

“FATIGUE ANALYSIS AND OPTIMIATION OF CONNECTING ROD USING FINITE ELEMENT ANALYSIS”

Prof. Pushpendra Kumar Sharma¹, Borse Rajendra R.²

¹Head Department Mechanical Engineering NRI-IST, RGPV University, Bhopal

²Research Scholar M.tech. NRI-IST, RGPV University, Bhopal

Email- pksoumy@yahoo.in

Abstract

The main aim of the present research is to determine total Deformation, Fatigue Analysis and Optimization in the existing Connecting rod. If the existing design shows the failure, then suggest the minimum design changes in the existing Connecting rod. In this research, only the static FEA of the connecting rod has been performed by the use of the software. The research identified fatigue strength as the most significant design factor in the optimization process. Then the combination of finite element technique with the aspects of weight reduction is to be made to obtain the required design of connecting rod.

Keywords: Pro/E Wildfire 3.0: For Solid Modeling, ANSYS WORKBENCH 9.0: For Finite Element Analysis

Material and method.

James R. Dales (Jan. 2005) presented connecting rod evaluation, a new steel, C-70, has been introduced from Europe as a crackable forging steel. Recently the American Iron and Steel Institute's (AISI) Bar and Rod Market Development Group has promoted C-70 as an improved material over P/F alloys on the basis of optimization work. Ali Fatemi and Mehrdad Zoroufi presented to increase the usage and competitiveness of forged ferrous components. Yoo *et al.* (1984) used variational equations of elasticity, material derivative idea of continuum mechanics and an adjoint variable technique to calculate shape design sensitivities of stress. Sarihan and Song (1990), for the optimization of the wrist pin end, used a fatigue load cycle consisting of compressive gas load corresponding to maximum torque and tensile load corresponding to maximum inertia load. Sonsino and Esper (1994) have discussed the fatigue design of sintered connecting rods. Athavale and Sajanpawar (1991) modeled the inertia load in their finite element model. In a study reported by Repgen (1998), also based on fatigue tests carried out on identical components made of powder metal and C-70 steel (fracture splitting steel).

Total Deformation

Fig.1 show the Total deformation of the connecting rod under static axial loading. After considering the appropriate regions of the connecting rod, under the tensile loading, the critical regions in the order of decreasing stress intensity are the oil hole, the surface of the pin end bore, the piston pin end transition, the extreme end of the cap and the crank end transition of the connecting rod.

Table 1 Total Deformation Result for Load Case 1 and Case 2

Name	Figure	Scope	Orientation	Minimum	Maximum	Minimum Occurs On	Maximum Occurs On	Alert Criteria
Case 1	5.4	Model	Global	0.0 mm	1.85×10^{-2} mm	Solid	Solid	None
ase 2	5.8	Model	Global	0.0 mm	1.85×10^{-2} mm	Solid	Solid	None

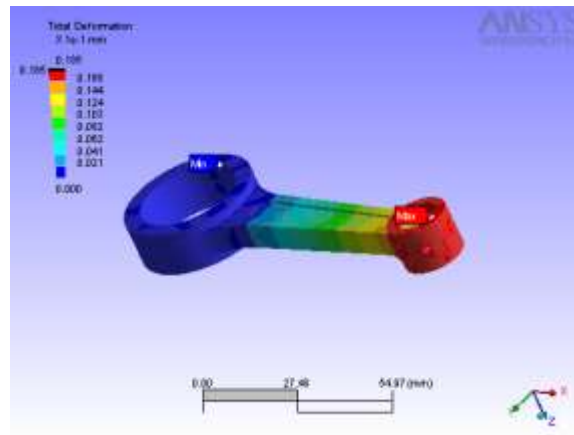


Fig. 1 Total Deformation for Load Case 1

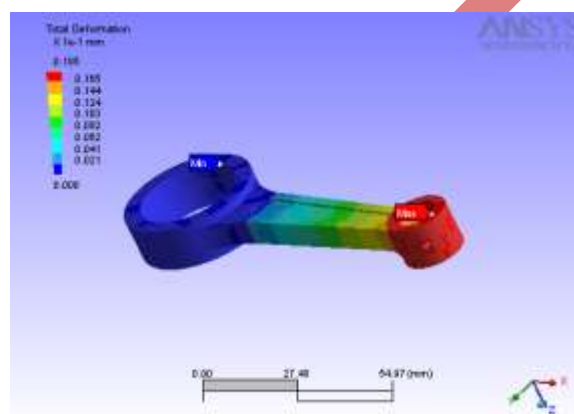


Fig. 2 Total Deformation for Load Case 2

Fatigue Analysis:

It is estimated that 50-90% of structural failure is due to fatigue, thus there is a need for quality fatigue design tool. The focus of fatigue in ANSYS is to provide useful information to the design engineer when fatigue failure may be a concern.

