Global Journal of Researches in Engineering: A
Mechanical and Mechanics Engineering
Volume 14 Issue 3 Version 1.0 Year 2014
Type: Double Blind Peer Reviewed International Research Journal
Publisher: Global Journals Inc. (USA)
Online ISSN: 2249-4596 & Print ISSN: 0975-5861

Review of Stress Analysis of Connecting Rod using Finite Element Analysis

By Swatantra Kulkarni, Ashwani Mishra, Himanshu Arora, Rajinder Singh, Prabhjot Singh & Ramanpreet Singh
Lovely Professional University, India

Abstract- Connecting rod is one of the important components of the whole engine assembly as it acts as a mediator between piston assembly and crankshaft. Also it faces a lot of tensile and compressive loads during its life time. So, a detailed analysis is the need of hour. For quick, better and accurate analysis CAD and FEA have proved very useful. This paper presents review on account of the developments done in the field of analysis, weight and cost reduction opportunities and better materials for connecting rod.

Keywords: finite element analysis, connecting rod, composite material, optimization, modelling.

GJRE-A Classification : FOR Code: 290501

REVIEWOFSTRESSANALYSISOFCONNECTINGRODUSINGFINITEELEMENTANALYSIS

Strictly as per the compliance and regulations of:

Review of Stress Analysis of Connecting Rod using Finite Element Analysis

Swatantra Kulkarni [α], Ashwani Mishra [σ], Himanshu Arora [ρ], Rajinder Singh [Ѡ], Prabhjot Singh [¥] & Ramanpreet Singh [§]

Abstract- Connecting rod is one of the important components of the whole engine assembly as it acts as a mediator between piston assembly and crankshaft. Also it faces a lot of tensile and compressive loads during its life time. So, a detailed analysis is the need of hour. For quick, better and accurate analysis CAD and FEA have proved very useful. This paper presents review on account of the developments done in the field of analysis, weight and cost reduction opportunities and better materials for connecting rod.

Keywords: finite element analysis, connecting rod, composite material, optimization, modelling.

I. Introduction

A connecting rod (Fig.1) acts as a link between the piston assembly and crankshaft thereby converting the reciprocating motion of piston into the rotary motion of crankshaft.

Around the globe connecting rod is produced in large quantity and furthermore it works under high tensile and compressive loads. So a connecting rod should be designed in such a way that it can withstand high stresses that are imposed on it. So its analysis is necessary.

It has mainly three parts namely- a pin end, a shank region and a crank end. Pin end is connected to the piston assembly and crank end is connected to crankshaft.

However the stress analysis can be performed easily by modelling it in any CAD software and analysing it by using FEA.

Discovering new techniques and methods for weight and cost reduction can definitely increase the engine performance and economy, thereby decreasing the inevitable centrifugal and inertial forces.

Moreover a search for new material can also be made for better results.

Authors α σ ρ Ѡ ¥: B. Tech Students, Department of Mechanical Engineering, Lovely Professional University, Phagwara, Punjab.
e-mail: swatantrakulkarni@gmail.com
Author §: Lecturer, Department of Mechanical Engineering, Lovely Professional University, Phagwara, Punjab.

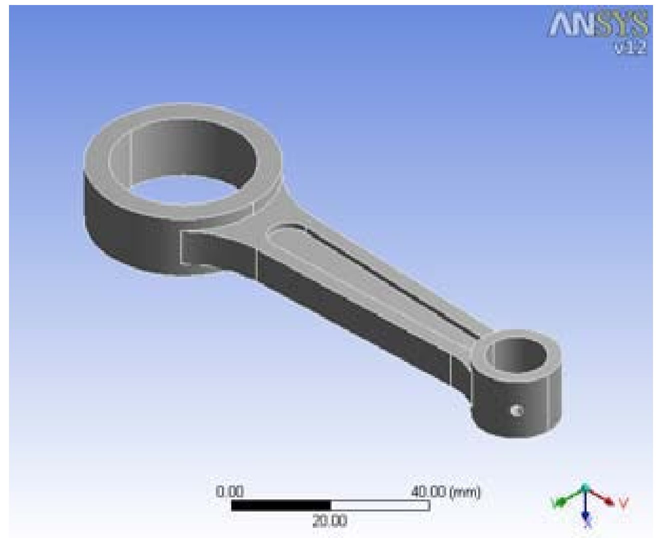

Figure 1 : Model of connecting rod in ANSYS

II. Literature Review

Anusha B et al. (2013) carried out the analysis and comparison of the cast iron connecting rod and structural steel connecting rod.Pro/E was used to model the rod and ANSYS Workbench was used for analysis. Hero Honda Splendor was taken as case study. However 3.15MPa of pressure was applied at piston end. Big end was kept fixed. It was observed that stresses induced in structural steel connecting rod were lesser than cast iron connecting rod and structural steel connecting rod showed improved results than connecting rod of other material. Moreover design was obtained safe for both materials. Finally it was recommended to use structural Steel connecting rod.

