Review Article

Investigation of Stresses at Critical Location for Shape Optimization of Connecting Rod using Finite Element Analysis

Anant Prakash Agrawal[1] and Shahazad Ali[1]

[1]Department of Mechanical Engineering,

Noida Institute of Engineering and Technology, Greater Noida, UP- 201306, India

Corresponding Author: anant.agraawal@gmail.com

Abstract

Connecting rod is an element that is used to transmit power from the piston to the crankshaft in an IC Engine. The behaviour of connecting rod is affected by the fatigue phenomenon because of reversible cyclic loadings. Fatigue is the primary cause of catastrophic failure of a connecting rod in an IC Engine. In the present work, shape optimization of the connecting rod is carried out by finite element analysis under reversible cyclic loading. In every phase of reversible cyclic loading, the stresses are generated and the critical locations on the connecting rod are located. The modelling of the connecting rod is carried out on CATIA, and ANSYS workbench is used for the FEA. The effects of design parameters investigated, such as fillet radius, groove depth, and groove length, based on maximum stresses generated at their critical location. With an increase in fillet radius, and decreasing groove depth and length, the maximum stresses generated were 15.07%, 6.31%, and 7.55% reduced, respectively, due to a decrease in stress concentration. Finally, the optimized model has reduced maximum stress at the bigger end of the connecting rod up to 26.44%. This connecting rod now has better longevity during the operation.

Keywords: Connecting rod, shape optimization, cyclic fatigue loading, FEM, FEA.

1. Introduction

In an IC engine, connecting rod is an intermediate link between the piston and the crank, which converts reciprocating motion to rotating motion. The connecting rod is subjected to a fatigue phenomenon of reversible cyclic loading of an order of 10^8 to 10^9 cycles as these forces are transmitted to the connecting rod through the piston, a significant compressive force act due to the combustion of fuels and a large tensile force due to inertia (Agrawal & Srivastava, 2012).

Manufacturing of connecting rods through casting process; however, from fatigue failure and durability point of view of connecting rods, castings could have some defects such as blow-holes and porosity (Agrawal & Srivastava, 2012). By forging, blow-hole and porosity can be eliminated, which gives an advantage over the cast rod (Tevatia, Lal & Srivastava, 2011). Generally, the shape of connecting rods is designed as I-section to provide maximum strength with minimum weight. The maximum stresses generated near the piston end of connecting rod can be reduced by giving excess material near the smaller end.

Parkash Gupta & Mittal, (2013) identified critical locations on connecting rods under the static and dynamic analysis of universal tractor (U650) under fatigue loading. The connecting rod is modeled using CATIA, Pro-E, and analyzed through ANSYS workbench. They optimized the model of connecting rods through weight reduction and improved life and manufacturability for better performance.

Fatigue strength is the most important consideration in the shape optimization of connecting rods (Tiwari, Tiwari & Chandrakar, 2014). Fatigue analysis and life prediction can be performed using three approaches: stress-life theory, strain-life theory, and crack growth (Agrawal & Srivastava, 2012). Since connecting rod of an engine bears static and dynamic fluctuation under loadings, it is one of the most critical components. Kumar & Kumar, (2015) improved the strength-to-weight ratio and optimized the shape of connecting rods of forged steel, grey cast iron, aluminum, and titanium connecting rod under varying loads by changing their cross-sectional area as the stress induced at the smaller end more significantly than, the larger end of connecting rod (Pathade & Ingoel, 2013). Some researchers analyzed connecting rods made of composite material Al6061 as highly deformable than Al6061+B_4C (Kumar, 2015).

Due to repetitive tensile and compressive stresses (reversible cyclic loadings), leads to the developing fatigue phenomenon, which causes dangerous ruptures and damage to CR (Agrawal & Srivastava, 2012; Roy, 2012). Agrawal and Srivastava (2012) modeled a forged steel crankshaft on Pro/E, whereas the ANSYS workbench is used for the FEFA under cyclic loading. They investigated the effects of crank-pin fillet radii, crank-pin oil hole dia., crank-web thickness, inner dia., and depth of drilled hole at the back of crankshaft based on maximum von-mises stresses generated at the critical location and predicts 13.5% lower Von Mises stresses as compared to the initial design.

Bharti, Singh, and Hussain (2013) investigated that maximum stress was generated at the crank end (Desai, Jagtap & Deshpande, 2014). They reduced 11.23%, 12.65%, and 10.56% weight of the I, +, and ellipsoidal sections of the C-70 connecting rod, respectively. Anusha and Reddy (2013) performed finite element analysis on the two-wheeler Hero Honda Splendor's connecting rod and observed that maximum stresses developed at the piston end of the connecting rod. Tiwari, Tiwari, and Chandrakar (2014) investigated 10% weight and 25% cost reduction optimization for C-70 steel than forged steel connecting rods. The optimized model was efficient in design as the stresses are lesser than the existing model. A weight reduction of about 3.5% was achieved in the material steel's optimized design (Shanmugasundar et al., 2021).

Tevatia, Lal, and Srivastava (2011) analyzed I, +(plus), and H sections connecting rods of equal masses and also investigated the effects of critical dimensions such as f_R, D, and H based on maximum von-mises stresses generated at their critical locations. They concluded from their research that H section CR is unsuitable against fatigue failure for the entire range of fillet radius, inner dia., and height of the big end compared to I and + section (Tevatia *et al*., 2011; Lal, Tevatia & Srivastava, 2010). Saxena and Ambikesh (2021) modeled and analyzed the stresses developed on the Splendor motorbike connecting rod and observed that Aluminium alloy 7475 had the best material than Carbon steel and Titanium alloy Ti-6Al-4V.

