Load Analysis and Multi Body Dynamics Analysis of Connecting Rod in Single Cylinder 4 Stroke Engine

Santhosh Reddy.A[1] Mr. Shivraj J[2] Dr.Maruthi B H[3]
[1]P.G. Student [2]Assistant Professor [3]Professor & HOD
[1,2,3]Department of Mechanical Engineering
[1,2,3]East West Institute of Technology, Bangalore

Abstract— The main objective of this thesis is performing the detailed load analysis including Multi Body Dynamics(MBD) analysis and to explore the weight reduction opportunities of connecting rod used in Bajaj Pulsar 150c.c engine. This has been carried out by considering the two cases. The first case includes the static load stress analysis to determine various stress and fatigue parameters acting on the connecting rod and Multi body dynamics analysis to find out the behaviour of the connecting rod with its associated parts while in relative motion. The second case includes performing the optimisation by looking at the possible weight reduction opportunities for connecting rod. The geometric modelling of connecting rod is done by using CATIA V R20 and saved in IGES format and imported to ANSYS workbench, using ANSYS 14.5 workbench the connecting rod analysed for various stresses by applying loads and boundary conditions by using different modules of ANSYS workbench 14.5. Finite element analysis of the connecting rod is done by considering Carbon steel material. Von misses stresses, frequencies with respect to modes; total deformation, buckling analysis, MBD analysis etc. are done. Optimization by weight reduction is carried out by providing elliptical cut out at I-section of connecting rod and analysed for various loads. Finally results are obtained for various load conditions and future scope for the project is defined.

Key words: ANSYS, FEM, CATIA, MBD, Connecting Rod

I. INTRODUCTION

Connecting rod is an important part in internal combustion engine. It acts as a linkage between piston and crank shaft. The main function of connecting rod is to transmit the reciprocating motion of piston into rotational motion of crank shaft. The connecting rod also transmits the thrust from piston to the crankshaft.

In some cases, reason for the failure of the engine is because of the failure of connecting rod. This leads us to study the behavior of connecting rod under various loading conditions. So load analysis and Multi Body Dynamics analysis is considered to know the various stresses acting and to study the behavior of connecting rod under various loading conditions.

In recent years, automobile industries are striving to achieve the fuel economy standards set by the various countries throughout the world. To achieve these stringent fuel economy and emission standards, automobile companies are looking for designing and engineering of light weight vehicles. Now a days automobile company are considering every possible option to reduce the overall weight of the vehicle to achieve the fuel economy standards set by government. Our study in this thesis mainly towards the option to reduce the weight of the connecting rod which will contribute to the reduction of overall weight of the vehicle. This promotes us to look into the option of optimisation by weight reduction of the connecting rod which will intern contribute to the overall weight reduction of the vehicle.

A. Connecting Rod Nomenclature:

A limiting factor in a gas turbine engine is the temperature of the turbine section.

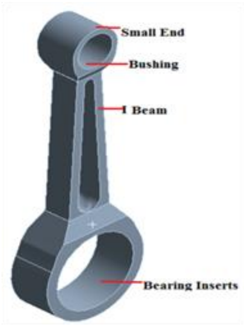


Fig. 1: Schematic representation of connecting rod

Schematic representation of connecting rod is as shown in figure1. In automobile engine, connecting rod connects the crankshaft to the piston. The connecting rod has flexibility at both ends and not fixed. This leads to change in angle between connecting rod and piston when rod makes movement up and down..

A typical connecting rod is subjected to combination of axial and bending stresses. Connecting rod is made up of I section to provide maximum rigidity along with minimum weight. Connecting rods are classified based on type of cross section i.e. I section, H section, Tabular section, circular section, In low speed applications, circular cross section is used with flat sides. In high speed applications H section or tabular section is generally used as it light weight. These types of connecting rods have big end and small end.

Failure of connecting rod is one of the important factors for engine failure. Hence connecting rod of Bajaj pulsar 150 C.C bike engine is used for the study the behavior under various loading conditions.

II. OBJECTIVE

The objective of this project to study the behavior of modified connecting rod subjected to various loading. Thus connecting rod is subjected to various types of analysis such as analytical design and analysis of Connecting Rod, Linear and bi linear analysis of Connecting Rod., Harmonic analysis of Connecting Rod, Thermal Analysis., High cycle Fatigue life analysis of Connecting Rod, Multi Body Dynamics (MBD) analysis of connecting rod.

