Design, Analysis and Optimization of Gas/petrol Engine Components

Dr T. Nageswara Rao[1], K. Vasudev Rao[2], V. Ananta Rahul[3], G. Rajesh[4], P. Surya Sowgandh[5]
[1,2,3,4,5] *Department of Mechanical Engineering, GITAM (Deemed to be University), Bengaluru, India*

***Abstract*-The main objective is to optimize gas/petrol engine components. In the initial study, a 3D model of piston, connecting rod, and crankshaft were developed in solid works and Finite element analysis (Structural) was carried out.FEA (Structural) was done on the assembly of engine parts, which was compared with reference model Yamaha FZ-16 for its optimization.**

***Keywords*: Piston, Connecting rod, Crankshaft, Finite element analysis, Temperature distribution.**

I.INTRODUCTION

1.1 Literature review

Jaimin Brahmbhatt et al. [1] analysed dynamic, harmonic analysis variation of stress magnitude at critical locations of crankshaft. The relationship between the frequency and the vibration modal is explained by the modal and harmonic analysis of crankshaft. **R.A Savanoor et al.** [2] performed FEA analysis, which was carried out by considering different materials. The parameters like von misses stress, and displacement were obtained from ANSYS software. **A.R. Bhagat et al.**[3] describes the thermal stress distribution of piston considering real engine condition during combustion process**. Sasi Kiran Prabhala et al.**[4] performed FEA (Modal and static analysis) which was carried out for both steel components as well as for aluminium alloyed component. It is found that weight of the engine got reduced because of aluminium alloy. Fan **Jiangpeng et al.** [5] performed model and static analysis on different sizes of the crankshaft, finally optimization was done. **Pranav G Charkha et al.** [6] performed FEA (static and fatigue analysis) for the connecting rod in ANSYS. Finally the component was optimized for fatigue life. **Zheng Bin Liu Yongqiet al.**[7] analysed stress distribution, safety factor, and fatigue life cycle of connecting rod. Finally structure of connecting rod was improved to increase the Safety factor and fatigue life cycle of connecting rod.

1.2 Reference Model: Yamaha Fz-16

Engine type	AirCooled,4Stroke,2valve,singlecylinder(153 cc) , SOHC
Type	SI Engine
Compression Ratio	9.5:1
Maximum Power	12.81HP@ 9.61KW @8000RPM
Maximum Torque	12.98N-M@6000
Bore	57.3mm
Stroke	57.9mm
Maximum gas pressure	8 MPa

Table no.1

II. CRANKSHAFT

This study was conducted on a single cylinder four-stroke gas/petrol engine. A solid three-dimensional parametric geometry of a single cylinder crankshaft of a four-stroke diesel engine is created using higher-end CAD software, i.e. solid works according to the detailed two-dimensional drawing. This solid geometry was imported in step format for finite element simulation purpose under structural simulation using ANSYS workbench software. The finite element analysis results were verified theoretically and numerically. However, these results were most important as a benchmark for further optimization study of the crankshaft. The optimization was studied by consideration andeffects of geometry and shape on the counter weight and crankpin without affecting mounting on the engine block and cylinder head in the existing design crankshaft. Hence, different feasibility optimization cases were evaluated in the present crankshaft design using computer-aided design and finite element analysis approach.

2.1 Methodology

- To design the crankshaft for a petrol engine.
- To geometrically model the crankshaft as per the dimensions generated from the process of design procedure followed.
- To analyse the equivalent stress which is bending stress.
- To analyse the equivalent stress using FEA approach for study. To plot the results for equivalent stress acting on reference model crankshaft and optimized crankshaft.

2.2 Cases considered for Crankshaft analysis:

*Case1:*The crank pin is at the top dead centre position and subjected to maximum bending moment and no torsion moment.

*Case2:*The crank is at an angle with the line of dead centre positions and subjected to maximum torsional moment.

2.3 Static structural analysis of the reference modeland optimized crankshaft(case1: Bending)

Static structural analysis of the existing crankshaft was done using the finite element analysis approach. The static structural analysis of the existing crankshaft was done using ANSYS workbench to evaluate the different stresses and deformations under static loading conditions. Finite element analysis involves four main steps to solve any physics problem using ANSYS software (http://www.ANSYS.COM).