Shenoy and Fatemi (2006) investigated the state of stress at various locations on connecting rods under service operating conditions. Due to the increase in inertia load, the maximum and mean stresses increase with increasing engine speed. The axial stresses are produced in the connecting rod due to gas pressure in the cylinder, whereas bending stresses are developed due to the centrifugal effects (Pathade & Ingoel, 2013). Agrawal, Ali, and Rathore (2022) investigated the effect of design parameters such as fillet radius and addendum on the maximum stresses generated at the fillet radius of the root of spur gear.

2. Problem Formulation:

2.1 Failure of Connecting Rod:

There are many causes of the failure of engine components. One of the causes of failure of an engine component connecting rod at the fillet areas due to reversible cyclic loading during its service life. Due to the geometry of connecting rod and engine mechanism, the connecting rod fillet has a maximum stress range. Due to the combustion of fuels inside the cylinder, the load is transmitted from the piston to connecting rod, causing a significant bending moment to develop on the entire geometry of connecting rod. Due to stress concentration at the fillet areas, a higher stress-activated and these locations/points act as a critical locations, where cyclic loads could cause fatigue crack initiation, which leads to fracture.

For designing and optimizing any engine component, identifying its causes of failure must be a critical factor. Fillets behave as stress raisers on the connecting rod surface; therefore, cracks may develop at their surface and grow inward due to combined cyclic bending and torsion loads.

2.2 Finite Element Modelling:

In the finite element parametric modeling phase, design parameters, components' dimensions, and their features are used and develop a relationship that captures the intended product behavior. The accuracy of models plays an important role in the Finite Element Analysis (FEA) of connecting rods to give the closest results. Modeling of any component represents a set of principles for mathematical and computer modeling of 3D solids. The basic dimensions of the I-section are 58 mm thickness connecting rod of 2500 mm length between big and small end centers. With the help of the geometry (2D) of the I-section connecting rod, a three-dimensional model of connecting rod is generated using CATIA V5 SOFTWARE, as shown in figure 1.

Figure 1. 3D solid model of Connecting Rod

2.3 Finite Element Analysis:

In this study, finite element analysis of connecting rod is carried out on ANSYS workbench software. The convergence is achieved, for the entire range of elemental length, using Ten–node tetragonal elements, having 3-DOF at each node, which are used for meshing the 3D model. A higher-order 3D element, having fine mesh, is used to model irregular shape components such as connecting rods. These fine-meshed 3D elements give more authentic results at critical locations. The 3D model of connecting rod meshed with 122,351 elements with elemental lengths varying from 6 mm to 2 mm in a step of 1 mm. For an elemental size of less than 4 mm, the variations in the magnitude of maximum stresses generated at the critical location become negligibly small. The meshed model of connecting rods with element sizes at different locations is shown in figure 2.

Figure 2. 3D meshed model of connecting rod

Boundary conditions in the FE model are based on the engine configuration. Figure 3 shows the boundary conditions applied in the FE model of the connecting rod. Boundary conditions change according to the direction of the load applied. At first, connecting rod is assumed to exert a tensile force, and corresponding stresses at each node are calculated; after that, the compressive stresses are determined by replacing the tensile force with a compressive force precisely of the same magnitude but opposite in direction. Figure 4 and Figure 5 show the defined loading conditions in the FE model of the connecting rod. In every phase of reversible cyclic loading, the Von-mises stresses are generated and locate the critical locations on the connecting rod, where the maximum stresses generated exceed the allowable limit.

To analyze the stresses on the connecting rod various boundary conditions were carried out on the connecting rod model. In the FEM analysis, the smaller end of the connecting rod is fixed, and the tensile and compressive load of 9500 N is applied on the big end.

Figure 3. Boundary Condition on Connecting Rod

Figure 4. Tensile Loading on Connecting Rod

Figure 5. Compressive Loading on Connecting Rod

The material properties of connecting rod used for linear elastic finite element analysis are obtained by (Tevatia *et al*., 2011; Lal *et al*., 2010), as listed in Table 1.

Table 1. Properties of connecting rod material (Tevatia *et al*., 2011; Lal *et al*., 2010)

In the present research work, it is obtained that the fillet radius at the big end of connecting rod experiences the highest Von Mises stress 3.1112e7 Pa; therefore, selected as a critical location, and the FEA underestimated by 7.32% maximum stress generated than FEA obtained by the (Tevatia *et al*., 2011). Therefore, the results obtained from the FEA of a connecting rod model can be assumed to be satisfactory and indicate the accuracy of the FE model used.

3. Shape Optimization of Connecting Rod:

The present work aims to optimize the shape of connecting rod to withstand high strength in terms of high tensile and compressive load. In this approach of shape optimization, the overall shape of the connecting is not changed; only the size is modified by varying some design parameters. Geometrical properties parameters are used as design variables in size optimization. In this approach, the design variables such as fillet radius, groove thickness, and groove length of connecting rod are varied.

In the process of optimizing the shape of the connecting rod, the various parameters such as the total length, radius, and thickness of the connecting rod are fixed. The parameters that are changed in the process of optimization process known as design variables. The present work following design parameters such as fillet radius, groove thickness, and groove length have been studied during the shape optimization.

4. Result and Discussion:

The finite element analysis investigates and analyzes the effects of their critical dimensions, such as fillet radius, groove thickness, and groove length of connecting rod, based on maximum Von-mises stresses generated at the critical location under reversible cyclic loading. The original model is analyzed by assuming the critical dimensions of connecting rod as fillet radius = 48 mm, groove thickness = 11.5 mm, and groove length = 165 mm. The Von-Mises stresses at the critical location have been presented, and the effects of design parameters are analyzed by taking the same masses of connecting rod.

