© INTERNATIONAL JOURNAL FOR RESEARCH PUBLICATION & SEMINAR
ISSN: 2278-6848 | Volume: 13 Issue: 04 | July - September 2022
Paper is available at http://www.jrps.in | Email : info@jrps.in
Refereed & Double Blind Peer Reviewed

VIBRATION AND STATIC ANALYSIS OF COMPOSITE 2 WHEELER CONNECTING ROD

Ravi V Singh
P.G Student, Department of Mechanical Engineering, D Y Patil University, School of Engineering & Technology
Ambi, Pune Maharashtra, India

Dr Shriramshastri Chavali
Professor, Department of Mechanical Engineering, D Y Patil University, School of Engineering & Technology
Ambi, Pune Maharashtra, India

Abstract-* Composite materials are now a day widely used in the engineering field. The general characteristics possessed by the composite materials are found to be the reason for using it in the automotive applications. The connecting rod is a major link inside of combustion engine. It connects the piston] to the crankshaft and is responsible for transferring power from the piston to the crankshaft. It has to work on high r.p.m. because of which it has to bear severe stresses which make its design vital for internal combustion engine. In this project, design, analysis of the 2 wheeler connecting rod will be perform. The CATIA V5 R20 software has been used for designing the connecting rod 3D model and then the designed connecting rod model is imported into the ANSYS software in which the design is meshed and analyzedby using the Finite Element Method (FEM) and the result is manipulated. Modal analysis of existing and composite connecting rod will be performed. Results of the analysis are also validated by using UTM.

***Keywords—* connecting rod, FEA, Carbon fiber, UTM,**

I. INTRODUCTION

The connecting rod is a major link inside a combustion engine. It connects the piston to the crankshaft and is responsible for transferring power from the piston to the crankshaft and sending it to the transmission. The objective of C.R. is to transmit push & pull from the piston pin to the crank pin and then converts reciprocating motion of the piston into the rotary motion of crank. The components are big shank, a small end and a big end. The cross section of shank may be rectangular, circular, tubular, I- Section, + -section or ellipsoidal-Section. It sustains force generated by mass & fuel combustion. The resulting bending stresses appear due to eccentricities, crank shaft, case wall deformation & rotational mass.

There are different types of materials and production methods used in the creation of connecting rods. The most common types of Connecting rods are steel and aluminium. The most common types of manufacturing processes are casting, forging and powdered metallurgy. Connecting rods are widely used in variety of engines such as, in-line engines, V-engine, opposed cylinder engines, radial engines and opposed-piston engines.

Fig. 1 Connecting rod

A connecting rod consists of a pin-end, a shank section, and a crank-end. Pin-end and crank-end pinholes at the upper and lower ends are machined to permit accurate fitting of bearings. The function of connecting rod is to transmit the thrust of the piston to the crankshaft. Figure shows the role of connecting rod in the conversion of reciprocating motion into rotary motion.

II. LITERATURE REVIEW

D.Gopinatha,Ch.V.Sushmab[1] “Design and Optimization of Four Wheeler Connecting Rod Using Finite Element Analysis”, The main objective of research was to explore weight reduction opportunities for the production of forged steel, aluminum and titanium connecting rods. This has entailed performing a detailed load analysis. Therefore, this study has dealt with two subjects, first, static load stress analysis of the connecting rod for three materials, and second, optimization for weight of forged steel connecting rod. In this research, firstly a proper geometrical model was developed using CATIA. Then the model is imported to the HYPERMESH which is a finite element pre-processor that provides a highly interactive and visual environment to analyze product design performance and the Finite Element model was developed. The stresses were found in the existing connecting rod for the given loading conditions using Finite Element Analysis software ANSYS 11.0.The topology optimization technique is used to achieve the objectives of optimization which is to reduce the weight of the connecting rod.

After carrying out static structural analysis the stresses in each loading conditions were studied and then area where excess material can be removed were decided. Optimization was performed to reduce weight of a forged steel connecting rod subjected to the peak compressive gas load and the peak tensile load. The shank region of the connecting rod offered the greatest potential for weight reduction. The rib and the web thicknesses were reduced, while maintaining forge ability.

Mohammed Mohsin Ali Ha , Mohamed Haneef b [2] “Analysis of Fatigue Stresses on Connecting Rod Subjected to Concentrated Loads At The Big End Connecting rod is modeled using CATIA software and FE analysis is carried out using ANSYS Software .Load distribution plays importantrole in fatigue life of the structure. Bush failure changes the loading direction and distribution. Present study is concentrated around the fatigue life due to concentrated load and cosine type load distribution on the bigger end. The connecting rod analysis is carried out to check the fatigue life and alternating stress development due to service and assembly loads with variation in load distribution. Initially the connecting rod is built to the actual dimensions using Catia software. Axi-symmetric analysis is carried out to find interference effect on the stress behavior in the joint. 8 nodedplane82 elements with quadratic displacement variation is used for accurate results. The contact pair is created with Targe169 and Contac172 elements. Interference is created through geometric built up. The result shows contact pressure development at the interface and higher compressive stress in the bush and tensile stress development in the small end. The results are plotted for radial, hoop and vonmises stresses. Also a three dimensional views are obtained through ansys ax symmetric options.

