Research Article *ISSN: 2319-507X*
Akhare AV, IJPRET, 2013; Volume 1(8): 250-260 *IJPRET*

INTERNATIONAL JOURNAL OF PURE AND APPLIED RESEARCH IN ENGINEERING AND TECHNOLOGY

A PATH FOR HORIZING YOUR INNOVATIVE WORK

STRESS ANALYSIS OF CONNECTING ROD

A. V. AKHARE, Y. B. DUPARE, A. S. WAGH

1. Principal, Dr. Rajendra Gode Polytechnic, Amravati (M.S.)
2. Lecturer in Mech. Engg. Dept., Priyadarshini College of Engineering, Nagpur.
3. Lecturer in Mech. Engg. Dept., Dr. Rajendra Gode Polytechnic, Amravati.

Accepted Date:

27/02/2013

Publish Date:

01/04/2013

Keywords

Finite element analysis (FEM),

Connecting rod (CR),

CAD,

Optimization.

Corresponding Author

Mr. A. V. Akhare

Abstract

In the present paper a case study of “STRESS analysis of connecting rod” is carried out. In this paper, finite element analysis of single cylinder four stroke petrol engines is taken as a case study. The work has to find out the stresses at various points on the connecting rod and the portion, which is more susceptible to failure and optimization of connecting rod. To evaluate the magnitude and location stresses in the existing connecting rod. This is of great interest to the auto manufactures that which is the portion of the connecting rod which mainly fails so that they can use various methods of hardening the specific area by using special hardening treatments. In response to an increasing demand for fuel-economy, more weight reduction techniques have been proposed to create an optimum connecting rod design. The research aims to maximize weight savings in a connecting rod, without sacrificing the structural performances such as bending strength, buckling strength, and torsional stiffness.

Introduction:

This submission shows the implementation of the FEM software for the assessment of the strength and distortion characteristics of a connecting rod. A combination of axial and bending stresses acts on the rod in operation. The axial stresses are produced due to cylinder gas pressure (compressive only) and the inertia force arising in account of reciprocating action (both tensile as well as compressive), whereas bending stresses are caused due to the centrifugal effects.

To provide maximum rigidity with minimum weight the main cross section of the connecting rod is made an I-section is made to blend smoothly into two rod ends called the small end (piston end) and big end (crank end).

This work is to determine the Optimization in the existing Connecting rod. If the existing design shows the failure, then suggest the minimum design changes in the existing Connecting rod. The study 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.

2. Case Study

2.1 FE modeling of connecting rod

In geometric modeling, the graphic image of an object is generated on the graphic screen of the system by inputting three types of commands to the computer. A solid model of the connecting rod, as shown in Fig. 2.1 was generated using Pro/E Wildfire 3.0.

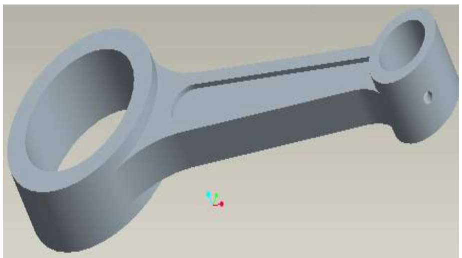

Fig. 2.1 Solid model of connecting rod

2.2 Stress analysis of connecting rod

2.2.1 Mesh Generation

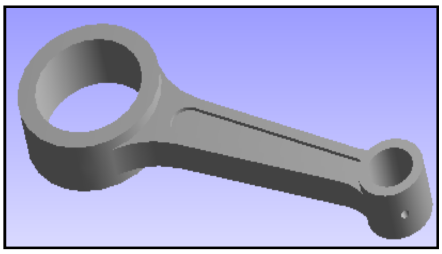

Fig. 2.2 Import model in Ansys Workbench 9.0.