4.1 Effect of Fillet Radius on Big End:

The overall decrease in stresses generated with an increase in fillet radius is due to a decrease in stress concentration at the big end fillet may be preferred. It is concluded that for both the loading tensile and compressive along an axis, a higher value of fillet radius 50 mm may be preferred at the optimum level. Increasing the fillet radius from 47 mm to 50 mm reduces the stresses generated by 15.07% compared to the original model of the connecting rod. Table 2 shows the magnitude of stress generated during the variation of the fillet radius of the big end of the connecting rod. Figure 6 shows the optimized connecting rod model at a fillet radius of 50 mm.

Table 2. Effect of big end fillet radius on the stress generated

Figure 6. Stress generated at fillet radius on connecting rod

4.2 Effect of Groove Depth (Thickness):

Another step towards shape optimization is the variation of the groove depth of connecting rod. The stresses at the critical location are obtained by assuming equal mass for all design parameters of the connecting rod. Table 3 shows the effect of groove depth on maximum stresses generated at the critical location for connecting the rod. Also, figure 7 shows the optimized model on which the magnitude of stress is generated at a groove depth of 10.0 mm on connecting rod. Therefore, this may be considered a safe value for both tensile and compressive loading. Moreover, the stress level at the critical location is also reduced by 6.31%, compared with the stresses generated during the optimization of a big end fillet radius of 50 mm.

Table 3. Effect of groove depth on the stress generated

Figure 7. Stress generated at groove depth on connecting rod

4.3 Effect of Groove Length:

Reducing the groove length is another applied optimization parameter performed on the connecting rod. The increase in stresses is due to higher stress concentration at this location, which depends not only on the big end fillet radius and the groove depth but also on the groove length, which is reduced in this optimization. Table 4 shows the magnitude of stress generated during the variation of the groove length of the connecting rod. Figure 8 shows the optimized connecting rod model at a groove length of 135 mm. Moreover, the stress level at the critical location is also reduced by 7.55 % compared to the stresses generated during the optimization of groove depth 10.0 mm.

Table 4. Effect of groove length on the stress generated

Figure 8. Stress generated at groove length on connecting rod

4.4 Comparison of Optimized and Original Connecting Rod:

Local geometry optimizations are applied separately to different design parameters of connecting rods based on dynamic loading and stress analysis results. Table 5 shows a comparison of optimized design parameters with the original connecting rod through their reducing the Von-Mises stresses at critical locations. Since the stresses are higher in the fillet area due to higher stress concentration, the first local optimization is increasing the fillet radius from 47 mm to 50 mm. and after that variation of groove depth (thickness) from 11.5 mm to 10.0 mm. Reducing the groove length from 165 to 135 mm is considered as next step of geometry optimization. The Von Mises stresses generated are reduced by optimizing the connecting rod.

Table 5. Comparison of optimized parameters with original connecting rod

Table 6 compares Von-Mises stresses of the optimized connecting rod with an original connecting rod of C-70 Alloy Steel. This optimization reduces the stresses reduced for reverse cyclic loading from 3.1112e7 Pa to 2.2884e7 Pa. Therefore, the result of the geometry optimization process is 26.44% Von Mises stress reduction compared with the original connecting rod at a critical location.

Table 6. Comparison of stresses in the optimized and original connecting rod at a critical location

Figure 9. Optimized model of a connecting rod

When the thickness of connecting rod at this critical location becomes less than the thickness of the remaining circular part of the big end, the connecting rod's section becomes weak. This fact leads drastic increase in the stress generated at a critical point. Figure 9 shows the finally optimized model of the connecting rod. This model of the connecting rod will have more durability. This can also be employed in some more stressful conditions.

5. Conclusions:

The finite element analysis is a very effective way to identify the stress distribution and make it easy to simulate the realistic loading conditions under complicated loading conditions on the connecting rod and locate the critical sections. The effect of critical dimensions, such as fillet radius, groove thickness, and groove length of the connecting rod was investigated based on maximum stresses generated at the critical location under the fully reversible cyclic loading. The following conclusions are drawn from the analysis:

1) The area near the fillet of the big end of the connecting rod is found to be a Critical (i.e., failure) location because it experiences the highest Von Mises stresses, which result in high-stress concentration factors.

2) Comparative performance on critical dimensions of the connecting rod against the fully reversible cyclic loading are:

a) With an increase in fillet radius, the maximum stresses generated decrease due to a decrease in stress concentration near the big end fillet radius on connecting rod. Increasing the fillet radius from 47 mm to 50 mm reduces the stresses generated by 15.07 % compared to the original model of the connecting rod.

b) With decreasing groove depth (thickness) on connecting rod from 11.5 mm to 10.0 mm, the stress level at the critical location is also reduced by 6.31 %, compared with the stresses generated during the optimization of the big end fillet radius of 50 mm.

c) By reducing the groove length from 165 mm to 135 mm, the magnitude of stress generated on the critical location of connecting rod is also reduced by 7.55 %, compared with the stresses generated during the optimization of groove depth 10.0 mm.

Therefore, the optimized connecting rod model predicts 26.44 % lower Von Mises stresses compared to the initial design at a critical location. This will provide more strength, and that will increase the longevity of the connecting rod.

