Chapter 1

Diesel Engine Connecting Rod Design, Development and Analysis Using CAE Tools

L.Karikalan [a*], S.Baskar [c], S.Venugopal [c] , S.Ramasubramanian [b] , V.S.Shaisundaram [c]

[a] *Professor and Head, Department of Automobile Engineering, Vels Institute of Science, Technology and Advanced Studies, Chennai-600117, India*
[b] *Associate Professor, Department of Automobile Engineering, Vels Institute of Science, Technology and Advanced Studies, Chennai-600117, India*
[c] *Assistant Professor, Department of Automobile Engineering, Vels Institute of Science, Technology and Advanced Studies, Chennai-600117, India*

** Corresponding Author: karilk1972@gmail.com*

Abstract

An important component of an engine is the connecting rod, which is constructed of C-70 alloy steel. by taking composite material into account. Finite element analysis is used to determine the ideal arrangement of parameters for connecting rods, including deformation, Von Mises stress, and weight reduction. To determine the best connecting rod design, I-section and H-section are subjected to finite element analysis. With the aid of the ANSYS 14 Workbench programme, finite element analysis is carried out. In CATIA V5.0 software, the appropriate CAD model is created for this function. By comparing the static analysis of connecting rods with various cross-sections made of the same material, it was possible to determine that the maximum stress for the connecting rod I-section is lower than that of H-section. To examine the fatigue strength of the connecting rod, fatigue failure analysis is also carried out. To calculate the safety factor, life, stress and damage of the connecting rod, a fatigue tool is utilised.

Keywords: Piston, Engine, Design, Connecting Rod, Analysis

ISBN 978-819620981-0

1. Introduction

Connecting rods are commonly employed in a variety of in-line engines, opposed cylinder engines, V-engines, and opposite-piston engines. A component that can be used for analysis and for validating the results is necessary because this study is based on finite element analysis. A universal tractor connecting rod was selected as the component for this project. Higher performance parts that need to be designed inexpensively present a problem on the market for master products. Despite the greater expense of raw materials, a few automakers have shifted from using forged steel to powder metal connecting rods [1-3]. However, a composite material can be significantly improved by using other materials, such as E-glass/Epoxy. Engine connecting rods must be strong while remaining reasonably lightweight.

The majority of researchers investigated different types of connecting rod fatigue, static stability, and dynamic stability. Different cross-section connecting rod must be thoroughly evaluated since connecting rod design is extremely significant from a cross-section point of view. the choice of material and the manufacturing procedures discovered in earlier literature [4]. The existing methods of making connecting rods via casting, drop forging, and powder forging are compared using available data. The primary objective of this assignment is to lower the weight of connecting rods while preserving their strength. The purpose of comparing E-glass/Epoxy with C-70 alloy steel connecting rod materials is to reduce connecting rod stress and weight based on the aforementioned literature gaps. Here, fatigue research was conducted to evaluate fatigue resistance.

Today's primary objective is to expand the structural modelling of tractor connecting rods using CATIA V5 software, which is subsequently loaded into ANSYS software utilising the ANSYS 15 workstation for further analysis. Connecting rods come in a variety of sizes and forms and are used for the same thing. Finding. The primary objectives of this work are to calculate Von-Mises stresses for H-section and I-section

ISBN 978-819620981-0
9 788196 209810

connecting rods and select the optimal design for long-term durability and engine performance.

2. Literature Review

The optimization of connecting rods and the dynamic load studies have been discussed by many academics. This project's primary objective was to demonstrate the potential for weight and cost reductions in forged steel connecting rods. After this work's analysis is complete, the connecting rod could be optimised for a load that included extreme loads caused by peak gas pressure and tensile loads comparable to 360" crank angles at the highest engine speeds. Additionally, it was detailed how an iterative process was used to achieve the weight loss [5-7]. This study optimises weight under cyclic loads with two excessive loads: static compressive load and dynamic tensile load. The results indicate that an optimised connecting rod is 25% less expensive and 10% lighter than the current model.

