IJCRT.ORG ISSN : 2320-2882

INTERNATIONAL JOURNAL OF CREATIVE RESEARCH THOUGHTS (IJCRT)
An International Open Access, Peer-reviewed, Refereed Journal

MULTI BODY DYNAMICS ANALYSIS OF CONNECTING ROD IN SINGLE CYLINDER 4 STROKE ENGINE

Raj Kumar K[1], Dr. Guruprasad H L[2],Dr. Maruthi B H[3]
1 PG Student, 2 Associate Professor, 3 Professor & HOD
Deportment of Mechanical Engineering
East West Institute Of Technology, Bengaluru-91,
Karnataka, India

Abstract- The main objective is performing the static and dynamic load analysis, and to explore the weight reduction opportunity of connecting rod. This has been carried out by two cases. The first case includes the static load stress analysis and material optimization by looking at the possible weight reduction of connecting rod. Second case includes dynamic load mode frequencies. The geometric modeling of connecting rod is done by using CATIA. And imported to Ansys work bench. The connecting rod analyzed for various stress by applying load and boundary condition. Finite element analysis of connecting rod is done three materials, finally results are obtained for applied load of connecting rod to check the strength and reduction of weight through material optimization of connecting rod.

Keywords- Ansys, FEM, Material optimization, connecting rod.

I. INTRODUCTION

The automobile engine connecting rod is a high volume production, critical component. It connects reciprocating piston to rotating crankshaft, transmitting the thrust of the piston to the crankshaft. Every vehicle that uses an internal combustion engine requires at least one connecting rod depending upon the number of cylinders in the engine. Connecting rods for automotive applications are typically manufactured by forging from either wrought steel or powdered metal. They could also be cast. However, the cost of the blank is high due to the high material cost and sophisticated manufacturing techniques. With steel forging, the material is inexpensive and the rough part manufacturing process is cost effective. Due to its large volume production, it is only logical that optimization of the connecting rod for its weight or volume will result in large scale savings. It can also achieve the objective of reducing the weight of the engine component, thus reducing inertia loads, reducing engine weight and improving engine performance and fuel economy. A Connecting rod is the link between the reciprocating piston and rotating crank shaft.

The function of connecting rod is to translate the transverse motion to rotational motion. It is a part of the engine, which is subjected to millions of repetitive cyclic loadings. It should be strong enough to remain rigid under loading, and also be light enough to reduce the inertia forces which are produced when the rod and piston stop, change directions and start again at the end of each stroke. The connecting rod should be designed with high reliability. It must be capable of transmitting axial tension, axial compression, and bending stresses.

At first the theoretical study of connecting rod is done. Here the model created using 3- Dimensional software is subjected to stress conditions, strain displacements. The three Dimensional created using CATIA is imported to FE software ANSYS and is meshed and that model is called a Finite elemental model. The meshed model is subjected to certain boundary conditions and sensitivity analysis is done using Ansys. It is done in three different steps, 1) Linear static analysis, 2) Dynamic analysis, 3) Life estimation.

To do static structural and modal analysis of engine mounting bracket for three different materials viz. aluminum (Al), Magnesium (Mg) and Cast Iron (CI) and suggest best material for the bracket.

II. LITERATURE REVIEW

Folgar et al [1] Developed a fiber FP/Metal matrix composite connecting rod with the aid of FEA, and loads obtained from kinematic analysis. Fatigue was not addressed at the design stage. However, prototypes were fatigue tested. The investigators identified design loads in terms of maximum engine speed, and loads at the crank and piston pin ends. They performed static tests in which the crank end and the piston pin end failed at different loads.

Balasubramaniam et al [2] Reported computational strategy used in Mercedes Benz using examples of engine components. In their opinion, 2D FE models can be used to obtain rapid trend statements, and 3D FE models for more accurate investigation. The various individual loads acting on the connecting rod were used for performing simulation and actual stress distribution was obtained by superposition. The loads included inertia load, firing load, the press fit of the bearing shell, and the bolt forces.