1. Preliminary decisions:
   a. Analysis type : static structural analysis
   b. CAD data : three-dimensional solid model
   c. Element type : Solid183
2. Pre-processing
   a. Define material: structural steel
   b. Import geometry
3. Solution
   a. Apply load: load and boundary conditions applied as per engine specification
4. Post processing
   a. Total deformation
   b. Von Mises stress

The original crankshaft is optimized by removing the material from the crankpin geometry with the technical assumptions and manufacturing aspects by considered 17 mm drilled diameter and 2nd optimization is done by reducing the thickness of the crank web by 5mm.

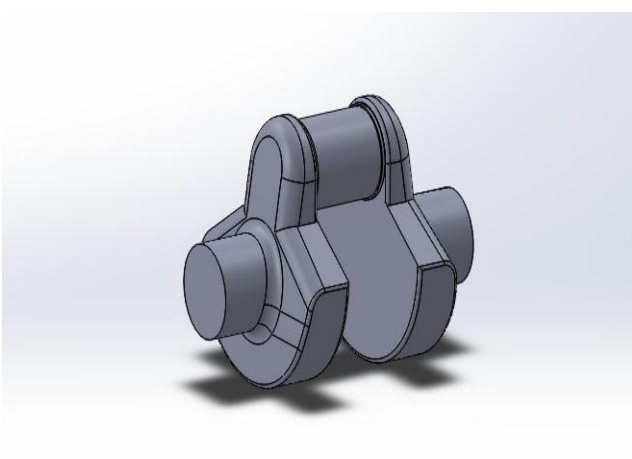

Fig: 1 Reference model Crankshaft

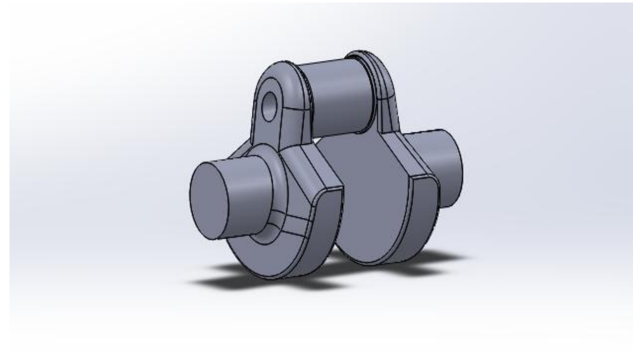

Fig: 2 optimized Crankshaft

Comparison between reference model crankshaft and optimized crankshaft (case 1: Bending)

Parameters	Reference model Crankshaft (Bending)	Optimised Crankshaft (Bending)
Equivalent Stress (N/mm^2)	52.195max	62.133max
Equivalent Strain	0.0002976 max	0.00035842 max
Deformation(mm)	0.0070234 max	0.0077859 max
Safety Factor	4.7897min	4.0236min

Table no.2

2.4 Static structural analysis of the reference model and optimized crankshaft (case 2: Bending and torsion)

FEA (structural) analysis was done on reference and optimized crankshaft considering both bending and torsional load.

Comparison between reference model crankshaft and optimized crankshaft (case 2:Bending and torsion)

Parameters	Reference model Crankshaft (Bending & Torsion)	Optimised Crankshaft (Bending & Torsion)
Equivalent Stress (N/mm^2)	112.89max	141.52max
Equivalent Strain	0.00065054 max	0.00073728 max
Deformation(mm)	0.023319 max	0.025911 max
Safety Factor	2.2146min	1.7665min

Table no.3

2.5 Weight analysis of crankshaft:

Parameters	Reference model crankshaft	Optimized crankshaft
Volume(mm^3)	6.7538×10^5	5.9001×10^5
Density(kg/mm^3)	7.850×10^{-6}	7.850×10^{-6}
Mass(kg)	5.301733	4.63157
Weight(N)	52.0100	45.4357

Table no.4

Percentage reduction in weight= 21.5909%

III.CONNECTING ROD

In this project the conventional material used for manufacturing of connecting that is steel is replaced by aluminium alloy. The lateral bending of connecting rod occurring are evaluated analytically and using FEA approach.