By using the finite element method, designers were able to achieve the greatest stresses in various tractor connecting rod sections. In this work, a comprehensive load study for the connecting rod was completed before using the finite element approach. The total forces were calculated, meshes, models, and loads were made in ANSYS V-9 software in order to estimate connecting rod stress. The highest stresses were firmly applied to different connecting rod parts [8-9]. For better fatigue life, the connecting rod design for heavy duty applications was produced using a different technique. The fundamental objective of this study is to restore the connecting rod by including production process influences into the research and achieving superior fatigue performance.

Numerous studies using the finite element approach for connecting rod parameter optimization and design assessment were published [10-11]. The first portion of this experiment involved the connecting rod being subjected to static loads, while the second half was concerned with safe design. The connecting rod's mass is additionally

ISBN 978-819620981-0

compressed by 0.477g. Since the strained area of the piston is where it is most noticeable, material there is expanded to reduce stress. According to this analysis, tensile stresses are greater than compressive stresses. The connecting rod's shank area experienced lesser stresses than the pin end or crank end, whereas the transition between the two ends experienced the highest stress. The stress analysis identifies the maximum stress point, deformation, and problematic region. The mesh used in this investigation is varied. To determine the connecting rod's service load, a dynamic load analysis was conducted, and FEA was used to calculate stresses at key places. The big end centre and small end bearing inner surfaces show the most distortion.

3. CAE Tools

Software simulation is used in CAE to advance product design or assist in the resolution of engineering issues. These consist of process and contemporary tools, as well as validation, simulation, and optimization. It involves optimization, computational fluid dynamics, multi-body dynamics, and finite element analysis. The performance and strength of the components are examined using CAE tools. The phrase "CAE" refers to the application of computer expertise in manufacturing for engineering research. When analysing the robustness of assemblies and components, CAE tools are referred to as software developed to prevent these actions. The analytical capacity of CAE tools has increased to the point where a significant amount of design validation is now performed through computer simulation rather than physical model testing.

The primary objectives are to analyse and evaluate the mechanical stress, thermal stress, and distribution of connecting rods during combustion. This study describes the FEA approach for predicting the high stress and crucial placement of a component. The connecting rod structural model will be created using the CATIA V5 programme, and stress analysis and simulation will be carried out using the ANSYS software.

ISBN 978-819620981-0
9 788196 209810

4. Result and Discussion

In the figures 1, 2, 3, 4, 5, 6, 7, and 8, respectively, the FEA findings for the static analysis of connecting rods with I-sections and H-sections are displayed. The Von-Misses stress maximum for the H-section is determined by comparing the two cross-sections of the identical material (C-70 alloy steel). As a result, the connecting rod with an I-section will be used in the design below the various forces and stresses. The greatest Von-Mises stress for compression load at the crank end is 22.682MPa for I-section and 40.167MPa for H-section for I-section. The huge end fillet of the connecting rod is where you might feel the most stress.

The maximum von-misses stress for compression loading at the pin end is 34.423MPa for I-section and 50.432MPa for H-section. At the small length of the connecting rod at the end of the pin, the stress is most apparent. Figure.1 depicts the profile of a comparable Von-Misses stress. The crank end of the connecting rod endured the most stress, whilst the piston pin end experienced the least. As illustrated in figure. 4, shear stress is greatest on one side of the crank end and least on the other. At the end of the crank and the end of the piston, the overall deformation and elastic strain are greatest. In the specific figures, blue represents the lowest stresses, strain, and overall deformation, while red represents the highest stress portion. The colours reflect the contour of stresses, strain, and total deformation.

