ISSN 2395-1621

Structural Analysis & Optimization of Connecting Rod

#1Swapnil A Narkhede, #2Prof. D. H. Burande

1snarkhede01@gmail.com
2dhburande.scoe@sinhgad.edu

#12Department of Mechanical Engineering

Sinhgad College of Engineering
Pune 411 041.

ABSTRACT

In this paper, the connecting rod is designed and optimized under a load range comprising tensile load corresponding to 360o crank angle at the maximum engine speed as one extreme load, and compressive load corresponding to the peak gas pressure as the other extreme load. In this paper a study has been done to model the connecting rod using CAD tool (CATIA V5) to do static structural analysis for static load to check stresses, deformation, strain locations & second part of project is topology optimization for mass reduction without much altering strength using CAE tool (Hypermesh, Ansys).

Keywords: Connecting Rod, Topological Optimization, CATIA, Hypermesh, Ansys

ARTICLE INFO

Article History

Received: 22nd July 2016

Received in revised form :

22nd July 2016

Accepted: 25th July 2016

Published online :

25th July 2016

I. INTRODUCTION

Connecting rod is the intermediate link between the piston and the crank. And is responsible to transmit the push and pull from the piston pin to crank pin, thus converting the reciprocating motion of the piston to rotary motion of the crank.

Connecting rods manufactured by forging either from wrought steel or powdered metal. They could also be cast. However, castings could have blow-holes which are detrimental from durability and fatigue points of view. The fact that forgings produce blow-hole-free and better rods gives them an advantage over cast rods. Between the forging processes, powder forged or drop forged, each process has its own pros and cons. Powder metal manufactured blanks have the advantage of being near net shape, reducing material waste. 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. Bringing the part to final dimensions under tight tolerance results in high expenditure for machining as the blank usually contains more excess material.

II. PROBLEM STATEMENT

Connecting rods are widely used in variety of engine. Currently, connecting rods contains excess material, leads to increase in weight of the vehicle. Directly affects the mileage and cost. In this thesis, modeling existing connecting rod in CAD software and analyzing it for induced stresses and deformation in CAE software. Then using Topology optimization material will be removed. Again, analysis will be done on optimized model for stresses and deformation. It is also tested experimentally and results were correlated it with analysis results.

III. OBJECTIVE

1. Modeling current connecting rod.
2. Analyzing for stresses and deformation.
3. Topological optimization for the model.
4. Analyzing for stresses and deformation on optimized model.
5. Experimental testing and correlating results.

IV. LITERATURE REVIEW

Webster et al. [1] 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.

Yoo et al. [2] used variational equations of elasticity, material derivative idea of continuum mechanics and an adjoint variable technique to calculate shape design sensitivities of stress. The results were used in an iterative optimization algorithm, steepest descent algorithm, to numerically solve an optimal design problem. The focus was on shape design sensitivity analysis with application to the example of a connecting rod. The stress constraints were imposed on principal stresses of inertia and firing loads. But fatigue strength was not addressed. The other constraint was the one on thickness to bound it away from zero. They could obtain 20% weight reduction in the neck region of the connecting rod.

Hippoliti [3] reported design methodology in use at Piaggio for connecting rod design, which incorporates an optimization session. However, neither the details of optimization nor the load under which optimization was performed were discussed. Two parametric FE procedures using 2D plane stress and 3D approach developed by the author were compared with experimental results and shown to have good agreements. The optimization procedure they developed was based on the 2D approach.

For their optimization study, Serag et al. [4] developed approximate mathematical formulae to define connecting rod weight and cost as objective functions and also the constraints. The optimization was achieved using a Geometric Programming technique. Constraints were imposed on the compression stress, the bearing pressure at the crank and the piston pin ends. Fatigue was not addressed. The cost function was expressed in some exponential form with the geometric parameters.

Balasubramaniam et al. [5] 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. No discussions on the optimization or fatigue, in particular, were presented.

V. FEA OF CONNECTING ROD

Engine Specification:
Engine: Cummins 6BTAA, DI Turbocharged, with Viscous fan
Emission Norms: BSIII
Engine Cylinders: 6
Displacement (cc): 5883
Max Power: 135 bhp @ 2400 rpm
Max Torque: 490 Nm @ 1400-1600rpm
Transmission: Manual
Gearbox: 6speed
Clutch: 352 mm dia., Spicer Puller type.

Fig.1 Connecting Rod

Dimensions of connecting rod are found physically with help of measuring instruments. Connecting rod is modeled with help of CATIA (V5R20) software.

Computer Aided Design (CAD):
CATIA (computer aided three-dimensional interactive application) is a multi – platform computer - aided design (CAD) / computer - aided manufacturing (CAM) / computer - aided engineering (CAE) software suite developed by the French company Dassault Systems.