Geometrically modelled connecting rod in Solidworks16.The calculation of equivalent stress due to inertia is first calculated using analytical approach and the equivalent stress are calculated by FEA software named as ANSYS which are compared to provide a platform for evaluation and validation of design.

Methodology used for the design and development of connecting rod is same as crankshaft as mention above.

Defined material: Aluminium alloy

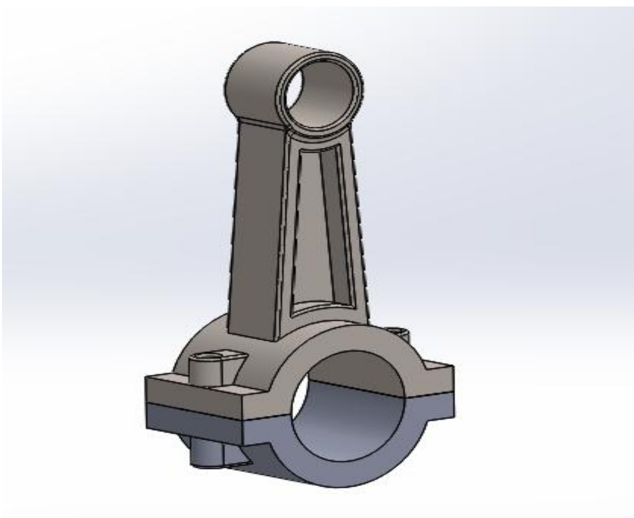

Fig: 3Reference model Connecting rod

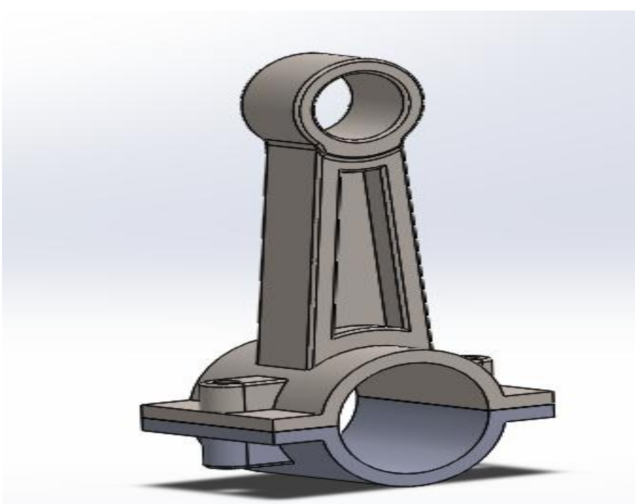

Fig: 4 Optimized Connecting rod

3.1 Static structural analysis of the reference model and optimized connecting rod

Static structural analysis of the existing connecting rod was done using the finite element analysis approach in ANSYS workbench to evaluate the different stresses and deformations under static loading conditions. Defined material: Aluminium alloy

The original connecting rod is optimized by removing the material from the bigend bearing geometry with the technical assumptions and manufacturing aspects by reducing the thickness and 2nd optimization is done by reducing the thickness of the I-section thickness.

Comparison between reference model connecting rod and optimized connecting rod.

Parameters	Reference model Connecting Rod	Optimised Connecting Rod
Equivalent Stress (N/mm^2)	170.09max	165.41max
Equivalent Strain	0.00233max	0.002266max
Deformation(mm)	0.065347max	0.069262max
Safety Factor	1.8814min	1.9346min

Table no.5

3.2 Weight analysis of connecting rod:

Parameters	Reference model connecting rod	Optimized connecting rod
Volume(mm^3)	140057	951161
Density(kg/mm^3)	2.6×10^{-6}	2.6×10^{-6}
Mass(kg)	0.3641	0.2474
Weight(N)	3.572	2.4271

Table no.6

Percentage reduction in weight= 32.05%

IV.PISTON

4.1 Static structural analysis of the Reference model and optimized piston

Static structural analysis for the reference model piston was done using the finite element analysis approach in ANSYS workbench to evaluate the different stresses and deformations under static loading conditions

The reference model piston is optimized by removing the material from the piston skirt with the technical assumptions and manufacturing aspects.

