Figure 3.1: (a) Structured steel

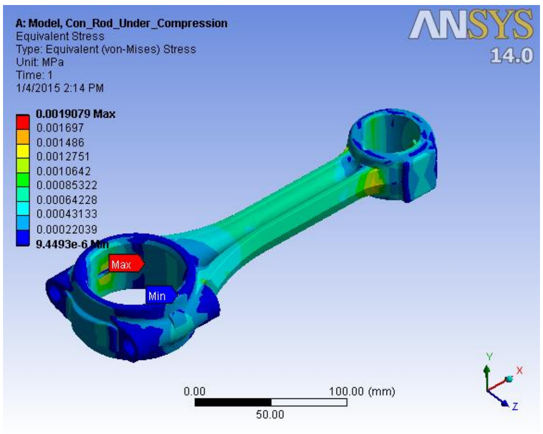

Figure 3.1: (b) Aluminum- Lm6

- MAXIMUM SHEAR STRESS

  Shear stresses are minimum in both cases; Very less region has shear stresses concentrated so Al-lm6 is more superior to structured steel.

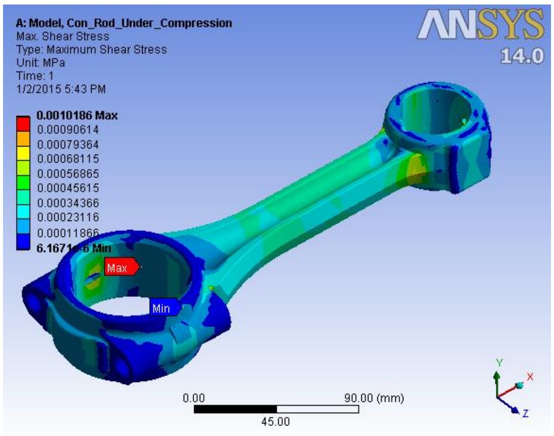

Figure 3.2: (a) Structured steel

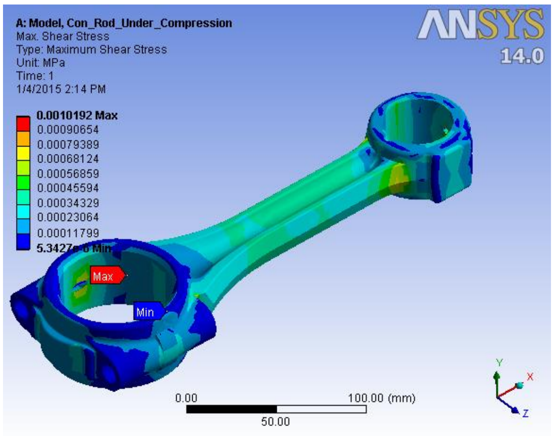

Figure 3.2: (b) Aluminum- Lm6

- EQUIVALENT ELASTIC STRAIN

When an external stress is applied to a body, the body tends to pull itself apart. This induces strain in the body. In our case the induced strain due to stresses is minimum in Al-Lm6.

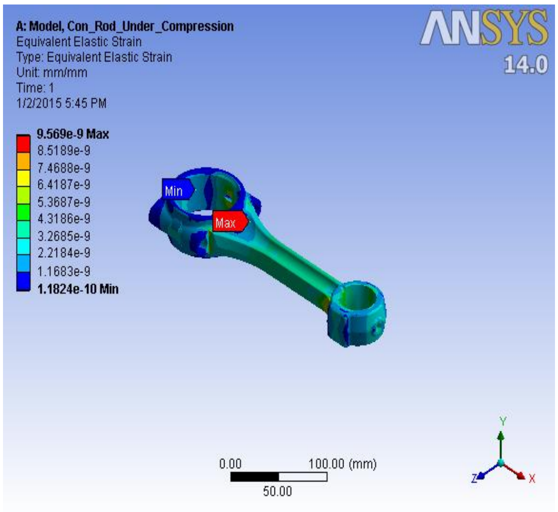

Figure 3.3: (a) Structured steel

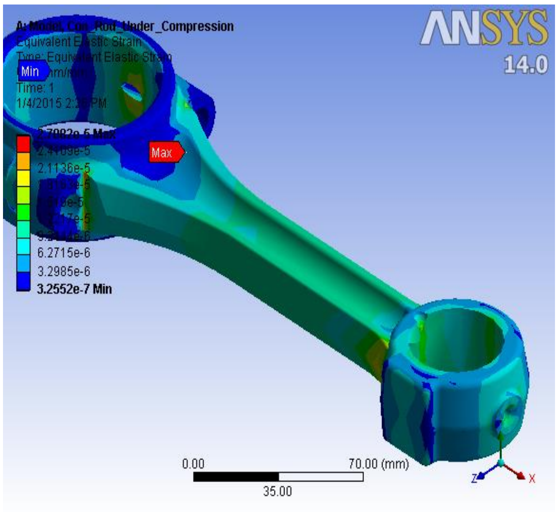

Figure 3.3: (b) Aluminum- Lm6

- MINIMUM PRINCIPAL STRESS

Minimum Principal Stress is found out for the structural steel and Al-lm6 by applying given loading conditions and compared. Overall stresses in Al-Lm6 are less.

