Experimental Stress Analysis and Optimization of Connecting Rod

Ghadoje Sagar S[1], Gite Hemant S[2], Gawali Dipak P[3] , Gayke Dnyaneshwar S.[4]

Mechanical Engineering [1,2,3,4], SND COE & RC, Yeola[1,2,3,4]
Email: ghadoje4engg@gmail.com[1],hemant.gite95@gmail.com[2]
dipakgawali59@gmail.com[3], gaikednyan025@gmail.com[4]

---***---

Abstract - *The main objective of this study was to explore weight and cost reduction Opportunities for a production forged steel connecting rod. This has entailed performing a detailed load analysis. Therefore, this study has dealt with two subjects, first, stress analysis of the connecting rod, and second, optimization for weight and cost. It is the conclusion of this study that the connecting rod can be 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.*

***Key Words*: Connecting Rod, Finite Element Analysis, Stress, optimization, Design .**

1. 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.

The connecting rod can be designed and optimized under a load range, 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 project we are going 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 (Ansys). Experimental is carried out by machining existing connecting rod as per topological optimization using EDM. Strain gauge will be mounted at high strain location on the connecting rod defined from FEA to get deformation while testing on UTM.

2. OBJECTIVES

(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) Results from topological optimization will be implemented on existing model.
(6) Machining the existing connecting rod as per optimization result.
(7) Mounting strain gauge on same portion.
(8) Preparing fixtures to hold connecting rod for experimental testing.
(9) Correlating results of both CAE and experimental

3. 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.

3.1 Engine & Transmission 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
- Clutch: 352 mm dia., Spicer Puller type
- Gearbox: 6speed
- Fuel: Diesel
- Fuel Tank: 250 Litres

Fig -1 : Existing component of TATA1612 heavy vehicle

4. MODELING OF CONNECTING ROD

Dimensions of cross-section of the connecting rod.
Dimensions of the crankpin at the big end and the piston pin at the small end.
Size of bolts for securing the big end cap.
Thickness of the big end cap.
As per existing connecting rod (i.e, TATA 1612 vehicle connecting rod),
Small hole inner dia. = 35mm
Small hole outer dia. = 50mm
Big hole inner dia. = 60mm
Big hole outer dia. = 80mm
Thickness = 35mm
Center to center distance = 225mm
Spine width = 30mm
Spine thickness = 21mm
Slot on spine length107x width13x depth7mm
Bolt dia. = 14mm
Length = 67mm
Head dia. = 21mm
Length = 8mm
Washer outer dia. = 28mm
Inner dia. = 14mm
Length = 4mm.

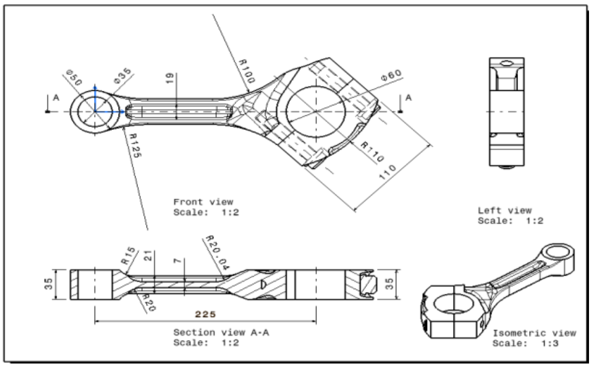

Fig - 2 : 2D Drafting of Connecting Rod

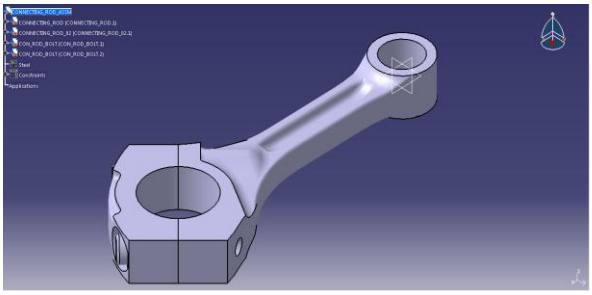

Fig - 3 : CAD Model of Connecting Rod

4.1 Calculation

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

Pressure (Pb) exerted due to combustion on piston head,
Pressure (Pb) = 855947.1 N/m2
Therefore,
Force (F) = Pressure * C/S Area

= 855947.1*π/4*(98.76*10-3)2
= 6556.64 ~ 6560N.

