Optimization and Fatigue Analysis of Modified Connecting Rod Using FEA Tool

Dnyandeep K. Nikam **Rupesh Tiwari**

Abstract **-The main aim of the present research is to determine total deformation, fatigue analysis and optimization 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, ANSYS. 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: - **CATIA V5 R20: For solid modelling,
ANSYS WORKBENCH 12: For Finite Element Analysis**

I. MATERIAL AND METHOD

Mechanical properties evaluated mainly include strength, ductility and fatigue. Similar alloys and heat treatment conditions are evaluated and benefits of forgings (in particular regarding fatigue properties) are compared to other products. A specific literature survey is also conducted for vehicle steering knuckle, which is used as an example part in this study. This survey includes material selection and manufacturing, stress analysis, fatigue analysis and life prediction, and optimization analysis. As mentioned earlier, some of the results from the literature survey are presented, after the analytical and experimental methods to be employed in the overall project are described.

Park et al. (2003) investigated micro structural behaviour at various forging conditions and recommend fast cooling for finer grain size and lower network ferrite content. From their research they concluded that laser notching exhibited best fracture splitting results, when compared with broached and wire cut notches. They optimized the fracture splitting parameters such as, applied hydraulic pressure, jig set up and geometry of cracking cylinder based on delay time, difference in cracking forces and roundness. They compared fracture splitting high carbon micro-alloyed steel (0.7% C) with carbon steel (0.48% C) using rotary bending fatigue test and concluded that the former has the same or better fatigue strength than the later. From a comparison of the fracture splitting high carbon micro-alloyed steel and powder metal, based on tension-compression fatigue test they noticed that fatigue strength of the former is 18% higher than the latter. [1]

Yoo, Y. M. (1984) had viewpoint of dependent feature-based modelling in computer-aided design is developed for the purposes of supporting engineering design representation and automation. This has two stages of mapping between models, and the multi-level model approach is implemented in three-level architecture. Top of this level is a feature-based description for each viewpoint, comprising a combination of form features and other features such as loads and constraints for analysis. The middle level is an executable representation of the feature model. The bottom of this multi-level modelling is an evaluation of a feature-based CAD model obtained by executable feature representations defined in the middle level. [2]. Athavale, S. and Sajanpawar, P. R., (1991) the mappings involved in the system comprise firstly, mapping between the top level feature representations associated with different viewpoints, for example for the geometric simplification and addition of boundary conditions associated with moving from a design model to an analysis model, and secondly mapping between the top level and the middle level representations in which the feature model is transformed into the executable representation. Because an executable representation is used as the intermediate layer, the low level evaluation can be active. The example will be implemented with an analysis model which is evaluated and for which results are output. This multi-level modelling approach will be investigated within the framework by Kwangchow Lee, Chris A. McMahon1 and Kwan H. Lee aimed for the design automation with a feature-based model. [3]

II. FINITE ELEMENT METHOD (FEM)

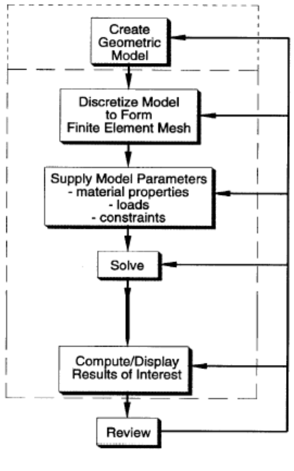

Fig.1 Graphical representation of FEM

Total Deformation - In Load Case 1 and load Case 2 Structural Loads and Supports are same but Condition is Axial Tensile and Compressive Load at Piston Pin End and Crank end restrained. Fig.2 and Fig.3 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	Fig.	Scope	Orientation	Min	Max	Min Occurs On	Max Occurs On	Alert Criteria
CASE 1	2	Mode 1	Global	0.0 mm	2×10^{-2} mm	Solid	Solid	None
CASE 2	3	Mode 1	Global	0.0 mm	2.25×10^{-2} mm	Solid	Solid	None