A fatigue analysis can be separated into 3 areas: materials, analysis, and results evaluation. A large part of a fatigue analysis is getting an accurate description of the fatigue material properties. These properties are included as a guide only with intent for the user to provide his/her own fatigue data for more accurate analysis. Fatigue results can be added before or after a stress solution has been performed. To create fatigue results, a fatigue tool must first be inserted into the tree. This can be done through the solution toolbar or through context menus. The details view of the fatigue tool is used to define the various aspects of a fatigue analysis such as loading type, handling of mean stress effects and more. Several results for evaluating fatigue are available to the user. Outputs include fatigue life, damage, factor of safety, stress biaxiality, fatigue sensitivity.

Name	Figure	Scope	Type	Design Life	Minimum	Maximum	Alert Criteria
Life	None	Model	Life		1,000,000.0	1,000,000.0	None
Damage	None	Model	Damage	1.0×10^5	1,000.0	1,000.0	None
Safety Factor	5.21	Model	Safety Factor	1.0×10^5	1.13	15.0	None
Biaxiality Indicator	5.22	Model	Biaxiality Indicator		-1.0	0.97	None
Equivalent Alternating Stress	5.23	Model	Equivalent Reversed Stress		0.01 Mpa	76.22 Mpa	None

Table 2 Fatigue Results

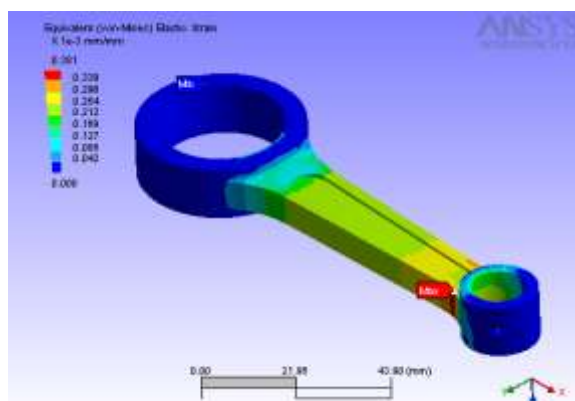


Fig.3 Equivalent (Von-Mises) Elastic Strain

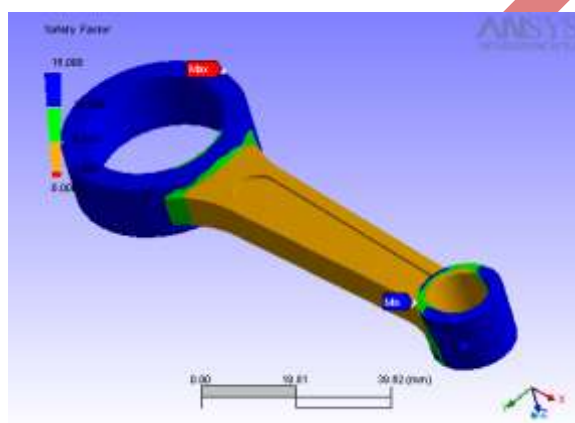


Fig. 4 Safety Factor

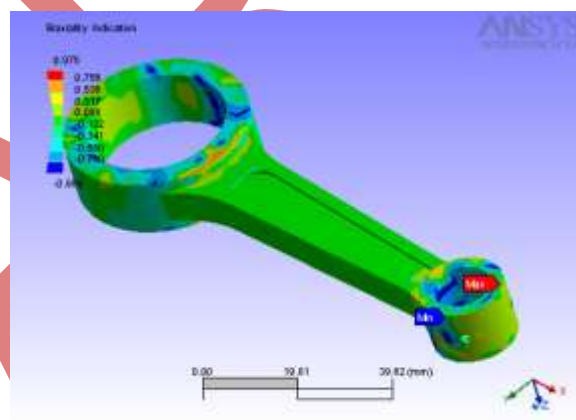


Fig. 5 Bixiality Indication

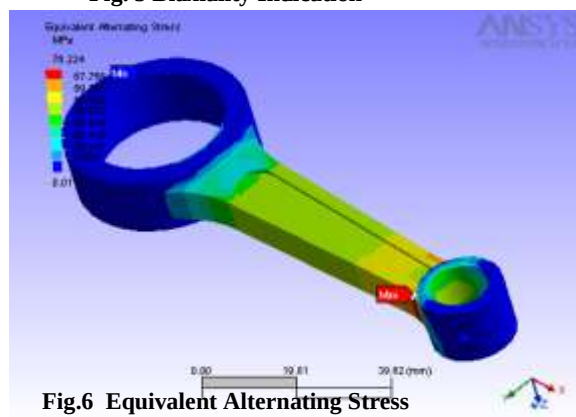


Fig.6 Equivalent Alternating Stress

Optimization

The objective is to optimize the connecting rod for its weight and manufacturing cost, taking into account the recent developments. The weight of the new connecting rod or the 'optimized connecting rod' is definitely lower than the existing connecting rod. The following factors have been addressed during the optimization: the buckling load factor, the stresses under the loads, bending stiffness, and axial stiffness.