Finite element mesh was generated using quadratic tetrahedral elements with element lengths as shown Table 2.2. First the connecting rod solid model is imported from PRO/Engineer wildfire 3.0 software in ANSYSWORKBENCH 9.0 as shown in fig.2.2. Once you have ensured regular shapes and assigned the appropriate divisions, generating the mesh is easy. Just press the Mesh button in the Mesh Tool, then press [Pick All] in the picker or choose the desired entities. After satisfyingabovecriteria, mesh generation has been done on each surface of component, then check for free edges of elements (equivalence of nodes), Apply quality checks for each surface and editing of meshing if it fails, Go on meshing for adjacent surfaces and check for nodal connectivity, any duplication, Finish the meshing of each surface of components.

The meshing model satisfying all quality requirements is prepared, final meshed model is shown in Fig. 2.3.

Table 2.1 Mesh Sizing Controls

Name	Type	Element Size	Associated Bodies
"Sizing"	Volume Sizing	5.0×10^{-3} m	"Solid"

Table 2.2 Element Types Summary

Generic Element Type Name	ANSYS Name	Description
10-Node Quadratic Tetrahedron	Solid187	10-Node Tetrahedral Structural Solid

Table 2.3 Bodies

Name	Material	Bounding Box(mm)	Mass (kg)	Volume (mm^3)	Nodes	Elements
"Solid"	"Structural Steel"	124.25, 40.0, 14.0	0.13	16,247.59	2680	1330

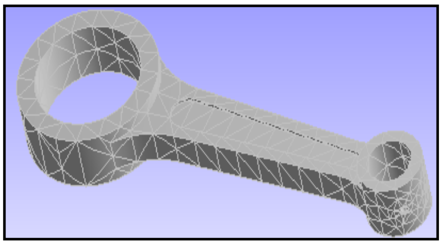

Fig. 2.3 Meshed Model of Connecting Rod.

2.2.1. Boundary and Loading Condition

Finite element analysis is to examine how a structure or component responds to certain loading conditions. Specifying the

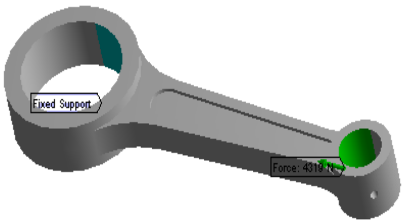

Fig. 2.4 FEA model of the connecting rod with axial compressive load at the piston pin end and crank end restrained.

The load analysis was carried out to obtain the loads acting on the connecting rod at any given time in the loading cycle and to perform FEA. Most investigators have used static axial loads for the design and analysis of connecting rods. However,

proper loading conditions is, therefore, a key step in the analysis. You can apply loads on the model in a variety of ways in the ANSYS program.

The word loads in ANSYS terminology includes boundary conditions andexternally or internally applied forcing functions, as illustrated Fig 2.4.

lately, some investigators have used inertia loads (axial load varying along the length) during the design process.

As a result, FEA was carried out under axial static load with no dynamic/inertia loads. The results of the above mentioned analyses are presented and discussed in this chapter with a view to use them for optimization. Static FEA results showed high stresses in the regions of the transitions to the shank at the crank end and piston pin end, the oil hole, and the cap.

Table 2.4 Stress Analysis Result

Name	Figure	Scope	Orientation	Minimum	Maximum	Minimum Occurs On	Maximum Occurs On	Alert Criteria
Equivalent Stress	5.1	Model	Global	0.01 MPa	76.22 MPa	Solid	Solid	None
Maximum Principal Stress	5.2	Model	Global	-9.72 MPa	35.68 MPa	Solid	Solid	None
Shear Stress	5.3	Model	XY Plane	-15.84 MPa	16.44 MPa	Solid	Solid	None
Total Deformation	5.4	Model	Global	0.0 mm	1.85×10^{-2} mm	Solid	Solid	None

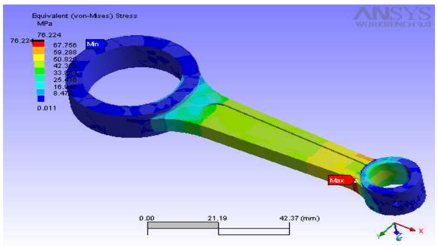

Fig. 2.5 Equivalent (Von-Mises) Stress.

