e-ISSN: 2582-5208
International Research Journal of Modernization in Engineering Technology and Science
(Peer-Reviewed, Open Access, Fully Refereed International Journal)
Volume:06/Issue:05/May-2024 Impact Factor- 7.868 www.irjmets.com

CONNECTION ROD OPTIMIZATION FOR WEIGHT AND COST REDUCTION

Mr. V. Keerthivi ME.*[1], Mr. Sivaraj. K*[2], Mr. Karthik. M*[3], Mr. Hageeali. A*[4]

*[1]Assistant Professor Of Mechanical Engineering, MRK Institute Of Technology, Kattumanarkoil, Tamil Nadu, India.

*[2,3,4]UG Student Of Mechanical Engineering, MRK Institute Of Technology, Kattumanarkoil, Tamil Nadu, India.

ABSTRACT

The aim of this project is to reduce weight and costs by optimizing the design and manufacturing of a connecting rod. The method consists of two main steps: optimization and load and stress analysis. To guide material removal, initial analysis estimates loads over time to locate areas of minimal stress. Stress distribution is evaluated at different crank angles using finite element analysis. In order to identify natural frequencies, the connecting rod is put through rigorous testing under stresses ranging from the tensile load of the engine running at maximum speed to the compressive load of the peak gas pressure. In order to guarantee component lifetime and enable weight and cost reduction, a new composite material with improved fracture crack ability is also taken into consideration for replacement. This study attempts to improve connecting rod performance by merging finite element analysis, load analysis, and material optimization.

Keywords: Weight Reduction, Cost Reduction, Connecting Rod, Load Analysis, Stress Analysis, Optimization.

I. INTRODUCTION

The optimization of connecting rods for weight and cost reduction is a critical aspect of modern engineering, particularly in automotive and aerospace industries where efficiency and performance are paramount. This study aims to explore methodologies to achieve these objectives through a comprehensive analysis of loads, stresses, and material properties.

The initial phase of this study involves a detailed load analysis of the connecting rod. By examining the loads acting on the rod over time, areas of minimum stress can be identified, allowing for targeted material removal to reduce unnecessary weight without compromising structural integrity. Additionally, the relationship between load and acceleration of the connecting rod is investigated, providing insights into dynamic behavior under different operating conditions.

Subsequently, finite element analysis (FEA) is employed to assess stress distribution and deformation at various crank angles, providing valuable data for optimization. Extreme loads, such as maximum engine speed tensile load and peak gas pressure compressive load, are considered to ensure the connecting rod's robustness under real-world conditions. Modal analysis is conducted to determine natural frequencies, further informing the design process. Moreover, this study explores the potential benefits of utilizing advanced composite materials for connecting rod construction. The introduction of a new composite material with enhanced fracture crack ability offers the opportunity to reduce weight while maintaining component durability. This aspect is integral to the optimization process, where weight and cost reduction are key objectives.

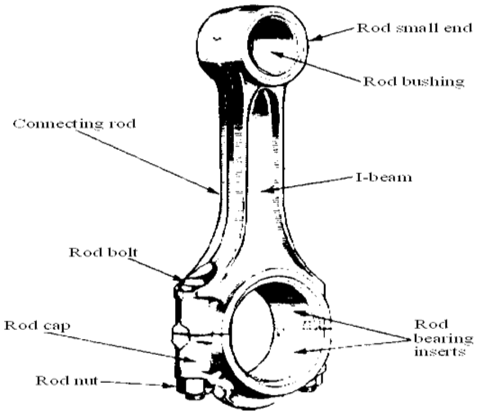


Fig 1: Connecting rod

[Fig. 1: Image of Connecting Rod Parts] This figure illustrates the various components of a connecting rod, providing a visual reference for the subsequent analysis.

II. METHODOLOGY

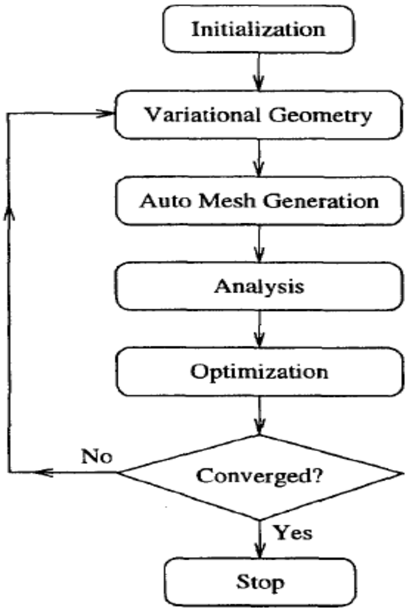
2.1 ANALYTICAL EVALUATIONS

Digitizing Connecting Rod Geometry

Stress (FEA) Analysis

Modal Analysis and Life Predictions

Optimization Analysis



2.2 STEPS INVOLVED IN METHODOLOGY

Step 1: Modeling of connecting rod using 3D modeling software.