Mathematically stated, the optimization statement would appear as follows:

Objective: Minimize Mass and Cost

Subject to:

- Compressive load = peak compressive gas load.
- Maximum stress < Allowable stress.
- Side constraints (Component dimensions).
- Manufacturing constraints.
- Buckling load > Factor of safety x the maximum gas load

Shape Results

Table 3 Total Weights

Name	Figure	Scope	Target Reduction	Predicted Reduction
"Shape Finder"	5.33	"Model"	20.0%	9.24% to 9.51%

Table 4 Total Weights

Name	Original	Optimized	Marginal (Discretionary)
"Shape Finder"	0.13 kg	0.12 kg	3.46×10^{-4} kg

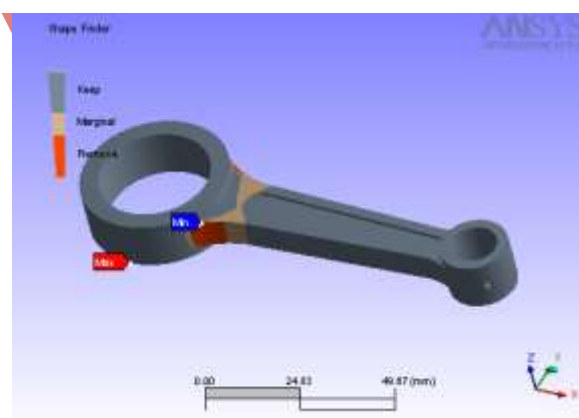


Fig. 7 Shape Finder

SUMMARY AND CONCLUSIONS:

This research investigated weight and cost reduction opportunities that steel forged connecting rods offer. This research is concentrated on the calculation of the stresses developed in the connecting rod and to find region more susceptible to failure. First the Cad Modeling of connecting rod with the help of Cad software Pro/E Wildfire 3.0 and then Load analysis was performed with different cases consideration. The analysis was carried out with computer aided simulation. The tool used for analysis is ANSYS WORKBENCH 9.0. The following conclusions can be drawn from this study:

- 1) The Optimization carried out in analysis gives deep insight by considering optimum parameter for suggestion of modification in the existing connecting rod.
- 2) Optimization was performed to reduce weight. Weight can be reduced by changing the material of the current forged steel connecting rod to crackable forged steel (C-70).
- 3) The parameter consideration for optimization are its 20 % reduction in weight of connecting rod, while reducing the weight, the static strength, fatigue strength, and the buckling load factor were taken into account.
- 4) The optimized geometry is 20% lighter than the current connecting rod. PM connecting rods can be replaced by fracture splittable steel forged connecting rods with an expected weight reduction of about higher than existing connecting rod, with similar or better fatigue behavior
- 5) By using other fracture crackable materials such as micro-alloyed steels having higher yield strength and endurance limit, the weight at the piston pin end and the crank end can be further reduced. Weight reduction in the shank region is, however, limited by manufacturing constraints
- 6) The software gives a view of stress distribution in the whole connecting rod which gives the information that which parts are to be hardened or given attention during manufacturing stage.

REFERENCES

- [1] Prayardhan S. Shenoy, "Dynamic load analysis and optimization of connecting rod", 2004, Master's thesis, University of Toledo.
- [2] 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.
- [3] Webster, W. D., Coffell R., and Alfaro D., 1983, "A Three Dimensional Finite Element Analysis of a High Speed Diesel Engine Connecting Rod," SAE Technical Paper Series, Paper No. 831322.
- [4] ANSYS WORKBENCH 9.0 Reference Manual.
- [5] R. C. Patel, "Elements of Heat engine", vol-I, II.
- [6] Bhandari V. B., 1994, "Design of Machine Elements," Tata McGraw-Hill.
- [7] Pro/E Wildfire 3.0 Reference Manual.

[8] James R. Dale, "Connecting Rod Evaluation", January 2005.

[9] Ali Fatemi and Mehrdad Zoroufi, "Fatigue Performance Evaluation of Forged Versus Competing Process Technologies: A Comparative Study", 24th Forging Industry Technical Conference, Cleveland, Ohio, 2002.

[10] 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

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

IJARSE