Amit Kumar, Bhingole P.P. Dinesh Kumar [3] Dynamic Analysis of Bajaj Pulsar 150cc Connecting Rod Using ANSYS 14.0. This paper deals with analysis of Bajaj Pulsar 150cc connecting rod in dynamic loading conditions. In this study the connecting rods modulate and simulated for the dynamic analysis by using catia software for modeling-design of connecting rod and ansys 14.0 for dynamic analysis. Using available high strengthen alloy is used for the connecting rod of Bajaj pulsar 150cc for the weight reduction to reduce moment of inertia. Dynamic analysis is carried out for determine the von mises stress, strain, and total deformation is calculated under loading conditions of compression and tension at crank end and pin end of connecting rod.

Pravardhan S. Shenoy, Ali Fatemi, [4] “Connecting Rod Optimization for Weight and Cost Reduction” The objective of this study was to optimize a forged steel connecting rod for its weight and manufacturing cost, taking into account recent developments. An optimization study was performed on a steel forged connecting rod with a consideration for improvement in weight and production cost. Since the weight of the connecting rod has little influence on its total production cost, the cost and the weight were dealt with separately. Reduction in machining operations, achieved by change in material, was a significant factor in manufacturing cost reduction. Weight reduction was achieved by using an iterative procedure. Literature survey suggests cyclic loads comprised of static tensile and compressive loads are often used for design and optimization of connecting rods. However, in this study weight optimization is performed under a cyclic load comprising dynamic tensile load and static compressive load as the two extreme loads. Constraints of fatigue strength, static strength, buckling resistance and manufacturability were also imposed. The fatigue strength was the most significant factor in the optimization of the connecting rod. An estimate of the cost savings is also made. The study results in an optimized connecting rod that is 10%lighter and 25% less expensive, as compared to the existing connecting rod. Optimization was performed to reduce weight and manufacturing cost of a forged steel connecting rod subjected to cyclic load comprising the peak compressive gas load and the peak dynamic tensile load at 5700 rev/min, corresponding to 360o crank angle. The structural factors considered for weight reduction during the optimization process included fatigue strength, static strength, buckling resistance, bending stiffness, and axial stiffness. Additional constraints imposed during the optimization process included maintaining the forge ability as well as interchangeability of the optimized connecting rod with the existing one.

Anil Kumar, Kama deep Grover, Balvinder Budania [5] “Optimization of Connecting Rod Parameters using CAE Tools”, Aim of this work is to optimize weight and reduce inertia forces on the existing connecting rod, which is obtained by changing such design variables in the existing connecting rod design. The model was developed in Pro/E wildfire 5.0 and then imported as paranoid (xt) form in ANSYS workbench. In this work finite element analysis of the single cylinder four stroke petrol engine connecting rod is consideredas case study. The Von Mises stress, strain and total deformation determined for the same loading conditions and compared with the existing results. Based on the observation of static FEA and the load analysis result, the load for the optimization study was selected same as on existing connecting rod. The current work consists of static structural analysis. The static analysis was carried out under axial and buckling load. The model is also selected for fatigue analysis to determine the fatigue strength. The stress was found maximum at the piston end. This can be reduced by increasingthe material near the piston end. The weight of the connecting rod was also reduced by 0.004 kg which was not significantbut reduces the inertia forces.

Satish Wable, Dattatray S. Galhe, Rajkumar L. Mankar [6] The main objective of this study is to review the weight optimization of a connecting rod in an automobile engine. To get the idea about designing the connecting rod, various stresses to be considered while designing the connecting rod and different materials used and comparing the result of all materials. To know the different software and Finite Element Method (FEM) packages useful for the modeling and analysis of connecting rod.

All researches mentioned in this study give the idea about designing of the connecting rod. It explains about the various stresses to be considered while designing the connecting rod and different materials used and comparing the result of all material. Also most of the researchers used the CATIA software for the modeling and ANSYS software for analysis. These can be used for designing any connecting rod in Automobile. Connecting rod can be designed for weight and cost reduction also to increase the life time of connecting rod. Upton some level of extent the weight of the connecting rod islighter and having more strength as compared to the original design.

Mr. Sahel, Mr. Jiten Saini [7] In this study static and modal analysis is performed. The S-N approach by modifiedGoodman criterion to the fatigue life prediction of the connecting rods is also presented. The model is developed using Solid Modeling software-Solidworks2013 .Further finite element analysis is done using Ansys14 Workbench to determine the von-misses stresses and strains, fatigue life and modal frequencies under different loading conditions. Thisstudy is based on the static structural module. Further analysis of connecting rod can be done under dynamic environment.

Samper Nasir Momin, R.J. Gowanda [8] This study incorporates FEA modal analysis and experimental modal analysis of connecting rod. A parametric model of Connecting rod is modeled using CATIA V5 R19 software and finite element analysis is carried out by using ANSYS Software. Finite element method is used to determine natural frequencies of a connecting rod and compare results with FFT analyzer. FFT analysis is done by hanging the connecting rod at small end and experimental results were compared with FEM.