Ishida et al [3] Measured the stress variation at the column center and column bottom of the connecting rod, as well as the bending stress at the column center.1.5 and 1.6 indicate that at the higher engine speeds, the peak tensile stress does not occur at 360 degree crank angle or top dead center. It was also observed that the R ratio varies with location, and at a given location it also varies with the engine speed. The maximum bending stress magnitude over the entire cycle (0 degree to 720 degree crank angle) at 12000 rev/min, at the column center was found to be about 25% of the peak tensile stress over the same cycle.

Prateek Joshi et al [4] conducted FEM Analysis of Connecting Rod of different materials using ANSYS. In this work High Strength Carbon Fiber connecting rod was compared with connecting rod made up of Stainless Steel. The results were used for optimization for weight reduction and for design modification of the connecting rod. Pro-E software was used for modeling and analyses were carried out in ANSYS software. The results archived helped us identify the spot or section where chances of failure were high due to stress induced.

III. FE MODELING OF THE CONNECTING ROD

The FEM is a method of piece insightful figure in which the structure or body is disconnected into little FE estimations called finite elements and a conservative cross later the standard body or the structure is considered as a social affair of these elements related at finite number of joints called nodal focuses or center focus interests. Since the true blue mix of field elements like dislodging, stress, temperature, weight or speed inside the continuum are not known, the party of the field variable inside a finite element can be approximated by an essential inspiration driving confinement. These estimation limits called expansion models are portrayed in like route as the estimations of the field parts of the inside interests. The nodal estimations of the field variable are acquired by understanding the field conditions, which are all around as structure conditions. methodologies of general continuum issues by the finite element method constantly take after a benefit especially requested process.

Procedural steps in FEM

- Modeling
- Description of Continuum
- Selection of proper interpolation model
- Derivation of element stiffness matrix
- Assemblage of element equations to obtain the equilibrium equations
- Enforcing the boundary condition
- Solution of system equation to find nodal values of displacement
- Computation of element strains and stresses

3.1 ENGINE SPECIFICATION AND CALCULATIONS

Engine Capacity: 109.2 cc
Power developed: 5.91kw @ 700rpm
Torque developed: 8.94 Nm @ 5500 rpm
Stroke: 55.6 mm
Bore: 50 mm
Compression ratio: 9.5:1
Density of petrol: =737.22kg/m3=737.22E-9 kg/mm3
Flash point for petrol (Gasoline) Flash point = -43°c (-45°F) Auto ignition temp. = 280°c (536°F) = 553°k Mass = Density x volume = 737.22E-9 x 109.1E3 = 0.08kg Molecular weight of petrol = 114.228g/mole = 0.11423 kg/mole
From gas equation, PV=m * R specific * T Where, P = Pressure, MPa V = Volume m = Mass, kg R specific = Specific gas constant T = Temperature, °k R specific = R/M R specific = 8.3143/0.114228 R specific = 72.76 Nm/kg K
P = m.R specific.T/V P = (0.08 x 72.757 x 553) / 109.2e3 = 29.4771 MPa

IV. FINITE ELEMENT ANALYSIS

Fig 4.1: Connecting Rod with Cut out.

The above figure 4.1 shows the Meshed Model of connecting rod, has the Maximum number of elements (262888)

4.1 CASE 1: STATIC ANALYSIS OF STAINLESS STEEL

Table 4.1: Mechanical properties of Stainless steel

PROPERTIES	MPa
Tensile Strength	105
Yield Strength	55
Elongation	55
Rockwell	90
Brinell	185

Table 4.2: Stainless steel composition

MATERIAL	In %
Carbon	0.12
Manganese	7.5/10
Silicone	0.9
Chromium	14/16
Nickel	0.5/2.0
Molybdenum	0.2
Phosphorus	0.06
Nitrogen	0.25

4.1.1 Total Deformation

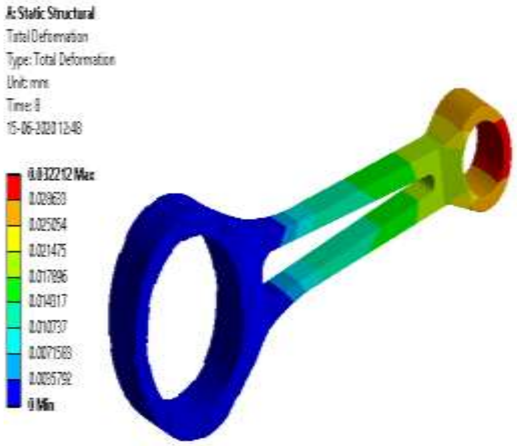

Fig 4.2: Total deformation

The above figure 4.2 shows total deformation obtained is 0.032212 mm for the applied load condition and maximum stress is less than the yield stress.