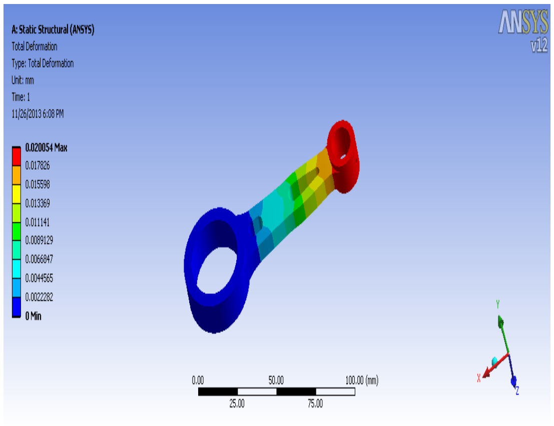

Fig. 2 Total Deformation for Load Case 1

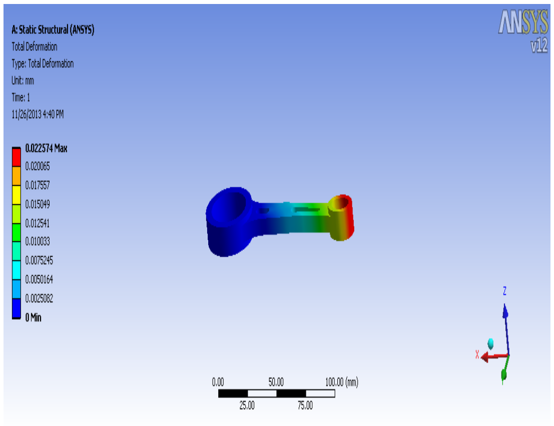

Fig. 3 Total Deformation for Load Case 2

III. FATIGUE ANALYSIS

Sonsino, C. M. and Esper, F. J., (1994) was 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 [4]. Ali Fatemi and Mehrdad Zoroufi ,according to them 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 Sused 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 [5].

Table 2 Fatigue Results

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	1.0×10^9	1,000.0	1,000.0	None
Safety Factor	5	Model	Safety Factor	1.0×10^9	0.2	15.0	None
Biaxiality Indication	6	Model	Biaxiality Indication		-0.9	0.94	None
Equivalent Alt Stress	7	Model	Equivalent Rev Stress		0.03 Mpa	83.585 Mpa	None

