Stress analysis and optimization of connecting rod using finite element analysis

Yogesh Kumar Bharti[1], Vikrant Singh[1], Afsar Hussain[1], Dipanshu Singh[1],
Shyam Bihari Lal[2], Satish Kumar Dwivedi[2]

[1]B. Tech. Students, Department of Mechanical Engineering, Buddha Institute of Technology
[2]Asst. Professor, Mechanical Engineering, Buddha Institute of Technology, GIDA, Gorakhpur - 273209 (U.P.) India

ABSTRACT

Design is process by which human desired fulfill. Design of connecting rod is very complex because the connecting rod works in very complicated condition. 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. For this purpose Pro/Engineer wildfire 4.0 develop a solid modeling and ANSYS v13.0 for finite element analysis. Optimization is process which provides best suited option and result. After observation and modeling the analysis for optimization was performed. It relates with weight comparison & reducing of size of connecting rod by analysis various factors which affect the performance like stress.

Key words- 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, I-section and ellipsoidal section. The failures of existing design suggest the minimum design changes in the existing connecting rod.

II.   MATERIAL STUDY:

The C-70 materials have been widely present in culture. Alloying elements in the material enables hardening of forged connecting rod when they undergo controlled cooling after forging. The properties of material are initial input for optimization task thus it play a very important role in optimization task. Connecting rod was design & modeled by using Pro/E 4.0 v. It was then imported to ANSYS for analysis.
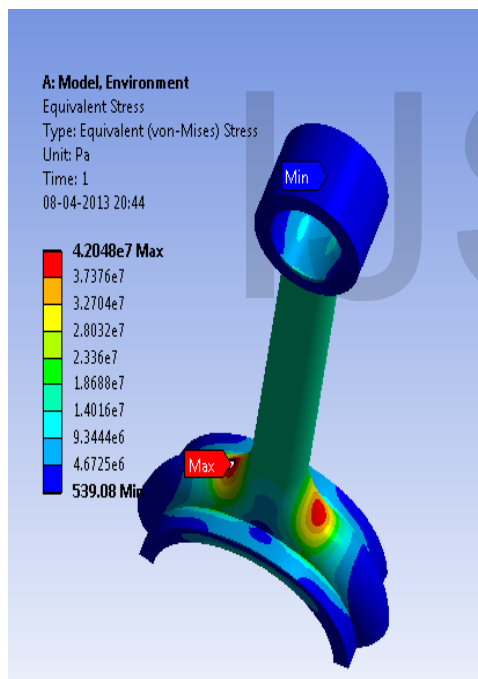

Fig. 1: Equivalent (Von-Mises) stress   Fig.2 : Shape Optimization
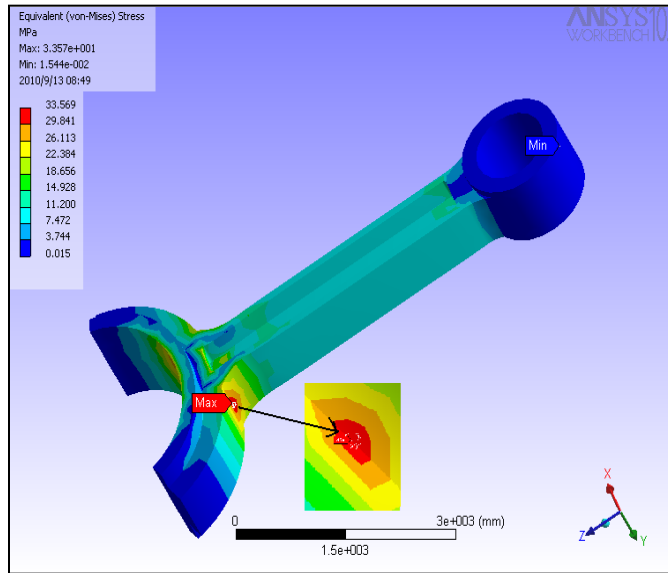
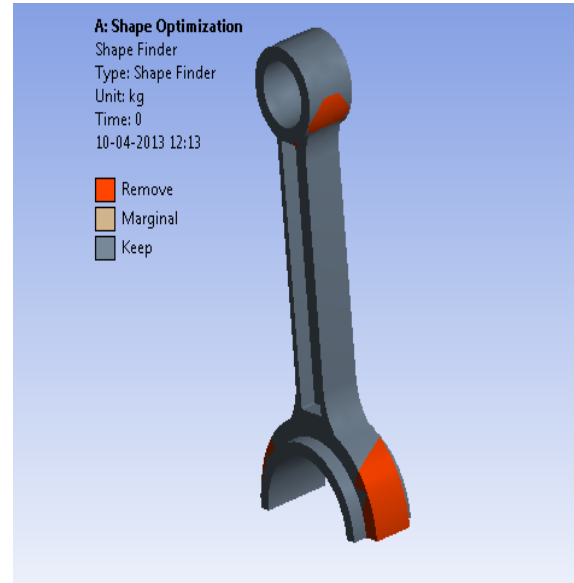Fig. 3: Equivalent (Von-Mises) Stress