Step 2: Finite element modeling of the connecting rod.

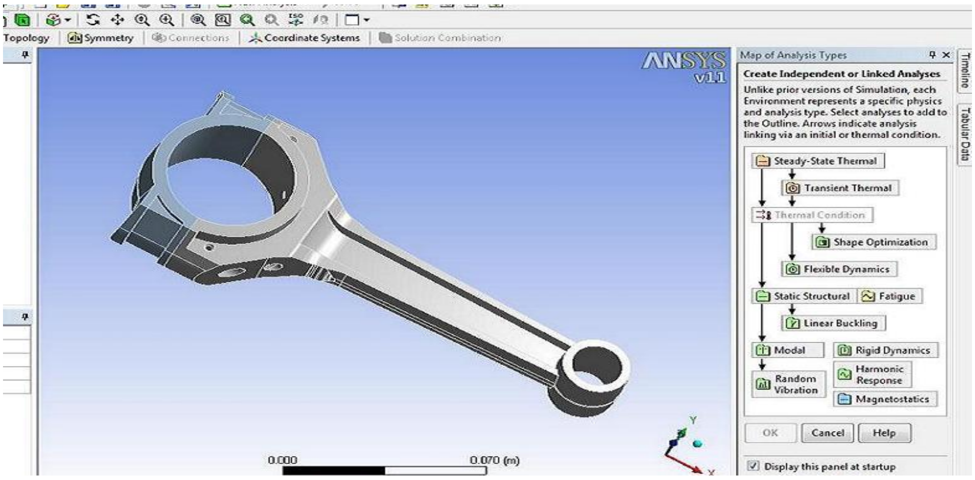
Step 3: Analysis of connecting rod using Ansys software.

1. Element selection.
2. Discretization.
3. Mesh generation.

Step 4: Finite element stress analysis.

Step 5: Modal analysis.

Step 6: Optimization.



Unlike prior versions of Simulation, each Environment represents a specific physics and analysis type. Select analyses to add to the Outline. Arrows indicate analysis linking via an initial or thermal condition.

☑ Display this panel at startup

Fig 2: Map of analysis data

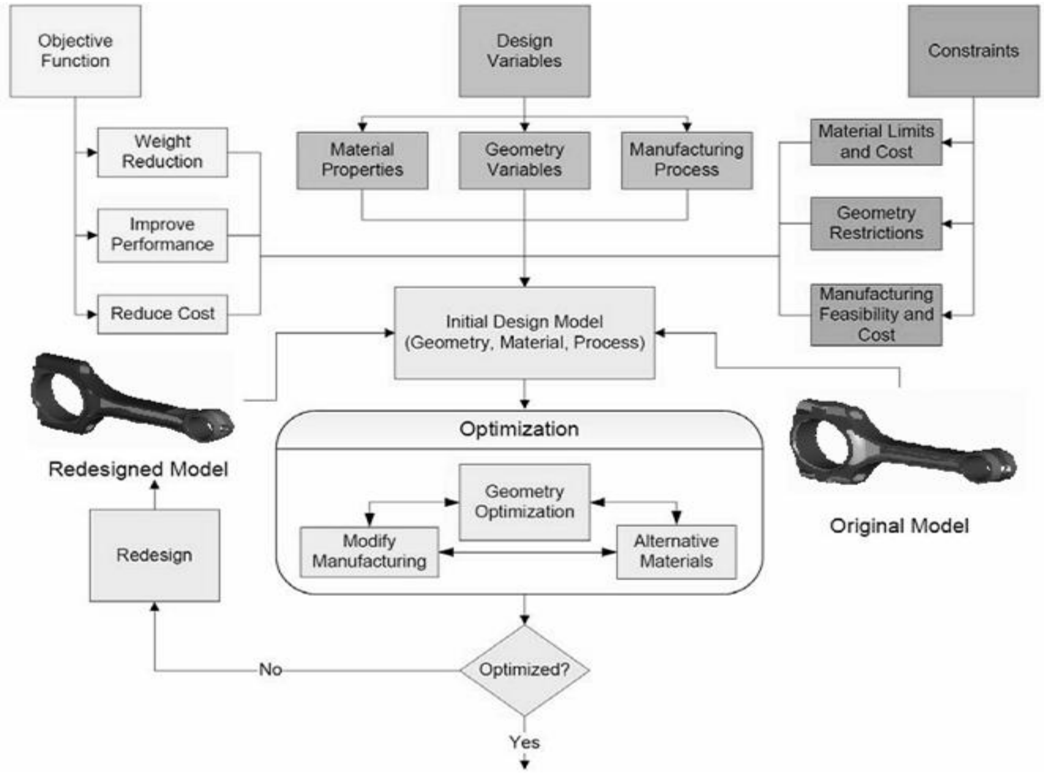
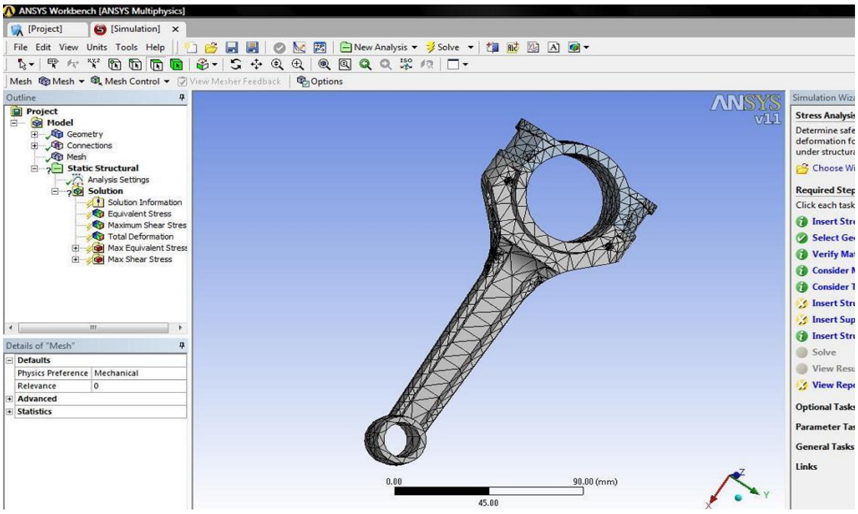