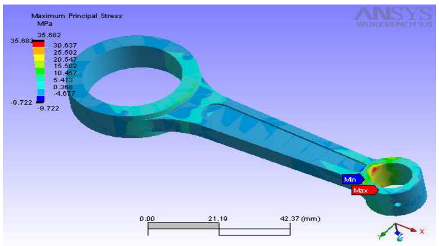

Fig. 2.6 Maximum Principle Stress

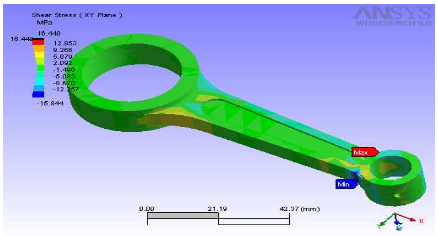

Fig. 2.7 Shear Stress (XY Plane).

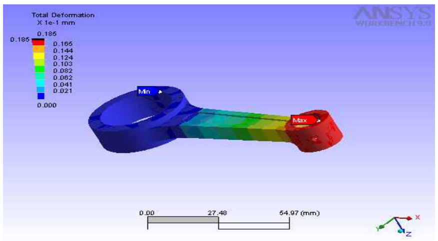

Fig. 2.8 Total Deformation.

2.3 Optimization

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.

This chapter discusses the constraints under which weight was reduced. It should be noted that the assembly-induced stresses are not included in the analysis.

2.3.1 Optimization Statement

Objective of the optimization task was to minimize the mass of the connecting rod under the effect of a load range comprising the two extreme loads, the peak compressive gas load, such that the maximum, minimum, and the equivalent stress amplitude are within the limits of the allowable stresses. The production cost of the connecting rod was also to be minimized. Furthermore, the buckling load factor under the peak gas load has to be permissible. 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 (Recommended FOS, 3 to 6).

2.3.2 Constraints

2.3.2.1 Applied loads

The load range under which the connecting rod was optimized is comprised of the compressive load of 4319 N, as discussed in the previous chapter. The compressive load of 4319 N is independent of the geometry of the connecting rod. The tensile load is, however, dependent upon the specific geometry, as it is a function of the mass, moment of inertia, and location of C.G

2.3.2.2 Allowable stress

Allowable stress is the ratio of yield strength to the factor of safety. A concept similar to factor of safety, the failure index (FI), was used in this work due to ease of processing the FEA results. The ratio of the von Mises stress (76.224 Mpa) in the existing geometry to the yield strength (330) of the existing material, referred to as the failure index (FI) (4.33), was obtained. The material chosen for the optimized connecting rod was structural Steel. As a result, the FI was defined with respect to the yield strength, rather than the ultimate tensile strength. As the name implies, failure index will be an indication of the failure possibility. The closer the FI to one, the higher the possibility of failure. A FI or factor of safety was assumed based on work by previous researchers. Either the assumed FI or the FI in the existing component, whichever was higher, was used for obtaining the allowable stress at a given location or region of the connecting rod.

One such critical location is the oil hole at the piston pin end of the connecting rod. FEA results predict a higher FI at this region than the assumed one. Since the new material has lower yield strength and fatigue limit, the region was redesigned to maintain the same FI as the existing connecting rod. At other locations the FI predicted by FEA for the existing geometry and material (connecting rod) was lower than the assumed value. As a result, the FI at these locations was raised closer to the assumed value in the optimized part.

In this paper calculated Factor of Safety is the ration of Yield strength to Maximum Von-Mises stress is 4.33 (higher than 1) and recommended FOS, (3 to 6). Then there is having scope for weight reduction. So unnecessary increase in weight of Connecting Rod is avoided. The Optimized figure is obtained when Solution done for optimization as shown in Fig. 2.9 indicates that the fillet side of existing connecting rod at least 9.24 % can be reduced. During the optimization result the weight of the connecting rod target reduction is 20 %, detailed result is given in table 2.5 and 2.6.

