ISSN 2395-695X (Print)
ISSN 2395-695X (Online)

International Journal of Advanced Research in Biology Engineering Science and Technology (IJARBEST)
Vol. 2, Special Issue 13, April 2016

MODELING AND STATIC ANALYSIS ON A TWO-WHEELER CONNECTING ROD WITH DIFFERENT MATERIALS

Praveen Kumar D[1], Vedha Narayanan V[2], Vignesh R[3], Venkateshwaran S[4]

[1,2,3,4] *B.E Student, Department of Mechanical Engineering, Velammal Engg College, Chennai*

---***---

Abstract: Connecting Rod is one of the important part in an internal combustion engine. It is in an intermediate link between the piston and the crankshaft of an engine. It converts the reciprocating motion of piston in to rotary motion of the crank. The Objective of the paper is to create a model of a two wheeler connecting rod and to conduct a static analysis on the connecting rod with different materials. The Model of a two wheeler connecting rod which is used in a single cylinder 4-stroke petrol engine is developed using a Solid modeling software i.e. CATIA V5 R19. Further a Static Structural Analysis (Finite Element Analysis) is done using ANSYS Workbench 15.0 in order to determine the Von-misses stresses, Shear stress Elastic Strains and Total deformation for the given loading conditions on different materials like Structural steel, Cast Iron, Carbon Fibre. Then material suggestions are made for the connecting rod material based on the comparative results of the analysis.

***Key Words*:** Static Structural Analysis, ANSYS Workbench15, CATIA V5 R19, Connecting Rod, Finite Element Analysis, Carbon Fibre

I. INTRODUCTION

Connecting Rods are generally used in all varieties of automobile engines. Acting as an intermediate link between the piston and the crankshaft of an engine of an automobile. It is responsible for transmission the up and down motion of the piston to the crankshaft of the engine, by converting the reciprocating motion of the piston to the rotary motion of crankshaft. The small end of the connecting rod is connecting to the piston of the engine by the means of piston pin, the other end, the bigger end being connected to the crankshaft with lower end big end bearing using bolts.

A. *Stresses on the Connecting Rod*

Forces generated on the connected rod are generally by weight and combustion of fuel inside cylinder acts upon piston and then on the connecting rod, which results in both the bending and axial stresses. The major stresses induced in the connecting rod are a combination of axial and bending stresses in operation. The cylinder gas pressure (compressive only) and the reciprocating action is obtained by the inertia force (both tensile as well as compressive) are causes for axial stresses and the bending stresses are caused due to the centrifugal forces.

B. *Manufacturing methods and Importance*

The most common type of manufacturing processes is casting, forging, and powdered metallurgy. Connecting rod is subjected to a complex state of loading. It undergoes high cyclic loads of the order of 10^8 to 10^9 cycles, which range from high compressive loads due to combustion, to high tensile loads due to inertia. Therefore, durability of this component is critical importance. Due to these factors, the connecting rod has been the topic of research for different aspects such as production technology, materials, performance, simulation, fatigue etc.

II. METHODOLOGY OF WORK

The Connecting rod is modelled and analyzed under static load conditions with respect to the strain intensity, stress concentration and deformation. Firstly based on the working parameter and the vehicle is chosen and the design parameter or dimensions of the connecting rod is calculated, then the model of the connecting rod part is prepared and it is analyzed using Finite Element Method and results thus achieved will provide us the required outcome of the work done. ANSYS being an analysis system which stands for "Advanced Numerical System Simulation". It is a CAE software, which has many capabilities, ranging from simple static analysis to complex non-linear, dynamic analysis, thermal analysis, transient state analysis, etc. By using a solid modeling software (CATIA V5R19), the geometric model is created, and then the ANSYS program is used for meshing the geometry for nodes and elements. In order to obtain the desirable results at each and every point of the model, the fine meshing is done which also results in accurate results output. In this study the elements formed after meshing are tetrahedral in shape. Loads and boundary constrains in the

ISSN 2395-695X (Print)
ISSN 2395-695X (Online)
International Journal of Advanced Research in Biology Engineering Science and Technology (IJARBEST)
Vol. 2, Special Issue 13, April 2016

ANSYS WORKBENCH can be applied on the surfaces and volume as required for Structural steel, Cast iron and Carbon Fibre. Finally the results and calculation are obtained from ANSYS Workbench and the desired output results can achieved.