References:

Agrawal, A. P., Ali, S., & Rathore, S. (2022). Finite element stress analysis for shape optimization of spur gear using ANSYS. Materials Today: Proceedings, 64, 1147–1152. https://doi.org/10.1016/j.matpr.2022.03.404

Agrawal, A. P., & Srivastava, S. K. (2012). Finite element fatigue analysis for shape optimization of crankshaft. ISST Journal of Mechanical Engineering, 3(2), 1-6. Retrieved from https://www.researchgate.net/publication/354100264_Finite_Element_Fatigue_Analysis_for_Shape_Optimization_of_Crankshaft

Agrawal, A. P., & Srivastava, S. K. (2012). Fatigue life prediction of crankshaft based on strain life theories. International Journal of Engineering Research & Technology, 1(8), 1-5. Retrieved from https://www.ijert.org/fatigue-life-prediction-of-crankshaft-based-on-strain-life-theories

Anusha, B., & Reddy, C. V. B. (2013). Modeling and Analysis of two wheeler connecting rod by using Ansys. IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE), 6(5), 83-87. Retrieved from https://www.academia.edu/download/32110227/K0658387.pdf

Bharti, Y. K., Singh, V., & Hussain, A. (2013). Stress analysis and optimization of connecting rod using finite element analysis. International Journal of Scientific & Engineering Research, 4(6), 1796-1803. Retrieved from https://www.ijser.org/ResearchPaperPublishing_June2013_Page4.aspx

Desai, F., Jagtap, K. K., & Deshpande, A. (2014). Numerical and experimental analysis of connecting rod. International Journal of Emerging Engineering Research and Technology, 2(4), 242-249. Retrieved from http://www.ijeert.org/pdf/v2-i4/31.pdf

Kumar, A. P. (2015). Design & analysis of connecting rod by composite material. IJRDO-Journal of Mechanical and Civil Engineering, 1(7), 1-5. Retrieved from http://www.ijrdo.org/index.php/mce/article/view/1084

Kumar, P. S., & Kumar, K. (2015). Stress analysis and shape optimization of connecting rod using different materials. REST J. Emerg. Trends Model. Manuf., 1(2), 20-28. Retrieved from https://secureservercdn.net/50.62.90.29/d8a.8cf.myftpupload.com/wp-content/uploads/2016/02/Stress-Analysis-and-Shape-Optimization-of-Connecting-Rod-using-Different-Materials.pdf

Lal, S. B., Tevatia, A., & Srivastava, S. K. (2010). Fatigue analysis of connecting rod using ansys code, International Journal of Mechanics and Solids, 5(2), 143-150. Retrieved from http://www.ripublication.com/ijms/ijmsv5n2_6.pdf

Parkash, O., Gupta, V., & Mittal, V. (2013). Optimizing the design of connecting rod under static and fatigue loading. International Journal of Research in Management, Science & Technology, 1(1), 39-43. Retrieved from http://www.academia.edu/34940934/Optimizing_the_Design_of_Connecting_Rod_under_Static_and_Fatigue_Loading

Pathade, V. C., & Ingole, D. S. (2013). Stress analysis of IC engine connecting rod by FEM and photoelasticity. IOSR Journal of Mechanical and Civil Engineering, 6(1), 117-125. doi:10.9790/1684-061117125

Roy, B. K. (2012). Design Analysis and optimization of various parameters of connecting rod using CAE softwares. International Journal of New Innovations in Engineering and Technology, 1(1), 52-63. Retrieved from http://www.ijniet.org/wp-content/uploads/2013/07/9.pdf

Saxena, S., & Ambikesh, R. K. (2021). Design and finite element analysis of connecting rod of different materials. In AIP Conference Proceedings (Vol. 2341, No. 1, p. 020034). AIP Publishing LLC. https://doi.org/10.1063/5.0049989

Shanmugasundar, G., Dharanidharan, M., Vishwa, D., & Kumar, A. S. (2021). Design, analysis and topology optimization of connecting rod. Materials Today: Proceedings, 46, 3430-3438. https://doi.org/10.1016/j.matpr.2020.11.778

Shenoy, P. S., & Fatemi, A. (2006). Dynamic analysis of loads and stresses in connecting rods. Proceedings of the Institution of Mechanical Engineers, Part C: Journal of Mechanical Engineering Science, 220(5), 615-624. https://doi.org/10.1243/09544062JMES1

Tevatia, A., Lal, S. B., & Srivastava, S. K. (2011). Finite element fatigue analysis of connecting rods of different cross-sections. International Journal of Mechanics and Solids, 6(1), 45-53. Retrieved from http://www.ripublication.com/ijms/ijmsv6n1_5.pdf

Tiwari, A., Tiwari, J. K., & Chandrakar, S. K. (2014). Fatigue, analysis of connecting rod using finite element analysis to explore weight and cost reduction opportunities for a production of forged steel connecting rod. International Journal of Advanced Mechanical Engineering, 4(7), 782-802. Retrieved from https://www.ripublication.com/ijame-spl/ijamev4n7spl_09.pdf