2.3.3 Shape Results

Table 2.5 Values

Name	Figure	Scope	Target Reduction	Predicted Reduction
"Shape Finder"	2.9	"Model"	20.0%	9.24% to 9.51%

Table 2.6 Total Weights

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

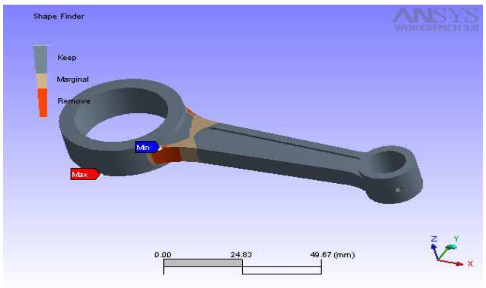

Fig. 2.9 Shape Finder

2.3.4 Observations from the Optimization Exercise

With manual optimization under static axial loading, at least 9.24 % weight reduction could be achieved for the same fatigue performance as the existing connecting rod as shown in fig. 5.33. This is in spite of the fact that C-70 steel has 18% lower yield strength and 20% lower endurance limit. Clearly, higher weight reduction may be achieved by automating the optimization and more accurate knowledge of load distributions at the connecting rod ends. The axial stiffness is about the same as the existing connecting rod and the buckling load factor is higher than that for the existing connecting rod.

C-40 has lower yield strength and endurance limit, As a result it was essential to increase weight in the pin end region. New fracture cracking materials are being developed (such as micro-alloyed steels) with better properties (Repgen, 1998). Using these materials can help significantly reduce the weight of the connecting rod in the pin end and crank end cap. However in the shank region, manufacturing constraints such as minimum web and rib dimensions for forge ability of the connecting rod present restrictions to the extent of weight reduction that can be achieved.

Considering static strength, buckling load factor, and fatigue strength, it was found that the fatigue strength of the connecting rod is the most significant and the driving factor in the design and optimization of connecting rod.

CONCLUSION

The following conclusions can be drawnfrom this study:

1) There is considerable difference in the structural behavior of the connecting rod between axial fatigue loading. The results obtained with the analysis tool are quite comfortable and can be used to optimize the model.
2) The Optimization carried out in analysis gives deep insight by considering optimum parameter for suggestion of modification in the existing connecting rod.
3) 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).
4) Fatigue strength was the most significant factor (design driving factor) in the optimization of this connecting rod.
5) 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.
6) The optimized geometry is 20% lighter than the current connecting rod. PM connecting rods can be replaced by fracture splitable steel forged connecting rods with an expected weight reduction of about higher than existing connecting rod, with similar or better fatigue behaviour.
7) By using other facture 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.
8) The stresses developed in the four load cases of connecting rod are below the yield value.
9) The stress multiaxiality is high, especially at the critical region of the crank end transition. Therefore, multiaxial fatigue analysis is needed to determine fatigue strength. Due to proportional loading, equivalent stress approach based on von Mises criterion can be used to compute the equivalent stress amplitude. Outputs include fatigue life, damage, factor of safety, stress biaxiality, fatigue sensitivity.

10) The buckling occurs in the rod is basically maximum at the piston pin end.
11) 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.
12) The software also reveals the importance of the varying I- cross section which is provided for uniform stress distribution over the entire web of the connecting rod.

REFERENCES

1. Pravardhan S. Shenoy, “Dynamic load analysis and optimization of connecting rod”, 2004, Master’s thesis, University of Toledo.

2. 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.

3. Repgen, B., 1998, “Optimized Connecting Rods to Enable Higher Engine Performance and Cost Reduction,” SAE Technical Paper Series, Paper No. 980882.

4. ANSYS WORKBENCH 9.0 Reference Manual.

5. Pro/E Wildfire 3.0 Reference Manual.

6. “DESIGN DATA”, Faculty of Mech. Engg., P.S.G. College of Technology, Coimbtore,

7. R. S. Khurmi and J. K. Gupta, “A Textbook of Machine Design”, 2005, S chand and Comp. Ltd., Delhi.

8. Hero Honda Splendor Company Manual.

9. Bhandari V. B., 1994, “Design of Machine Elements,” Tata McGraw-Hill.