5. FINITE ELEMENT METHOD

5.1 Mesh Generation:

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.

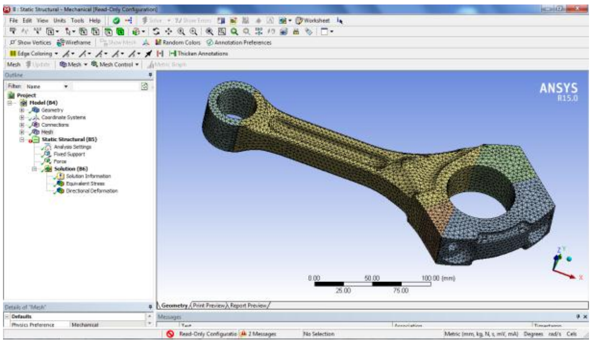

Fig- 4: Discretized Model

Element type- Wedge
No. of Elements- 82105
No. of Nodes- 52748

6. FEA PROCESSING

6.1 Pre Processing

The values of young's modulus, poisons ratio, density, yield strength for steel are taken from material library of the FEA PACKAGE.
Material- Structural Steel
Young's Modulus- 200 GPa
Poisons Ratio- 0.3
Density- 7850 kg/m^3
Yield Strength- 250 MPa.

The nodes around the connecting rod holes have a rigid element connecting them to the centre of the hole which has of its degree of freedom fixed. The element which is used to fix connecting rod and crank shaft is fixed and used as a rigid element. Fig shows connecting rod is fixed at crank end and load is applied from the piston end.

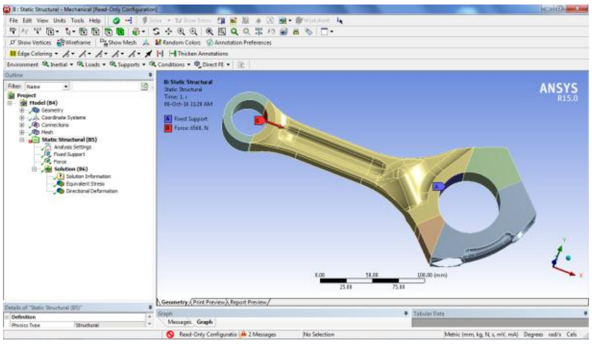

Fig - 5: Boundary Condition

6.2 Post Processing

Model acceptance criteria: the maximum von-Mises stress must be less than the material yield strength for the duration of the component. The deflection is considered and the maximum Von-Mises stress must be less than the yield strength for abuse load case.