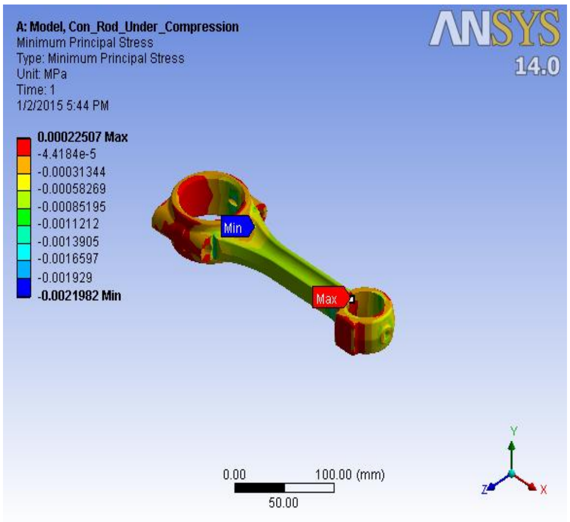

Figure 3.4: (a) Structured steel

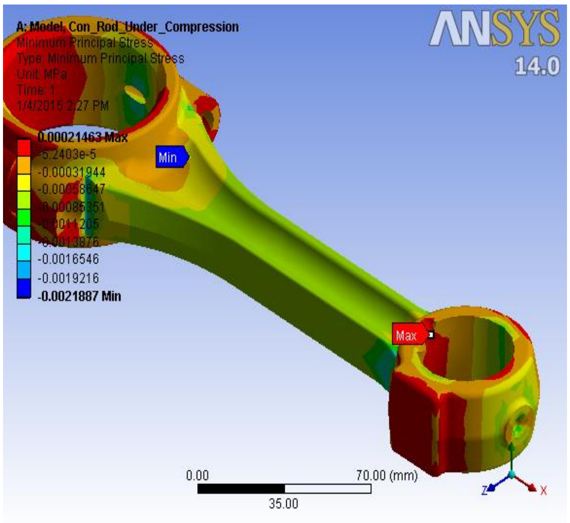

Figure 3.4: (b) Aluminum- Lm6

3.2 FATIGUE ANALYSIS

- Safety Factor

It is measure of factor of safety for design. Safety factor of 15 is recorded at both the ends which is maximum. Factor of safety for material tested for 1.e+006 cycles is less than 15 for all materials

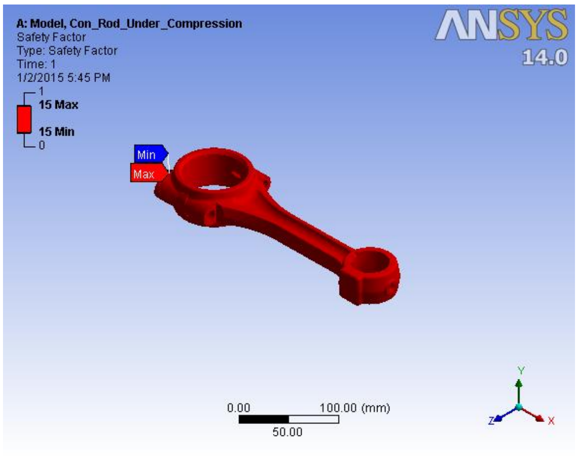

Figure 3.5: (a) Structured steel

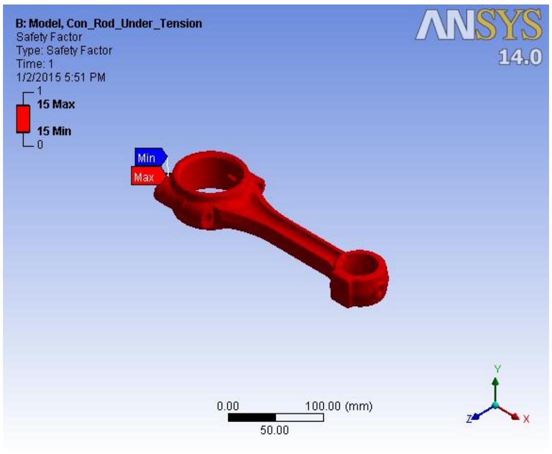

Figure 3.5: (b) Aluminum- Lm6

- BIAXILITY INDICATION

  It is qualitative measure of stress. Three mutually perpendicular principal stresses are less in Al- lm6.