III. SCOPE OF WORK

Before performing optimization of connecting rod, modeling of connecting rod needs to be developed by using Computer aided design (CAD) software. The structural modeling developed will be imported into the Computer aided engineering (CAE) software to carry out analysis work. Initially Meshing of the connecting rod will be done. Finite element modeling process was performed by using ANSYS. The boundary conditions and the type of loading is selected and placed at connecting rod. The Finite element analysis is carried out at connecting rod to solve the analysis equations and thus producing the result of stress, strain and displacement which will be helpful to analyze the critical area of the connecting rod. Finally Optimization will take place and the behavior of optimized connecting rod under various loadings is studied. Thus results will be used for the development of new design of connecting rod.

IV. Material Selection

Micro alloyed high carbon steels (such as C70 S6, SMA4 and FRACTIM) have been thought to be efficient different options for powder metal and conventional steel, having been utilized as main connecting rod materials. Therefore in this study C70 is the material used in the design and analysis of the connecting rod. C70 material is also referred as annealed and cold drawn C70 carbon steel. The material considered is of Isotropic nature. Its chemical composition physical properties, mechanical properties and thermal properties are represented in table (1), (2), (3) & (4) respectively.

Column1	Column2
Iron (Fe)	98.3 to 98.8 %
Manganese (Mn)	0.6 to 0.9 %
Carbon ©	0.65 to 0.75 %
Sulfur (S)	0 to 0.05 %
Phosphorus (P)	0 to 0.04 %

Table 1: Chemical composition of C70 material.

Elastic (Young's, Tensile) Modulus	210 Gpa
Elongation at Break	10%
Specific Heat Capacity	450 J/kg-K
Strength to Weight Ratio	82 kN-m/kg
Tensile Strength: Ultimate (UTS)	640 Mpa
Tensile Strength: Yield (Proof)	495 Mpa
Thermal Expansion	11.9 µm/m-K

Table 2: Mechanical properties of C70 material

Density	**7.8 g/cm3**

Table 3: Physical properties of C70 material

V. Design of Connecting Rod:

The specifications of Bajaj 150
c.c Engine are indicated in below.
Bore Diameter = 57 mm
Stroke = 58.6 mm
Displacement = 149.5 cc
Volume = 150 ×10 ^{3} mm3
Maximum Power = 13.8 bhp @ 8500 rpm
Torque = 13.4 Nm @ 6000 rpm

The dimensions of connecting rod is calculated as per design procedure followed by data hand book and cross checked by reverse engineering i.e the physical measurements of connecting rod is measured with the assistance of measuring instruments and cross verified with the designed dimensions. Diameter of big end is Dc = 43 mm Inner diameter of big end is d_g = 38 mm, Inner diameter of small end is d_i=16 mm (assumed 1 mm clearance),Outer diameter of small end Do = 22 mm

The calculated dimensions in previous section are used in the development of geometric model of connecting rod using CATIA V5 software. In this study connecting rod is designed with two aspects considering weight reduction as stated in objectives part. Out of all sections in connecting rod. I-beam section subjected to least stresses along its axis. So, connecting rod is also developed with cut out to make along I section of beam. Then developed connecting rod is subjected to various loading conditions to study the behaviour and various analysis is carried out.

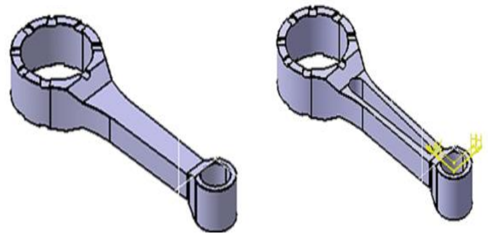


Fig. 2: Connecting Rod

Dimensions of the connecting rod with cut out are same as that of the connecting rod without cut out. Geometric modelling of the connecting rod with cut out is done using same CATIA V5 software and represented to show features and dimensions.