III. FINITE ELEMENT METHOD - ANALYSIS

The finite element method (FEM) is a numerical technique for solving problems to find out approximate solution of a problem which are described by the partial differential equations or can also be formulated as functional minimization. A principle of interest is to be represented as an assembly of finite elements. Approximating functions in the finite elements are determined in the terms of the nodal values of a physical field which is sought. FEM subdivides a whole problem or entity into numbers of smaller simpler parts, called finite elements, and solve these parts for the problems. The main advantage of FEM is that it can handle complicated boundary and geometries with very ease.

STATIC STRUCTURAL ANALYSIS (Steps):

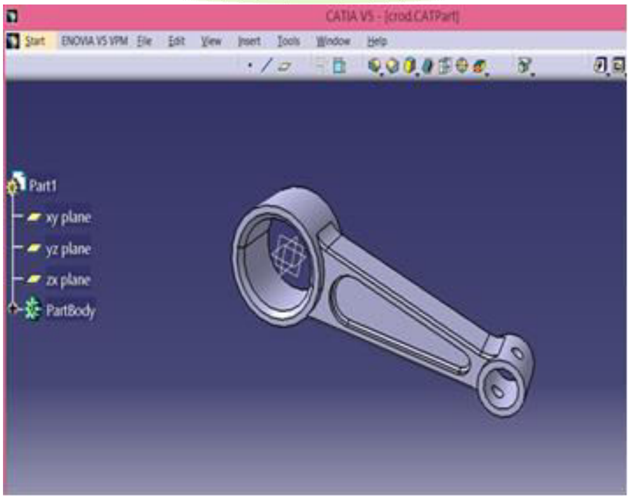
- Solid Modelling based on geometry
- Importing the model into ANSYS Workbench
- Defining element type
- Defining material properties
- Meshing of model
- Applying boundary constrains (Fixed Support)
- Applying load (Loading Conditions with direction)
- Results of Analysis
- Repeat for different Material
- Comparative Results

A. *Modeling*

Connecting rod of Hero Honda splendor, market available is selected for the present investigation. The dimensions of the selected connecting rod are found using vernier calipers, screw gauge and are tabulated in the table1. According to the dimensions the model of the connecting rod is developed using CATIAV5R19. The modeled connecting rod is shown in Fig-1. It is imported into design modeler of ANSYS WORKBENCH for analysis using three materials.

TABLE I: DIMENSIONS OF CONNECTING ROD

S.no	*Parameters*	*Values*
1.	Length of connecting rod	94.27mm
2.	Outer Diameter of Big end	39.02mm
3.	Inner Diameter of Big end	30.19mm
4.	Outer Diameter of Small end	17.75mm
5.	Inner Diameter of small end	13.02mm



ISSN 2395-695X (Print)
ISSN 2395-695X (Online)
International Journal of Advanced Research in Biology Engineering Science and Technology (IJARBEST)
Vol. 2, Special Issue 13, April 2016

Fig -1: CATIA V5 Model of a Connecting Rod

B. Materials

The material used for selected connecting rod is cast iron, Structural steel and Carbon Fibre. The properties of the material are presented in the Table 2.

TABLE - II: PROPERTIES OF CAST IRON

Material Selected	*Cast Iron*
Young's Modulus(E)	1.78e+005MPa
Poisson's Ratio	0.3
Density	7.197e-006Kg/mm^3
Tensile Strength	100 to 200 MPa
Compressive Strength	400 to 1000 MPa
Shear strength	120 MPa

TABLE - III: PROPERTIES OF STRUCTURAL STEEL

Material Selected	*Structural Steel*
Young's Modulus(E)	2.0e+005MPa
Poisson's Ratio	0.3
Density	7.85e-006Kg/mm^3
Tensile Ultimate Strength	460 MPa
Tensile Yield Strength	250 MPa
Compressive Yield Strength	200 MPa

TABLE - IV: PROPERTIES OF CARBON FIBRE

Material Selected	*Carbon Fibre*
Young's Modulus(E)	7.e+004 MPa
Poisson's Ratio	0.1
Density	1750 kg m^-3
Tensile Ultimate Strength	1600 MPa
Tensile Yield Strength	800 MPa
Compressive Yield Strength	450 MPa

C. Meshing the Model

The next stage of the analysis is to create meshing of the created model. The below said parameters in Table-5 are used for meshing. The mesh model of connecting rod is as shown in Fig- 2.

