Stress analysis and optimization of connecting rod of + and Ellipsoidal sections using finite element analysis

R. K. Vishwkarma

***Abstract*— To obtain a best & suitable design of connecting rod to sustain various stress & forces, finite element analysis suggest the minimum design specification with the help of ANSYS v13. For this purpose Pro/Engineer wildfire 4.0 develop a solid modeling and ANSIS Workbench 13.0 for finite element analysis After observation and modeling the analysis for optimization was performed. It relates with weight comparison & reducing of size by analysis various factors which affect the performance. The intermediate component between crank and piston is known as CONNECTING ROD. The objective of C.R. is to transmit push & pull from the piston pin to the crank pin and then converts reciprocating motion of the piston into the rotary motion of crank. The components are big shank, a small end and a big end. The shank may be rectangular, circular, tubular, I- Section or H-Section. It sustains force generated by mass & fuel combustion. The resulting bending stresses appears due to eccentricities, crank shaft, case wall deformation & rotational**

***Index Terms*— Pro-E, ANSYS, Stress analysis, Connecting rod, FEA**

I. INTRODUCTION

The intermediate component between crank and piston is known as connecting rod**.** The objective of C.R. is to transmit push & pull from the piston pin to the crank pin and then converts reciprocating motion of the piston into the rotary motion of crank. The components are big shank, a small end and a big end. The cross section of shank may be rectangular, circular, tubular, I**-** Section, **+** -section or ellipsoidal-Section. It sustains force generated by mass & fuel combustion. The resulting bending stresses appear due to eccentricities, crank shaft, case wall deformation & rotational mass.

FEA approach deals with structural analysis along with various parameters which affects its working & define best solution to overcome the barriers associated with it. The structural analysis allows stresses & strains to be calculated in FEA, by using the structural model. The structural analysis performed to create high & low stresses region from the input of the material, loads, boundary condition. FEA approach was adopted in structural analysis to overcome the barriers associated with the geometry & boundary condition. It is used to improve optimize design.

The main objective of this work is to determine shear stresses and optimization in the existing connecting rod, which are in different cross-section as plus (+) section and ellipsoidal section. The failures of existing design suggest the minimum design changes in the existing connecting rod.

R. K. Vishwkarma, Dept. name of organization (Mechanical Engineering), Eshan College Of Engg. (Mathura)

II. FINITE ELEMENT ANALYSIS:

FEA approach deals with structural analysis along with various parameters which affects its working &define best solution to overcome the barriers associated with it. The structural analysis allows stresses & strains to be calculated in FEA, by using the structural model.

The structural analysis performed to create high & low stresses region from the input of the material, loads, boundary condition. FEA approach was adopted in structural analysis to overcome the barriers associated with the geometry &boundary condition. The structural analysis allows stresses & strains to be calculated in FEA by using structural model. It is used to improve optimize design.