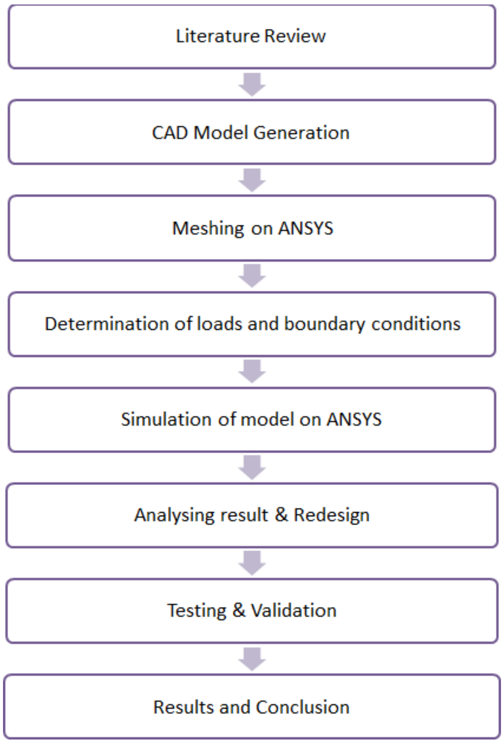
III. PROBLEM STATEMENT

Each day consumers are looking for the best from the best. That’s why the material optimization is really important in industry. Material Optimization of the component is to make the less time to produce the product that is stronger, lighter and less cost. The design and weight of the connecting rod influence on performance. Hence, it is effect on the manufacture credibility. The tensile and compressive stresses are produced due to pressure, and bending stresses are produced due to centrifugal effect & eccentricity. So the connecting rods are designed generally of I-section to provide maximum rigidity with minimum weight. Change in the structural design and also material will be significant increments in weight and performance

IV. OBJECTIVES

- Modeling Two-wheeler connecting rod in CATIA V5 software.
- To perform static analysis for existing 2-wheeler connecting rod with composite reinforcement to determine the enhancement in mechanical properties under buckling analysis in ANSYS 19 software.
- Manufacturing of carbon fibre reinforced connecting rod by using hand lay -up method on connecting rod.
- To perform experimental testing of new carbon fibre reinforced connecting rod on UTM.
- Validation of experimental testing and FEA results.

V. METHODOLOGY



CATIA MODEL

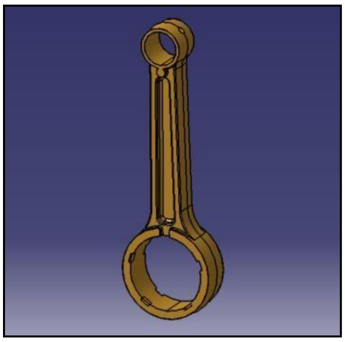
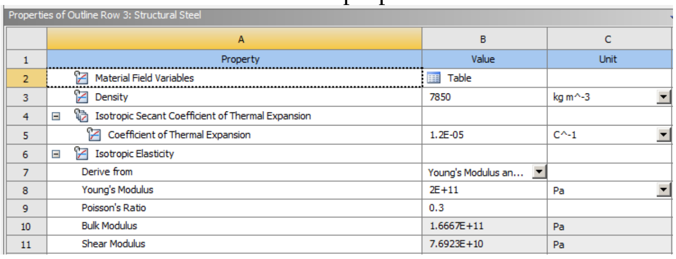


Fig. 2 CATIA model of 2-wheeler connecting rod

VI. STATIC ANALYSIS OF EXISTING CONNECTING ROD

1. Material Properties

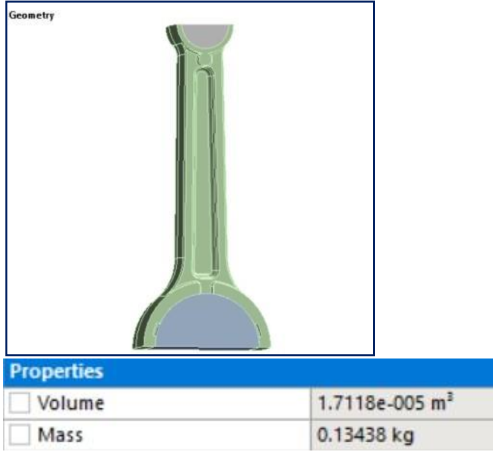
Table 1 Material properties of SS



Properties of Outline Row 3: Structural Steel

	A	B	C
1	Property	Value	Unit
2	Material Field Variables	Table	
3	Density	7850	kg m^-3
4	⊟ Isotropic Secant Coefficient of Thermal Expansion		
5	Coefficient of Thermal Expansion	1.2E-05	C^-1
6	⊟ Isotropic Elasticity		
7	Derive from	Young's Modulus an...	
8	Young's Modulus	2E+11	Pa
9	Poisson's Ratio	0.3	
10	Bulk Modulus	1.6667E+11	Pa
11	Shear Modulus	7.6923E+10	Pa

2. Finite Element Analysis:

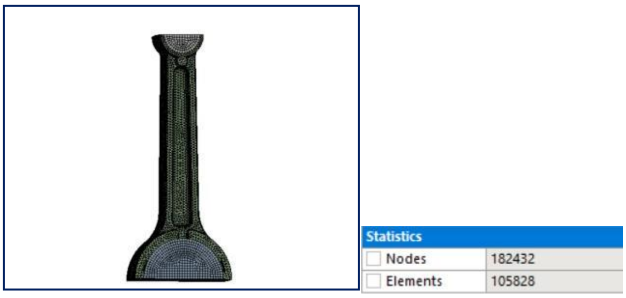


Properties	
Volume	1.7118e-005 m³
Mass	0.13438 kg

Fig.3 CATIA model imported in ANSYS

3. Mesh

ANSYS Meshing is a general-purpose, intelligent, automated high-performance product. It produces the most appropriate mesh for accurate, efficient Multiphysics solutions. A mesh well suited for a specific analysis can be generated with a single mouse click for all parts in a model. Full controls over the options used to generate the mesh are available for the expert user who wants to fine-tune it.