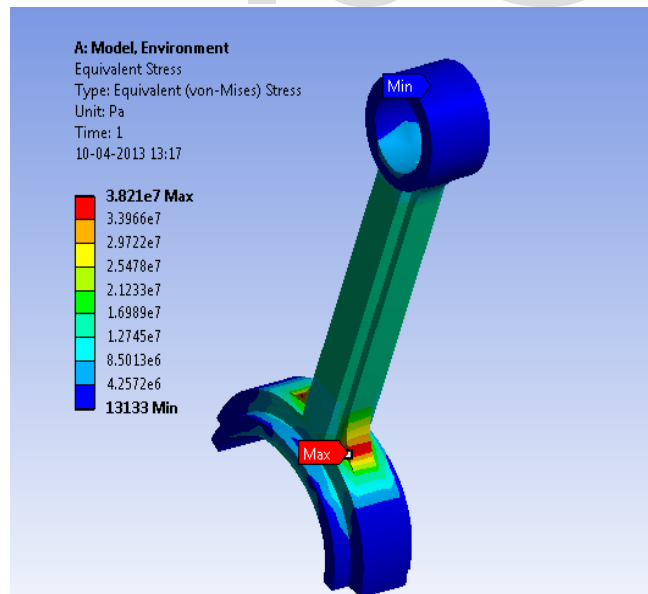

Fig.4: Shape Optimization

Fig. 5: Equivalent (Von-Mises) Stress

Fig.6: Shape Optimization

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 I-section, 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 (Ellipsoidal Section):

Table: 1

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

Table: 2

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

Shape Results (I-Section):

Table: 3

Name	Scope	Target Reduction	Predicted Reduction
"Shape Finder"	"Geometry"	20%	11.23%

Table: 4

Name	Original	Optimized	Marginal
"Shape Finder"	1.7385 kg	1.5433 kg	1.22e-02 kg

Shape Results (+ Section)

Table: 5

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

Table: 6

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

RESULT AND DISCUSSION: As compared to table no. 1 to 6. It will be the most significant finding was find from the table no 1 to 6 according to mass wise analysis shown I table. When target reduction kept that ellipsoidal has removed less amount of mass(10.56%) and more mass will be removed shown in + section in table no 5. The main theme at + section more superior than the other to the masses among them but the target is shifted on weight among all the section than will find that + section superior to other section. At a glance after the optimization of connecting rod it will be predicated equivalent stress is more suitable in I section. Therefore I section will be consider for connecting rod design under the sustain various stresses and forces.

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 (I-section) is reduced by 0.1952 kg, which is about 11.23% of original weight of connecting rod.

4. 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.

5. 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.

REFERENCES:

[1] A. Mirehei, M.Hedayati Zadeh, A. Jafari and M.Omid, 2008 "Fatigue analysis of connecting rod of universal tractor through finite element method (ANSYS)" Journal of Agriculture Technology, v.4(2):21-27.

[2] Lal,S.B. Tevatia,A. and Srivastava,S.K.,"Fatigue Analysis of Connecting Rod using ANSYS Code",International Journal of Mechanics and Solids.v.5 Number 2(2010),pp.143 -150

[3]   Tevatia,A. , Lal,S.B. ,and Srivastava,S.K.," Finite Element Fatigue Analysis of Connecting Rods of Different Cross-Sections" International Journal of Mechanics and Solids.v.6 Number 1(2011),pp.45-53

[4] Suraj Pal and Sunil Kumar, Dec 2012 "Design Evaluation Optimization of Connecting Rod Parameters Using FEM" International Journal of Engineering and Management Research, vol.-2, Issue-6, pages 21-25.

[5] P S Shenoy and A Fatemi, 2006 "Dynamic analysis of loads and stresses in connecting rods" Journal of Mechanical Engineering Science, vol.220 part C: IMechE.