A FIGURE FILE

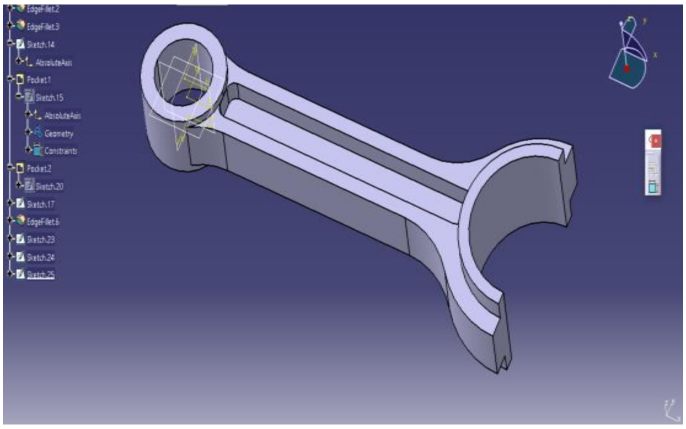

Figure 1. 3D solid model of Connecting Rod

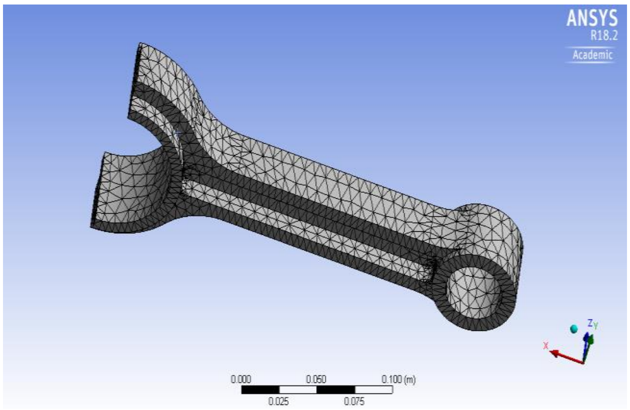

Figure 2. 3D meshed model of connecting rod

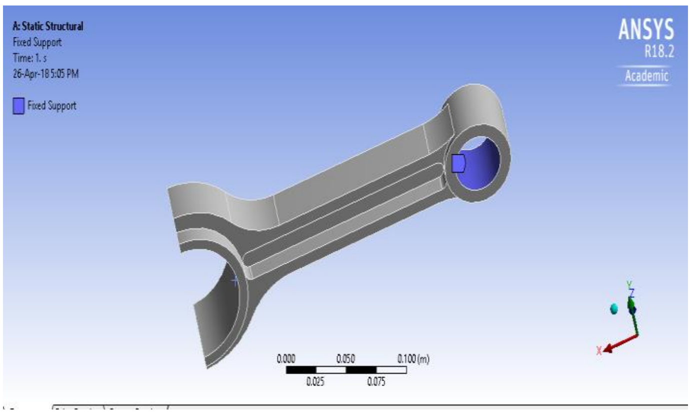

Figure 3. Boundary Condition on Connecting Rod

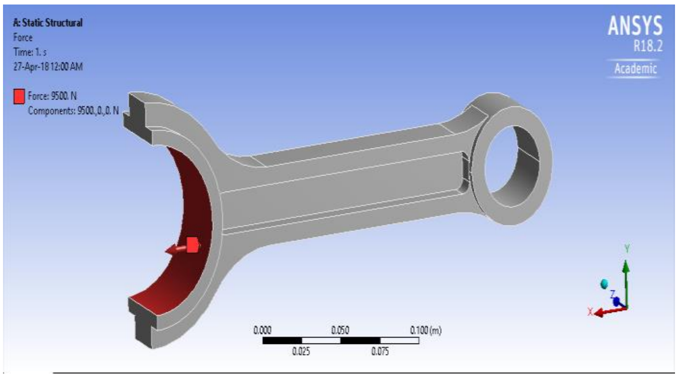

Figure 4. Tensile Loading on Connecting Rod

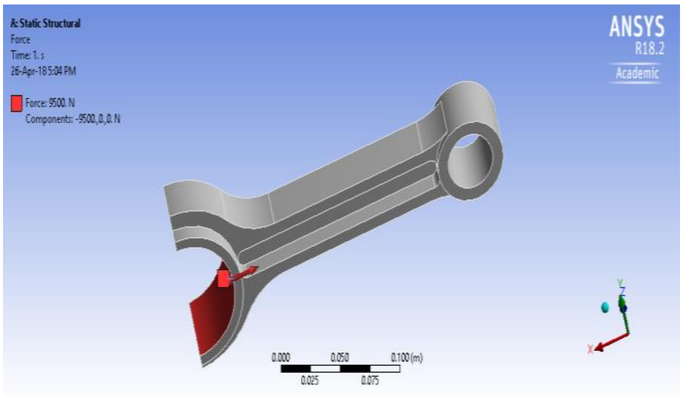

Figure 5. Compressive Loading on Connecting Rod

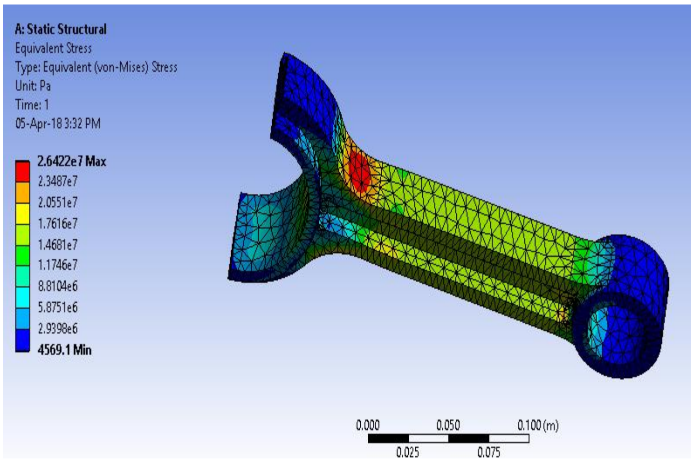

Figure 6. Stress generated at fillet radius on connecting rod

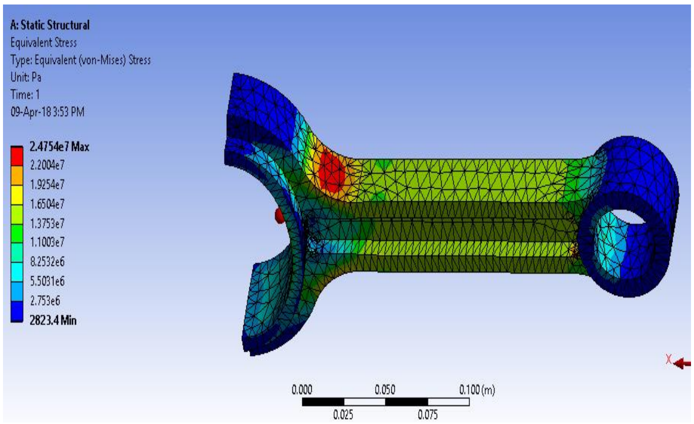

Figure 7. Stress generated at groove depth on connecting rod

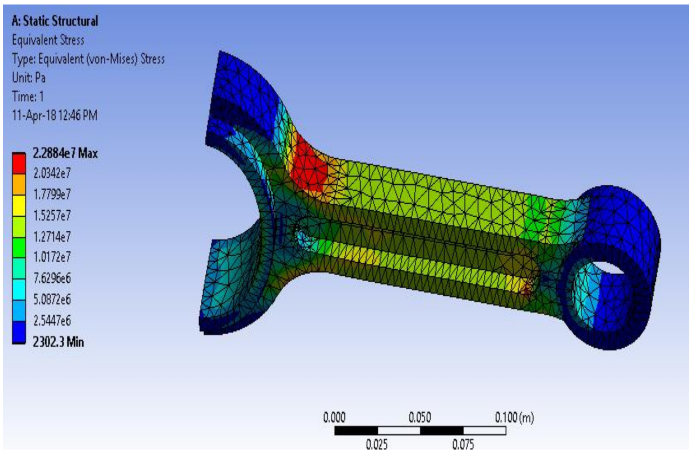

Figure 8. Stress generated at groove length on connecting rod

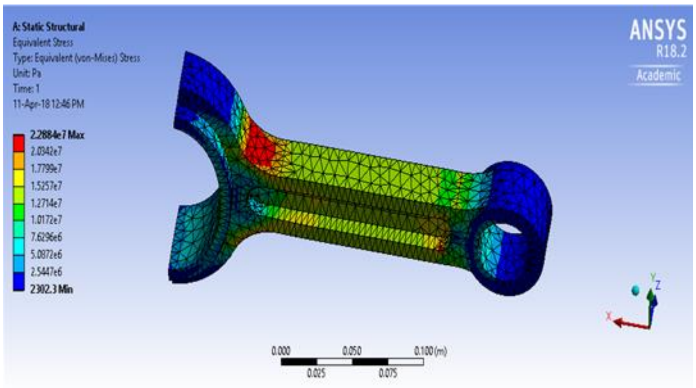

Figure 9. Optimized model of a connecting rod

A TABLE FILE

Material Property	C-70 Alloy Steel
Tensile strength	621 MPa
Yield strength	483 MPa
Young's modulus	207 GPa
Density	7700 Kg/m^3
Poison ratio	0.30

Table 1. Properties of connecting rod material (Tevatia *et al*., 2011; Lal *et al*., 2010)

Fillet Radius (mm)	Max. Stress (Pa)	Min. Stress (Pa)	Status
47	3.6679e7	2199.8	
48	3.1112e7	3710.8	Original
49	2.6715e7	4267.8	
50	2.6422e7	4569.1	Optimized

Table 2. Effect of big end fillet radius on stress generated

Groove Depth (mm)	Max. Stress (Pa)	Min. Stress (Pa)	Status
10.0	2.4754e7	2823.4	Optimized
10.5	2.4985e7	2803.7	
11.0	2.5723e7	4362.5	
11.5	2.6422e7	4569.1	Original

Table 3. Effect of groove depth on stress generated

Groove Length (mm)	Max. Stress (Pa)	Min. Stress (Pa)	Status
165.00	2.4754e7	2823.4	Original
155.00	2.3022e7	2685.9	
145.00	2.2978e7	2414.4	
135.00	2.2884e7	2302.3	Optimized