Statistics	
Nodes	182432
Elements	105828

Fig. 4 meshing of connecting rod

After meshing of 2 wheeler connecting rod the Number of nodes 182432 and Elements 105828

4. Boundary conditions

To determine the buckling strength initial 1000 N compressive force is applied along with piston end fixed.

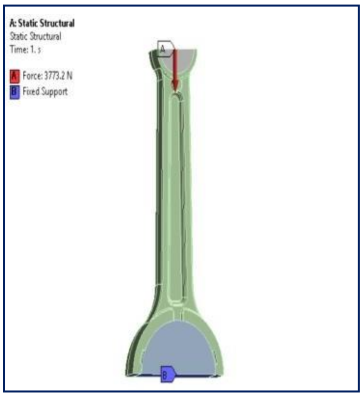
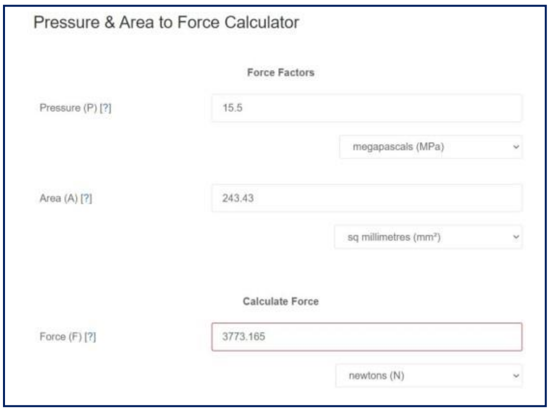


Fig. 5 Boundary condition for static analysis



Pressure & Area to Force Calculator

Force Factors

Pressure (P) [?] 15.5 megapascals (MPa)

Area (A) [?] 243.43 sq millimetres (mm²)

Calculate Force

Force (F) [?] 3773.165 newtons (N)

5. Results

5.1 Total deformation

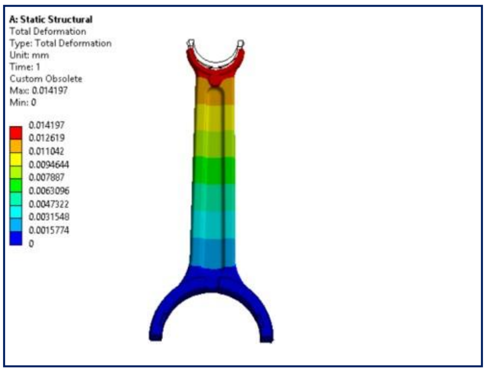


Fig. 6 Total deformation plot for existing connecting rod

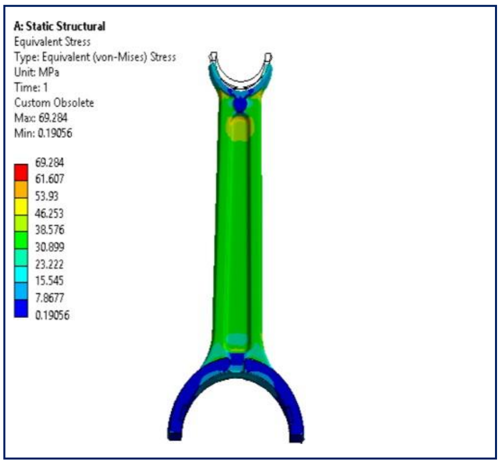
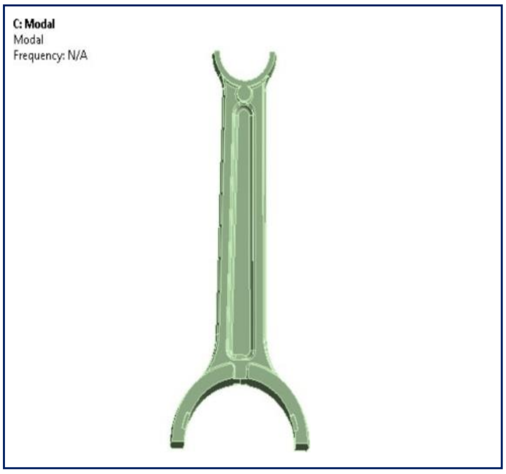
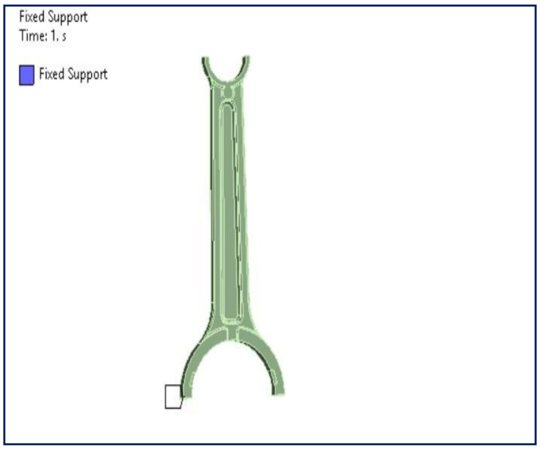


Fig. 7 Equivalent stress plot for existing connecting rod

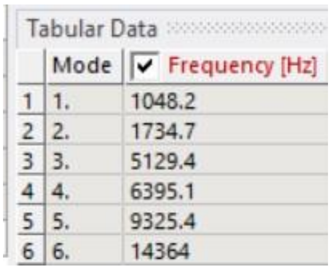
VII. MODAL ANALYSIS OF EXISTING CONNECTING ROD