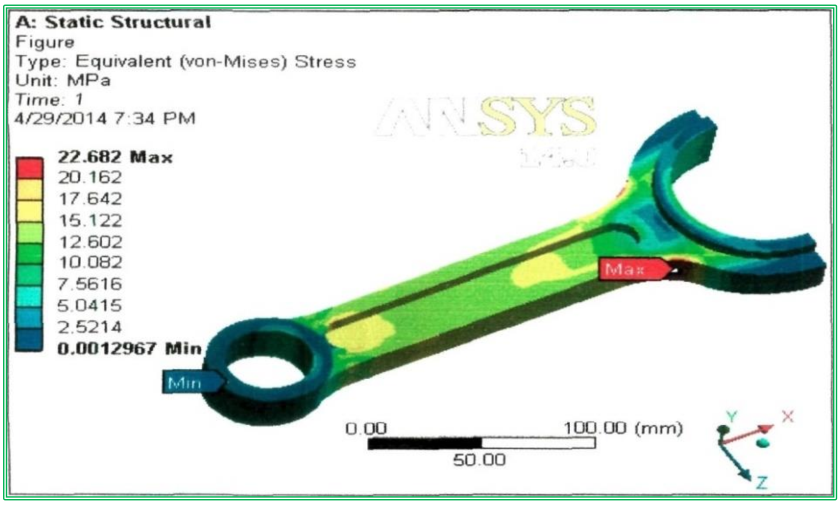

Figure. 1: Von-Mises Stress at 'I' section

ISBN 978-819620981-0

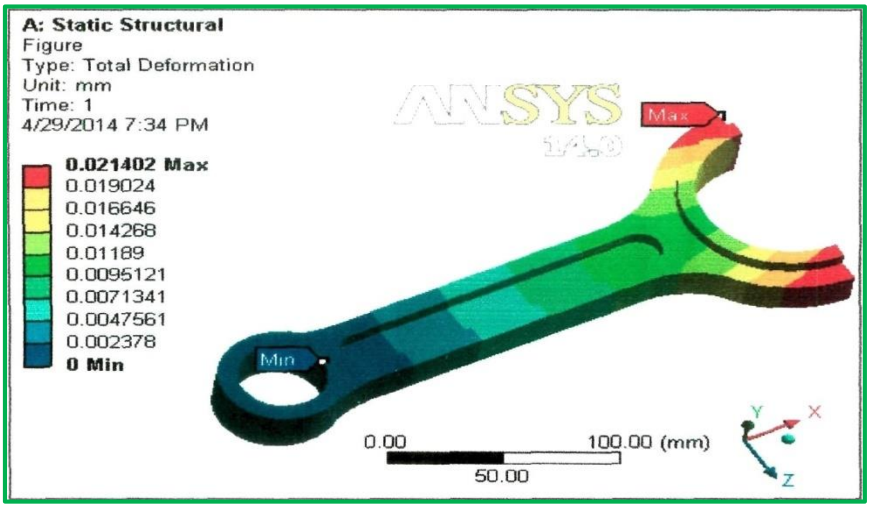

Figure. 2: Total deformation at 'I' section

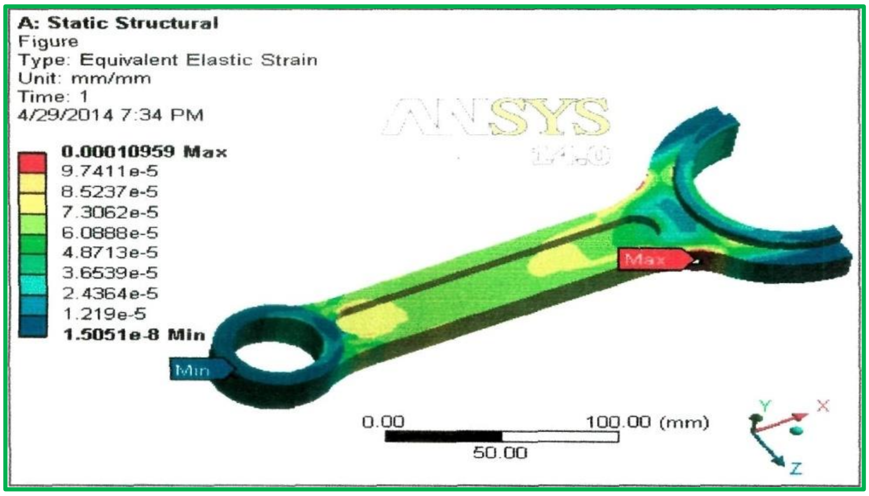

Figure. 3: Elastic Strain at 'I' section

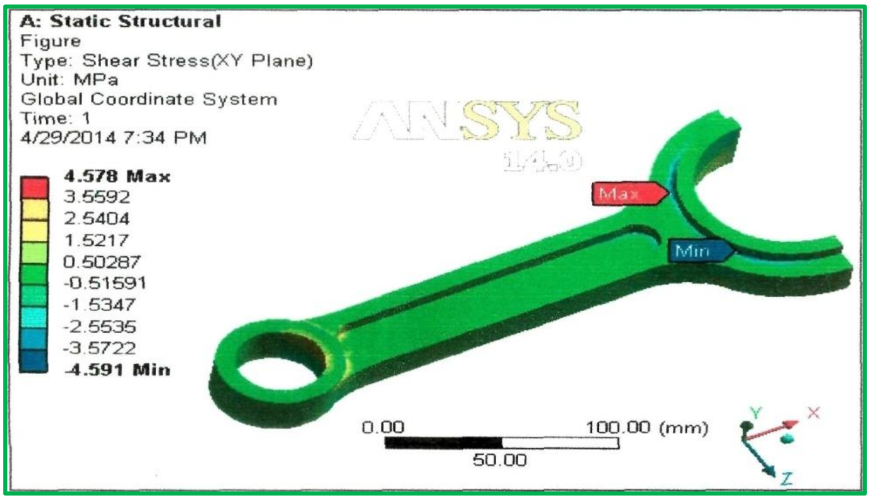

Figure. 4: Shear Stress at 'I' section

ISBN 978-819620981-0

Figure. 5: Von-mises Stress at 'H' section

Figure. 6: Total deformation at 'H' section

Figure. 7: Elastic Strain at 'H' section