Anusha B. et al (2013) took Hero Honda Splendor connecting rod as case study. For investigation purposes 3.15MPa of pressure was applied at piston end. Big end was kept fixed. Pro/E (Creo Parametric) was used to model the rod and ANSYS was used for its analysis. As a result piston end was identified as the region under maximum stresses.

Rao G N M (2013) compared genetic steel, Aluminum, Titanium and Cast Iron for weight reduction opportunities of a connecting rod. Firstly load analysis was carried out then FEA and optimization were performed. After comparing the results, the study discovered that genetic steel connecting rod showed less deflection and stress as compared to other materials like Titanium, Cast Iron and Aluminum.

Pathade V.C. et al. (2013) used three methods for performing the stress analysis of connecting rod. The three methods were Finite element analysis, photoelasticity and theoretical method. For analysis through experimental method (photoelasticity) the casting of photoelastic sheet using Resin AY103 and Hardener HY951 was done and then model of photoelastic sheet was prepared. In FEA, Pro/E was used to model the rod and ANSYS Workbench 11.0 was used for analysis.The results from all the three methods were investigated and it was found that effect of stress concentration exists at both ends and it was found negligible in the middle portion of connecting rod. Moreover, small end was observed to be under more stresses than big end.

Prakash O. et al. (2013) carried out the case study of the connecting rod of Universal Tractor (U650). They re-optimized the connecting rod. Furthermore they also performed static and fatigue analysis. CATIA was used to model the rod and ANSYS V12 was used for its analysis. Critical areas were improved. Optimization led to the decrease in weight by 5gm and hence decreased the inertial and centrifugal forces.

Sarkate T. S. et al. (2013) took a case study of alloy aluminum 7068 and AISI 4340 alloy steel. Static analysis and comparison of both the above materials were performed. Pro/E Wildfire 4.0 was used to model the rod and ANSYS V12 was used for its analysis. As a result of analysis 63.95% weight of the rod was lowered and a decrement in stresses by 3.59% was identified with aluminum 7068 alloy.

Singh R (2013) took two materials for connecting rod–one was isotropic and other was orthotropic and carried out linear static stress analysis using Finite Element Analysis. The isotropic material was steel and orthotropic material was E-glass /Epoxy. CATIA V5R10 was used to model the rod and MSC.PATRAN was used for its analysis. The comparison indicated that E-Glass/ Epoxy showed a 33.99% stress reduction and 0.026% reduction of displacement. Furthermore, Mesh TET 10 was suggested for better results.

Vazhappilly C V et al. (2013) discovered weight and manufacturing cost reduction opportunities by taking into account the recent developments in several fields like finite element modeling techniques, optimization techniques, and developments in production technology etc. Additionally it also accounted the importance of CAD and FEA for the optimization purposes.

Kumar et al. (2012) used CAE tools for performing the optimization of connecting rod's parameters. Pro/E was used to model the rod and ANSYS Workbench 11.0 was used for its analysis. Static FEA was performed. Load was applied on big end and small end alternatively. As a result the piston end was observed to be subjected to maximum stresses. However the study suggested that by modifying design parameters, improvement in the existing results can be achieved. Furthermore increase in the material near the piston end was suggested to reduce the stresses and use of other materials like C-70 steel for the optimization was suggested.

Pal et al. (2012) used FEA for design evaluation and optimization purpose of connecting rod. Pro/E Wildfire 4.0 was used to model the rod and ANSYS V12 was used for its analysis.The study resulted in a reduction of weight by 0.477g and small end was observed under maximum stresses. It was concluded that change in design parameters can yield better results and increase in material in the stressed region can reduce stresses. Fatigue strength was identified as important parameter during designing and optimization purposes.

Pathade V.C. et al. (2012) performed theoretical as well as finite element analysis of I. C. Engine's connecting rod. Big end was kept fixed. Different loads were applied at small end. The rod was modeled in Pro/E and ANSYS was used for its analysis. As a result it was concluded that small end of connecting rod was observed under more stresses than big end.

Ranjbarkohan et al. (2011) carried out a case study of Nissan Z24 engine. Kinematic and kinetic analysis of slider-crank mechanism and stress analysis of connecting rod of Nissan Z24 engine was performed. Modeling was done in Solid Works and ANSYS software was used for analyzing purpose. Additionally the simulation of engine was performed in MSC/ADAMS/engine software. It was concluded that pin end faced maximum tensile stress. Moreover fatigue analysis of connecting rod was also recommended.