CATIA delivers the unique ability not only to model any product, but to do so in the context of its real-life behaviour: design in the age of experience. Systems architects, engineers, designers and all contributors can define, imagine and shape the connected world.

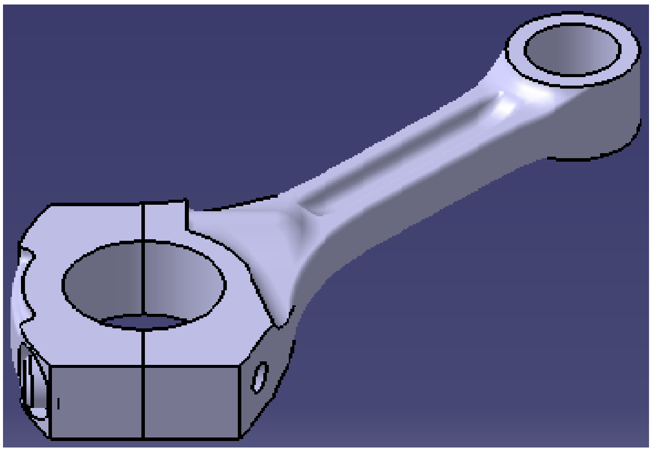

Fig.2 Modeled in CATIA (V5R20) software

Finite Element Modeling (FEM):
It is a numerical technique for finding approximate solutions to boundary value problems for partial differential equations. It is also referred to as finite element analysis (FEA). FEM subdivides a large problem into smaller, simpler, parts, called finite elements. The simple equations that model these finite elements are then assembled into a larger system of equations that models the entire problem. FEM then uses variational methods from the calculus of variations to approximate a solution by minimizing an associated error function.

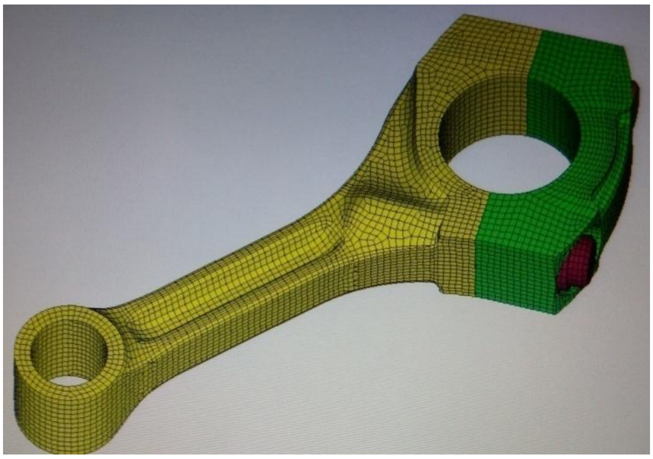

Fig.3 Solid Meshing

Discretization:

A solid element mesh is required to be generated. The meshing of the connecting rod is done in HYPERMESH software.

Type of Elements – Hexahedron, Pyramid, Wedge
Element Order- First order
Number of Elements- 23132
Number of Nodes- 28360

Static Structural Analysis:

A static structural analysis is done to analyze displacements, stresses, strains and forces on structure or a component due to load application. The structures response and loads are assumed to vary slowly with respect to time. There are various types of loading that can be applied in this analysis which are externally applied forces and pressures, and temperatures.

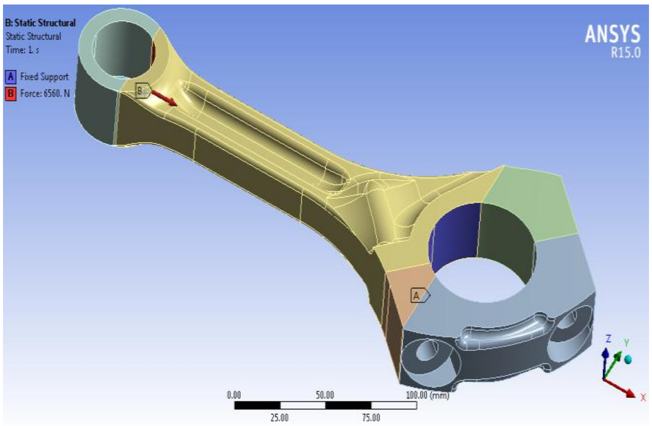

Fig.4 Load & Constraint

Material Properties:
Material- Structural Steel
Young's Modulus- 200 GPa
Poisons Ratio- 0.3
Density- 7850 kg/m^3
Yield Strength- 250 MPa

Loads Calculations:
Crank throw (t') = 64mm
Dia. of Bore (d) calculate as follows,

$$d = \sqrt{\frac{cc}{3\pi * t'}}$$

$d = 9.876cm$

Pressure (Pb) exerted due to combustion on piston head,

$$Pb = \frac{BHP * 746 * 60}{L * A * N * K * n}$$

Pressure (Pb) = 855947.1 N/m2
Therefore,
Force (F) = Pressure * C/S Area
= 6556.64 ~ 6560N