1. Boundary conditions



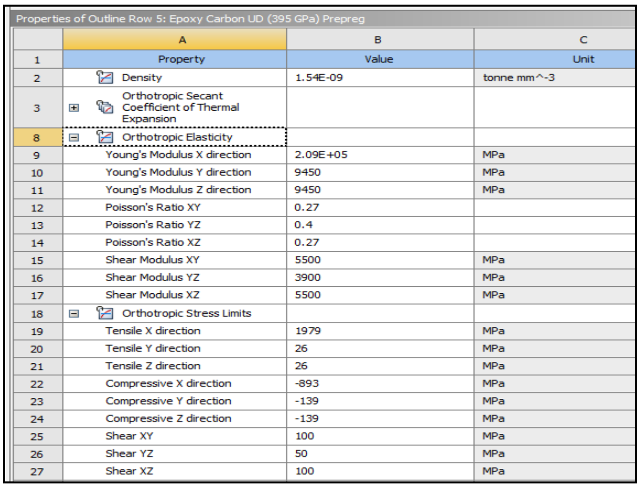
2. Results



	Mode	☑ Frequency [Hz]
1	1.	1048.2
2	2.	1734.7
3	3.	5129.4
4	4.	6395.1
5	5.	9325.4
6	6.	14364

VIII. CONNECING ROD WITH COMPOSITE MATERIAL REINFORCEMENT

Table. Material properties of carbon epoxy



Properties of Outline Row 5: Epoxy Carbon UD (395 GPa) Prepreg

	A	B	C
1	Property	Value	Unit
2	Density	1.54E-09	tonne mm^-3
3	⊞ Orthotropic Secant Coefficient of Thermal Expansion		
8	⊟ Orthotropic Elasticity		
9	Young's Modulus X direction	2.09E+05	MPa
10	Young's Modulus Y direction	9450	MPa
11	Young's Modulus Z direction	9450	MPa
12	Poisson's Ratio XY	0.27	
13	Poisson's Ratio YZ	0.4	
14	Poisson's Ratio XZ	0.27	
15	Shear Modulus XY	5500	MPa
16	Shear Modulus YZ	3900	MPa
17	Shear Modulus XZ	5500	MPa
18	⊟ Orthotropic Stress Limits		
19	Tensile X direction	1979	MPa
20	Tensile Y direction	26	MPa
21	Tensile Z direction	26	MPa
22	Compressive X direction	-893	MPa
23	Compressive Y direction	-139	MPa
24	Compressive Z direction	-139	MPa
25	Shear XY	100	MPa
26	Shear YZ	50	MPa
27	Shear XZ	100	MPa

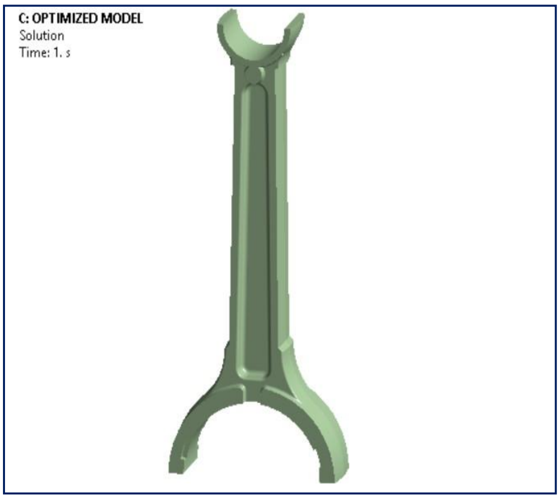
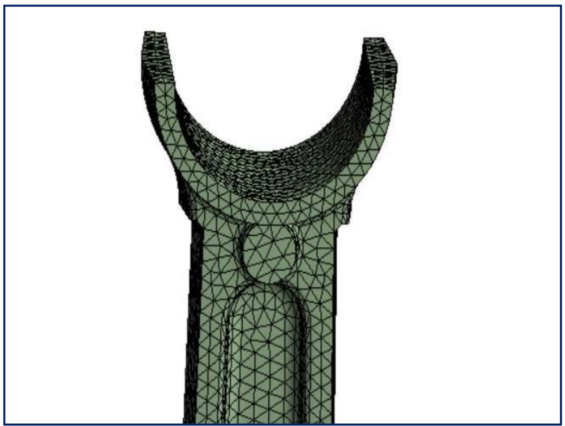
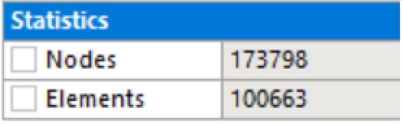