[6] Atish Gawale, A.A. Shaikh and Vinay Patil, 2012 "Nonlinear static finite element analysis and optimization of connecting rod" World journal of science and Technology, vol.2 (4):01-04.

[7] Wisam Abdul-Lateef, 2012 "Finite Element Analysis of Connecting Rod Using NASTRAN Software" The Iraqi Journal For Mechanical And Material Engineering, Vol.12, No.2.

[8] K.Sudershn Kumar, Dr. Tirupathi Reddy and Syed Altaf Hussain, Sep-Oct.2012 "Modeling and Anaysis of Two Wheeler Connecting Rod" International Journal of Modern Engineering Research, Vol.2, Issue.5, pp-3367-3371

[9] 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.

[10] Gupta, R. K., 1993, "Recent Developments in Materials and Processes for Automotive Connecting rods," *SAE Technical Paper Series,* Paper No. 930491.

[11] Hancq, D.A., 2003. Fatigue Analysis Using ANSYS. ANSYS Inc., 22 P. http://www.ansys.com/

[12] Sonsino, C. M., and Esper, F. J., 1994, "*Fatigue Design for PM Components,*" European Powder Metallurgy Association (EPMA).

[13] Fermer, M. and H. Svensson, 2001. Industrial experiences of FE-based fatigue life predictions of welded automotive structures. J. Fatigue Fract. Eng. Mater Struct., 24: 489-500.

[14] Del Llano-Vizcaya, L., C. Rubio-González, G. Mesmacque and T. Cervantes-Hernández, 2006. Multiaxial fatigue and failure analysis of helical compression springs. J. Eng. Failure Anal., 13: 1303-1313.

[15] Lu, P.C., 1996. The optimization of a connecting rod with fatigue life constraint. Int. J. Mat. Prod. Technol., 11: 357-370.

[16] Rahman, M.M., A.K. Arffin, N. Jamaludin, S. Abdullah and M.M. Noor, 2008. Finite element based fatigue life prediction of a new free piston engine mounting. J. Applied Sci., 8: 1612-1621.

[17] Nanaware, G.K. and M.J. Pable, 2003. Failures of rear axle shafts of 575 DI tractors. J. Eng. Failure Anal., 10: 719-724.

[18] Bhaumik, S.K., 2002. Fatigue fracture of crankshaft of an aircraft engine. J. Eng. Failure Anal., 9: 255-263.

[19]   Sarihan, V. and Song, J., 1990, "Optimization of the Wrist Pin End of an Automobile Engine Connecting Rod With an Interference Fit," *Journal of Mechanical Design*, Transactions of the ASME, Vol. 112, pp. 406-412.

[20]   Yoo, Y. M., Haug, E. J., and Choi, K. K., 1984, "Shape optimal design of an engine connecting rod," *Journal of Mechanisms, Transmissions, and Automation in Design*, Transactions of ASME, Vol. 106, pp. 415-419.

[21]   Sonsino, C. M., and Esper, F. J., 1994, "*Fatigue Design for PM Components,*" European Powder Metallurgy Association (EPMA).

[22]   Bayrakceken, H., S. Tasgetirena and F. Aksoya, 2007. Failures of single cylinder diesel engines crank shafts. J. Eng. Failure Anal., 14: 725-730.

[23]   Whittaker, D., 2001. The competition for automotive connecting rod markets. J. Metal Powder Report, 56: 32-37.

[24]   Park, H., Ko, Y. S., Jung, S. C., Song, B. T., Jun, Y. H., Lee, B. C., and Lim, J. D., 2003, "Development of Fracture Split Steel Connecting Rods," *SAE Technical Paper Series*, Paper No. 2003-01-1309.

[25]   Ishida, S., Hori, Y., Kinoshita, T., and Iwamoto, T., 1995, "Development of technique to measure stress on connecting rod during firing operation," SAE 951797, pp. 1851-1856.

[26]   Raymond Browell, 2006, "Calculating and Displaying Fatigue Results", Product Manager New Technologies ANSYS, Inc.

[27]   Dr. NWM Bishop and Dr. F. Sherratt, 2000, "Finite Element Based Fatigue Calculations" NAFEMS, RLD Ltd., Hutton Roof, Eglinton Rod, Tilford Farnham, Surrey, GU102DH.

[28]   Richard C.Rise, Brian N.Leis, Drew V.Nelson, Henry D.Berns, Dan Lingenfelser, M.R. Mitchell, 1988. Fatigue Design Handbook, Society of Automotive Engineers, Inc.