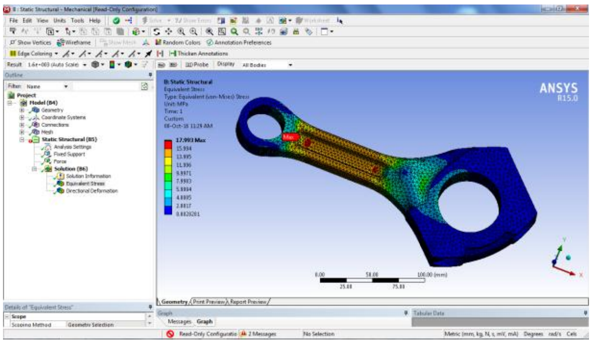

Fig- 6: Von - Mises stress of existing model

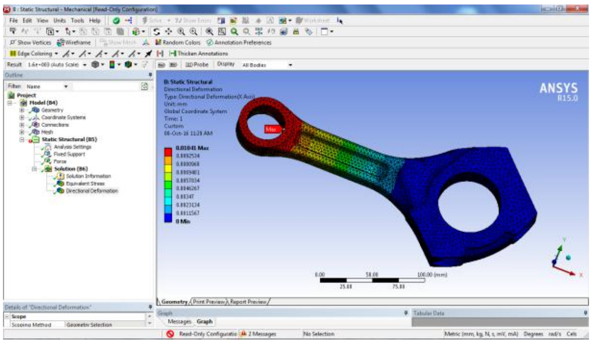

Fig- 7: Deformation of existing model

7. OPTIMIZATION

7.1 Topology 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

Three optimization methods correspond to the three stages of the product design process, namely the detailed design, basic design and conceptual design. Size optimization keeps the structural shape and topology structure invariant, to optimize the various parameters of structure, such as thickness, section size of beam, material's properties; shape optimization maintains the topology structure, to change the boundary of structure and shape, seek the most suitable structure boundary situation and shape; topology optimization is to find the optimal path of material's distribution in a continuous domain which meet the displacement and stress conditions in structure, make a certain performance optimal. Thus, compared to size and shape optimization, topology optimization with more freedom degree and greater design space, its greatest feature is under uncertain structural shape, according to the known boundary condition and a given load to determine the reasonable structure, both for the conceptual design of new products and improvement design for existing products, it is the most promising aspect of structural optimization.