Fig 3: Optimization Procedure

III. FINE ELEMENT ANALYSIS OF CONNECTIN ROD

3.1 STATIC ANALYSIS

The effects of constant loading conditions on the connecting rod are computed in this analysis. This approach can also be used to assess the effects of time-varying loads, such as inertia loads, which are represented as static equivalent loads. It is employed to ascertain the forces, displacements, strains, and stresses on the structures. Externally applied forces and pressures, steady-state inertial forces like gravity or rotational velocity, imposed displacements, and temperatures are examples of the loading types that can be used in a static analysis.



Physics Preference: Mechanical
Relevance: 0

Determine safe
deformation fc
under structura

Fig 4: Finite element model of the connection rod

3.2 APPLYING LOADS TO THE FINITE ELEMENT MODEL

The connecting rod's finite element model was fitted with the axial and bending stresses determined previously.

The model's centre of gravity is where the bending stress is applied transversely, and the small end of the lower half of the connecting rod is where the axial load is applied.

The two load types mentioned above, which were computed at various crank degrees, were applied to the model one at a time and each load set was examined independently.

3.3 APPLYING BOUNDARY CONDITIONS

For every node on the geometry symmetry plane, there is a restriction on the displacement perpendicular to the plane. Axial loads impose constraints on the displacement in the direction of the load. All nodes on the bearing surface of the big end bore top half are restrained if the axial forces' compressive in character; in contrast, the lower half is restrained if the load is tension-based.

All of the big end and small end bearing surface nodes are restricted in their displacement in the same direction for transverse loads. Analytical calculations were used to determine the loads that will be applied to the finite element model.

Tension and compression loads applied as pressure on bearing surfaces,

Four cases considered,

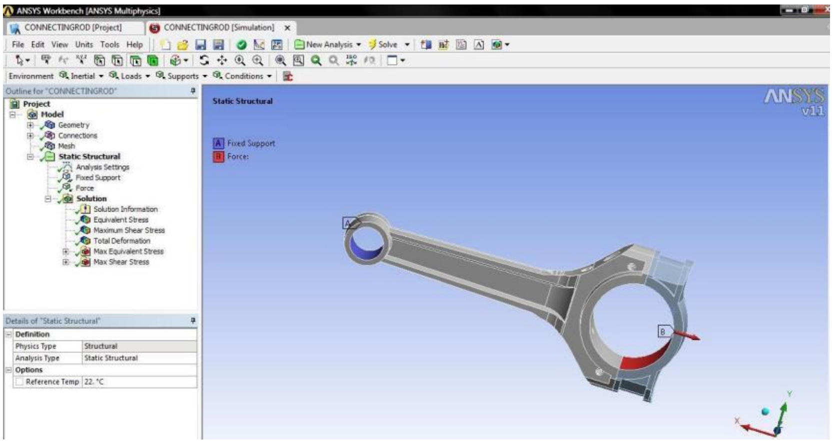
Crank end	**Pin end**
Tension	Restrict
Compression	Restrict
Restrict	Tension
Restrict	Compression

3.4 ANALYSIS OF THE CONNECTING ROD

The finite element issue is solved by the ANSYS solution module when the loads and boundary conditions are applied.