TABLE - V: MESH PARAMETERS

ISSN 2395-695X (Print)
ISSN 2395-695X (Online)
International Journal of Advanced Research in Biology Engineering Science and Technology (IJARBEST)
Vol. 2, Special Issue 13, April 2016

Types of elements	*Tetrahedron*
Nodes	9119
Elements	5146

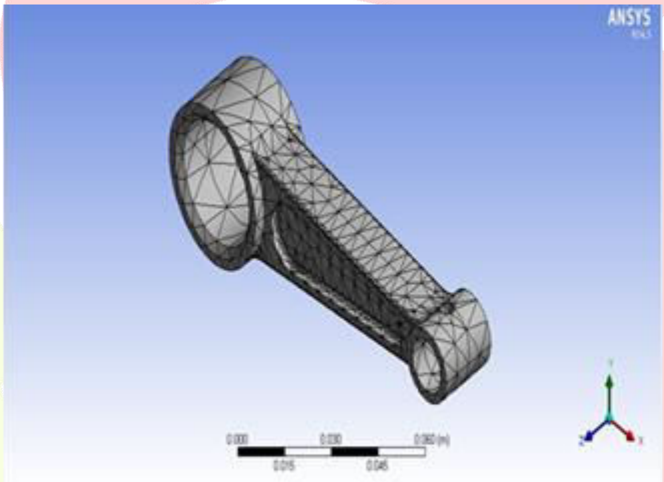


Fig -2: Meshing the model into nodes and elements

D. Load Diagram of Connecting Rod

The next step is to specify the Fixed Support and the Pressure for the Static Structural analysis. The Big end (crank end) of the connecting rod is fixed and at the Small end (piston end), a pressure of 3.15Mpa load is applied as shown in Fig-3.

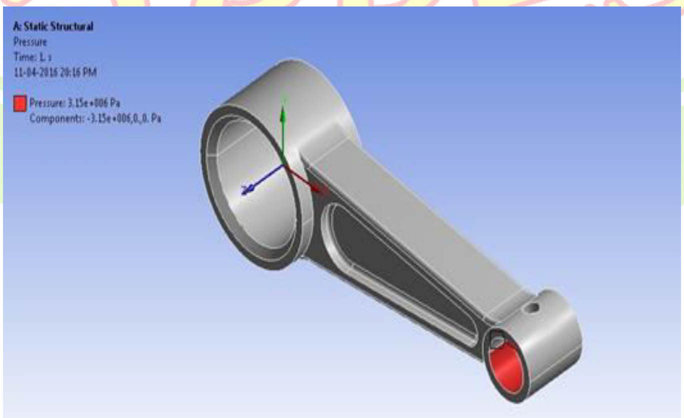


Fig -3: Meshing the model into nodes and elements

IV. RESULTS AND TABULATION

A. *Cast Iron - Results*

Fig -4: Equivalent Von-misses Stress – Cast Iron

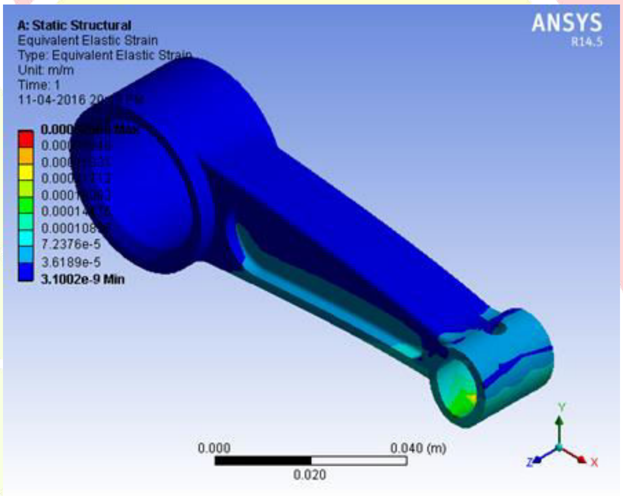


Fig -5: Equivalent Elastic Strain – Cast Iron

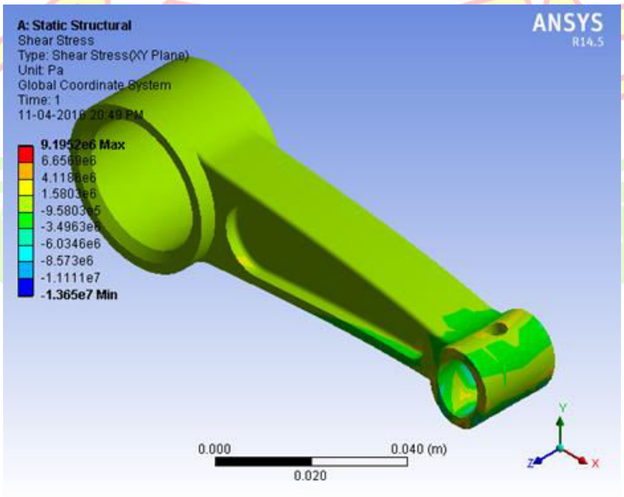


Fig -6: Shear Stress – Cast Iron

ISSN 2395-695X (Print)
ISSN 2395-695X (Online)
International Journal of Advanced Research in Biology Engineering Science and Technology (IJARBEST)
Vol. 2, Special Issue 13, April 2016