Table 4. Effect of groove length on stress generated

Sr. No.	Parameters	Original Connecting Rod (mm)	Optimized Connecting Rod (mm)
1.	Fillet Radius	48	50
2.	Groove Depth	11.5	10.0
3.	Groove Length	165	135

Table 5. Comparison of optimized parameters with original connecting rod

Stresses generated in original connecting rod		Stresses generated in optimized connecting rod		Percentage reduction in stresses generated
Max. Stress (Pa)	Min. Stress (Pa)	Max. Stress (Pa)	Min. Stress (Pa)	
3.1112e7	3710.8	2.2884e7	2302.3	26.44

Table 6. Comparison of stresses in optimized and original connecting rod at critical location

Detailed Response to Reviewers

Ref.: Ms. No. SJST-D-21-00434
Article Title: **"Investigation of Stresses at Critical Location for Shape Optimization of Connecting Rod using Finite Element Analysis"**

To
The Editor,
Songklanakarin Journal of Science and Technology

Dear Sir,
Thank you for the valuable suggestions that are been provided for the improvement of the article. The comments are addressed in the modified manuscript and the same is appended below in this letter for your knowledge.

***Please find the detailed response to Reviewer #2:**

Reviewer #2: The author has performed the Investigation of Stresses at Critical Location for Shape Optimization of Connecting Rod using Finite Element Analysis. This paper is presenting with proper FEM results but the following are suggestions and comments which need to be incorporated before publishing it.

Comment 1: Modify the abstract with concise numerical results for better understanding.

Response: - Thank you for the valuable feedback, as per the suggestion, The abstract has been revised and added numerical results.