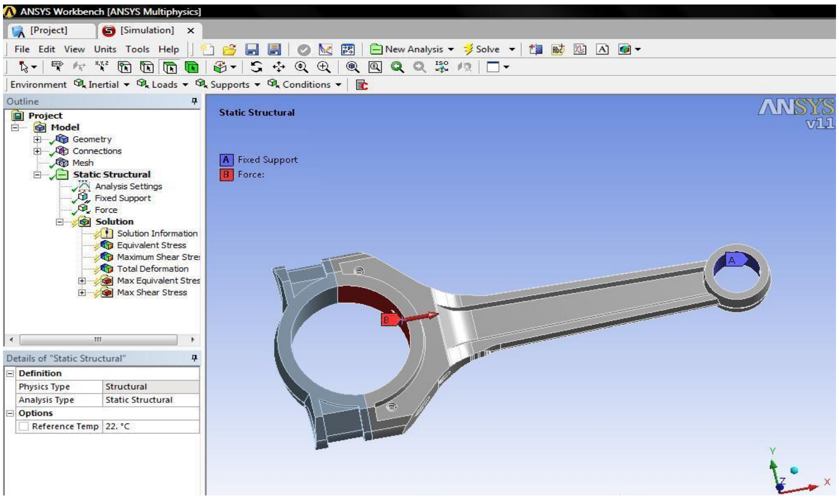
The model was solved by applying the forces discovered at various crank angles. The combined axial compressive and inertia bending loads at various crank angles are what cause the stress contours in the model. Considering the model's stress profiles as a result of coupled bending loads at various crank degrees caused by inertia and tensile stresses.

The highest values of induced stress that emerged in the model due to the application of loads. The values are obtained from the finite element analysis of the connecting rod



Definition	
Physics Type	Structural
Analysis Type	Static Structural
Options	
Reference Temp	22. °C

Fig 5: Tension On Crank End And Pin End Restricted



Details of "Static Structural"

Definition	
Physics Type	Structural
Analysis Type	Static Structural
Options	
Reference Temp	22. °C

Fig 6: Compression On Crank End And Pin End Restricted

IV. RESULTS AND DISCUSSION

A high-speed automobile compression ignition engine's connecting rod was modeled using modeling technique and examined using finite element analysis. The final dimensions produced after finite element analysis satisfy the requirements, and several possibilities are presented for the postulated conditions in the above modeling and analysis of the connecting rod.

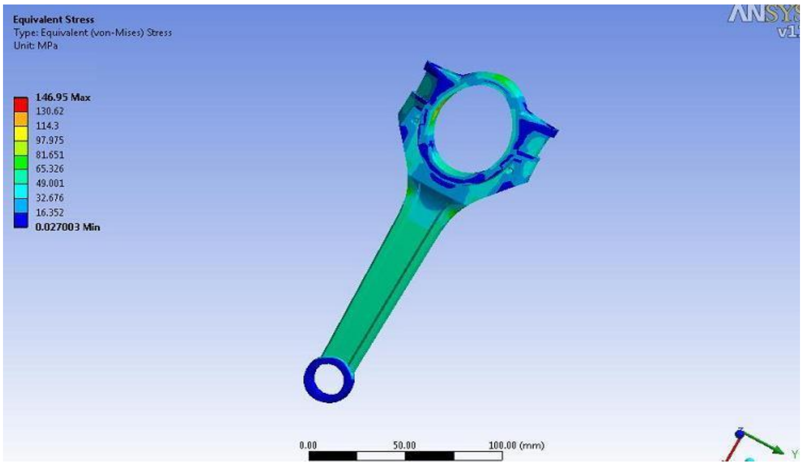


Fig 7: Structural Analysis On Existing Connecting Rod

The axial and bending stresses have been considered in the static analysis of the connecting rod. The axial and bending loads are computed analytically at various crank angles, applied to the connecting rod's finite element model, and then examined. Analytical stress values are compared with the finite element analysis result. The study reveals that the stress levels are significantly lower than the connecting rod material's yield strength.

