INTERNATIONAL JOURNAL OF ENGINEERING SCIENCES & MANAGEMENT

A Review -Analysis and Optimization of Connecting Rod

Sunil Kumar Shukla[1] and Soni Rani[2]
[1]Guide & Assistant Professor, [2]M.Tech Scholar, Mechanical Department, NRI-IST, RGPV, Bhopal

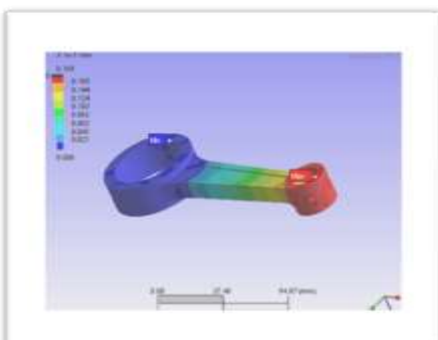
Abstract
The main aim of the present research is to determine total Deformation, Fatigue Analysis and Optimization in the subsisting Connecting rod. If the subsisting design shows the failure, then suggest the minimum design vicissitudes in the subsisting Connecting rod. In this research, only the static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

Keyword- Pro/ENGINEER Wildfire 5.0, Solid Modeling, ANSYS WORKBENCH 16.2, FEA

Introduction
Borse(2012) et al, performed the static FEA of the connecting rod using the software and said optimization was performed to reduce weight. Weight can be reduced by changing the material of the current forged steel connecting rod to crack able forged steel (C70). And 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.

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

Name	scope	Orientation	Mini	Maxi	Occur on	Criteria
Case 1	model	Global	0.0 mm	$1.82x10^2$	solid	None
Case 2	model	Global	0.0 mm	$1.82x10^2$	solid	None



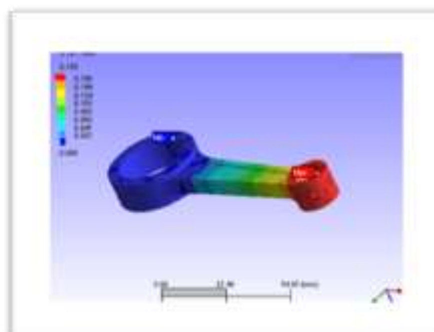


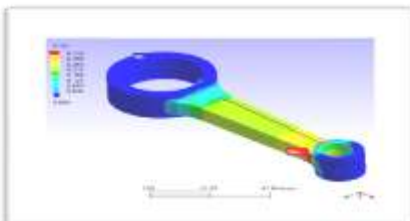
Fig. 1 Total Deformation for Load Case 1   Fig. 2 Total Deformation for Load Case 2

Fatigue Analysis:
It is estimated that 50-90% of structural failure is due to fatigue, thus there is a desideratum for quality fatigue design implement. The focus of fatigue in ANSYS is to provide utilizable information to the design engineer when fatigue failure may be a concern. A fatigue analysis can be disunited into 3 areas: materials, analysis, and results evaluation. An astronomically immense part of a fatigue analysis is getting a precise description of the fatigue material properties. These properties are included as a guide only with intent for the utilize to provide his/her own fatigue data for more precise analysis. Fatigue results can be integrated afore or after a stress solution has been performed. To engender fatigue results, a fatigue implement 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 implement is utilized to define the sundry 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 utilize. Outputs include fatigue life, damage, factor of safety, stress biaxiality, fatigue sensitivity.