Defined material: Aluminium alloy

Comparison between reference model piston and optimized piston

Parameters	Reference model Piston	Optimised Piston
Equivalent Stress (N/mm^2)	159.17max	179.88max
Equivalent Strain	0.0025632max	0.0024648max
Deformation(mm)	0.03004max	0.018346max
Safety Factor	2.0104min	1.779min

Table no.7

4.2 Thermal analysis of Reference model and optimized Piston

The piston is divided into the areas defined by a series of grooves for sealing rings. The boundary conditions for mechanical simulation were defined as the temperature loads acting on the entire pistonsurface.It is necessary to load certain data on material that refer to thermal properties. The temperature load is applied on different areas. The regions like piston head and piston ring regions are applied with large amount of heat (180°C-270°C). The convection values on the piston wall ranges from 350 W/mK to 600 W/mK.

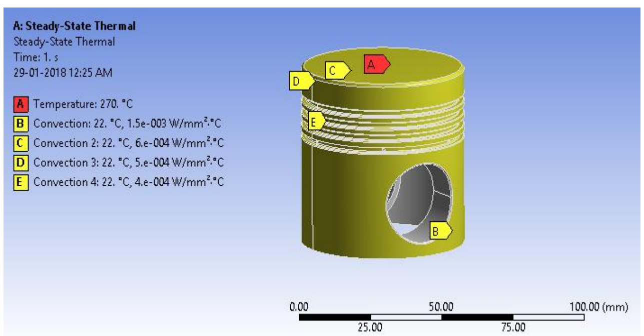

Fig.5 Thermal boundary conditions (Reference model)

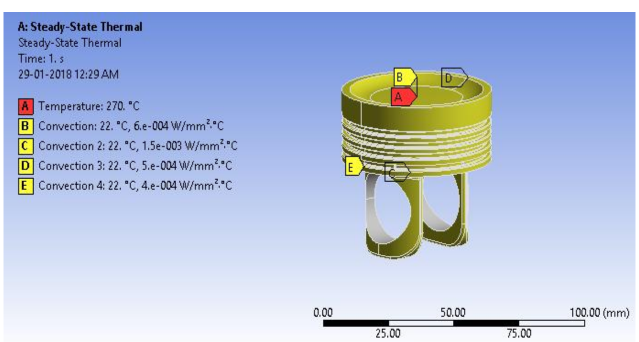

Fig.6 Thermal boundary conditions (Optimized)

Comparison between reference model piston and optimized piston

Parameters	Reference model Piston	Optimized Piston
Temperature Distribution (^0C)	270 - 59.515	270-30.381
Heat Flux (w/mm^2)	1.985 - 5.0246×10^{-5}	3.4856 -0.0016915

Table no.8

4.3 Weight analysis of piston

Parameters	Reference model Piston	Optimized Piston
Volume(mm^3)	61561	50128
Density(kg/mm^3)	2.6×10^{-6}	2.6×10^{-6}
Mass(kg)	0.1600	0.13033
Weight(N)	1.5701	1.2785

Table no.9

Percentage reduction in weight=18.57%

V. ICENGINE ASSEMBLY

Assembly is joining of all the components of different parts together to a single assembled component. The assembly of component is created by using align, mate component for automatic assembly and for mechanism the assembly of components by using different types of joints depending upon their mechanism.

5.1 Static structural analysis of the Reference model assembly

Static structural analysis of the reference model assembly was done using the finite element analysis approach in ANSYS workbench to evaluate the different stresses and deformations under static loading conditions.