The connecting rod's shank area has the greatest potential for weight reduction, according to the comparison of the data. Only a limited amount of rib and web thickness reduction occurred in the shank region.

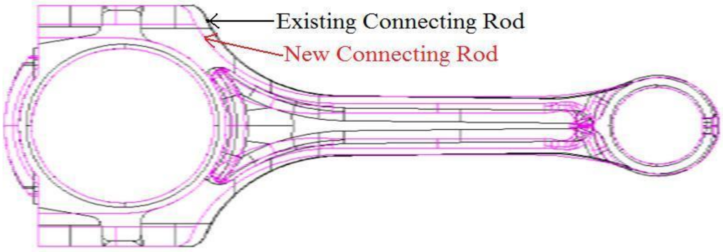
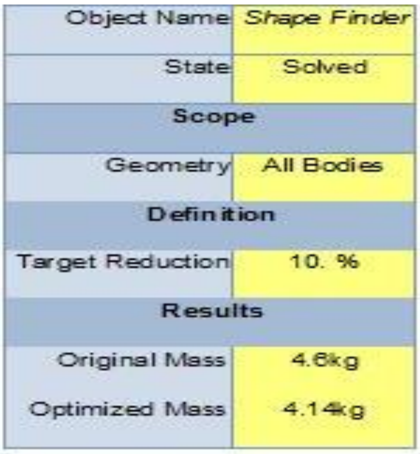


Fig 8: Shape optimization

Once loads at various speeds are calculated and analysis is carried out multiple times, a constrained model is produced.



Object Name	*Shape Finder*
State	Solved
Scope	
Geometry	All Bodies
Definition	
Target Reduction	10. %
Results	
Original Mass	4.6kg
Optimized Mass	4.14kg

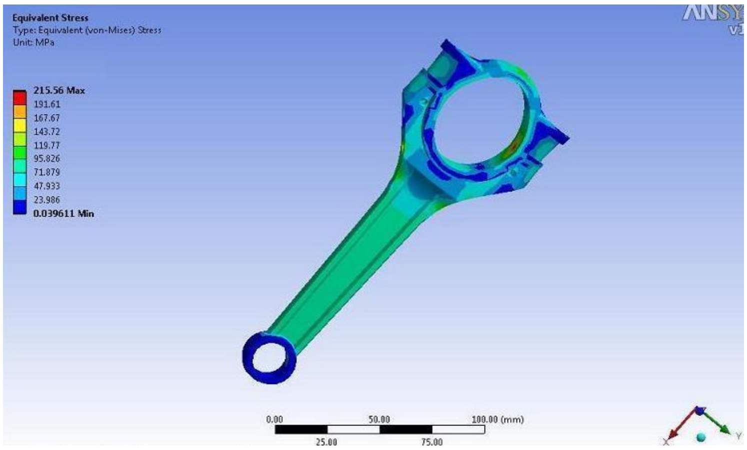
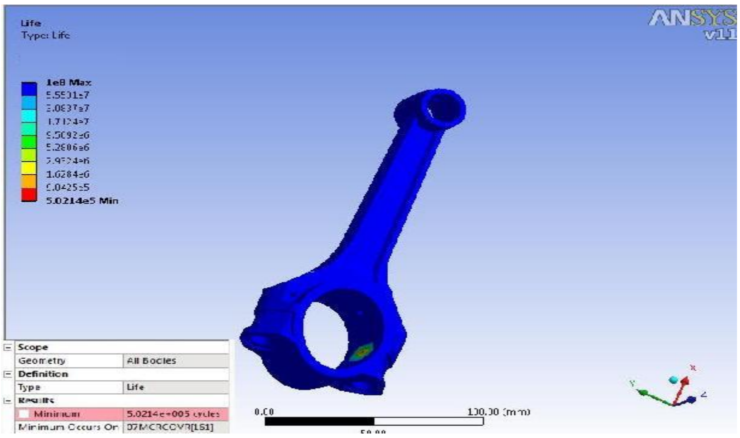


Fig 9: Structural analysis of new connection rod

This work examines weight optimization under two cycle loads, with static compressive and dynamic tensile loads serving as the two extreme loads. For life prediction analysis, take into account the cyclic load conditions as well. In this field of study, fatigue strength plays a major role in the optimization process. An improved connecting rod that is roughly 10% lighter and 25% less expensive than the current connecting rod is the outcome of the study.