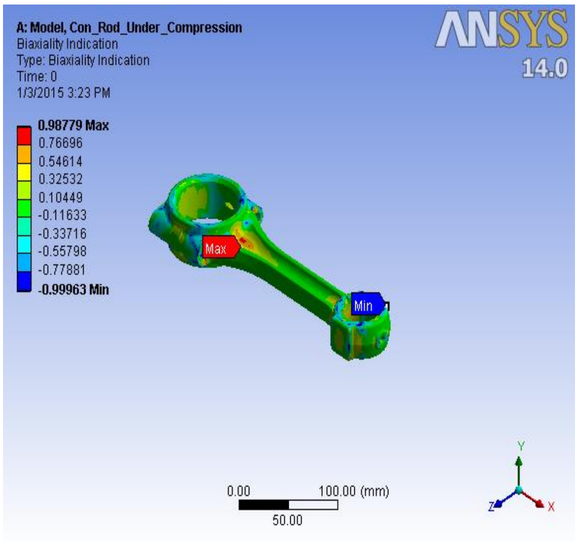

Figure 3.6: (a) Structured steel

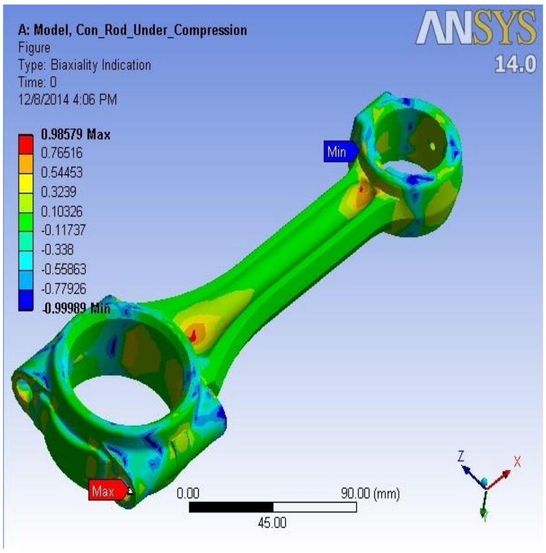

Figure 3.6: (b) Aluminum- Lm6

3.3 RESULTS FOR STATIC LOADING

Table 3.1: Maximum and Minimum Values of Stress Parameters

Parameter	Structural Steel		Aluminum Lm6	
	Min	Max	Min	Max
Equivalent stress(MPa)	1.1077e-005	1.8973e-003	9.4468e-006	1.9079e-003
Maximum Shear stress(MPa)	6.1685e-006	1.0186e-003	5.3405e-006	1.0192e-003
Equivalent elastic strain(mm/mm)	1.1819e-010	9.5685e-009	3.2539e-007	2.7082e-005
Minimum principal stress(MPa)	-2.1982e-003	2.2506e-004	-2.1887e-003	2.1463e-004

3.4 FATIGUE TOOL RESULTS

Table 3.2: Maximum and Minimum Values of Fatigue Parameters

Parameter	**Structural Steel**		**Aluminum Lm6**	
	Min	**Max**	**Min**	**Max**
Safety Factor	15	15	15	15
Biaxility Indication	-0.99948	0.9878	-0.99989	0.98579

4. CONCLUSIONS

- The analysis performed in this project gave scope for optimization. Analysis of different parameters it has Suggested modification in existing connecting rod.
- From the analysis we came to know that composite material can be a best alternate of conventional structural steel connecting rod due to light weight and high strength to weight ratio.
- The stress multiaxiality is high, therefore multiaxial fatigue analysis is needed to determine fatigue strength.
- The maximum stresses occurred in static structural analysis are less than the yield strength of material. Hence the design is safe.

REFERENCES

[1] A Textbook of Machine Design by R.S.KHURMI AND K.GUPTA.

[2] Bhagat A.R. and Jibhakate Y.M. (2012) "Thermal Analysis and Optimization of I.C. Engine Piston Using finite Element Method" *International Journal of Modern Engineering Research (IJMER)*, Vol.2, Issue.4, pp-2919-2921

[3] Charkha G.P. and JajuB. S. (2009) "Analysis & optimization of connecting rod" *Second International Conference on Emerging Trends in Engineering and Technology* ICETET, pp 86-91.

[4] Prof. Doshi P. N. and Prof. Ingole K. N. (2013) "Analysis of Connecting Rod Using Analytical and Finite Element Method" *International Journal of Modern Engineering Research (IJMER),* Vol.3, Issue.1, pp-65-68.

[5] Prof. Sharma P. K. and Borse R. R. (2012) "Fatigue analysis and optimization of connecting rod using Finite Element Analysis" *International Journal Of advance research in Science and Engineering, (IJARSE)*, Vol. No.1, Issue No. I

[6] Om Prakash, Vikas gupta, Vinod Mittal, 'optimizing the design of connecting rod under static and fatigue loading' IJRMST Vol. 1; No. 1, June 2013

[7] Ansys workbench 14.0 *"Reference Manual"*

[8] Pro/E 5.0 "*Reference Manual"*