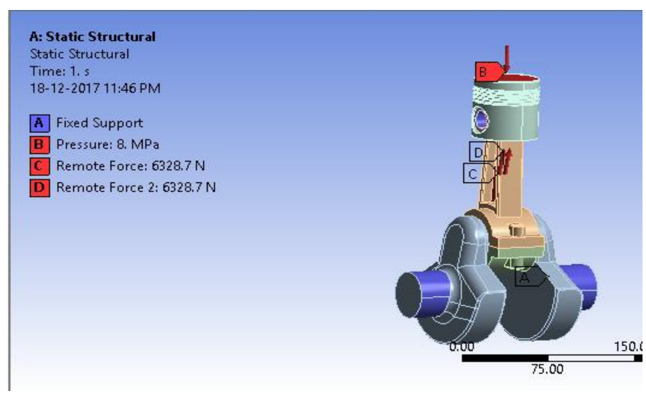

Fig.7 Boundary conditions

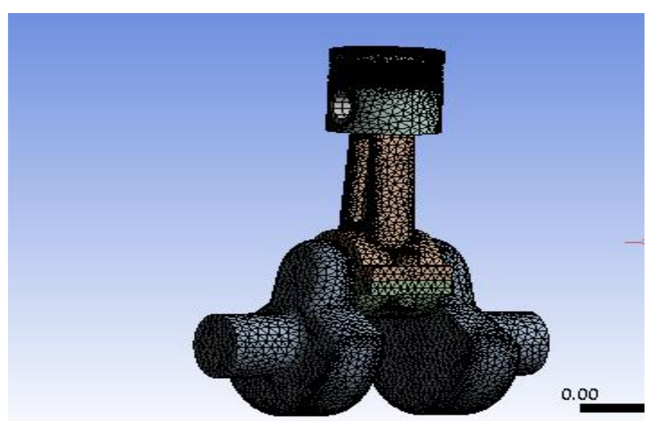

Fig.8 Geometry of mesh

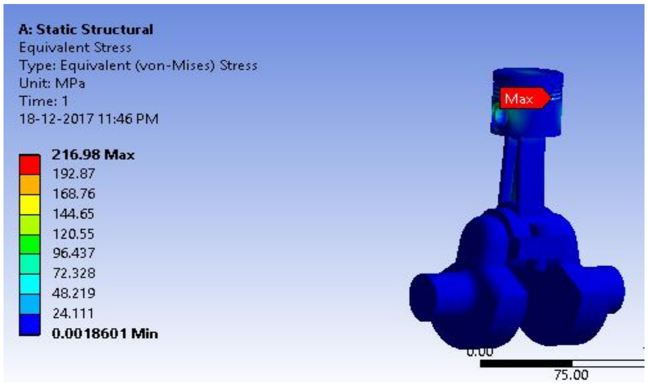

Fig.9 Equivalent stress

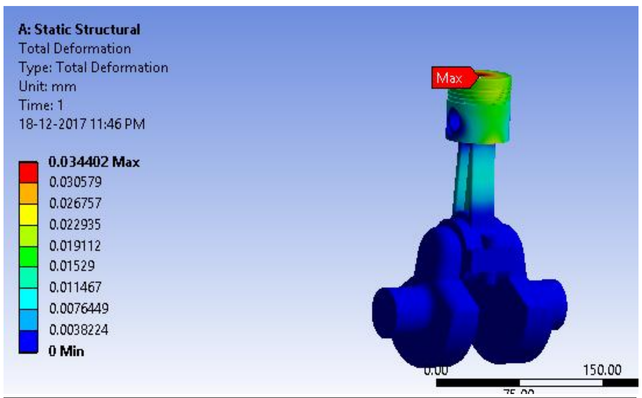

Fig.10 Total deformation

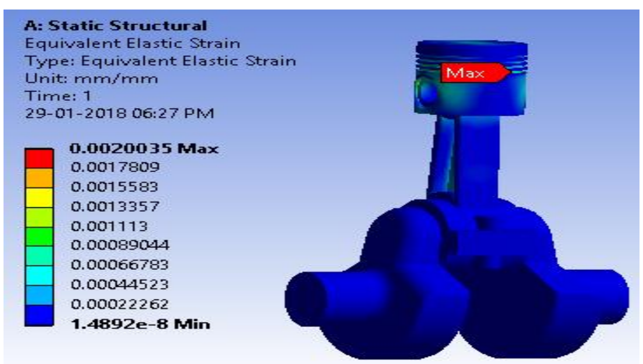

Fig.11 Equivalent elastic strain

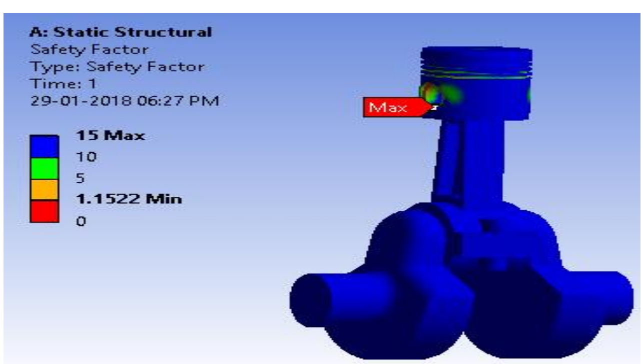

Fig.12 Safety Factor

5.2 Static structural analysis of the Optimized assembly

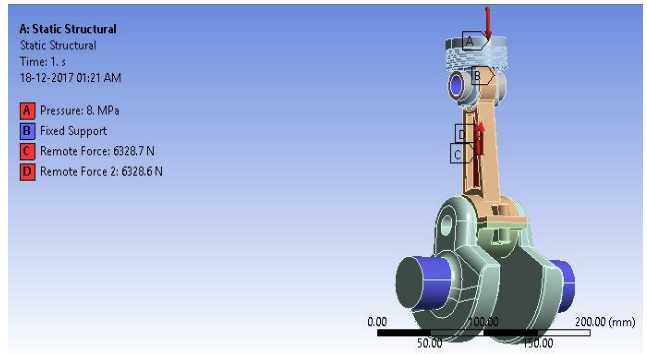