In the production of diesel engines, connecting rods are often made of C45 steel. In this study, I suggest using C70 steel, an alternate material, when making connecting rods in order to lower the rod's weight. It is unexpected to find that there is a weight decrease of about 10% when comparing C45 and C70 steel. Because the material's tensile strength and endurance limit are nearly identical, changing the material has no effect whatsoever on the engine's performance.



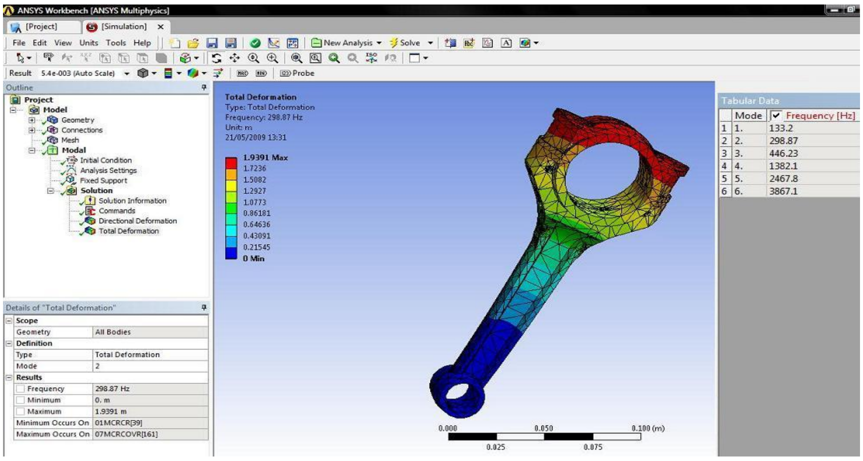
Scope	
Geometry	All Bodies
Definition	
Type	Life
Results	
Minimum	5.0214e+005 cycles
Minimum Occurs On	07MCRCOVR[151]

Fig 10: Life Predictions Analysis On Connecting Rod

Table 1: Material Properties Of C45 And C70 Steel

S. No	Material Properties	Values for C45 steel CR	Values for C70 steel CR
1.	E (GPa)	207	212
2.	Yield strength(MPa)	700	574
3.	% Elongation	24	27
4.	% Reduction in area	42	25
5.	Tensile strength(MPa)	938	966
6.	Endurance limit(MPa)	415	399
7.	Density (kg/m^2)	0.000782	0.000796
8.	Axial Displacement(mm)	0.206	0.208
9.	Weight(gms)	440	396
10.	Izz (kg-m^2)	0.00144	0.00139
11	Buckling load factor	7.8	9.6

A mechanical engineer's job includes studying vibration in order to reduce the impacts of vibration on mechanical components through appropriate design. We have looked for the system's natural frequency in an effort to prevent the rotating system from experiencing undue and unwanted stress.



Total Deformation
Type: Total Deformation
Frequency: 298.87 Hz
Unit: m
21/05/2009 13:31

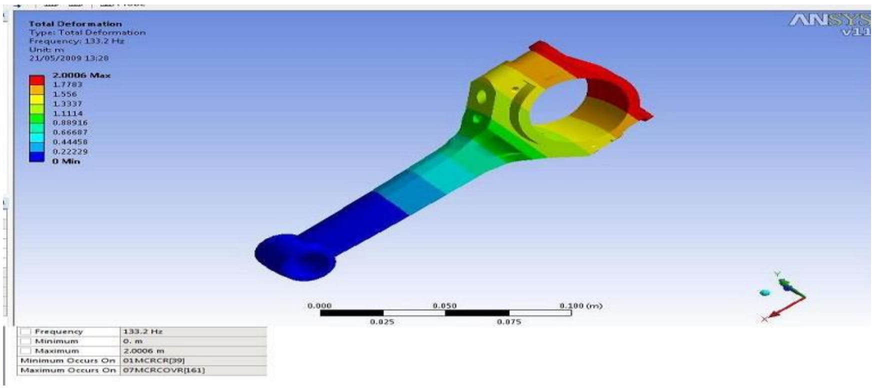
Tabular Data

	Mode	☑ Frequency [Hz]
1	1.	133.2
2	2.	298.87
3	3.	446.23
4	4.	1382.1
5	5.	2467.8
6	6.	3867.1

Details of "Total Deformation"

Scope	
Geometry	All Bodies
Definition	
Type	Total Deformation
Mode	2
Results	
☐ Frequency	298.87 Hz
☐ Minimum	0. m
☐ Maximum	1.9391 m
Minimum Occurs On	01MCRCR[39]
Maximum Occurs On	07MCRCOVR[161]