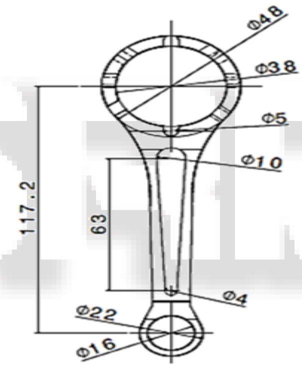


Fig. 3: Dimensions

VI. Finite Elements Model

The finite element meshed model of both before and after modified connecting rod are as shown in figure. Total number of nodes and elements here will be 454018 and 313018 respectively for connecting rod without cut out. Total number of nodes and elements for modified connecting rod will be 389061 and 262888 respectively.

Fig. 4: Finite Elements Models

A. Boundary Condition

Boundary conditions used are,

– Frictionless support at the big end and fixed all degrees of freedom.
– Downward pressure (16 Mpa) due to combustion gas load acting on small end of connecting rod.

16 MPa compression load is applied in the form of pressure, the load is applied in the Y-Axis of the Connecting Rod. The figure is shown below.

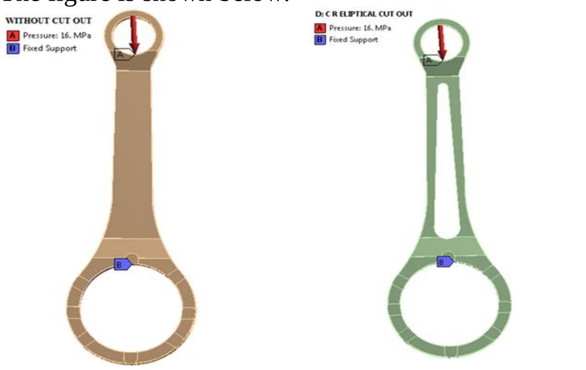


Fig. 5: Boundary Conditions

B. Results and Discussion:

1) Linear Static Analysis

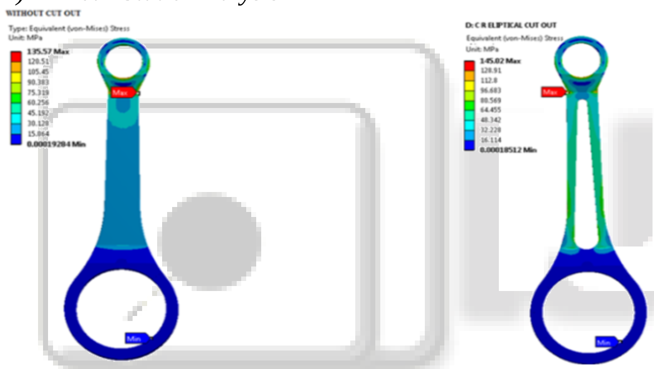


Fig. 6: Linear Static Analysis

From the above figure maximum stress obtained in connecting rod before modification is 135.57 Mpa and maximum stress value obtained after modification of connecting rod is 145.02 Mpa. The stress value obtained after modification of connecting rod 145.02 Mpa which is within the allowable yield stress of the materialli.e 420 Mpa. Hence the design is considered to be safe.

2) Bi Linear Static Analysis:

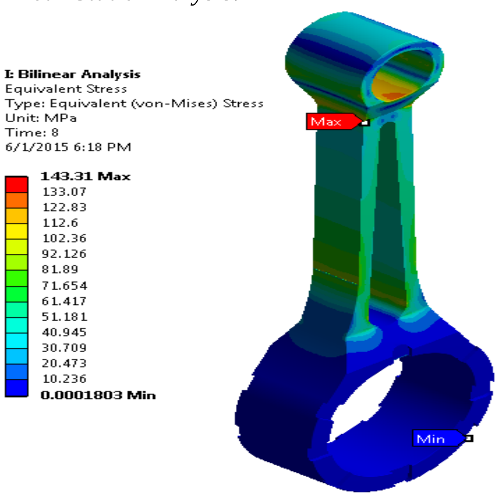


Fig. 7: Bi Linear Static Analysis

By observing the stress value, maximum stress 143.31 Mpa is obtained at the piston end of connecting rod. However we have yield strength of 495 Mpa. Hence the design is considered to be safe.