Fig -7: Total Deformation – Cast Iron

B. Structural Steel – Results

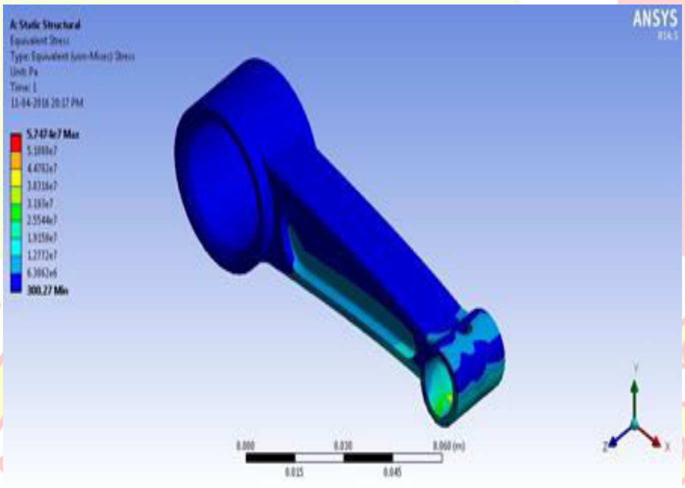
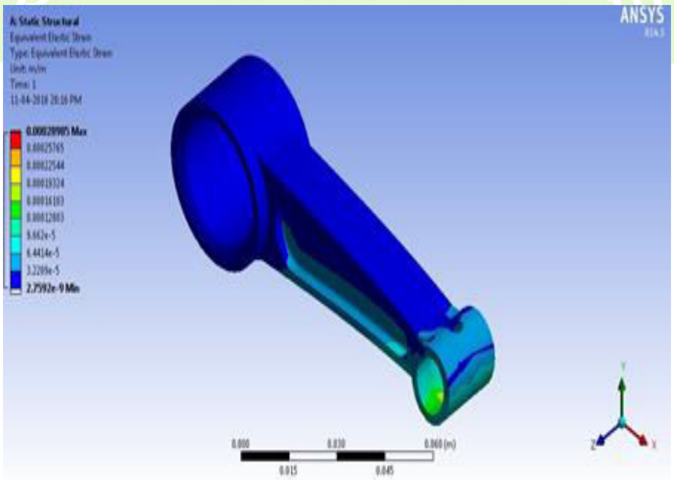


Fig -8: Equivalent Von-misses Stress – Structural Steel



ISSN 2395-695X (Print)
ISSN 2395-695X (Online)
International Journal of Advanced Research in Biology Engineering Science and Technology (IJARBEST)
Vol. 2, Special Issue 13, April 2016