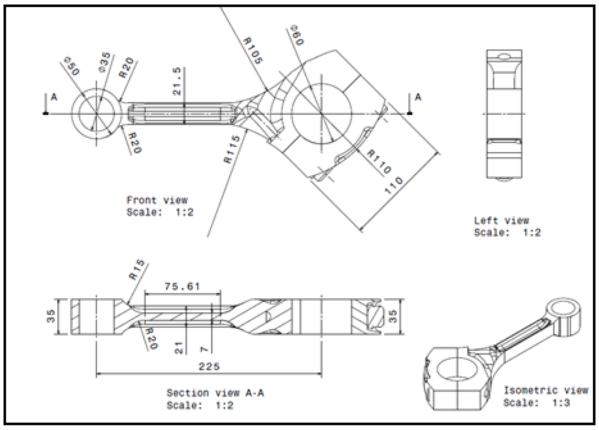

Fig-8: 2D drafting of optimized model

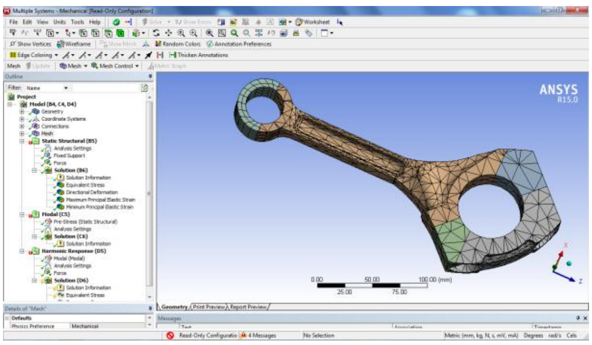

Fig- 9: Meshing for optimized model

Element type- Wedge
No. of Elements- 10531
No. of Nodes- 5373

7.2 Boundary Conditions:

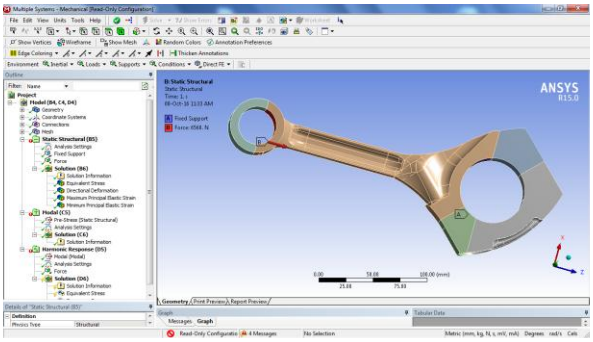

Fig - 10: Boundary condition

7.3 Von Misses:

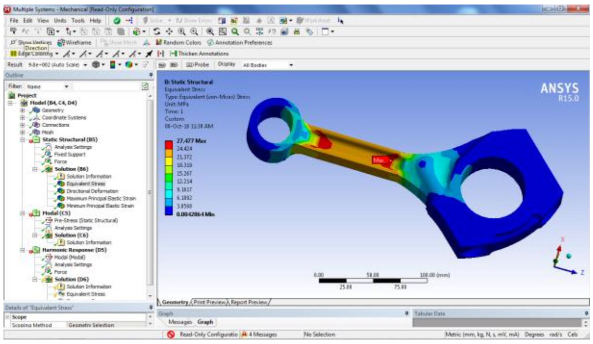

Fig-11: Von- Mises stress of Optimized model

8. EXPERIMENTAL ANALYSIS OF CONNECTING ROD

8.1 Installation of Strain Gauge:

The proper installation of a strain gauge requires a number of distinct steps. First, the surface must be properly cleaned and polished. After this, the gauge is installed and lead wires are connected. Finally, a protective coating is applied to protect the gauge.

8.2 Cleaning procedure:

The surface where the gauge is to be installed should be cleaned no more than 30 minutes prior to the installation of the gauge. If more time has elapsed, repeat this procedure.
1. Degrease the surface of interest with a small amount of isopropyl alcohol and a Kim Wipe. Take care not to contaminate the cleaned surfaces from this point on.
2. Apply one or two drops of M-prep Conditioner A to the cleaned area and lightly sand with 400 grit or higher emery paper to remove the paint in the gauge area. Use sand paper carefully to prevent damage to surface.

3. Use a Kim Wipe to remove any loose particles from the surface. Continue wiping until no trace of paint can be seen on the Kim Wipe.
4. Wet the exposed area with one or two drops of M-prep Conditioner A and lightly scrub the area with a Kim Wipe. Finish by using a fresh wipe and making a single stroke across the surface.
5. Wet the exposed area with one or two drops of M-prep Neutralizer 5A and lightly scrub the area with a Kim Wipe. Again finish by using a fresh wipe and making a single stroke across the cleaned surface.

Fig-12: Experimental Set-up

Deflection measured and found on optimized 0.0174 mm model is very less.

9. RESULT AND CONCLUSIONS

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 optimize, 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 found on existing (17.993MPa) and optimized (27.477 MPa) components are within the material yield strength.
- Deflection measured and found on existing (0.011mm) and optimized (0.0174mm) model is very less.
- With this topological optimization weight reduced from existing (2.351kg) to optimized (2.240kg) component
- Von-mises stress found on existing (17.993MPa) and optimized (27.477 MPa) components are within the material yield strength.
- Deflection measured and found on existing (0.011mm) and optimized (0.0174mm) model is very less.

10. REFERENCES

[1] Kuldeep B, Arun L.R, Mohammed Faheem , "Analysis and optimization of connecting rod using alfasic composites", International Journal of Innovative Research in Science, Engineering and Technology, Vol-2, Issue-6, June 2013
[2] AbhinavGautam, K Priya Ajit, " Static Stress Analysis of Connecting Rod Using Finite Element Approach", IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE) e-ISSN: 2278-1684, p-ISSN: 2320-334X, Volume 10, Issue 1 (Nov. - Dec. 2013), pp. 47-51
[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, Vol-1, Issue-3, pp-12-15, March 2012.
[4] J.D.Ramani, Sunil Shukla, Pushpendra Kumar Sharma, "FE-Analysis of Connecting Rod of I.C.Engine by Using Ansys for Material Optimization" Int. Journal of Engineering Research and Applications www.ijera.com ISSN : 2248-9622, Vol. 4, Issue 3(Version 1), March 2014, pp.216-220