Research Paper

Volume : 2 | Issue : 6 | June 2013 • ISSN No 2277 - 8179

Static Structure Analysis of Diesel Truck Engine Connecting Rod

Engineering

KEYWORDS : - Static structure analysis, Connecting rod, Stress analysis

Marmik M Dave | Lecturer, School of Diploma studies, RK University Rajkot, Gujarat, India

Manish P Vekariya | Asst. Technical Manager – Khodiyar CAD Center (I) Pvt. Ltd, Rajkot Gujarat, India

ABSTRACT

In recent years necessitates a detailed understanding of the applied loads and resulting stresses under in-service conditions of automobile components. The connecting rod is the most important parts used widely in various engines. Connecting rods transfer energy from pistons to crankshafts and convert the linear, reciprocating motion of a piston into the rotary motion of a crankshaft. In this paper, the connecting rod finite element analysis is proposed in ANSYS 14, and Critical location of Maximum Stress, Maximum Strain and total Deformation with Von-meshes criteria for the connecting rod has been identified. The structural model of the connecting rod was prepared by utilizing the (PRO-E Creo) and material is C 70 S6.

INTRODUCTION

Internal combustion engine connecting rod is a high volume production critical component. The Connecting Rod is the intermediate member between the piston and the crankshaft. Its primary function is to transmit the push and pull from the piston pin to the crankpin and thus converts the reciprocating motion of the piston into the rotary motion of the crank [1]. In the automobile industry the environment demands of lower emissions, lower fuel consumption, longer service intervals and longer lifecycles are growing tougher. The Connecting rod undergoes high cyclic loads of the order of 108–109 cycles, which range from high compressive loads because of combustion, to high tensile loads because of inertia. Therefore, durability of this component is of critical importance [2].

In 1983 Webster et al. performed three dimensional finite element analysis of a high-speed diesel engine connecting rod. For this analysis they used the maximum compressive load which was measured experimentally, and the maximum tensile load which is essentially the inertia load of the piston assembly

Mass. The load distributions on the piston pin end and crank end were determined experimentally. They modeled the connecting rod cap separately, and also modeled the bolt pretension using beam elements and multi point constraint equations [3].

Here, diesel engine Connecting Rod is made up from the C70 S6 material which is one of the ductile materials, so we apply von-meshes criteria. The solid model is Prepared in PRO-E Creo and For the purpose of static structural analysis we are using ANSYS 14 workbench. By using this software we show the maximum stress, maximum strain and total deformation.

MATERIAL:

- Material : C 70 S6
- Density : 7.85 g/cubic cm
- Young's Modulus : 210000 MPa
- Poisson's Ratio : 0.30
- Yield Strength : 550 MPa
- Compressive Strength : 600 MPa
- Ultimate Tensile Strength : 900 MPa

Parameters:

- Engine type: Turbo Chat (TC)
- Connecting rod Big End dia.= 45.128mm
- Connecting rod Small End dia.= 23.939mm
- Bore = 105 mm
- Connecting rod CD=133.55mm
- Stroke = 137 mm
- Pick Comustion Pressure = 18 (MPa)
- Mass of Connecting Rod = 0.285kg

MATHEMATICAL CALCULATIONS:

Engine Load, PA = $(\pi/4) \times (d^2) \times (P)$

Where, d = Bore Diameter of Engine in cm
P = Pick Combustion Pressure of Chamber in BAR
So, PA = $(\pi/4)*(d^2)*(P)$ Kgf
= $(\pi/4) \times (10.5)^2 \times (180)$ Kgf
= 15586.226 Kgf
= 15,588 Kgf
So, PA = 1, 52,866 N

ANALYSIS OF CONNECTING ROD:

For the purpose of Finite Element Analysis we are using the ANSYS WORKBENCH 14.0. For the purpose of analysis we follow the step by step procedure given below and generate the maximum stress, strain and total deformation of Connecting Rod under the loading conditions.

1, Draw the geometry of Connecting Rod in PRO-E 5.0/ Creo.
2, Generate Mesh in ANSYS Workbench. For any finite element analysis the total work piece divided in to finite number of small element.

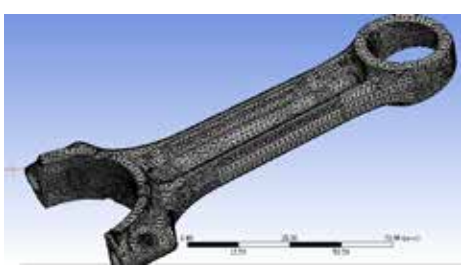

Figure :1 Mesh of Connecting Rod

Size of elements = 1mm.
Number of nodes = 150226
Element Type: Tetrahedron

3, After Generate Mesh in ANSYS Workbench. Applying displacement, force conditions on Connecting rod, and the Force Application region is Inner 110 degree surface from Crank Pin Side towards Piston Pin Side.