3) Modal Analysis:

a) Mode 1& 2

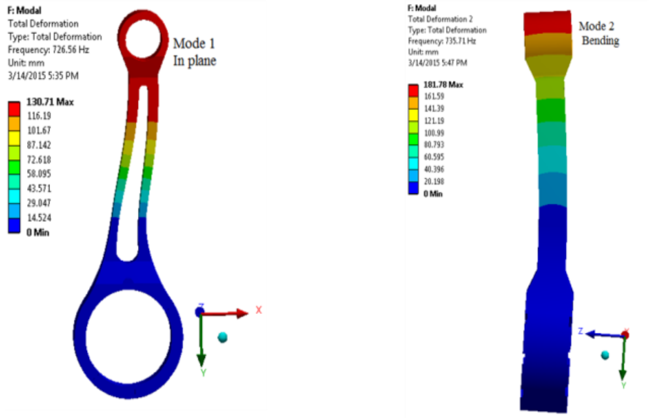


Fig. 8: Model 1 & 2

b) Mode 3 & 4:

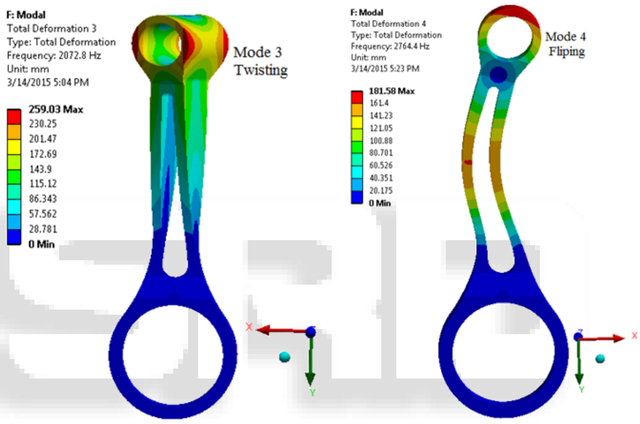


Fig. 9: Model 3 & 4

c) Mode 5 &6:

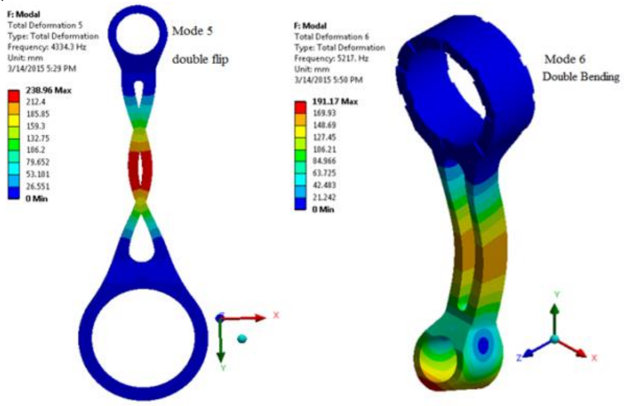


Fig. 10: Model 5 & 6

FREQUENCIES WITH RESPECT TO MODES:
Mode 1: In X-Direction: In plane
Mode 2: In Z- Direction: Bending
Mode 3: In Y- Direction: Twisting
Mode 4: In X- Direction: Flipping
Mode 5: In X-Direction: Double Flip
Mode 6: In Z-Direction: Double Bending

Mode	Frequency [Hz]
1.	726.56
2.	735.71
3.	2072.8
4.	2764.4
5.	4334.3
6.	5217.

Table 4: Frequencies with Respect To Modes

Figures show the Campbell diagram drawn for modal analysis. In which engine speed is constructed on abscissa and frequency values on ordinate. The condition for resonance in Campbell diagram is, all the three parameters such as frequency, speed and modes must coincide at a point. But in this study we verified for modes up to 45X, and we are not come across the resonance. Hence with this we conclude that as per modal analysis the connecting rod design is safe.