With an increase in fillet radius, and decreasing groove depth and length, the maximum stresses generated were 15.07%, 6.31%, and 7.55% reduced, respectively, due to a decrease in stress concentration. Finally, the optimized model has reduced maximum stress at the bigger end of the connecting rod up to 26.44%.

Comment 2: English errors in the manuscript should be carefully modified.

Response: - The entire manuscript is thoroughly checked for English errors and the same is reflected in the modified manuscript according to the reviewers' suggestion**.**

Comment 3: Some latest papers must be included in the literature.

Response: - We appreciate the reviewer's detailed and thoughtful comments on this manuscript. As suggested by the reviewer, following three latest articles were included in literature and addended in references.

1. Agrawal, A. P., Ali, S., & Rathore, S. (2022). Finite element stress analysis for shape optimization of spur gear using ANSYS. Materials Today: Proceedings, 64, 1147–1152. https://doi.org/10.1016/j.matpr.2022.03.404
2. Shanmugasundar, G., Dharanidharan, M., Vishwa, D., & Kumar, A. S. (2021). Design, analysis and topology optimization of connecting rod. Materials Today: Proceedings, 46, 3430-3438. https://doi.org/10.1016/j.matpr.2020.11.778
3. Saxena, S., & Ambikesh, R. K. (2021, May). Design and finite element analysis of connecting rod of different materials. In AIP Conference Proceedings (Vol. 2341, No. 1, p. 020034). AIP Publishing LLC. https://doi.org/10.1063/5.0049989

Comment 4: The introduction must be concise. The well-known historical events can be skipped.

Response: - Thank you for the valuable feedback, comments has been incorporated in introduction part and removed general historical events from manuscript.

Comment 5: How the author(s) identified fillet radius as the critical location. Justify this in the manuscript for better understanding.

Response: - Since fillet areas of connecting rod have maximum stress generated during power transmission from piston as obtained through FEM analysis, therefore these fillet radius locations/points act as a critical location. The details explanations were discussed under section 2.3 in the manuscript.

Comment 6: Dimension of 3D modelled I-section connection rod need to include (if possible) in the manuscript for better understanding.

Response: - A 3D model of I-section connecting rod presented in figure 1 and dimension were included and details were discussed under section 2.2 in the manuscript.

The basic dimensions of the 3D modelled I-section are 58 mm thickness connecting rod of 2500 mm length between big and small end centers.

Comment 7: Clearly define the applied boundary conditions for analysis in this study.

Response: - In order to analyse the stresses on the connecting rod, the boundary conditions carried out on the model of connecting rod. In the FEM analysis smaller end of the connecting rod are fixed and a tensile and compressive cyclic load of 9500 N is applied on the big end. The details explanations were discussed under section 2.3 in the manuscript.

To analyse the stresses on the connecting rod various boundary conditions were carried out on the connecting rod model. In the FEM analysis, the smaller end of the connecting rod is fixed, and the tensile and compressive load of 9500 N is applied on the big end.

Comment 8: The numerical value of Von Mises stresses 3.1112e7 in section 2.3 Finite Element Analysis, should with the proper unit (Pa or MPa).

Response: - Thank you for your feedback. I have corrected it. The numerical value of Von Mises stresses is 3.1112e7 Pa.

Comment 9: The author needs to be discussed how to validate the results of the model with experiments or other models.

Response: - The model of connecting rod were validated through the FEA results were obtained at critical location (fillet radius) which underestimate by 7.32% maximum stress generated than FEA obtained by the (Tevatia *et al.*, 2011). Therefore, the results obtained from FEA of model of connecting rod can be assumed to be satisfactory and it indicates the accuracy of FE model used. The details explanations were discussed under section 2.3 in the manuscript.

***Please find the detailed response to Reviewer #3:**

Reviewer #3: In this paper, the investigation of stresses at the critical location for shape optimized of connecting rod by varying fillet radius and other parameters using FEM. The article is nicely written however some observations may be incorporated to improve the article. The comments and some suggestion as below:

Comment 1: The manuscript needs to be revised especially in some sentence formation and grammar. There are some corrections required concerning Grammarly.

Response: - The entire manuscript is thoroughly checked for grammar mistakes and the same is reflected in the modified manuscript according to the reviewers' suggestion**.**

Comment 2: The sequence of references in the literature review section.

Response: - Thank you for your feedback. I have corrected it.

Comment 3: Since the accuracy of the numerical results of the simulation depends upon the meshing quality. The authors need to clarify the connecting rod break into how many numbers of elements are in the manuscript.

Response: - The model of connecting rod is messed and broken into 122,351 elements. The details explanations were discussed under section 2.3 in the manuscript.

The 3D model of connecting rod meshed with 122,351 elements with elemental lengths varying from 6 mm to 2 mm in a step of 1 mm. For an elemental size of less than 4 mm, the variations in the magnitude of maximum stresses generated at the critical location become negligibly small.

Comment 4: Authors need to be specified the proper loading with constraints that are applied for FE analysis.

Response: - In the FEM analysis smaller end of the connecting rod are fixed and a tensile and compressive cyclic load of 9500 N is applied on the big end. In process of optimization of shape of connecting rod the various following parameters are fixed (constraints) such as the total length, radius, and thickness of connecting rod. The details explanations were discussed under section 2.3 and 3 in the manuscript.

Comment 5: In this study, why connecting rod shape is optimized through the von-Mises stress is on fillet radius?

Response: - Since fillet areas of connecting rod have maximum stress generated during power transmission from piston as obtained through FEM analysis, therefor these fillet radius locations/points act as a critical location and shape of connecting rod were optimized through the von-Mises stress is on fillet radius. The details explanations were discussed under section 2.2 and 2.3 in the manuscript.