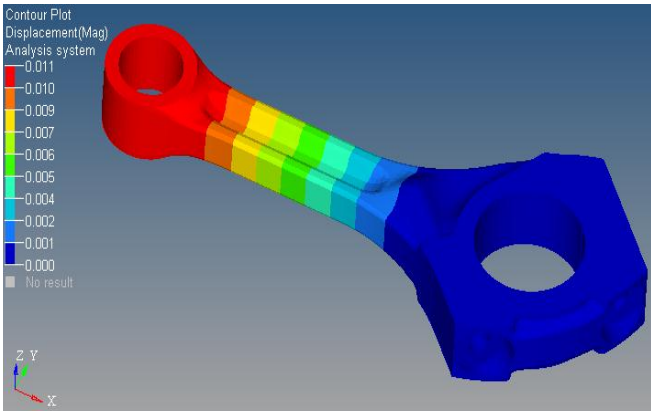

Fig.5 Maximum Deflection of existing Design

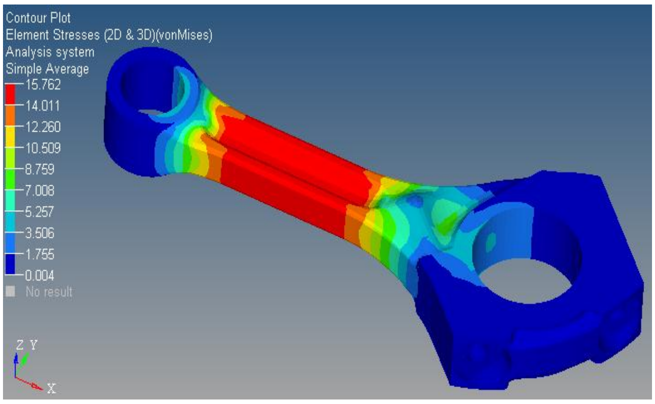

Fig.6 Maximum Von-Mises stress for given boundary

VI. Topological Optimization

Topology optimization is a mathematical approach that optimizes material layout within a given design space, for a given set of loads and boundary conditions such that the resulting layout meets a prescribed set of performance targets.

There are three kinds of structure optimization,

- Size Optimization
- Shape Optimization
- Topology Optimization

The topology optimization consists of the following sequence of steps.

- Define the design space
- Define optimization parameters
- Material removal process and detail design