ISBN 978-819620981-0

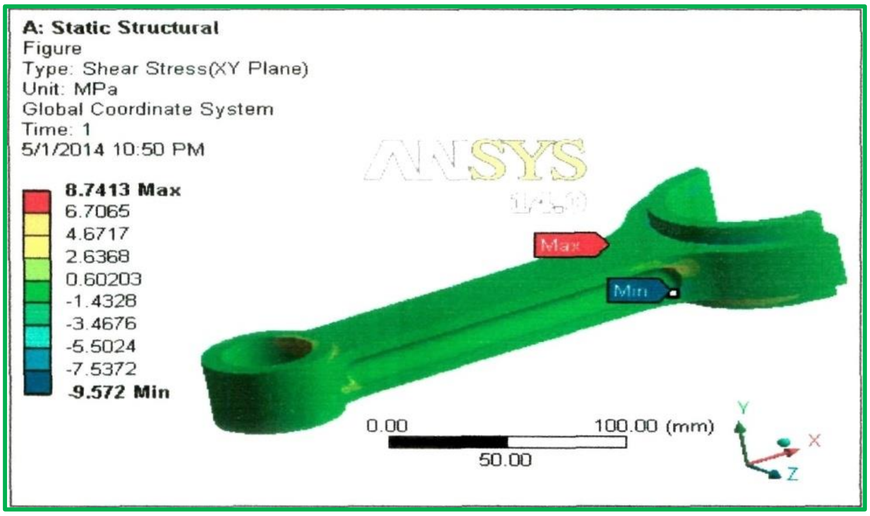

Figure. 8: Shear Stress at 'H' section

4.1 FEA Comparison for Different Cross-Section of Connecting Rod

The figure above depicts the FEA results for the static analysis of I- and H-section connecting rods. The Von-Mises stress maximum for the H-section is determined by comparing the two cross-sections of the identical material (C70 alloy steel). I-section connecting rods are superior to H-section connecting rods for a variety of loads and forces.

