

Design Optimization and Comparative Stress Analysis of Connecting Rod using Finite Element Analysis

Himanshu Arora¹

¹ Lovely Professional University, Punjab, India

Received: 10 December 2013 Accepted: 3 January 2014 Published: 15 January 2014

Abstract

A connecting rod is an important member of the engine assembly accompanied by mass production all over the world. Moreover it possesses great opportunity for research and development in terms of its material and its design parameters. In the present research work initially modelling of connecting rod was done in ANSYS 14.5 software and then analysed in again ANSYS 14.5 software itself. Analysis was mainly focussed on design parameters and its material. Then best design parameters and best material was chosen.

Index terms— connecting rod, finite element analysis, ortho- tropic material.

1 Introduction

connecting rod acts as a bridge between piston assembly and crankshaft. By acting as a bridge it changes the motion of piston assembly which is nothing but a reciprocating motion to the rotating motion of the crankshaft. Therefore it is an important member of the engine assembly. Moreover it also witnesses high cycle of tensile and compressive loading during its operation and researchers have found out that piston end witnesses maximum stress and that can be decreased by either increasing the material near the piston end or by changing the design parameters of the connecting rod. Also being an important member of the engine assembly its proper functioning and safety from failure is important because its failure can lead to the replacement cost of the engine assembly. Additionally we are witnessing huge production of connecting rod as its demand is increasing day by day. Therefore there is a need for research and development in the field of its design parameters, material and a lot more to widen our horizon for the betterment of connecting rod and engine performance. a) Literature Review Anusha B et al. (2013) did a case study on a Hero Honda splendor connecting rod. In this study piston end was made to bear a pressure of 3.15 Mpa whereas big end was kept fixed. Modeling and analysis were performed in Pro/E and ANSYS respectively. The whole study was focused on the comparison of Author ? ? ? ??: Department of Mechanical Engineering, B. Tech Students, Lovely Professional University Phagwara, Punjab. e-mail: arorahimanshu63@gmail.com Author ¥: Lecturer, Lovely Professional University Phagwara, Punjab. connecting rod made up of structural steel and cast iron which resulted in the fact that lesser stresses were induced in the structural steel connecting rod than cast iron connecting rod. Therefore connecting rod made up of structural steel was advised to use.

Vazhappilly C V et al. (2013) presented a literature survey of various research and developments that took place in the field of optimization technique, fatigue modelling, manufacturing cost analysis etc. Furthermore importance of CAD and Finite Element Analysis for optimization process was also presented.

Pathade V.C. et al. (2013) made three loads, 69kg, 85kg, and 99 kg, to be applied over the small end. Three approaches namely experimental, theoretical and numerical (Finite Element Analysis) were taken to do stress analysis of connecting rod. Comparison of the results of these methods was done and then it was noticed that small end was prone to more stresses than the other end.

2 b) Modeling

The connecting rod on which the present investigation is done is taken from the research paper of Pal S, Kumar S(2012), "Design Evaluation and optimization of connecting rod parameters using FEM" IJEMR, Vol. 2, issue 6 [7]. Modeling of the connecting rod is performed in ANSYS 14.5.

3 A c) Validation

The rod with the dimensions same as that of reference paper [7] was modeled and even the same compressive loads and fixed support were applied on the same portion of the connecting rod as in the reference paper [7]. Hence our model is validated as we have got approximately same results as in the reference paper.

After validation Changes in design parameters of model taken from reference paper [7] was made and 3 more models of connecting rod were prepared and static FEA was performed on all of them including existing model from reference paper by applying 2 different orthotropic materials on all of them one by one and they were compared. The drawing of existing and best model is shown below.

4 Material Properties

Two orthotropic materials were used and compared.

Table ?? : Properties of E-glass/Epoxy [8] Properties E-Glass/ Epoxy Longitudinal modulus E1(GPa) The crank end was kept fixed in every analysis and a compressive bearing load of 4319N [7] was applied.

41Gpa

5 III.

6 Results

All the models and materials were compared on the basis of maximum equivalent von-mises stress which was noted after every analysis.

7 Conclusion

Static FEA was performed on a connecting rod by applying a compressive bearing load on its piston end making the crank end fixed. The design of existing connecting rod taken from reference paper [7] was modified and all the designs were tested with 2 different orthotropic materials.