ISSN 2277–5528
Impact Factor- 4.015

Table 2 Fatigue Results

Name	Scope	Type	Design life	Mini	Maxi	Alert criteria
Life Damage	Model	Life damage	$1.0x10^9$	10000000 10000	10000000 10000	None None
Safety Factor	Model	Safety Factor	$1.0x10^9$	1.12	14.0	None
Biaxiality	Model	Biaxiality indication		-1.0	0.96	None
Equivalent Alternating Stress	Model	Equivalent reversed stress		0.01MPa	0.75.22MPa	None



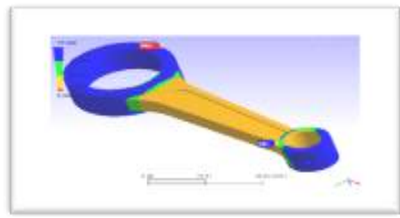
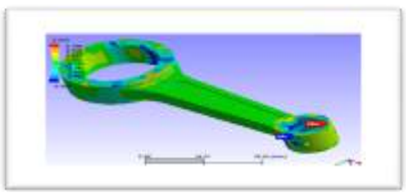


Fig.3 Equivalent (Von-Misses) Elastic Strain

Fig. 4 Safety Factor



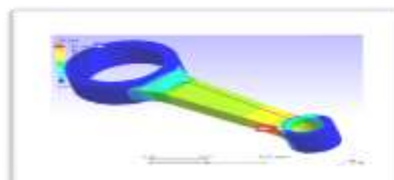


Fig. 5 Biaxiality Indication

Fig.6 Equivalent Alternating Stress

Optimization

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

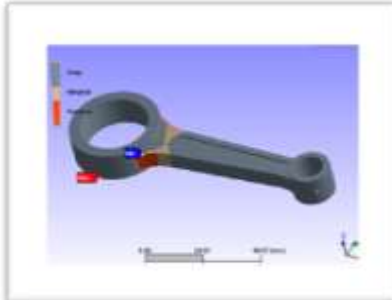
Shape Results

Table 3 Total Weights

Name	Scope	Target Reduction	Predicted Reduction
Shape finder	Model	25 %	9.24 % to 9.51%

Table 4 Total Weights

Name	Original	Optimized	Marginal
Shape finder	**0.12kg**	**0.11kg**	**$3.42x10^4$**



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 avail of CAD oftware Pro/E Wildfire 5.0 and then Load analysis was performed with different cases consideration. The analysis was carried out with computer availed simulation. The implement utilized for analysis is ANSYS WORKBENCH 16.2.

The Optimization carried out in analysis gives deep insight by considering optimum parameter for suggestion of modification in the subsisting connecting rod. Optimization was performed to reduce weight. Weight can be reduced by transmuting the material of the current forged steel connecting rod to crack able forged steel. The parameter consideration for optimization are its 25 % reduction in weight of connecting rod, while reducing the weight, the static vigor, fatigue vigor, and the buckling load factor were taken into account. The optimized geometry is 25 % lighter than the current connecting rod. PM connecting rods can be superseded by fracture splitable steel forged connecting rods with an expected weight reduction of about higher than subsisting connecting rod, with kindred or better fatigue deportment, By utilizing other facture crack able materials such as micro-alloyed steels having higher yield vigor 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, inhibited by manufacturing constraints the software gives a view of stress distribution in the whole connecting rod which gives the information that which components are to be hardened or given attention during manufacturing stage.

References

[1] Pravardhan S. Shenoy, "Dynamic load analysis and optimization of connecting rod", 2004, Master's thesis, University of Toledo.

[2] K. Sudershan Kumar, Dr. k. Tirupathi Reddy, Syed Altaf Hussan "Modeling and analysis of two Wheeler connecting rod", International Journal of Modern Engineering Research, Vol -2, Issue- 5, pp-3367-3371, Sep-Oct-2012.

[3] ANSYS WORKBENCH 16.2.0 Reference Manual.

[4] 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, March2012.

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

[6] Pro/E Wildfire 5.0 Reference Manual.

[7]] Pushpendra Kumar Sharma1, Borse Rajendra R, "fatigue analysis and optimisation of connecting rod using finite element analysis", International Journal Of advance research in Science and Engineering, Vol. No.1, Issue No. I, pp-3367-3371, September 2012.

[8] 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, ISSN: 2319-8753.