Fig. 8 Detailed view of connecting rod with carbon fibre reinforcement

Properties	
☐ Volume	1.9626e-005 m³
☐ Mass	0.1298 kg

1. Meshing





Statistics	
☐ Nodes	173798
☐ Elements	100663

Fig. 9 meshing of connecting rod with carbon fiber reinforcement

2. Boundary conditions

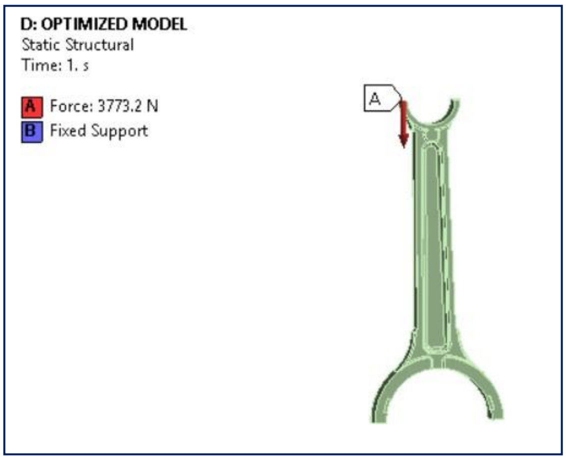


Fig. 10 Boundary condition for static analysis of connecting rod with carbon fiber reinforcement

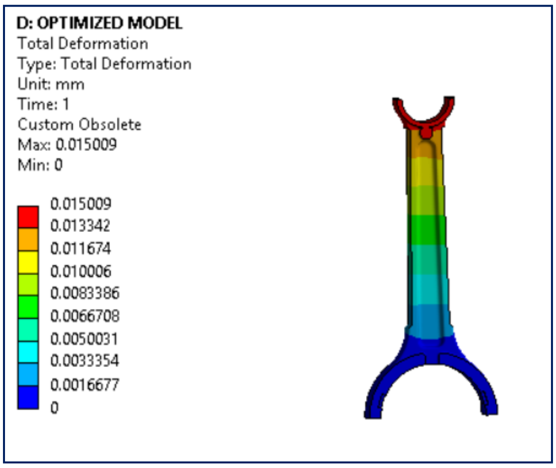


Fig. 11 Total deformation plot for connecting rod with carbon fiber reinforcement

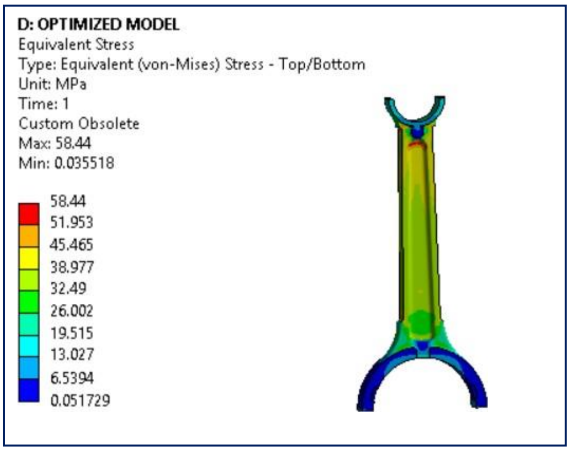


Fig. 12 Equivalent stress plot for connecting rod with carbon fiber reinforcement

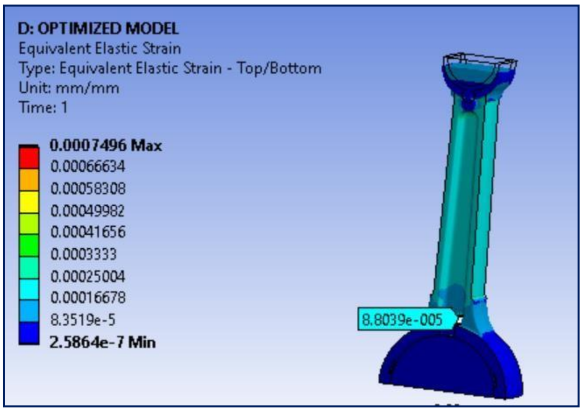
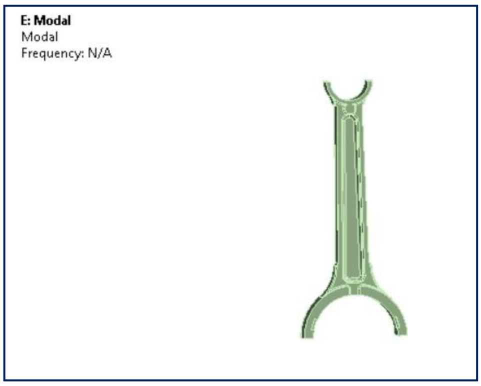


Fig. 12 Equivalent strain plot for connecting rod with carbon fiber reinforcement

IX. MODAL ANALYSIS OF CONNECING ROD WITH COMPOSITE MATERIAL REINFORCEMENT



The 6th order mode vibration frequency and vibration model are obtained by solving the solver. The following is the mode shape of the 6th order mode.

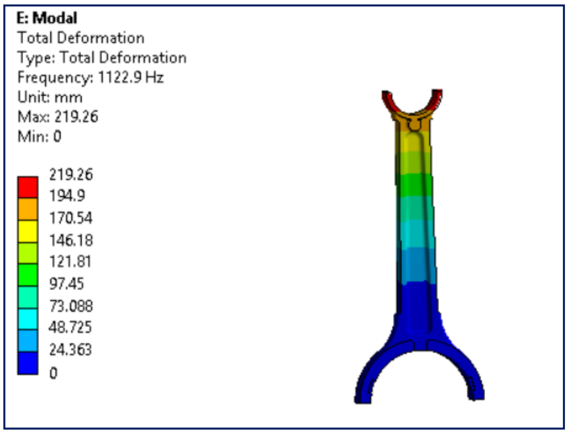


Figure 13. First mode shape

Mode 1 {Φ} 1 f1 = 1122.9 Hz

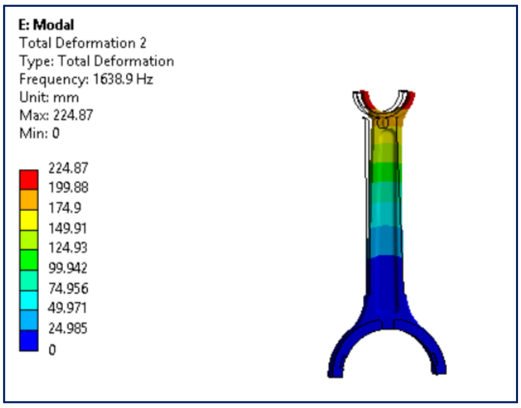


Figure 14. Second mode shape.