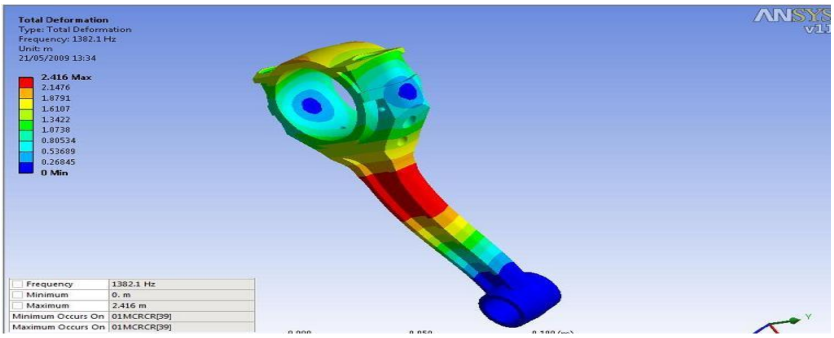
Fig 11: First 6 Natural Frequencies



Total Deformation
Type: Total Deformation
Frequency: 133.2 Hz
Unit: m
21/05/2009 13:28

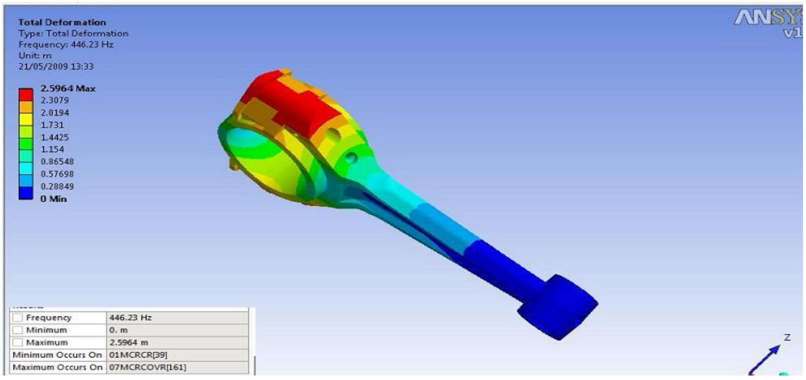
☐ Frequency	133.2 Hz
☐ Minimum	0. m
☐ Maximum	2.0006 m
Minimum Occurs On	01MCRCR[39]
Maximum Occurs On	07MCRCOVR[161]

Fig 12: Natural Frequency -Axial Mode



☐ Frequency	1382.1 Hz
☐ Minimum	0. m
☐ Maximum	2.416 m
Minimum Occurs On	01MCRCR[39]
Maximum Occurs On	01MCRCR[39]

Fig 13: Natural Frequency -Bending Mode



☐ Frequency	446.23 Hz
☐ Minimum	0. m
☐ Maximum	2.5964 m
Minimum Occurs On	01MCRCR[39]
Maximum Occurs On	07MCRCOVR[161]

Fig 14: Natural Frequency -Twisting Mode

We found that there was no possibility of resonance when we used modal analysis to examine the natural frequency. Resonance develops when the frequency of excitation matches one of the natural frequencies; this could lead to the system's mechanical breakdown.

APPENDIX 1

ENGINE SPECIFICATIONS

Table 2:

NAME OF THE ENGINE	S447 ENGINE
BORE DIAMETER	92 mm
STROKE LENGTH	120 mm
MAXIMUM POWER	112 Hp at 2800 rpm
MAXIMUM TORQUE	30 kgf at 1900 rpm
CUBIC CAPACITY	4788cc
COMPRESSION RATIO	17:1

APPENDIX 2

CONNECTING ROD SPECIFICATION

Table 2:

SMALL END EYE DIAMETER	30 mm
BIG END BORE DIAMETER	60 mm
CONNECTING ROD LENGTH (CENTER TO CENTER)	230 mm

V. CONCLUSION

An earnest attempt is made in the development of a suitable, lighter-weight connecting rod for a multi-cylinder automotive compression-ignited engine using modeling and finite element techniques.

Optimization was performed to reduce the weight and manufacturing cost of a steel connecting rod subjected to cyclic loads comprising the compressive gas load and the dynamic tensile load at different speeds, corresponding to various crank angles. The cost was reduced by changing the material of the existing C45 steel connecting rod to C70 steel.

Optimization is performed to reduce the weight and manufacturing cost of a steel connecting rod subjected to different loads at different speeds, corresponding to various crank angles.

- Material removal has been introduced in places where minimum stress is acting.
- Modal analysis was performed to find out natural frequencies to reduce excitation noise and vibration.
- The cost is reduced by changing the material of the existing C 45 steel connecting rod to C 70 steel.