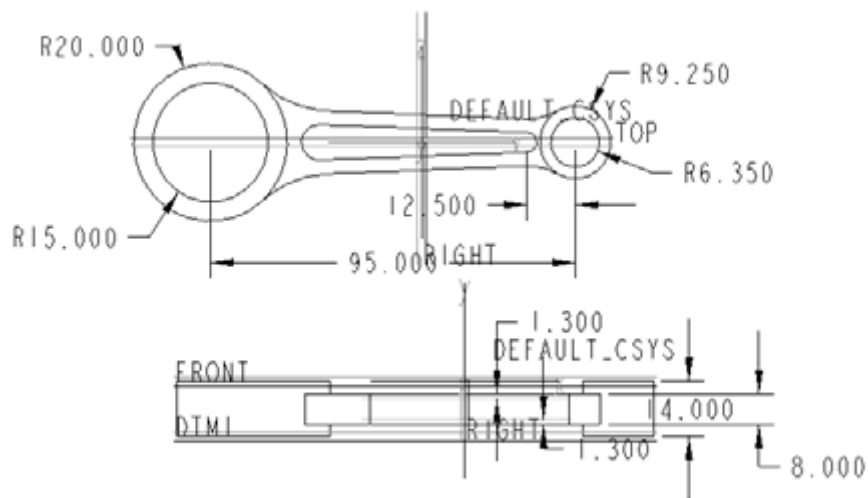
From analysis it was inferred that a) Modifying the design parameters can yield better results. Since modified model I gave best results. Therefore this design proved to be the best among others tested.

b) Among 2 orthotropic materials investigated and compared the E-glass/epoxy proved to be best. ¹



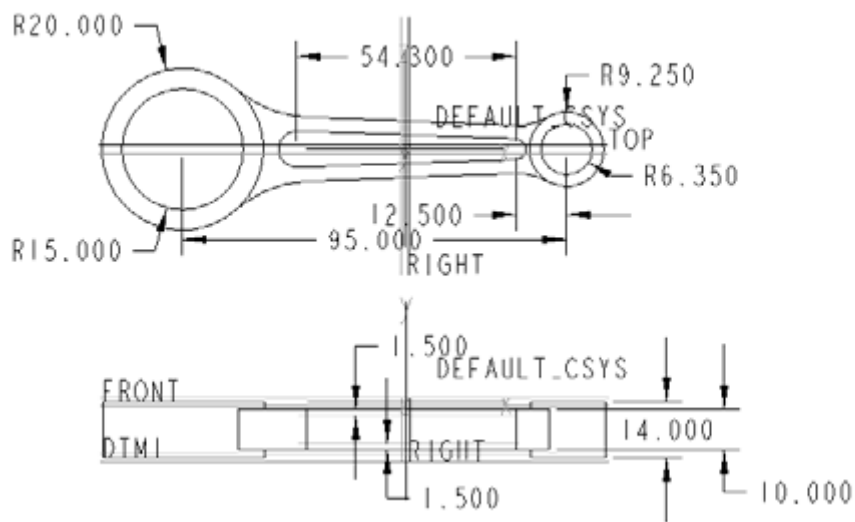
Figure 1:

¹© 2014 Global Journals Inc. (US)



1

Figure 2: Figure 1 :



2

Figure 3: Figure 2 :