Mode 2 {Φ} 2 f2 = 1638.9 Hz

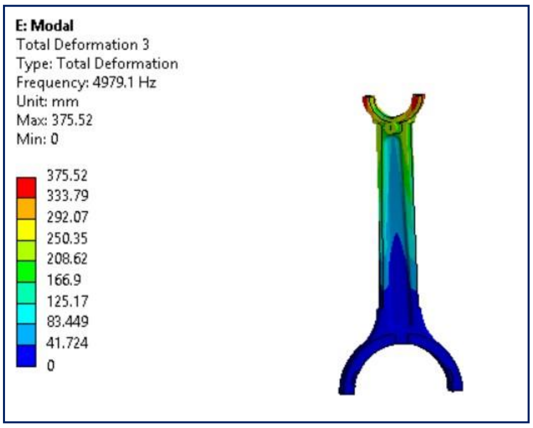


Figure 15. Third mode shape

Mode 3 {Φ} 3 f3 = 4979.1 Hz

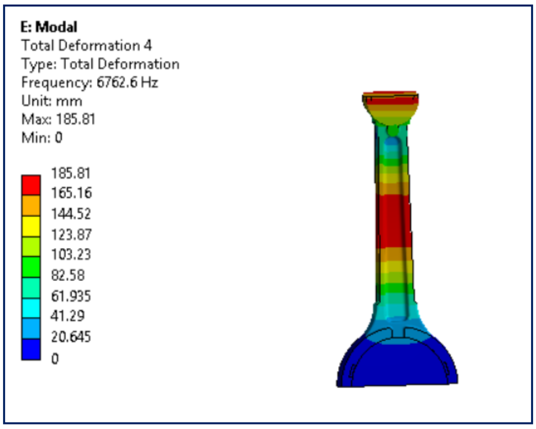


Figure 16. Fourth mode shape

Mode 4 {Φ} 4 f4 = 6762.6 Hz

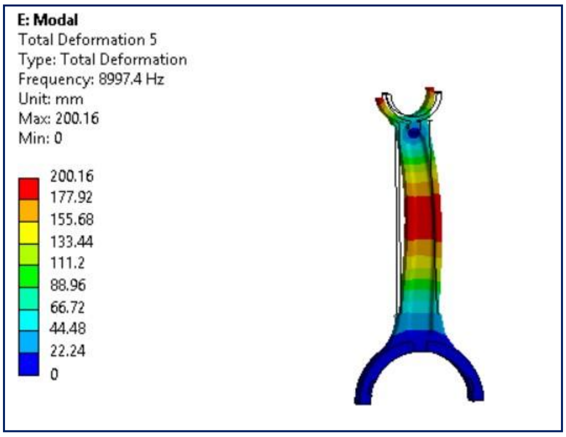


Figure 17. Fifth mode shape.

Mode 5 {Φ} 5 f5 = 64.212 Hz

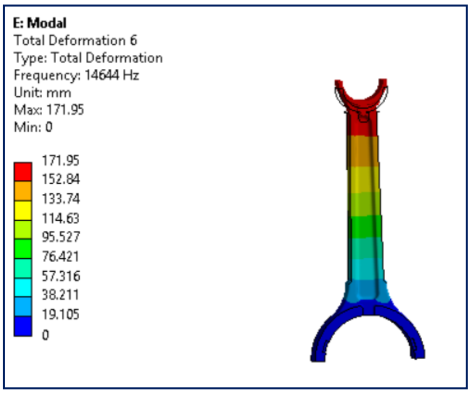
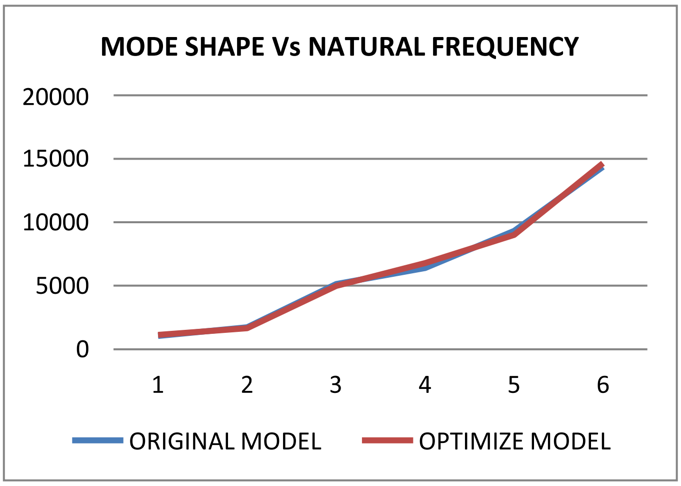