Fig -9: Equivalent Elastic Strain – Structural Steel

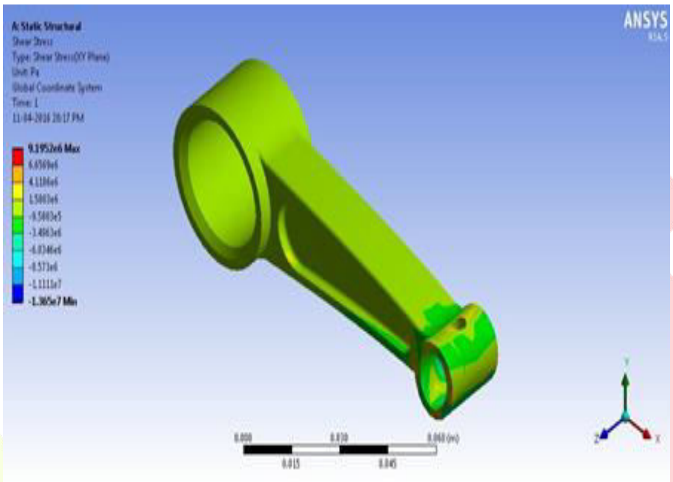


Fig -10: Shear Stress – Structural Steel

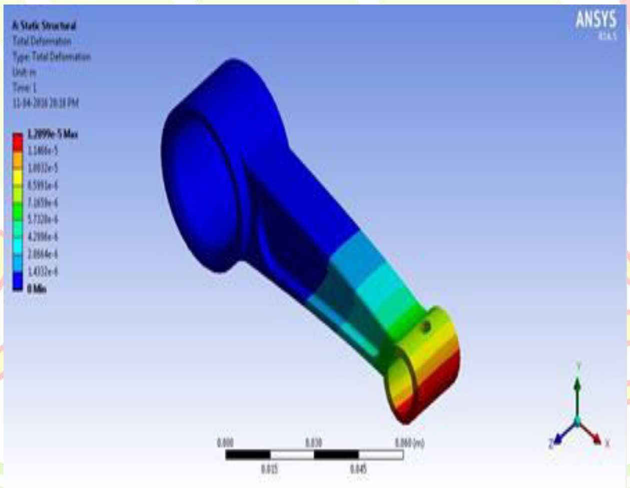
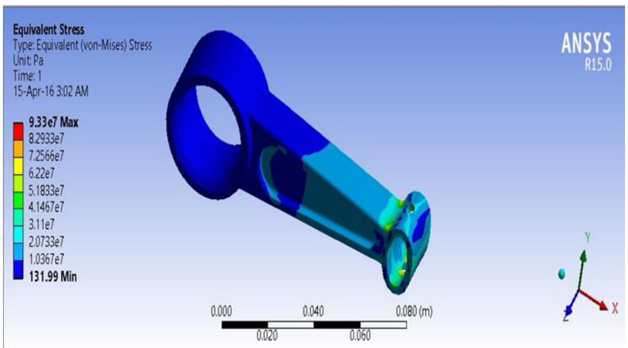


Fig -11: Total Deformation – Structured Steel

C. Carbon Fibre – Results



ISSN 2395-695X (Print)
ISSN 2395-695X (Online)
International Journal of Advanced Research in Biology Engineering Science and Technology (IJARBEST)
Vol. 2, Special Issue 13, April 2016

Fig -12: Equivalent Von-misses Stress – Carbon Fibre

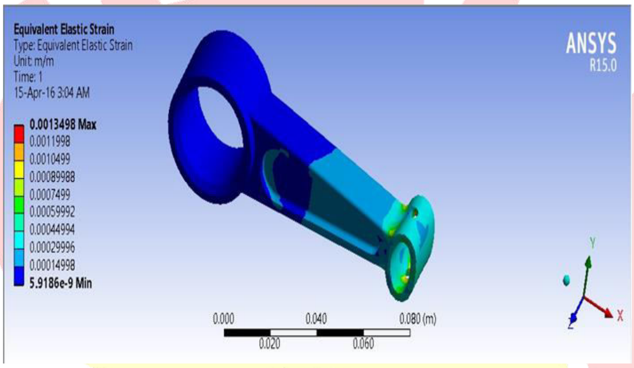


Fig -13 Equivalent Elastic Strain – Carbon Fibre

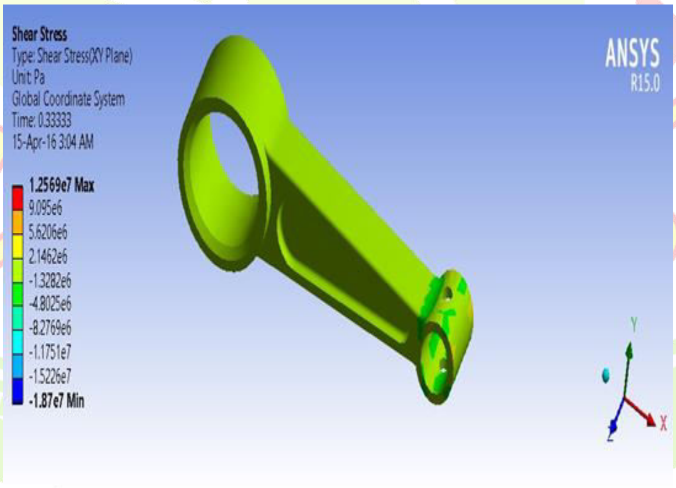
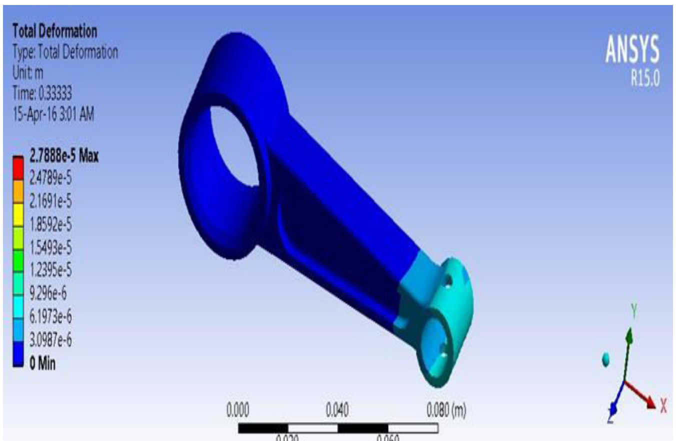