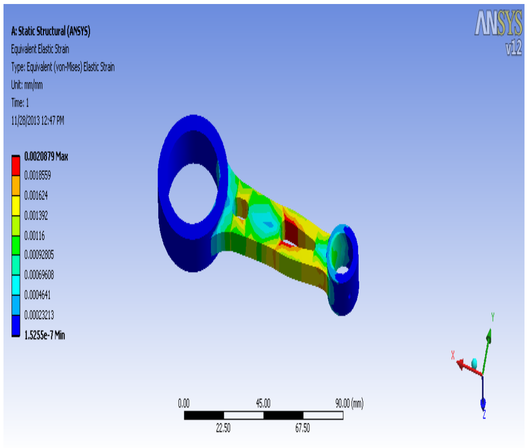

Fig.4 Equivalent (Von-Mises) Elastic Strain

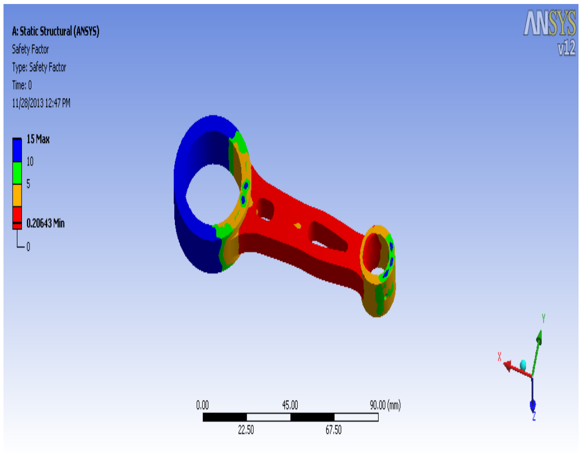

Fig. 5 Safety Factor

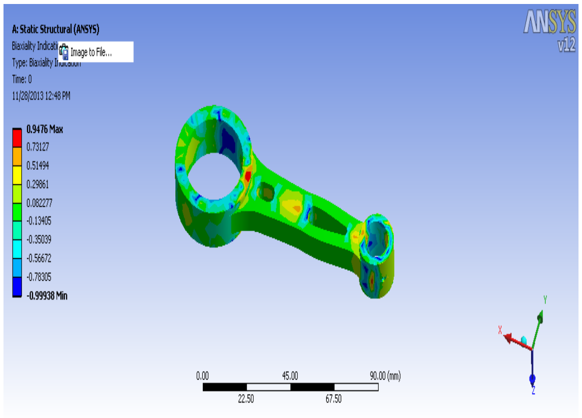

Fig. 6 Biaxiality Indication

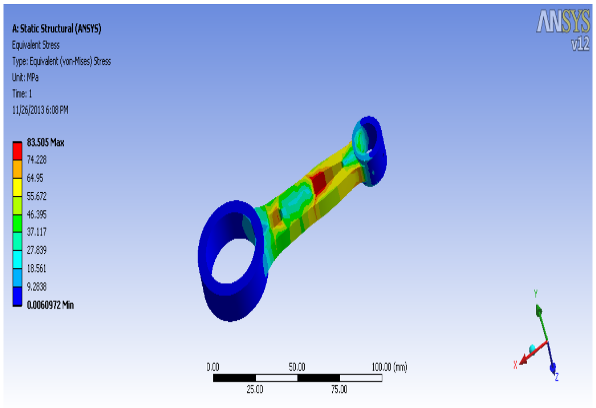

Fig.7 Equivalent Alternating Stress Optimization

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. The connecting rod design loads are peak gas load as the maximum compressive load. The connecting rod does have a potential for weight reduction. Due to high multiaxiality in a few regions of the connecting rod, equivalent multiaxial stress approach will be used for fatigue design during optimization. The load range for fatigue design will be the entire operating range as per the industry trend (Sarihan and Song, 1990). The entire operating range covers the maximum compressive gas load. It also highlights the fact that if the component is designed on the basis of axial static load or a load range based on the load variation at the crank end, it will be over designed. In actual operation, few regions of the connecting rod are stressed to much lower stress levels than under static load corresponding to the load at the crank end [6].

The objective is to optimize the connecting rod for its weight and manufacturing cost, taking into account the recent developments. Optimization carried out here is not in the true mathematical sense. Typically, an optimum solution is the minimum or maximum possible value the objective function could achieve under the defined set of constraints. This is not the case here. The weight of the new connecting rod or the 'optimized connecting rod' is definitely lower than the existing connecting rod. But this may not be the minimum possible weight under the set of constraints defined. What has been attempted here is an effort to reduce both the weight and the manufacturing cost of the component. Rather than using numerical optimization techniques for weight reduction, judgment has been used.

IV. SHAPE RESULTS

Table 3 Weight Reduction

Name	**Figure**	**Actual Reduction**
CONNECTING ROD	8	7.69 %

Table 4 Compared Weights

Name	**Original**	**Optimized**
CONNECTING ROD	0.13 kg	0.12 kg

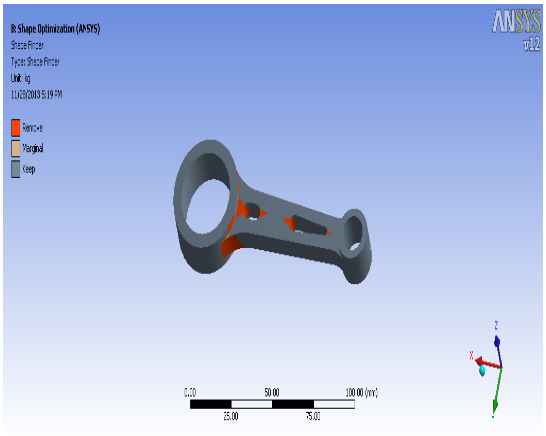

Fig. 8 Shape Finder

V. SUMMARY AND CONCLUSIONS

A lot has been done and still a lot has to be done in this field. In this project, only the static FEA of the connecting rod has been performed by the use of the software Catia V5R20 for cad modelling and ANSYS WORKBENCH for Finite Element Analysis. This work can be extended to study the effect of loads on the connecting rod under dynamic conditions. Experimental stress analysis (ESA) can also be used to calculate the stresses which will provide more reasons to compare the different values obtained.

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 7.69%

3) By using other facture crack able 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

4) 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] 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. 2003-01-13092]

[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] 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.

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

[5] 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.

[6] 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

[7] Bhandari V. B., 1994, *"Design of Machine Elements,"* Tata McGraw-Hill.

[8] ANSYS WORKBENCH 12.0 Reference Manual.