Figure 18. Sixth mode shape

Mode 6 {Φ} 6 $f6$ = 14644 Hz

FEA results of modal analysis

ORIGINAL MODEL	OPTIMIZE MODEL
1048.2	1122.9
1734.7	1638.9
5129.4	4979.1
6395.1	6762.6
9325.4	8997.4
14364	14644

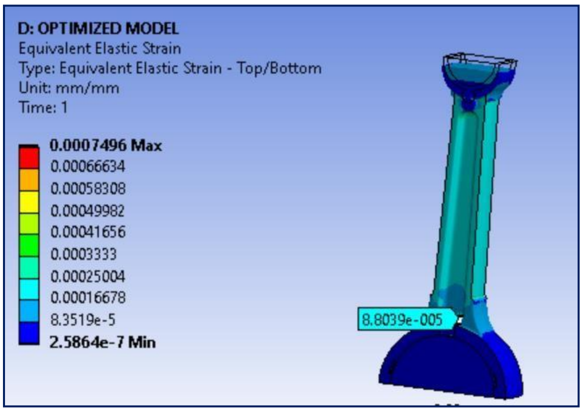


X. EXPERIMENTAL SETUP

A universal testing machine (UTM), also known as a universal tester, materials testing machine or materials test frame, is used to test the tensile strength and compressive strength of materials. The set-up and usage are detailed in a test method, often published by a standards organization.This specifies the sample preparation, fixturing, gauge length (the length which is under study or observation), analysis, etc. The specimen is placed in the machine between the grips and an extensometer if required can automatically record the change in gauge length during the test. If an extensometer is not fitted, the machine itself can record the displacement between its cross heads on which the specimen is held. However, this method not only records the change in length ofthe specimen but also all other extending / elastic components of the testing machine and its drive systems including any slipping of the specimen in the grips. Once the machine is started it begins to apply an increasing load on specimen. Throughout the tests the control system and its associatedsoftware record the load and extension or compression of the specimen.

1. Specification of UTM

1	Max Capacity	400KN
2	Measuring range	0-400KN
3	Least Count	0.04KN
4	Clearance for Tensile Test	50-700 mm
5	Clearance for Compression Test	0- 700 mm
6	Clearance Between column	500 mm
7	Ram stroke	200 mm
8	Power supply	3 Phase, 440 Volts, 50 cycle. A.C
9	Overall dimension of machine (L*W*H)	2100*800*2060
10	Weight	2300Kg



Strain is observed around 88 microns in connecting rod with carbon fiber reinforcement

Figure 19. Experimental Setup

XI. CONCLUSION

The design and static structural analysis of existing connecting rod and connecting rod with carbon fiber reinforcement has been carried out. Comparison has been made carbon fibre reinforced connecting rod with steel connecting rod having same design and same load carrying capacity.

The stress and displacements have been calculated using ANSYS for steel connecting rod and carbon fibre reinforced connecting rod. By analyzing the design, it was found that all the stresses in the connecting rod were well within the allowable limits and with good factor of safety.

From modal analysis it concluded that natural frequency of carbon fiber reinforced connecting rod higher than existing connecting rod.

Due to use of carbon fibre reinforcement on existingconnecting rod achieve 3.35 % weight optimization.

Strain measurement of connecting rod 88 microns and 92 microns by numerical and experimental testing respectively.

XII. REFERENCES

1) D.Gopinatha,Ch.V.Sushmab, “Design and Optimization of Four Wheeler Connecting Rod Using Finite Element Analysis”, 4th International Conference on Materials Processing and Characterization, ScienceDirect.

2) Mohammed Mohsin Ali Ha , Mohamed Haneef b, “Analysis of Fatigue Stresses on Connecting Rod Subjected to Concentrated Loads At The Big End”, 4th International Conference on Materials Processing and Characterization, ScienceDirect.

3) Amit Kumar, Bhingole P.P. , Dinesh Kumar, “Dynamic Analysis of Bajaj Pulsar 150cc Connecting Rod Using ANSYS 14.0”, Asian Journal of Engineering and Applied Technology

4) Pravardhan S. Shenoy, Ali Fatemi, “Connecting Rod Optimization for Weight and Cost Reduction”, SAE International

5) Anil kumar, Kamaldeep Grover, BalvinderBudania, “Optimization of Connecting Rod Parameters using CAE Tools”,

6) SatishWable, DattatrayS.Galhe, RajkumarL.Mankar, “REVIEW ON DESIGN AND OPTIMIZATION OF CONNECTING ROD”

7) Mr. Sahil, Mr. JitenSaini, “Static, Fatigue and Modal Analysis of Connecting Rod under Different Loading Conditions”, INTERNATIONAL JOURNAL OF ENGINEERING SCIENCES & RESEARCH TECHNOLOGY,

8) Sameer NasirMomin , R.J. Gawande, “Investigation of Natural Frequency by FEA Modal Analysis and Experimental Modal Analysis of Connecting Rod”, International Journal of Innovative Research in Science, Engineering and Technology, ISSN(Online)

9) Hitesh Kumar, Vijay Kumar Sharma, “Stress Analysis and Optimization of I. C. EngineConnecting Rod by Finite Element Analysis”, International Journal of Engineering Research &Technology (IJERT)

10) Gaba. Peeyush, Sethi APS, Design Evaluation of Connecting Rod using FEM for HighCycle Fatigue Strength, nternational Journal of Research in Advent Technology, Vol.2, No.5, May 2014