Fig.13 Boundary conditions

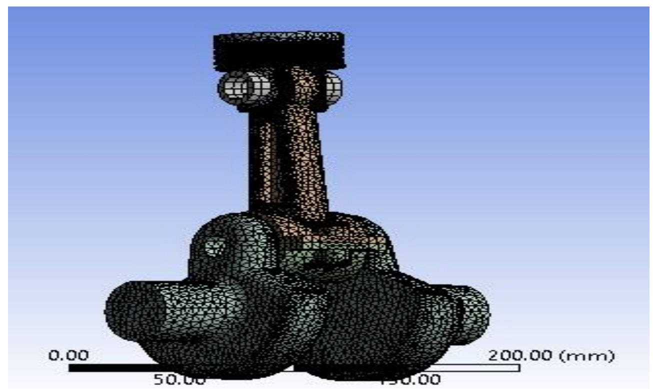

Fig.14 Geometry of mesh

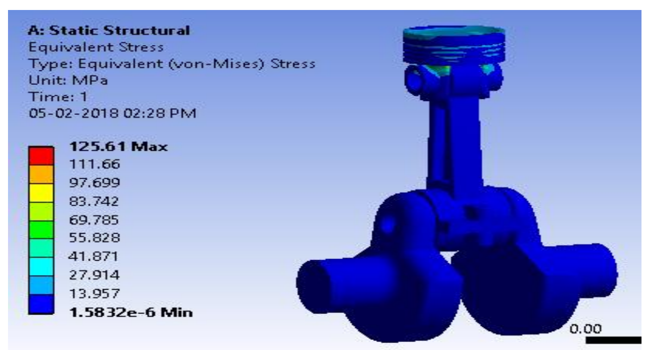

Fig.15 Equivalent stress

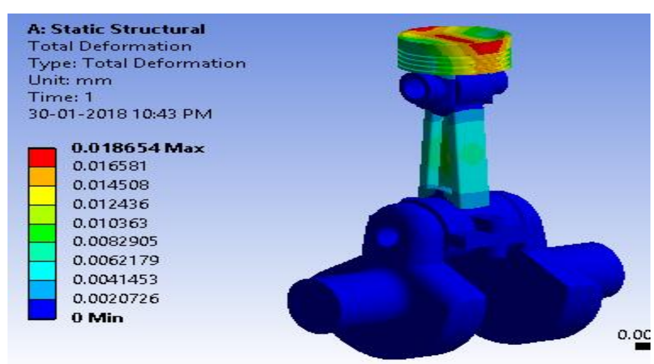

Fig.16 Total deformation

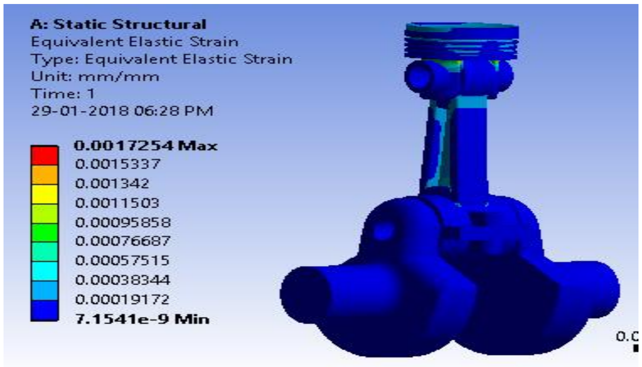

Fig.17 Equivalent elastic strain

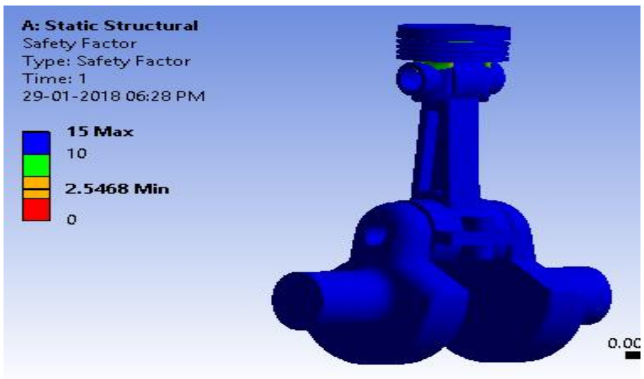

Fig.18 Safety Factor-(Optimized assembly)

From Fig.15 and Fig.16, It is understood that the deformation and equivalent stress were less for optimized assembly when compared to reference model assembly