4.2 CASE 2: STATIC ANALYSIS OF ALUMINIUM 360

Table 4.3: Aluminium 360 Compositions

ALLOY COMPOSITION	In %
Silicon	9-10
Iron	1.3
Copper	0.6
Manganese	0.35
Magnesium	0.4-0.6
Nickel	0.50
Zinc	0.50
Tin	0.15
Others	0.25
Aluminium	Balance

Table 4.4: Mechanical Properties of Aluminium

Tensile Strength (ksi)	46
Yield Strength (ksi)	24
Elongation (%)	3.5
Hardness (HB)	75
Shear Strength (ksi)	26
Fatigue Strength (ksi)	18
Density	0.095

4.2.1 Total Deformation

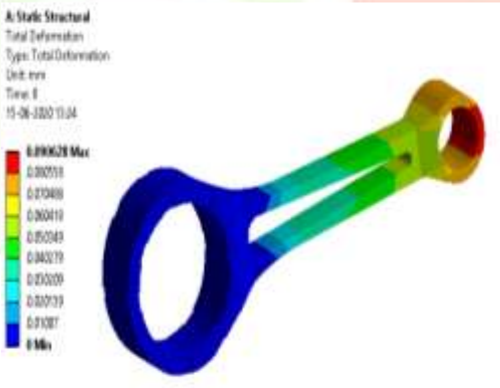

Fig 4.3: Total deformation

In the above figure 4.3 Total deformation obtained is 0.090079 mm for the applied load condition and maximum stress is less than the yield stress.

4.3 CASE 3: STATIC ANALYSIS OF E- GLASS EPOXY

Table 4.5: Mechanical Properties

Compressive strength-longitudinal (Mpa)	300
Compressive strength –transverse (Mpa)	415
Density (g cm-3)	1.9
Tensile strength-longitudinal (Mpa)	490

4.3.1 Total Deformation

Fig 4.4: Total deformation

Fig shows Total deformation in the connecting rod is 0.088329 mm for the applied load condition and maximum stress is less then the yield stress

Table 4.6 : weight and Stress of the Component

	Weight KG	Total deformation mm	Equivalent Stress MPa	Maximum stress Mpa	Minimum stress Mpa
Aluminium 360	1.201	0.09008	142.9	120.22	4.7009
Stainless Steel	1.5	0.0332	145.02	120.4	4.868
E Glass	0.7215	0.08833	143.88	127.34	5.7023

V. DYNAMIC ANALYSIS OF E-GLASS EPOXY

Dynamic analysis is used to analyze the structural vibration problems and damping effect. These processes include defining the vibration sensitivity. This can be used for relatively simple basic concept or complicated mechanical device with periodic loading. These systems will determine the natural frequencies and mode shape using technique finite element analysis.

VI. RESULT AND OBSERVATION

6.1 Six Modes and Corresponding Natural Frequency

Fig 6.1: First mode of Connecting rod

The above figure 6.1 shows First mode of Connecting rod and corresponding frequency is 677.56 Hz for the applied dynamic load condition.

Fig 6.2: Sixth mode of Connecting rod

The above figure 6.2 shows Sixth mode of connecting rod and corresponding frequency is 5203.7 Hz for the applied dynamic load condition.

Table 6.1: Six modes of frequencies

	Mode	Natural frequency (Hz)
01	1	677.56
02	2	747.47
03	3	2051.4
04	4	2700.0
05	5	4260.6
06	6	5203.7

VII. FATIGUE ANALYSIS

In materials science, fatigue is the wearying of a material caused by recurrently applied loads. It is the broad-minded and restricted structural damage that occurs when a material is subjected to cyclic loading. The nominal maximum stress values that cause such damage may be much less than the strength of the material classically quoted as the ultimate tensile stress limit, or the yield stress limit.