Figure:2 Force and displacement conditions

4, For solution select Von-mises theory and apply total deformation, maximum stress and maximum strain.
5, Solution for Connecting Rod

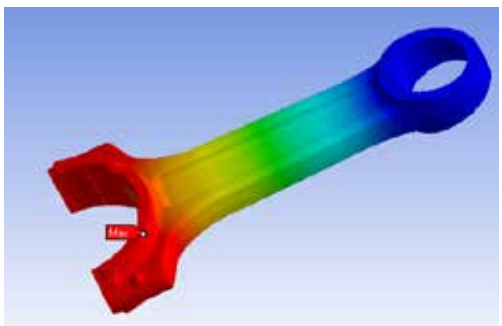

Figure: 3 (A) Total deformation

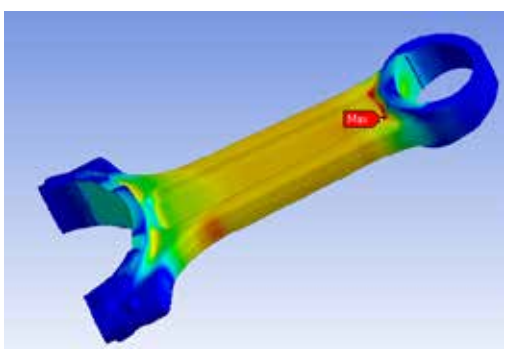

Figure: 3 (B) Maximum Strain

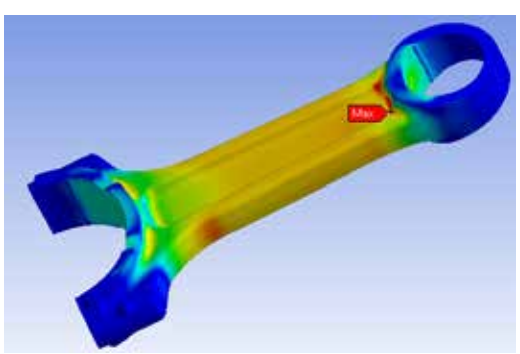

Figure: 3(C) Maximum Stress

6, Results for Connecting Rod:

Result	Connecting Rod
MAXIMUM STRESS (MPa)	397.6
MAXIMUM STRAIN	0.006
TOTAL DEFORMATION (mm)	0.225
FACTOR OF SAFETY FOR DESIGN	1.51

Table : 1 Result of Connecting Rod Analysis

All steps show the procedure for the finite element analysis for Connecting rod. And Table 1 shows the Result of Maximum stress, Maximum strain, Factor of Safety and total deformation for Given Connecting Rod.

CONCLUSION:

We can say that the material C70 S6 is more suitable for Fracture split process during Manufacturing of connecting rod. Due to Fracture Split Process we can reduce overall cost of production and labor. From this FEM based CAE analysis, we have analyze that factor of Safety for this Design is 1.51 so the value of Von meshes is under limit. As well as Strain induced in this material is quite negligible in comparison of its materials elongation capacity 8%. Deformation for this Design is also negligible against overall dimension of this model.

FUCTURE SCOPE:

The future scope for this Design will be useful for finding out the vibration Behavior of this design at its actual operating frequency. As well as the detection of Crippling Failure locations on this Design and the Value of Crippling load also.

ACKNOWLEDGEMENT:

We gratefully acknowledge Mechanical engineering department of RK University for technical support and providing the research facilities. Also we are thankful to Khodiyar CAD Centre for Providing software interface facilities.

REFERENCE

[1] Ram Bansal-"Dynamic Simulation of a Connecting Rod made of Aluminium Alloy using Finite Element Analysis Approach." IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE). Volume 5, Issue 2 (Jan. - Feb. 2013), PP 01-05. | [2] P S Shenoy and A Fatemi-"Dynamic analysis of loads and stresses in connecting rods." Proc. IMechE Vol. 220 Part C. Page no- 615-624. | [3] Vivek. C. Pathade, Bhumeshwar Patle, Ajay N. Ingale.- "Stress Analysis of I.C.Engine Connecting Rod by FEM". International Journal of Engineering and Innovative Technology (IJEIT) Volume 1, Issue 3, March 2012.