Fillets behave as stress raisers on the connecting rod surface; therefore, cracks may develop at their surface and grow inward due to combined cyclic bending and torsion loads.

Comment 6: Description of figs. maybe given in the text also.

Response: - Thank you for your feedback. I have included the suitable description of all figs. in the manuscript.

Comment 7: Starting paragraph of the conclusion should be rewritten with findings.

Response: - I have updated the conclusions as per your suggestion, focusing on the finding i.e., maximum stress reduction for critical dimensions at the critical location for higher strength.

The finite element analysis is a very effective way to identify the stress distribution and make it easy to simulate the realistic loading conditions under complicated loading conditions on the connecting rod and locate the critical sections. The effect of critical dimensions, such as fillet radius, groove thickness, and groove length of the connecting rod was investigated based on maximum stresses generated at the critical location under the fully reversible cyclic loading.

***Please find the detailed response to Reviewer #4:**

Reviewer #4: Overall, this is a clear, concise, and well-written manuscript. The introduction is relevant to the problem and its possible solution. Overall, the results are clear and compelling with the comparison of FEM results with other models. Some miner-specific comments need to be incorporated.

Comment 1: Need to properly define abbreviations of some notation.

Response: - Thank you for the valuable feedback on the manuscript as per the suggestion, the abbreviations were properly mentioned with notation in the manuscript.

Comment 2: References must be cited and arranged in sequence.

Response: - Thank you for your feedback. I have corrected it.

Comment 3: Please clearly mention what is meant by shape optimization, and what factors are taken into consideration for shape optimization.

Response: - In this shape optimization approach, the design variables (factors) such as fillet radius, groove thickness and groove length of connecting rod are varied and considered. The details explanations were discussed under section 3 in the manuscript.

The parameters that are changed in the process of optimization process known as design variables. The present work following design parameters such as fillet radius, groove thickness, and groove length have been studied during the shape optimization.

Comment 4: The author should follow uniformity in units throughout the manuscript. Follow any one standard for specifying units.

Response: - Thank you for your feedback. I have corrected it and used Pa.

Comment 5: Some location references need to be mentioned, i.e., from where data are taken.

Response: - Thank you for your feedback. I have incorporated comments and references were added in Table 1.

Table 1. Properties of connecting rod material (Tevatia *et al*., 2011; Lal *et al*., 2010)

Comment 6: The detailed mesh convergence analysis must be included in the paper to increase the knowledge on the subject.

Response: - Ten–node tetragonal elements, having 3-DOF at each node, are used for meshing the 3D model. A higher order 3D element, having fine mesh, is used to model irregular shape components such as connecting rod. The details explanations were discussed under section 2.3 in the manuscript.

***Please find the detailed response to Reviewer #5:**

Reviewer #5: This is an interesting study and the authors have collected own results in comparatively form through FEM analysis for shape optimization of connecting rod. The paper is generally well written and structured.

Comment 1: The manuscript needs to be revised, particularly in parts of the grammar and sentence structure.

Response: - Thank you for the valuable feedback on the manuscript as per the suggestion, the abbreviations were properly mentioned with notation in the manuscript.

Comment 2: The author should be questioned about how to use experiments or other models to verify the model's predictions.

Response: - The model of connecting rod were validated through the FEA results were obtained at critical location (fillet radius) which underestimate by 7.32% maximum stress generated than FEA obtained by the (Tevatia *et al.*, 2011). Therefore, the results obtained from FEA of model of connecting rod can be assumed to be satisfactory and it indicates the accuracy of FE model used. The details explanations were discussed under section 2.3 in the manuscript.

Comment 3: Clearly state the boundary conditions that were used in this study's analysis.

Response: - In order to analyze the stresses on the connecting rod, the boundary conditions carried out on the model of connecting rod. In the FEM analysis smaller end of the connecting rod are fixed and a tensile and compressive cyclic load of 9500 N is applied on the big end. The details explanations were discussed under section 2.3 in the manuscript.

Comment 4: The results should be discussed more rationally and in comparison, to other models.

Response: - The comparison of optimized design parameters with original connecting rod through their reducing the Von-Mises stresses at critical locations is presented in Table 5.

Comment 5: The introduction needs to be short.

Response: - Thank you for the valuable feedback, comments has been incorporated in introduction part. This introduction part written concise and general historical events removed from manuscript.

Comment 6: The connecting rod divide into how many elements are in the article needs to be made clear by the authors.

Response: - The model of connecting rod is messed and broken into 122,351 elements. The details explanations were discussed under section 2.3 in the manuscript.

The 3D model of connecting rod meshed with 122,351 elements with elemental lengths varying from 6 mm to 2 mm in a step of 1 mm. For an elemental size of less than 4 mm, the variations in the magnitude of maximum stresses generated at the critical location become negligibly small.

Comment 7: It is necessary to correctly define some notation's acronyms.

Response: - Thank you for the valuable feedback on the manuscript as per the suggestion, the abbreviations were properly mentioned with notation in the manuscript.

Comment 8: There should be some location references are missing.

Response: - Thank you for your feedback. I have corrected it.

Comment 9: The appropriate loading must be given by authors when constraints are used for FE analysis.

Response: - In the FEM analysis smaller end of the connecting rod are fixed and a tensile and compressive cyclic load of 9500 N is applied on the big end. In process of optimization of shape of connecting rod the various following parameters are fixed (constraints) such as the total length, radius, and thickness of connecting rod. The details explanations were discussed under section 2.3 and 3 in the manuscript.

Note: All the addressed comments are highlighted with red color in the revised manuscript.