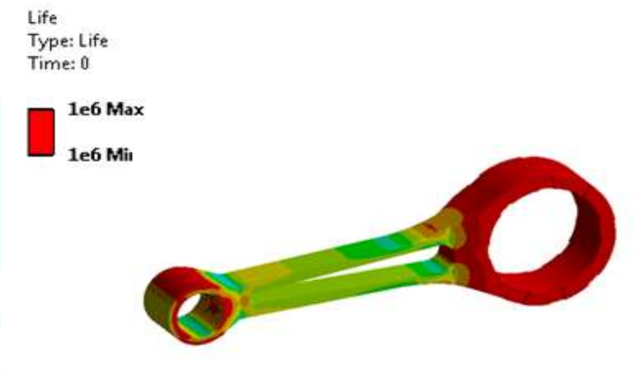

Fig 7.1: Fatigue analysis of Connecting rod

In the above figure 7.1 fatigue life estimation of connecting rod, it life estimated up to 1000000 cycles for the repeat load condition.

Goodman diagram

Mean Stress can be calculated from,

$$\sigma_{mean} = \frac{\sigma_{von}}{2}$$

Where

σ_{von} = Equivalent von-Misses Stress

$$\sigma_{mean} = \frac{\sigma_1 + \sigma_2}{2}$$

$$\sigma_{mean} = 66.32 \text{ MPa}$$

$$\sigma_a = \frac{\sigma_1 - \sigma_2}{2}$$

= 60.81 MPa

Where

σ_1= Maximum Principal Stress

σ_2= Minimum Principal Stress

Number of cycles:

$$N_f = \left\{ \frac{\left[\sigma_{ult} - \sigma_{ult}\left(\frac{1}{fos} - \frac{\sigma_a}{\sigma_e}\right)\right]}{\sigma_a} \right\}^{\frac{1}{0.08}}$$

Where,

Nf=Fatigue life σe=Endurance limit

σutl=Ultimate stress σa=Alternating stress

Fos=Factor of Safety

$$N_f = 1.22 \times 10^6$$

The resulting life of failure obtained by analytical method is greater than Cycles hence it is high cycle fatigue.

CONCLUSION

The free model analysis and static structural analysis of a verbalized urban connecting rod, with an aggregate length of 0.66 m, has been performed through Global Finite Elements Method. The structure behaviour towards four distinctive loading conditions, illustrative of its run of the mill duty cycle, has been broke down: the activity of gravitational acceleration, the braking at the upper deceleration limit of the vehicle,2g load and impact load condition to acquired stress, strain and displacement. Sensitivity investigations in order to assess the connecting rod performances have been done in order to get dependable outcomes in terms of stiffness and displacements of the connecting rod.

Linear static structural analysis has been carried out to estimate the maximum stress, strain and deformation in bus body. It is found that peak stress of 66.32 Mpa, total deformation of 0.088329 mm is obtained.

Fatigue analysis of connecting rod was carried out for 1000000 cycles of startup and shutdown, the fatigue life results obtained is more than 1000000 cycles, hence the design is safe.

Weight of 0.72154 kg in connecting rod which is Optimization of connecting rod to increase the life and efficiency**.**

REFERENCES

[1] Afzal, A., 2004, “Fatigue Behavior and Life prediction of Forged Steel and PM Connecting Rods,” Master’s Thesis, University of Toledo.
[2] 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.
[3] Balasubramaniam, B., Svoboda, M., and Bauer, W., 1991, “Structural optimization of I.C. engines subjected to mechanical and thermal loads,” Computer Methods in Applied Mechanics and Engineering, Vol. 89, pp. 337-360.
[4] Bhandari, V. B., 1994, “Design of Machine Elements,” Tata McGraw-Hill.
[5] Clark, J. P., Field III, F. R., and Nallicheri, N. V., 1989, “Engine state-of-the-art a competitive assessment of steel, cost estimates and performance analysis,” Research Report BR 89-1, Automotive Applications Committee, American Iron and Steel Institute.
[6] El-Sayed, M. E. M., and Lund, E. H., 1990, “Structural optimization with fatigue life constraints,” Engineering Fracture Mechanics, Vol. 37, No. 6, pp. 1149-1156.