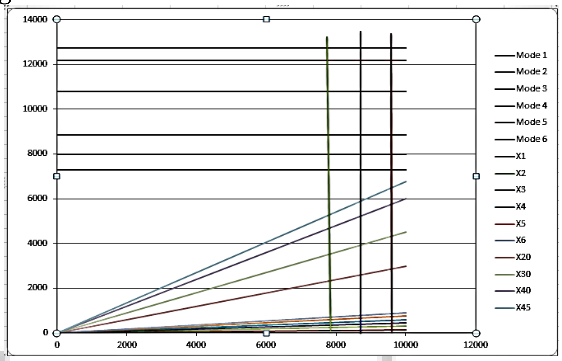


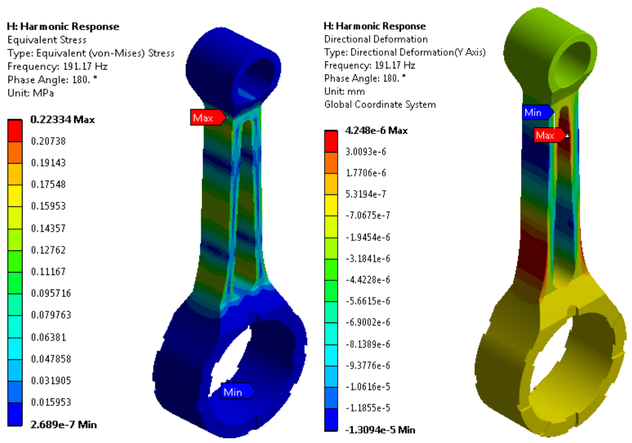
Fig. 11: Campbell diagram

4) *Harmonic Analysis:*

Harmonic analysis is done by using stimulus approach. Here the load considered is 0.16 Mpa, i.e. 1% of combustion gas pressure (16Mpa). This is because 1% of gas load is enough to excite the frequency. Below figure shows the results of harmonic analysis under various conditions.

a) Stress Tensor Plot for Piston in Harmonic Analysis:
The figure shows that the stress tensor obtained when the harmonic analysis is performed by applying definite boundary conditions. The vibratory stress resulted is peak is 0.22334 Mpa. The limiting stress for us is only 4Mpa. Hence it is safe.

b) Directional Deformation of Connecting Rod in Harmonic Analysis:
The figure 6.12 shows the directional deformation obtained when the harmonic analysis is performed by applying definite boundary conditions. Maximum displacement observed is 0.0000004248 mm.



(a) (b)
Fig. 12: Harmonic Analysis

5) *Fatigue Analysis:*

The fatigue life of connecting rod obtained is $1e^6$ cycles.

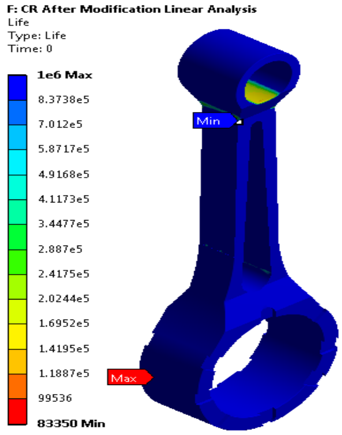


Fig. 13: Fatigue Analysis

This value of fatigue life obtained through hand calculation is order of 106 cycles which is closer to obtained value in ANSYS. Hence it is safe.

6) *Thermal Analysis of Modified Connecting Rod:*

Behavior of modified connecting rod under temperature variation is studied under this section. The below figure shows the boundary conditions and temperature distribution for thermal analysis of modified connecting rod at steady state thermal condition.

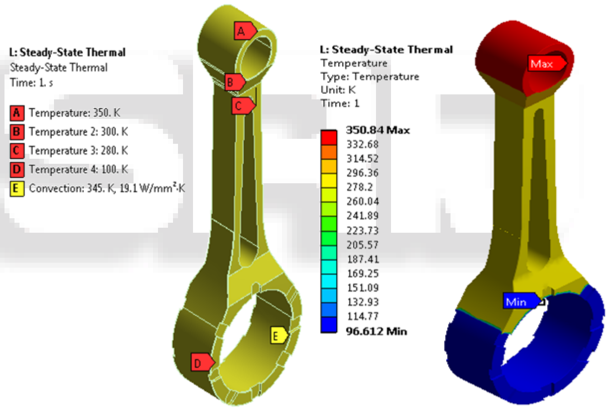


Fig. 14: Thermal Analysis

7) *Multi Body Dynamic Analysis:*

Finite element model and meshing for multi body dynamic analysis of connecting rod is as shown in the figure.