Fig -14: Shear Stress – Carbon Fibre



ISSN 2395-695X (Print)
ISSN 2395-695X (Online)
International Journal of Advanced Research in Biology Engineering Science and Technology (IJARBEST)
Vol. 2, Special Issue 13, April 2016

Fig -15: Total Deformation – Carbon Fibre

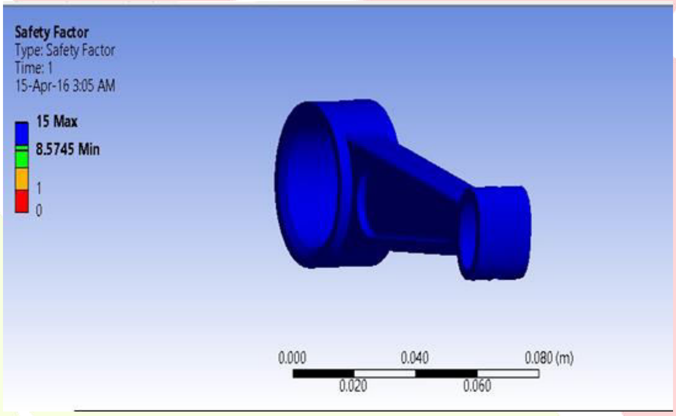


Fig -16: Safety Factor – Carbon Fibre

The Von misses Stress, Displacement, Shear Stress and Elastic Strain are tabulated in Table-4 for comparing the results.

TABLE -IV: COMPARATIVE RESULTS

S.no	Type	Cast Iron	Structural Steel	Carbon Fibre
1.	Von misses stress	91.59 MPa	82.593 MPa	9.33e+007Pa
2.	Shear stress	47.76 MPa	43.407 MPa	1.25+007 Pa
3.	Elastic strain	0.000514m/m	0.000414 m/m	0.0013498 m/m
4.	Total deformation	0.455 mm	0.456 mm	0.02788mm

V. CONCLUSION

- It is observed that the Stress and Strain Intensity induced in the Connecting Rod made up of Carbon Fibre is comparatively slightly greater than as compare to the Connecting Rod made up of Steel or cast iron. Thus with more advancement in technology in the field of Carbon Fibre, with higher Strength will be available in the near future for automobile industry.

- The factor safety of Carbon Fibre and also is found to be comparatively higher. Also lighter weight of connecting rod (made up of High Strength Carbon Fibre) can also help in reducing weight of engine block of the automobile, thus increasing fuel economy and thus also decreasing the emission from the automobile

- Hence from the above results the Carbon Fibre is suggested as a suitable material in the future but Structural steel is now preferred with the perspective of high production rates at low cost.

- It is concluded that Carbon Fibre is one of the future materials that can be used for the manufacturing of Connecting Rod, for being lighter and with comparable strength that Steel.

REFERENCES

[1] ANSYS WORKBENCH 15 manual. http://orange.engr.ucdavis.edu/Documentation12.1/121/ans_str.pdf
[2] Atish Gawale, "Non linear static finite element analysis and optimization of connecting rod", ISSN:2231-2587(2012).
[3] Anil Kumar, "Optimization of connecting rod parameters using CAE tools", International Journal of Latest Trends in Engineering and Technology, Vol- 1, Issue- 3
[4] M. RAVICHANDRAN, "Design and analysis of connecting rod by using cfrp material," J. Name Stand. Abbrev., in press.
[5] Prof. Vivek C. Pathade , Dr. Dilip S. Ingole "Stress analysis of i.c.engine connecting rod by fem and photoelasticity" IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE) e-ISSN:278-1684 Volume 6, Issue 1 (Mar. - Apr. 2013), PP 117-125.