5.3Parameter study

Assembly is subjected to different pressure loads and parameters like equivalent stress, total deformation, equivalent strain, safety factor.

Stress:

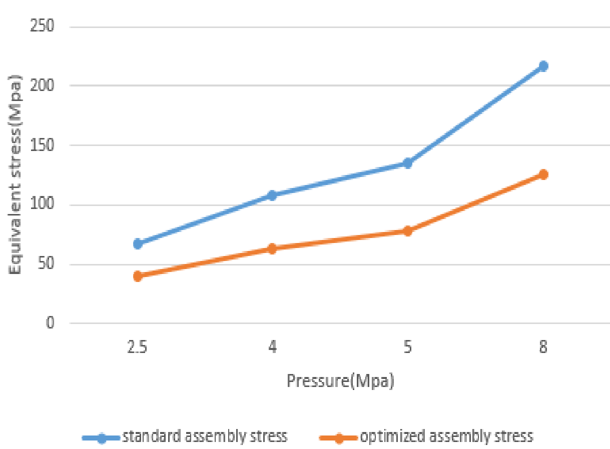

Fig.20 Equivalent stress at different pressure loads

Deformation:

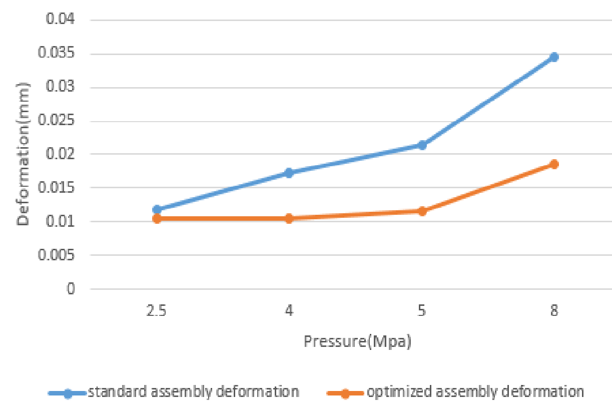

Fig.21 Deformation at different pressure loads

Strain:

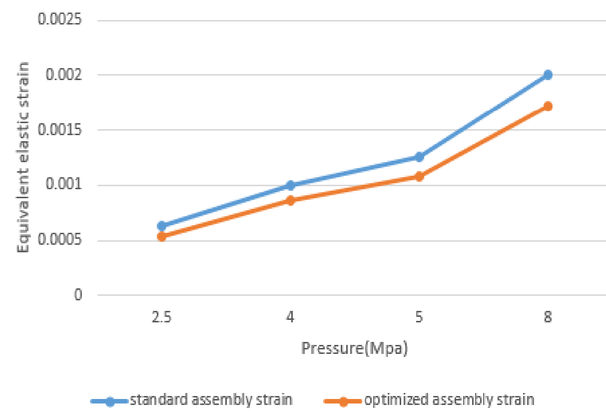

Fig.22 Equivalent strain at different pressure loads

Safety factor:

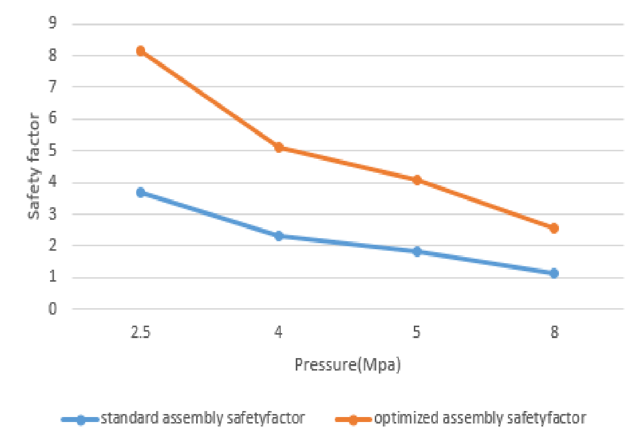

Fig.23 Safety factor at different pressure loads

5.4 Analysis of parameter study:

The values of von-missesstresses, deformation, equivalent strains in optimized assembly is lower than the reference model assembly, which explains that durability and reliability of optimized assembly is more.

Comparison between reference model assembly and optimized assembly

parameters	Reference model Assembly	Optimized Assembly
Equivalent Stress (N/mm^2)	216.98max	125.61max
Equivalent Strain	0.0020035max	0.0017254 max
Deformation(mm)	0.034402 max	0.018654 max
Safety Factor	1.1522min	2.5468 min

Table no.10

5.5 Weight analysis of Assembly

Parameters	Reference model assembly	Optimized assembly
Mass(kg)	5.93226	5.1156

Table no.11

Percentage reduction in weight = 13.7654%

VI. CONCLUSION

- In our first study, the value of Von-Misses Stresses of crankshaft, connecting rod, piston analysis was far less than material yield stress so our design is safe and we did optimization to reduce the material and cost.
- Even after optimizing the components, the von-misses stress was less than the maximum yield strength of the material.
- In our second study, the deformation value for the optimized assembly was less than that of reference model assembly.
- In our final study (weight analysis), there was 13.76 % reduction in weight for the optimized assembly of engine components.

ACKNOWLEDGEMENT

It is with profound gratitude that we express our deep indebtedness to our guide, **Dr. T. Nageswara Rao**, whose dedication to the project, whose knowledge and references got our job done in the shortest time frame possible .we would also like to thank **Dr. K. R. V.Subramanian** who helped us to clear our doubts and for guiding us at each and every step towards completion of this project. This research paper is a tribute to Professor Stephen hawking.

REFERENCES

[1]. DESIGN AND ANALYSIS OF CRANKSHAFT FOR SINGLE CYLINDER 4-STROKE DEISEL ENGINE (Department of Mechanical Engineering, Oriental Institute Of science and technology, Bhopal, India) – E-ISSN2249–8974

[2]. FINITE ELEMENT ANALYSIS OF IC ENGINE CONNECTING ROD BY ANSYS-(ISSN 2278 – 0149)

[3]. Thermal Analysis and Optimization of I.C. Engine Piston Using Finite Element Method-(Mechanical Engineering, Department, KITS College of Engineering Nagpur)

[4]. DESIGN AND WEIGHT OPTIMIZATION OF IC ENGINE-(Department of Mechanical Engineering SIR C.R.R.College of Engineering, (Affiliated to Andhra University)

[5]. .Finite Element Analysis of Crankshaft by ABAQUS- (The College of Mechanical and Electrical Engineering, China University of Mining and Technology Xuzhou 221008,Jiangsu, China)

[6]. Analysis & Optimization of Connecting Rod- (Department of Mechanical Engineering, Pankaj Laddhad Institute of Technology & Management Studies)

[7]. .Finite Element Analysis and Structural Improvement of Diesel Engine Connecting Rod- (Shandong University of Technology Zibo City, China)

[8]. Design And Material Optimization of the Piston by using PROE and ANSYS (M. SreeDivya-1, Dr K.RajaGopal-2,1-M.Tech (CAD/CAM) P.G Scholar, Dept. of M.E, KSRM College of Engineering, Kadapa. 2- PhD, Professor & HOD, Dept. of M.E, KSRM College of Engineering, Kadapa.)