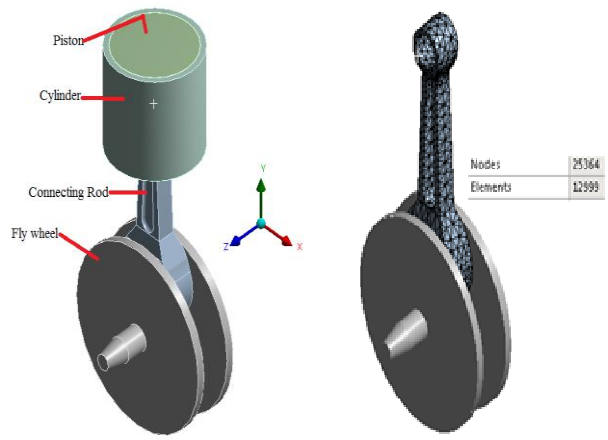


Fig. 15: Multi Body Dynamic Analysis

Total number of nodes and elements for modified connecting rod will be 25364 and 12999 respectively.

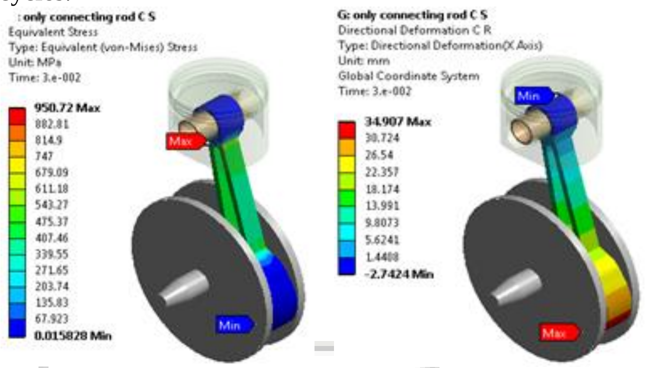
a) Boundary conditions:

Load acting on the piston is given as 16Mpa and rotational velocity at crankshaft is 8500 rpm. Fly wheel is subjected to rotational velocity of (Step load of 2000 -8500) 8500 RPM due to the pressure on the piston head.

- Results: Equivalent Stresses: During the transverse motion of the assembly the different components subjected to different stresses where the connecting rod is subjected to a stress nearer to big end of the connecting rod of 950.72 MPa.

b) The deformation of the connecting rod during the reciprocating motion of the assembly where the connecting rod deforms 34.907 mm at smaller end.

c) Fatigue life of the connecting rod obtained is $1e^6$ cycles.



(a) (b)

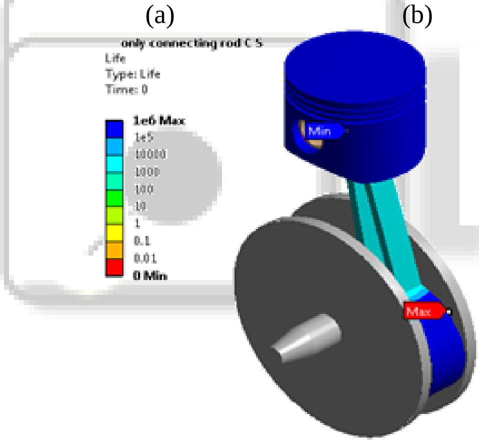


Fig. 16: Fatigue Life

VII. Conclusion

Static analysis is done on connecting rod and design of baseline model of connect ting rod is modified and obtained results well within the permissible values.

Linear and bilinear analysis is carried out on connecting rod and verified for assumed preliminary design consideration.

Modal analysis of connecting rod is performed and with the Campbell diagram, design of connecting rod is verified and considered as safe.

Harmonic analysis is carried away by following modal analysis, using the frequency values for different mode shapes which are obtained in modal analysis and verified that the component is safe at different frequency levels.

With the prediction of high cycle fatigue life, the life of connecting rod is evaluated and obtained a satisfactory result of 1e6 cycles.

MBD analysis is carried out and analyzed the results about critical stress regions as well as total deformation of the component in the assembly.

References

[1] Christy V vazhappilly et al "Stress analysis of connecting rod for weight reduction- A review", International Journal of scientific and research publications, volume 3, issue 2, ISSN 225 -3153.

[2] K. Sudarshan Kumar et al "Modeling and analysis of two wheeler connecting rod", International journal of modern engineering research (IJMER), volume 2, issue 5, ISSN; 2249-6645.

[3] Pravardan S. Shenoy et al "Dynamic load analysis and optimization of connecting rod.