Table 1: Compression Load Static Analysis at Crank end

S.No.	**Parameters**	**FEA outcome**	
		H-section	I-section
1	Elastic strain (mm/mm)	0.00019405	0.00010959
2	Shear stress (MPa)	8.7413	4.578
3	Equivalent Von Mises stress (MPa)	40.167	22.682
4	Total Deformation (mm)	0.020114	0.021402

Table 2: Compression Load Static Analysis at Pin end

S.No.	**Parameters**	**FEA outcome**	
		H-section	I-section
1	Elastic strain (mm/mm)	0.00024411	0.00016635
2	Shear stress (MPa)	13.79	7.314
3	Equivalent Von Mises stress (MPa)	50.432	34.423
4	Total Deformation (mm)	0.020072	0.021282

ISBN 978-819620981-0

5. Conclusion

The tractor connecting rod's finite element analysis was finished using the FEA tool ANSYS 14 Workbench. Numerous discussions have been created using the results of the FE analysis. The finding is in line with comparable outcomes that are already on hand. The current model is safe and within permissible stress boundaries. The results of the investigation for two different cross-sections revealed that the I-section is subjected to less stress than that of H-section. I-section could therefore be employed for improved design of connecting rod.

References

[1] A text book of “Machine Design”, R.S.Khurmi and J.K.Gupta. S Chand; 25th edition, 2020.

[2] A text book of “Automobile Mechanics”, Dr.N.K.Giri, Khanna Publishers; Special Edition, 1998.

[3] A text book of “Automobile Design”, R.N.Bahl, Dreamtech Press, 2019.

[4] B.K.Roy, “Design Analysis and optimization of various parameters of connecting rod using CAE softwares”, International Journal of New Innovations in Engineering and Technology, Vol.1(1), pp.52-63, 2012.

[5] A.Singh, A.Deb,.M.Mensi and R.S.Gunti, “A Unified CAE Framework for Assessing an IC Engine Design (No. 2015-01-1664)”, SAE Technical Paper, 2015.

[6] S.Wable, D.S. Galhe and R.L.Mankar, “A Review on Design and Optimization of connecting rod”, International Journal on Applied Research in Industrial Engineering, Vol. 2(1), pp.261-270, 2016.

[7] S.Kaliappan, S.M.D.P.Nagarajan and M.R.Kamal, “Analysis of an Innovative Connecting Rod by using Finite Element Method”, Taga Journal of Graphic Technology, Vol.14, pp.1147-1152, 2018.

[8] L.Karikalan and P.Sakthivel, “Assessing the diesel engine performance using piston with dissimilar material piston rings”, Materials Today: Proceedings, Vol.21(1), pp.796-799, 2019.

[9] M.Bulut and O.Cihan, “Stress and deformation analysis of a connecting rod by using ANSYS”, International Journal of Automotive Engineering and Technologies, Vol.9(3), pp.154-160, 2020.

[10] A.Muhammad, M.A.H.Ali and I.H.Shanono, “Finite Element Analysis of a connecting rod in ANSYS: An overview”, IOP Conf. Series: Materials Science and Engineering 736 (2020) 022119, doi:10.1088/1757-899X/736/2/022119, 2020.

[11] H.Zhou, S. Liu, G. Li, G. Tian, Z.Wang, and C.Wang, “ Machining stress analysis and deformation prediction of connecting rod based on FEM and GRNN”, Iranian Journal of Science and Technology, Transactions of Mechanical Engineering, Vol.44(1), pp.183-192, 2020.

ISBN 978-819620981-0

9 788196 209810