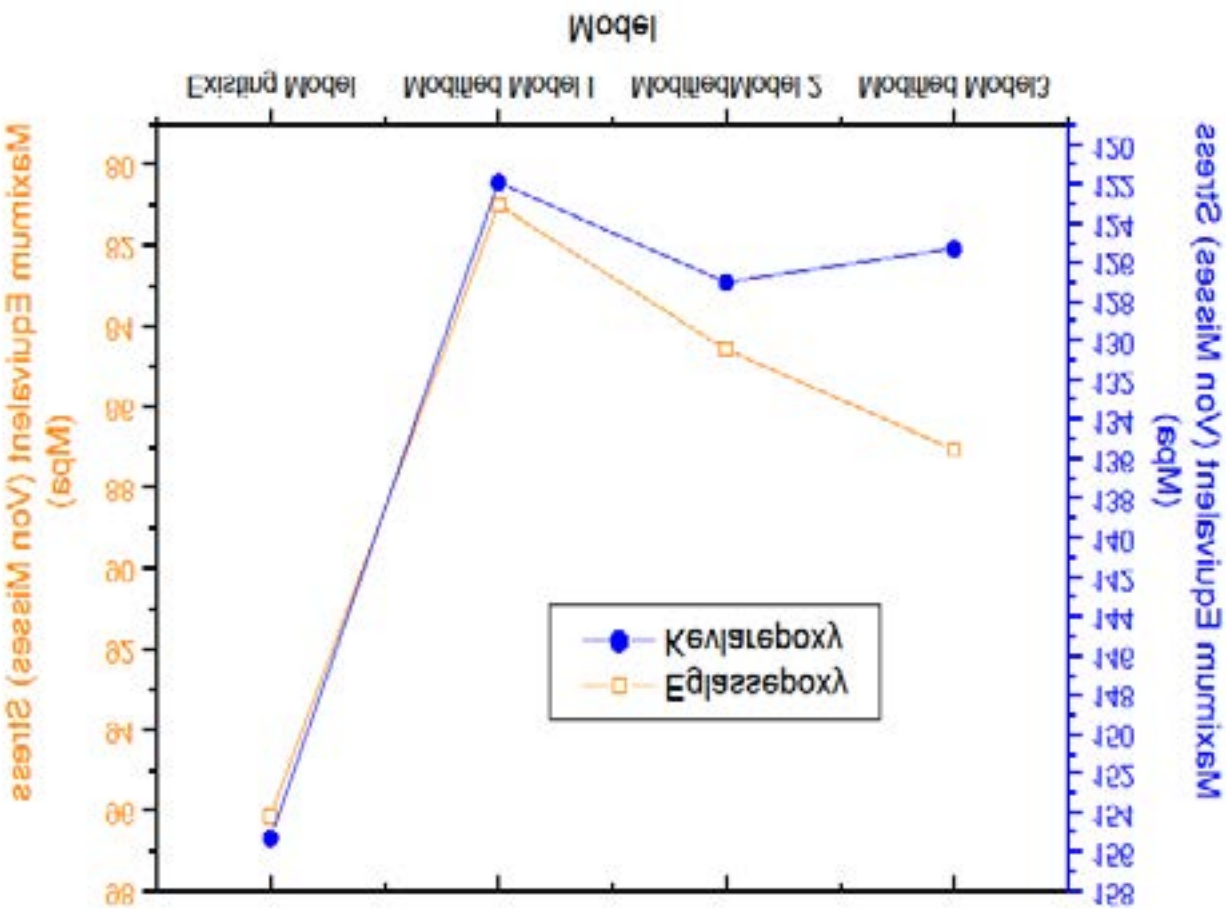


Figure 4:

1

von-mises stresses in Mpa are				
S. no.	Compressiv e Loads	Existing Model	Our Model	Variation (%)
1.	4319N	71.20	74.816	4.83

Figure 5: Table 1 :

4

Maximum equivalent von-mises stress Parameters	Eglass/ Epoxy (Mpa)	Kevlar49/ Epoxy (Mpa)
Existing Model	96.135	155.31
Modified Model I	80.995	121.95
Variation*(%)	15.75	21.479
Modified Model 2	84.567	127.03
Variation*(%)	12.033	18.20
Modified Model3	87.05	125.31
Variation*(%)	9.45	19.31

*[Note: *Variation (%) with existing model. The comparison presented in above table have also been presented by graph. Global Journal of Researches in Engineering]*

Figure 6: Table 4 :

.1 Graph for Table I

Following figures presents a comparison of static FEA between existing model from reference paper [7] and best modified design i.e. Modified Model I of connecting rod when compressive bearing load applied was 4319N and material used were E-glass/Epoxy and Kevlar49/Epoxy.

[Mechanical Materials Engineering Faculty Publications. Paper] , *Mechanical & Materials Engineering Faculty Publications. Paper 45.*

[Anusha and Reddy ()] ‘Comparison of Material for two-wheeler Connecting Rod Using ANSYS’. B Anusha , C V B Reddy . *IJETT* 2013. 4 p. 9.

[Pal and Kumar ()] ‘Design Evaluation and optimization of connecting rod parameters using FEM’. S Pal , S Kumar . *IJEMR* 2012. 2 (6) .

[Rao ()] ‘Design optimization and analysis of connecting rod using’. G N M Rao . *ANSYS* 2013. 2 (7) . (IJSR)

[Kasavajhala and Gu ()] *Fracture analysis of Kevlar-49/ epoxy and e-glass/epoxy doublers for reinforcement of cracked aluminum plates*, A R M Kasavajhala , L Gu . 2011.

[Sarkate et al. ()] *Optimization of Steel connecting rod by aluminum connecting rod using Finite Element Analysis*, T S Sarkate , S P Washimkar , S S Dhulekar . 2013. 1.

[Prakash et al. ()] *Optimizing the design of connecting rod under static and fatigue loading*, O Prakash , V Gupta , V Mittal . 2013. 1.

[Vazhappilly and Sathiamurthi ()] ‘Stress analysis of connecting rod for weight reduction-a review’. C V Vazhappilly , P Sathiamurthi . *IJSRP* 2013. 3 (2) .

[Pathade and Ingole ()] ‘Stress Analysis of I.C. engine connecting rod by FEM and Photoelasticity’. V C Pathade , D Ingole . *IOSR-JMCE* 2013. 6. (Issue 1)