With more informative data, we will be able to generate very good designs in a relatively short period of time, and we may use optimal designs. We believe that the techniques employed in this study will be of great use to the designer. Similar studies may be taken up for the effective design of the other internal combustion engine components. Static analysis is only carried out in this project. Further, it can be extended to do a quasi-dynamic analysis. It can also be studied by using various materials.

VI. REFERENCES

[1] Adila Afzal and Ali Fatemi, 2003, "A Comparative Study of Fatigue Behavior and Life Predictions of Forged Steel and PM Connecting Rods", SAE International.

[2] Adila Afzal and Pravardhan Shenoy, 2003, "Dynamic Load Analysis and Fatigue Behavior of Forged Steel v_s Powder Metal Connecting Rods", American Iron and Steel Institute, October Edition.

[3] Athavale, S. and Sajanpawar, P. R., 1991, "Studies on Some Modelling Aspects in the Finite Element Analysis of Small Gasoline Engine Components," Small Engine Technology Conference Proceedings, Society of Automotive Engineers of Japan, Tokyo, PP. 379-389.

[4] Augugliaro G. and Biancolini M.E., "Optimisation of Fatigue Performance of a Titanium Connecting Rod", ISPESL, Italy.

[5] Farzin H. Montazersadgh and Ali Fatemi, 2008, "Optimization of a Forged Steel Crankshaft Subject to Dynamic Loading", SAE International.

[6] Farzin h. Montazersadgh and Ali Fatemi, 2007, "Dynamic Load and Stress Analysis of a Crankshaft", SAE International.

[7] Giuseppe Sala, 2002, "Tecnology-Driven Design of MMC Squeeze Cast Connecting Rods", Science and Technology of Advanced Materials, No. 3, PP. 45-57.

[8] Hippoliti, R., 1993, "FEM Method For Design and Optimization of Connecting Rods for Small Two-Stroke Engines," Small Engine Technology Conference, PP. 217-231.

[9] James R. Dale, 2005, "Connecting Rod Evaluation", Metal Powder Industries Federation, January Edition.

[10] Pai, C. L., 1996, "The Shape Optimization of a Connecting Rod with Fatigue Life Constraint", Int. J. of Materials and Product Technology, Vol. 11, No. 5-6, PP. 357-370.

[11] Park, H., Ko, Y. S., Jung, S. C., Song, B. T., Jun, Y. H., Lee, B. C., and Lim, J. D., 2003, "Development of Fracture Split Steel Connecting Rods," SAE Technical Paper Series, Paper No. 2003-01-1309.

[12] R.J. Yang, D.L. Dewhirst, J.E. Allison and A. Lee, 1992, "Shape optimization of connecting rod pin end using a generic model", Finite Elements in Analysis and Design, No. 11, PP. 257-264.

[13] Rabb, R., 1996, "Fatigue Failure of a Connecting Rod", Engineering Failure Analysis, Vol. 3, No. 1, PP. 13-28.

[14] Repgen, B., 1998, "Optimized Connecting Rods to Enable Higher Engine Performance and Cost Reduction", SAE Technical Paper Series, Paper No. 980882.

[15] Rice, R. C., ed., 1997, "SAE Fatigue Design Handbook", Society of Automotive Engineers, Warrendale, 3rd Edition.

[16] Sarihan, V. and Song, J., 1990, "Optimization of the Wrist Pin End of an Automobile Engine Connecting Rod With an Interference Fit", Journal of Mechanical Design, Transactions of the ASME, Vol. 112, PP. 406-412.

[17] Serag, S., Sevien, L., Sheha, G., and El-Beshtawi, I., 1989, "Optimal Design of the Connecting Rod", Modelling, Simulation and Control, B, AMSE Press, Vol. 24, No. 3, PP. 49-63.

[18] Teruie Takemasu, Victor Vazquez, Brett Painter and Taylan Altan, 1996 "Investigation of Metal Flow and Perform Optimization in Flash Less Forging of a Connecting Rod", Journal of Materials Processing Technology, No. 59, 95-105.

[19] Webster, W. D., Coffell R., and Alfaro D., 1983, "A Three Dimensional Finite Element Analysis of a High Speed Diesel Engine Connecting Rod," SAE Technical Paper Series, Paper No. 831322.

[20] Yoo, Y. M., Haug, E. J., and Choi, K. K., 1984, "Shape Optimal Design of an Engine Connecting Rod," Journal of Mechanisms, Transmissions, and Automation in Design, Transactions of ASME, Vol. 106, PP. 415-419.