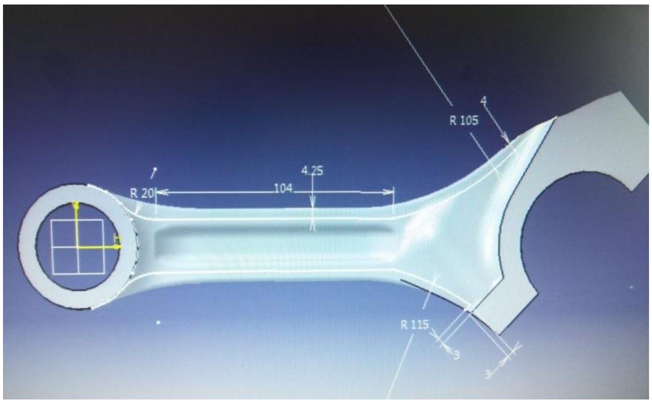

Fig.7 Connecting Rod With Extra Material

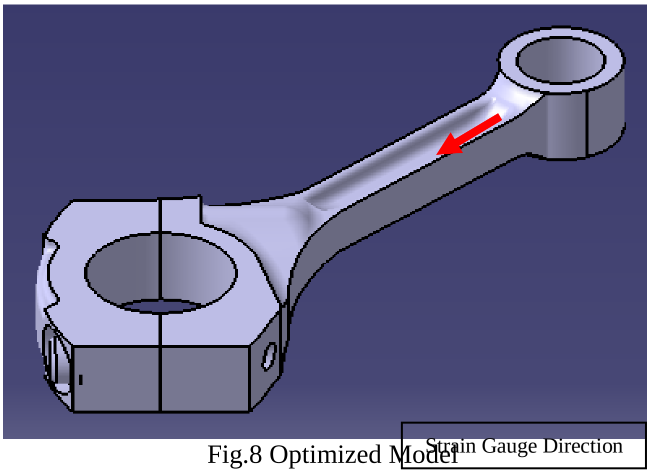

Fig.8 Optimized Model

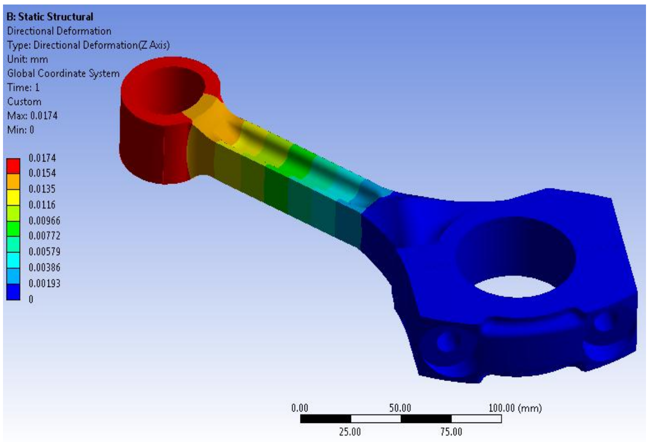

Fig.9 Maximum Deflection Of optimized model

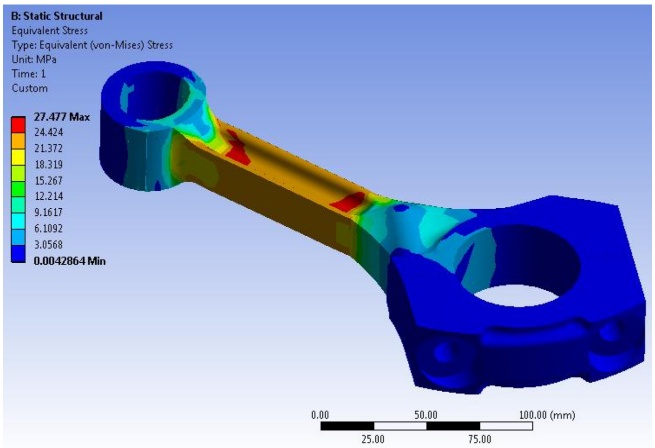

Fig.10 Maximum Von-Mises stress of optimized model for given boundary conditions

VII. EXPERIMENTAL

Existing connecting rod is machined on Electronic Discharge Machine (EDM) to get the required accuracy.

Fig.11 Machined Part as per Optimized Model

Fig.12 Experimental Setup

Experiment is done on UTM having capacity of 100ton. Fixture is designed as per testing feasibility. Load applied is 6560N. Strain from experimental is 160microns.

VIII. CONCLUSION

From results of finite element analysis it is observed that the maximum stress value is within the safety limit. There is a great potential to optimized this safety limit which can be done by removing material from low stressed region thus optimizing its weight without affecting its structural behavior. The maximum displacement value is also very less. From results of finite element analysis it is observed that the maximum stress value is within the safety limit. There is a great potential to optimized this safety limit which can be done by removing material from low stressed region thus optimizing its weight without affecting its structural behavior. The maximum displacement value is also very less. So, the material from low stressed region is can be removed without affecting its strength and is within the yield strength.

- Von-mises stress of existing and optimized components are having 4% variation compared to material yield strength.
- Deflection measured and found on existing (0.011mm) and optimized (0.017mm) model is very less.

- With this topological optimization a weight saving of 11% from existing (2.65kg) to optimized (2.35kg) component.

ACKNOWLEDGEMENT

It is my great pleasure to acknowledge sense of gratitude to all, who have made it possible for us to complete this seminar with success. It gives me great pleasure to express my deep gratitude to my seminar guide Prof. D. H. Burande for his support and help from time to time during seminar work.

It is my pleasure to acknowledge sense of gratitude to our Head of Department Prof. V. N. Kapatkar, Vice Principal Dr. Y. P. Reddy and Principal Dr. S. D. Lokhande for their great support and encouragement in seminar work.

Finally yet importantly I would like to thank all Staff Members and all our colleagues for their valuable suggestions and support.

REFRENCES

[1] Webster, W. D., Coffell R., and Alfaro D., "A Three Dimensional Finite Element Analysis of a High Speed Diesel Engine Connecting Rod," SAE Technical Paper Series, Paper No. 831322.

[2] Yoo, Y. M., Haug, E. J., and Choi, K. K., "Shape optimal design of an engine connecting rod," Journal of Mechanisms, Transmissions, and Automation in Design, Transactions of ASME, Vol. 106, pp. 415-419.

[3] Hippoliti, R., "FEM method for design and optimization of connecting rods for small two-stroke engines," Small Engine Technology Conference, pp. 217-231.

[4] Serag, S., Sevien, L., Sheha, G., and El-Beshtawi, I., "Optimal design of the connecting-rod", Modelling, Simulation and Control, B, AMSE Press, Vol. 24, No. 3, pp. 49-63.

[5] Balasubramaniam, B., Svoboda, M., and Bauer, W., "Structural optimization of I.C. engines subjected to mechanical and thermal loads," Computer Methods in Applied Mechanics and Engineering, Vol. 89, pp. 337-360.