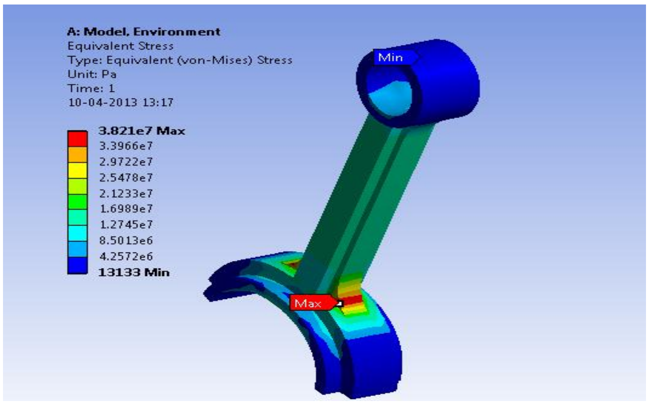

Fig. 1 Equivalent von-mises stress

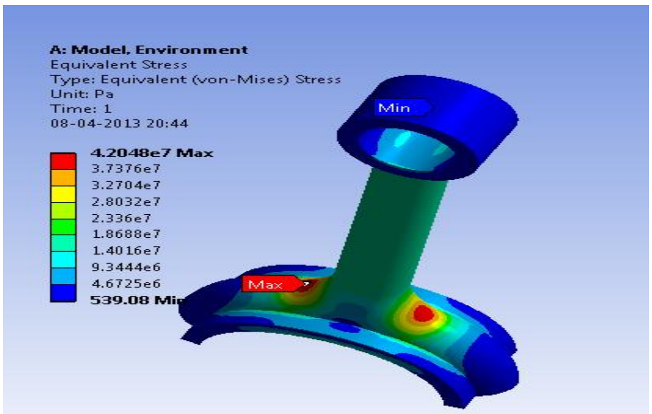

Fig. 2 Equivalent von-mises stress

Finite element analysis of connecting rod has been carried out for investigation of critical stresses, elastic strain and total deformation.

III. OPTIMIZATION

Objective of the optimization task was to minimize the mass of the connecting rod of various cross- section such as, plus(**+**)-section and ellipsoidal-section. The weight of optimized connecting rod is certainly lower than the weight of original connecting rod. The factors have been addressed during the optimization- load factor, stresses under the loads.

Objective: Minimize Mass and Cost:
Subject to:

a) Compressive load = peak compressive gas load.
b) Maximum stress < Allowable stress
c) Side constraints (Component dimensions)
d) Manufacturing constraints
e) Buckling load > Factor of safety x the maximum gas load
f) Optimized Model

Shape Results (+ Section)
Table: 1

Name	Scope	Target Reduction	Predicted Reduction
"Shape Finder"	"Geometry"	20%	12.65%

Table: 2

Name	Original	Optimized	Marginal
"Shape Finder"	2.1083 kg	1.8416 kg	1.034e-02 kg

Fig.3 Shape Finder

IV. Shape Results (Ellipsoidal Section)

Table: 3

Name	Scope	Target Reduction	Predicted Reduction
"Shape Finder"	"Geometry"	20%	10.56%

Table: 4

Name	Original	Optimized	Marginal
"Shape Finder"	1.9274kg	1.7192kg	1.344e-02 kg

Fig.4 Shape Finder

V. CONCLUSION

The result find out by FEA tool in this present work the value of Equivalent Stress, Shear Elastic Strain and Total Deformation. The weight optimization of connecting rod, consider following properties. This work investigates the weight and cost reduction opportunities that steel C-70 connecting rods offer. The maximum stresses developed near the big end connecting rod, it the region which is more susceptible for failure. The following conclusions can be drawn from this work.

1. The parameter taken in account to reduce 20% weight of steel C-70 connecting rod.
2. The stress is found maximum at the crank end so the material is increased in stressed portion to minimize the stress.
3. The weight of the steel C-70 connecting rod (+ section) is reduced by 0.2667 kg, which is about 12.65% of original weight of connecting rod.
4. The weight of the steel C-70 connecting rod (ellipsoidal-section) is reduced by 0.2082 kg, which is about 10.56% of original weight of connecting rod.

VI. REFERENCES

[1] A. Mirehei, M.Hedayati Zadeh, A. Jafari and M.Omid, "Fatigue analysis of connecting rod of universal tractor through finite element method (ANSYS)" Journal of Agriculture Technology 2008, v.4(2):21-27.
[2] Suraj Pal and sunil Kumar, "Design Evaluation and Optimization of Connecting Rod Parameters Using FEM" International Journal of Engineering and Management Research, vol.-2, Issue-6, December 2012, 21-25.
[3] P S Shenoy and A Fatemi, "Dynamic analysis of loads and stresses in connecting rods" Journal of Mechanical Engineering Science, vol.220 part C: IMechE 2006.
[4] Atish Gawale, A.A. Shaikh and Vinay Patil, "Nonlinear static finite element analysis and optimization of connecting rod" World journal of science and Technology 2012, vol.2 (4):01-04.
[5] Wisam Abdul-Lateef, "Finite Element Analysis of Connecting Rod Using NASTRAN Software" The Iraqi Journal For Mechanical And Material Engineering, Vol.12,No.2,2012
[6] K.Sudershn Kumar, Dr. Tirupathi Reddy and Syed Altaf Hussain, "Modeling and Anaysis of Two Wheeler Connecting Rod" International Journal of Modern Engineering Research, Vol.2, Issue.5, Sep-Oct.2012 pp-3367-3371.