Thomas et al. (2011) inspected fatigue life of heavy duty application's connecting rod and observed that shot peening increased the fatigue life of connecting rod by 72% and thus shot penning was suggested for improving fatigue life cycles of connecting rod.

CIOATĂ et al. (2010) carried out the static analysis of connecting rod's foot. The rod was modeled in Autodesk Inventor Software and analyzed in ANSYS V11. It was identified that 0.036mm deformation of foot of connecting rod was obtained with FEA and 0.073mm was obtained with classical method. Furthermore the use of CAD software and software for finite element analysis was given due importance.

Shenoy P.S et al. (2005) worked for the optimization of steel forged connecting rod to discover weight and cost reduction opportunities. The study discovered that crackable forged steel (C-70), if used in place of forged steel connecting rod, can reduce the production cost by 25% and weight by 10%. It was also observed that the shank region possesses maximum margin for weight reduction. Moreover it was also recommended to consider fatigue strength during designing.

III. Conclusion

For the stress analysis of the connecting rod, it can be easily modeled in any CAD software like CATIA, Pro/E etc. and then it can be analyzed in any FEA software like ANSYS. With FEA we can get accurate results. It has been observed that small end is exposed to maximum stresses whereas the middle region of rod is subjected to negligible stresses.

Moreover better results can be achieved with changed or better design parameters and for reducing stresses we can increase the material at the small end. But this can increase the inertial and centrifugal forces. Moreover we can remove some material from the shank portion as it is observed as the greatest region for weight reduction.

Additionally fatigue strength is an important parameter to be considered while designing and optimizing and fatigue analysis should be performed. However it has been found out that shot peening can improve the fatigue life of connecting rod.

Additionally it has been found out that structural steel connecting gives better results as compared to cast iron connecting rod and Aluminum 7068 alloy performs better than AISI 4340 alloy whereas Genetic steel proves to be best when compared with Titanium, Cast Iron and Aluminum.

However by choosing different materials like E-glass/Epoxy, a composite material, C-70 steel etc. significant improvement can be obtained.

References Références Referencias

1. Cioata V G, kiss I. (2010) "Computer Aided Design of the Connecting Rod" *Faculty of Technical Sciences Novi Sad, Machine Design*.
2. Vazhappilly C V, Sathiamurthi P (2013) "Stress analysis of connecting rod for weight reduction-a review" *IJSRP, Vol 3, issue 2.*
3. Pathade C V, Patle B, Ingale A N (2012) "Stress Analysis of I. C. Engine Connecting Rod by FEM*". IJEIT , Vol. 1, No 3.*
4. Ranjbarkohan M, Asadi M R and Dardashti B N (2011) "Stress Analysis of Connecting Rod of Nissan Z24 Engine by the Finite Elements Method" *AJBAS.*
5. Anusha B., Reddy C V B (2013) "Modelling and analysis of two wheeler connecting rod by using ANSYS". *IOSR-JMCE, Vol. 6, Issue 5.*
6. Pathade. V C, Ingole D S (2013) "Stress Analysis of I. C. engine connecting rod by FEM and Photoelasticity". *IOSR-JMCE*, *Vol 6, Issue 1.*
7. Pal S, Kumar S (2012), "Design Evaluation and optimization of connecting rod parameters using FEM". *IJEMR, Vol. 2, issue 6.*
8. Prakash O, Gupta V, Mittal V. (2013) "Optimizing the design of connecting rod under static and fatigue loading*. IJRMST*, *Vol. 1, No.1.*
9. Thomas T G, Srikari S, Suman M L J (2011) "Design of connecting rod for heavy duty Applications produced by different processes for enhanced fatigue life". *sasTech Journal, Vol. 10, issue 1.*
10. Shenoy P S, Fatemi A (2005) "Connecting Rod Optimization for Weight and Cost Reduction" *SAE International.*
11. Singh R (2013) "Stress Analysis of orthotropic and isotropic Connecting Rod using Finite Element Method". *IJMERR, Vol. 2 No. 2.*
12. Sarkate T S, Washimkar S P, Dhulekar S S. (2013) "Optimization of Steel connecting rod by aluminum connecting rod using Finite Element Analysis" *Vol. 1, Issue 1.*
13. Anusha B, Reddy C V B (2013) "Comparison of Material for two-wheeler Connecting Rod Using ANSYS". *IJETT, Vol.4, issue 9.*
14. Rao G N M (2013), "Design optimization and analysis of connecting rod using ANSYS" *IJSR, Vol. 2, issue 7.*
15. Kumar A, Grover K, Budania B. (2012) "Optimization of connecting rod parameters using CAE tools". *IJLTET, Vol. 1, Issue 3.*

This page is intentionally left blank